DESIGN AND SIMULATION FOR NATURAL VENTILATION

Please complete the electronic course evaluation form via the URL http://tiny.cc/st188x

or the appended QR code.

Organised by: BCA ACADEMY

200 Braddell Road
Singapore 579700
Tel: 6248 9999
Website: http://www.bcaa.edu.sg
Email:bca_academy@bca.gov.sg

Important Notes to Course Participants

On Attendance/Examination/Assessment/Assignment

Please remember to sign on the attendance sheet daily and promptly upon arriving at the class. Course participants are **not allowed** to sign for lesson(s) not attended or for someone else. Participants are to monitor their attendance throughout the whole course.

Participants must achieve at least **75% class attendance for lessons (excluding exam attendance)** to qualify to sit for all the assessment(s). Participants who do not achieve at least 75% class attendance may be debarred from all the assessment(s). In this case, the participant will also not be allowed to sit for the supplementary assessment(s). The participant would be deemed to have failed the course and will not be issued the Certificate of Successful Completion.

Submission of assignment/project report (if any) will have to be made by the given deadline. Failure to do so, will result in incompetency/unsuccessful completion of the course.

Late submission of assignment will not be accepted.

BCA Academy

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

Course Instructor:
Mr Owen Wee, Surbana International Consultants Pte.Ltd.
Dr. POH Hee Joo, 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** • CFD as a potential tool to optimize and deliver green buildings, and case studies • Natural ventilation requirements in Green Mark criteria • Strategies At Estate Level To Achieve Good Natural Ventilation	**Dr. Poh Hee Joo**
10:30am – 11:00am : Tea break		
11:00am - 12:30pm	**Introduction to CFD in Green Building Application** • BCA guidelines for running a ventilation simulation • Challenges and Opportunities in using CFD as a tool for design optimization • 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**
1:15pm - 2:15pm : Lunch		
1:30pm – 3:00pm	**Introduction to CFD process** **Demo of CFD software** • Brief demo and software usage	**Dr. Poh Hee Joo**
3:00pm – 3:30pm : Tea break		
3:30pm – 5:30pm	**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**
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** • Simplifying architecture data • Creating water tight model • Creating model from primitive data • Preparing the model for meshing • Using BIM for CFD simulation	**Dr. Poh Hee Joo**
10:30am – 11:00am : Tea break		
11:00am - 12:30pm	**Meshing computational model** • Avoiding highly skewed geometry • Ensuring good quality mesh • Mesh densities • Different meshing topology **Numerical Settings** **Post processing of simulation results**	**Dr. Poh Hee Joo**
12:30pm - 1:30pm : Lunch		
1:30pm – 3:00pm	**Hands on with CFD software** • Tutorial for Practice 2 - Simulation of air flow at unit level (step by step guideline)	**Dr. Poh Hee Joo**
3:00pm - 3:30pm : Tea break		
3:30pm – 5:00pm	**Hands on with CFD software** • Tutorial for Practice 2 - Simulation of air flow at unit level (step by step guideline) (continued)	**Dr. Poh Hee Joo**
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**
9:45am – 10:30am	**CFD case studies** • **CFD Case Studies for Urban and Built Environment – Researcher Perspective**	**Dr. Poh Hee Joo**
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 air flow around building development and adjacent buildings (cont…)	**Dr. Poh Hee Joo**
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 air flow around building development and adjacent buildings (cont…)	**Dr. Poh Hee Joo**
3:00pm - 3:30am : Tea break		
3:30pm – 5:00pm	**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 air flow around building development and adjacent buildings (end)	**Dr. Poh Hee Joo**
	Closure • Feedback session	
End of Day 4		

References (selected):

1. Nielsen, P.V., Computational Fluid Dynamics in Ventilation Design, Brussels, Belgium: REHVA/Federation of European Heating and Air-conditioning Associations, 2007.
2. Tominagaa, Y., Mochidab, A., Yoshiec, R., Kataokad, H., Nozue, T,. Yoshikawaf, M., and Shirasawac, T., AIJ Guidelines for Practical Applications of CFD to Pedestrian Wind Environment Around Buildings, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 96, 1749 – 1761, 2008.
3. Casey, M. and Wintergerste, T., ERCOFTAC Special Interest Group on "Quality and Trust in Industrial CFD" Best Practice Guidelines, 2000.
4. Franke, J., Hellsten, A., Schlunzen, K.H. and Carissimo, B., Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment: COST ACTION 732, 2007.
5. Blocken,B. and Gualtieri,C., Ten Iterative Steps for Model Development and Evaluation Applied to Computational Fluid Dynamics for Environmental Fluid Mechanics, Environmental Modelling & Software, Vol. 33, 1 – 22, 2012.
6. Blocken B, Stathopoulos T, Carmeliet J. 2007. CFD simulation of the Atmospheric Boundary Layer – wall function problems. ***Atmospheric Environment 41(2): 238-252.***

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.

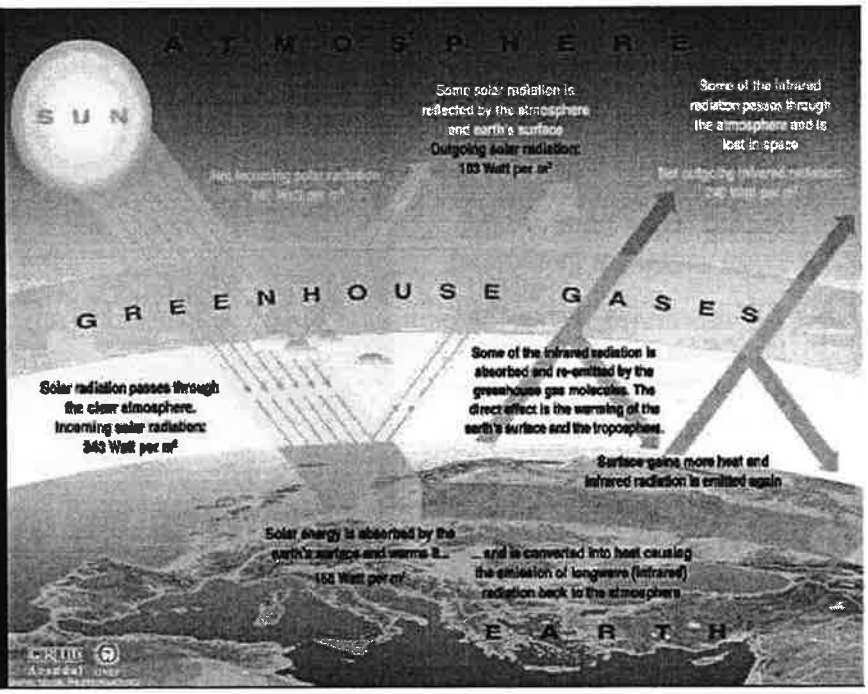
Source : Nottingham University : Dr Sergio A.

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

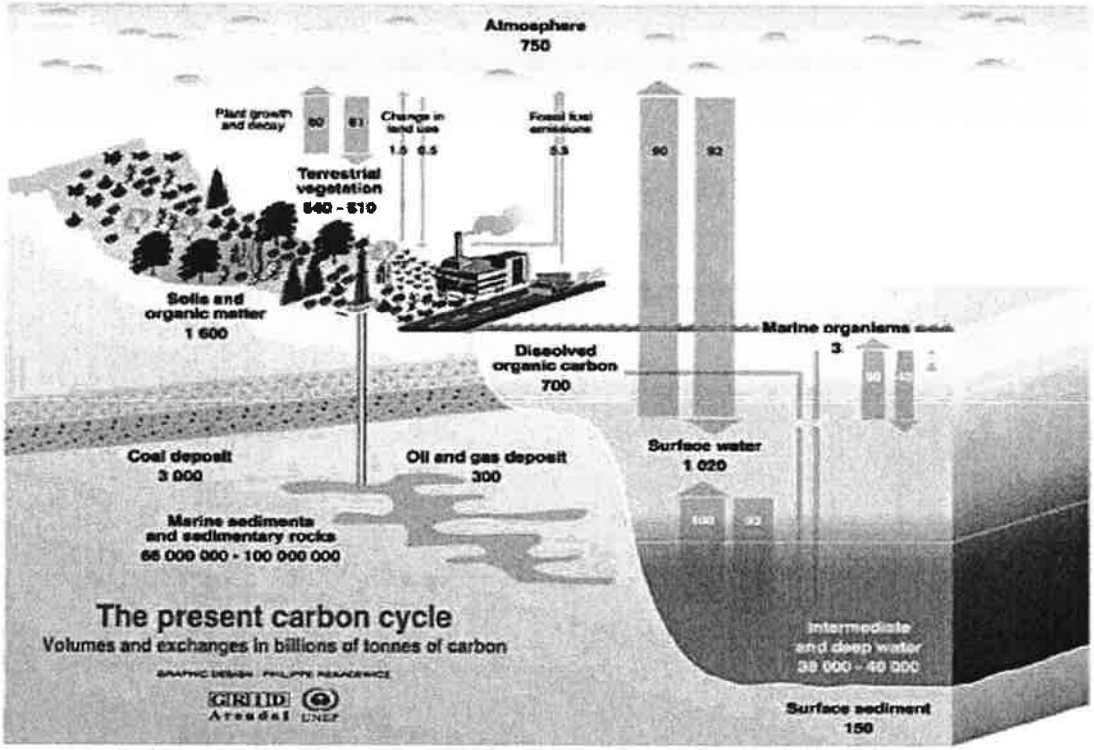
Without the natural greenhouse effect the Earth's surface would be about 30°C colder than the average +14°C we enjoy at present



The Greenhouse effect

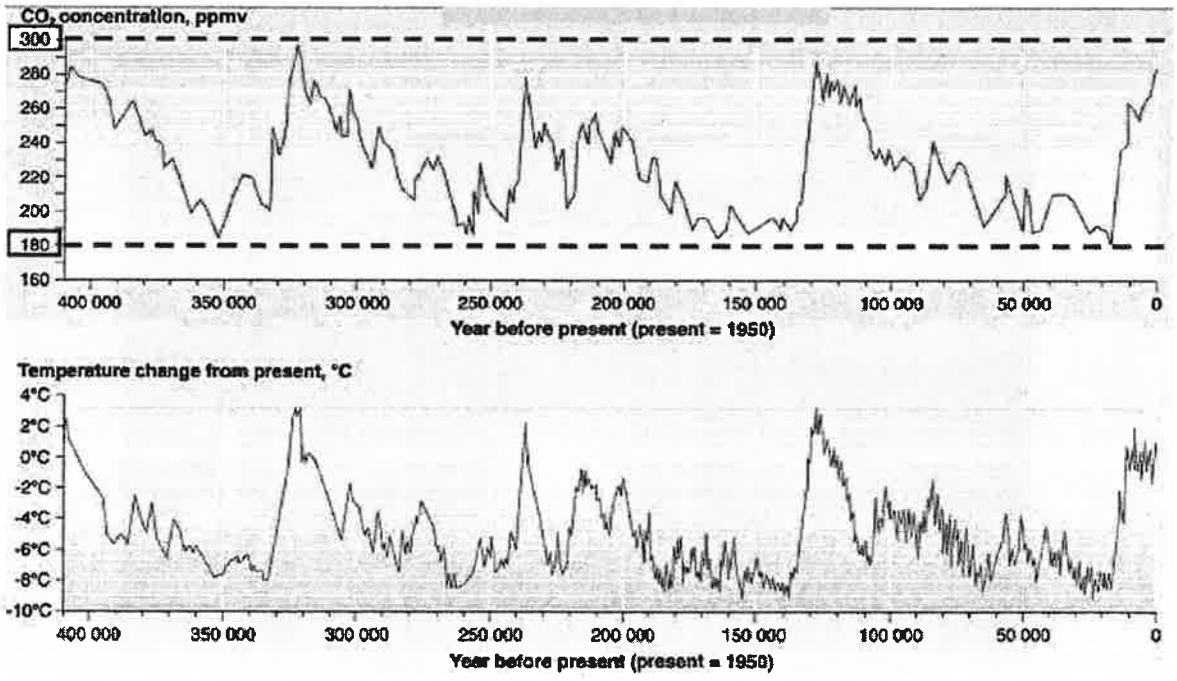
Source: IPCC, Climate Change 2007

Our Mother Earth



Sources: Center for climatic research, Institute for environmental studies, university of Wisconsin at Madison, Okanagan university college in Canada, Department of geography; World Watch, November-December 1998; Climate change 1995, The science of climate change, contribution of working group 1 to the second assessment report of the intergovernmental panel on climate change, UNEP and WMO, Cambridge press university, 1996.

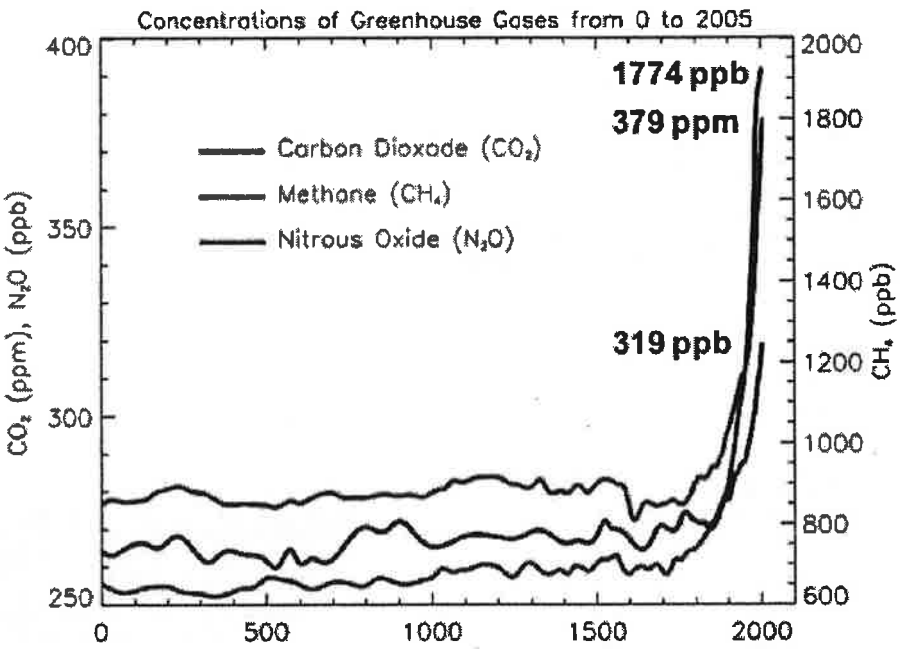
Our Mother Earth



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

Source: IPCC, Climate Change 2007

Our Mother Earth : Last 50 years



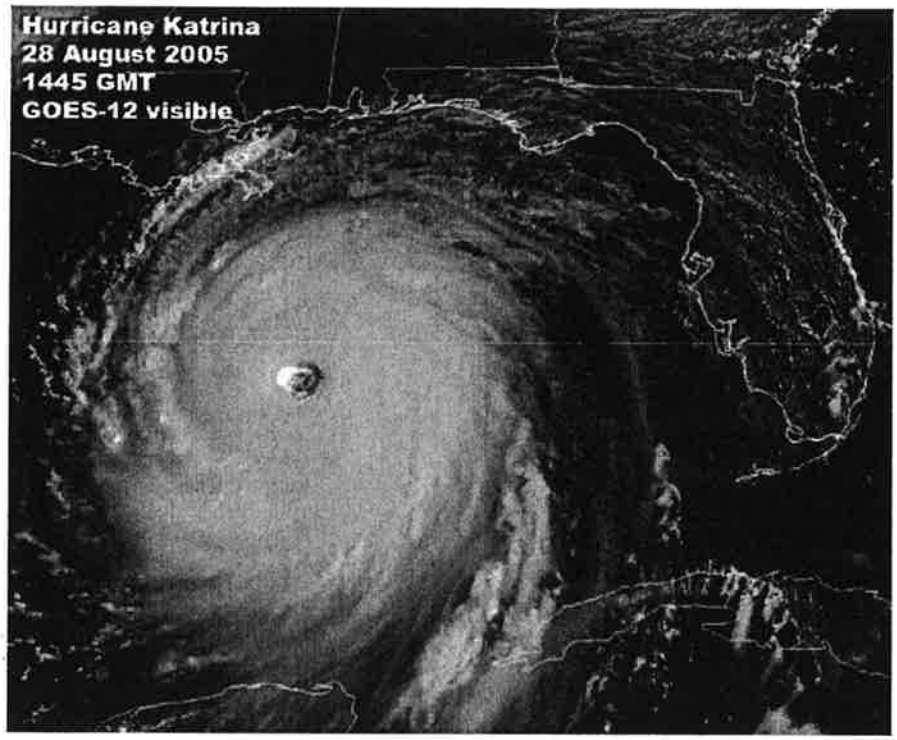
Atmospheric concentrations of long-lived GHG gases over the last 2,000 years

Source: IPCC, Climate Change 2007

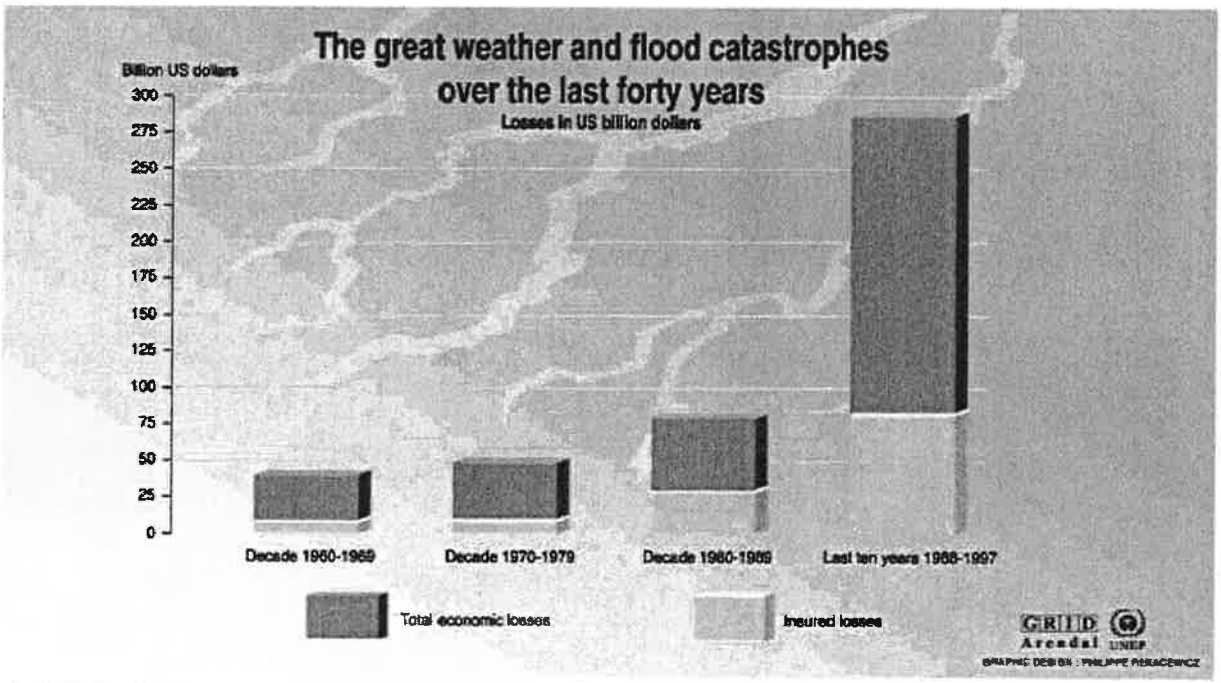
Our Mother Earth : Last 50 years



Our Mother Earth : Last 50 years



Our Mother Earth : Last 50 years

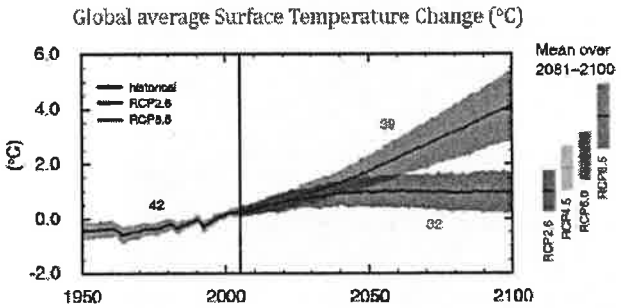


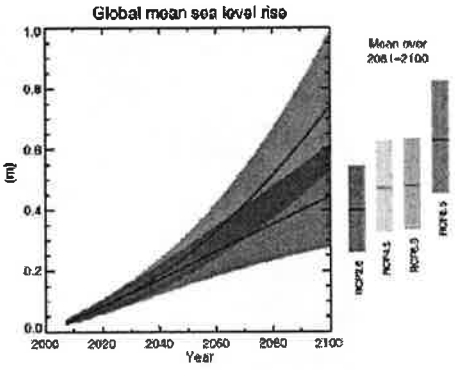
Source: Munich Re Group, 1998.

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

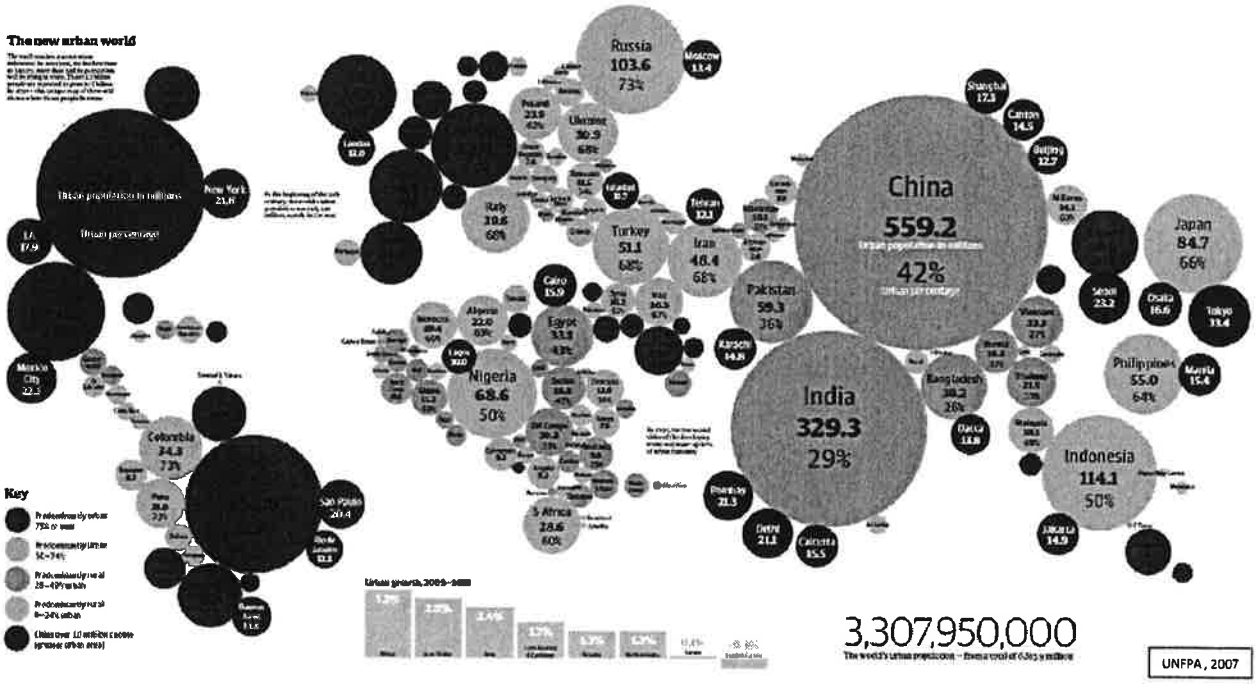
Conclusion

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.caIn 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	Recreation space 绿地与休闲	Slumps and pollution 平民区和污染	Resources 资源

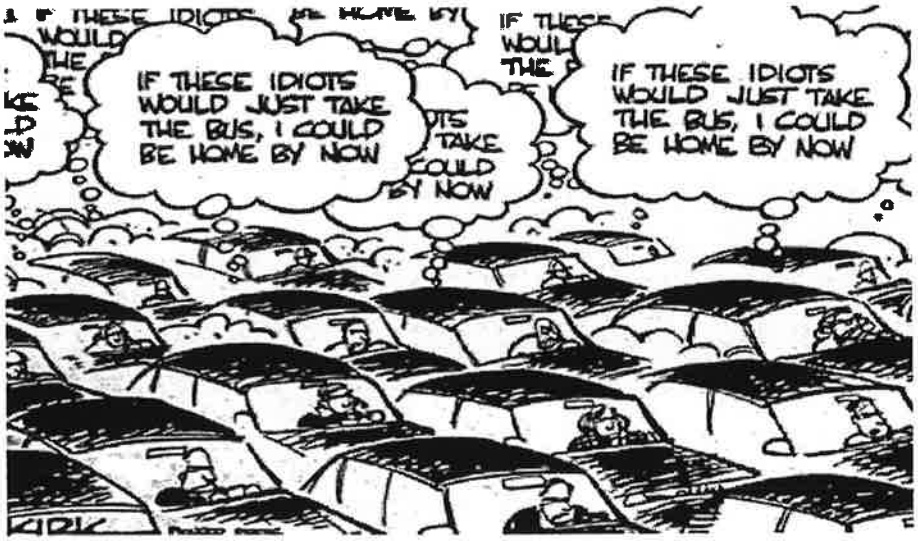


Our Mother Earth : Last 50 years

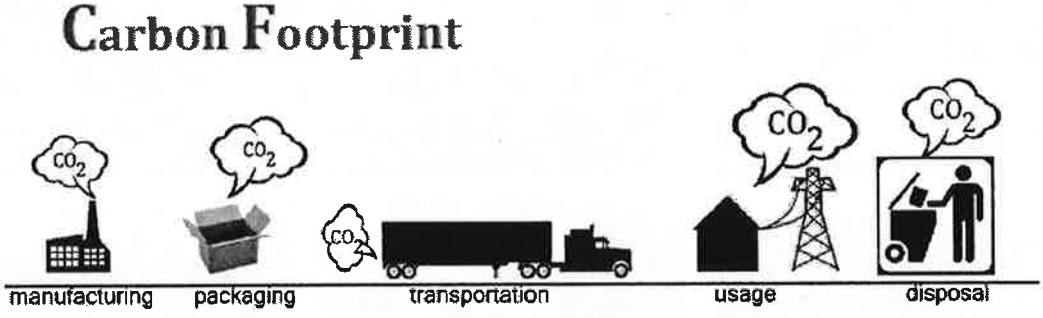


Life of high carbon foot print

Our Mother Earth : Last 50 years



Our Mother Earth : Last 50 years



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

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

Predicted Reserves of Fossil Fuels	
Oil	30 years
Natural Gas	50 years
Coal	190 years

Table Adapted from Vale, B&R (1991)

Our Mother Earth : Last 50 years

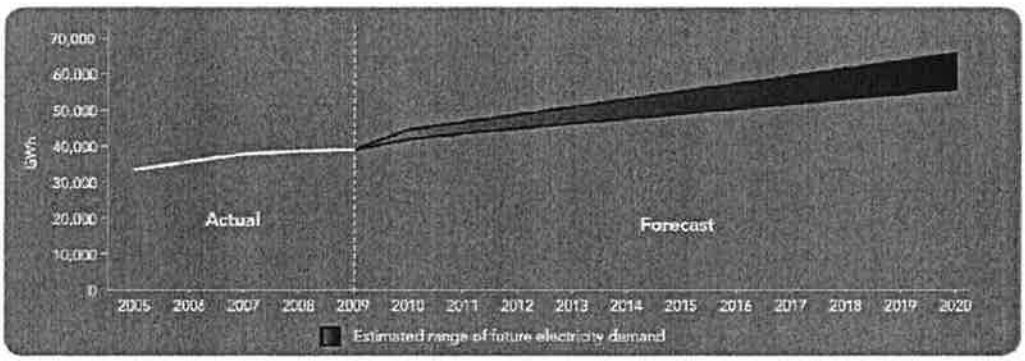
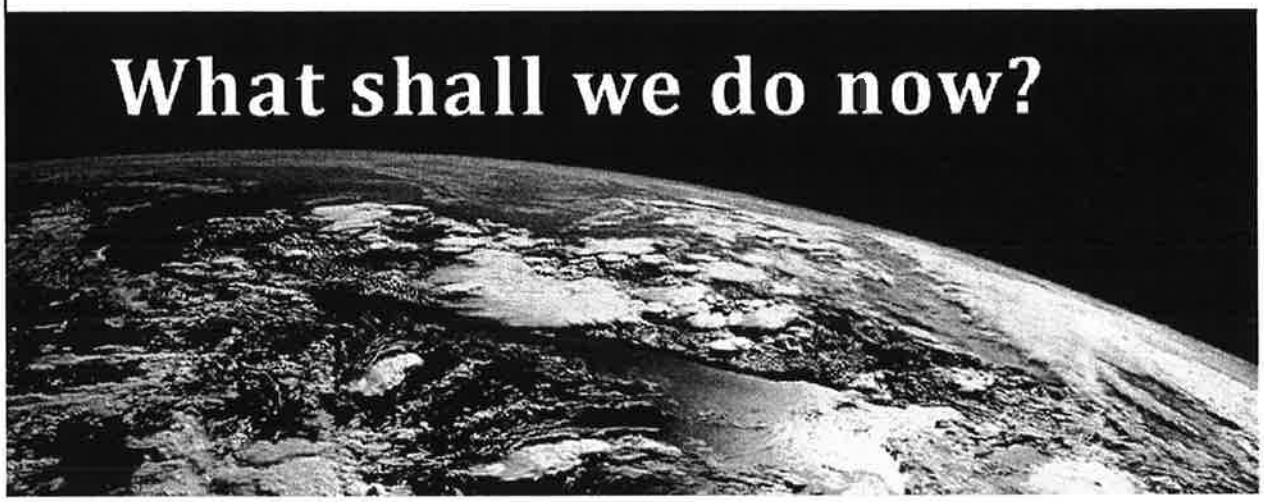


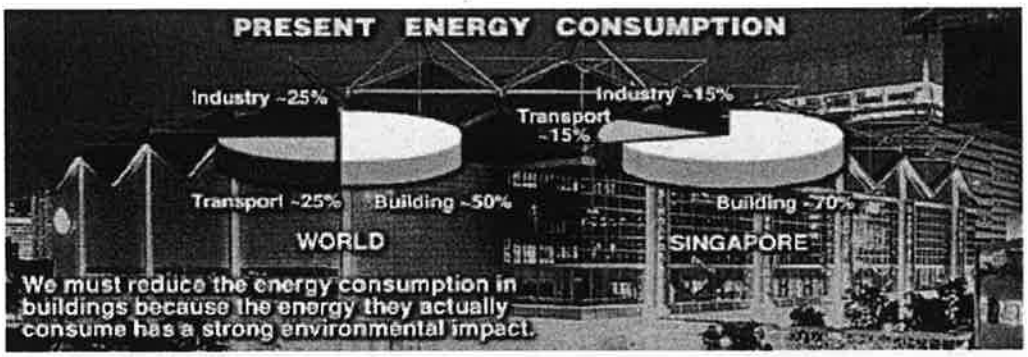
Fig 3.1- Estimated range of future electrical demand. Source:- EMA, 2010, pg 8

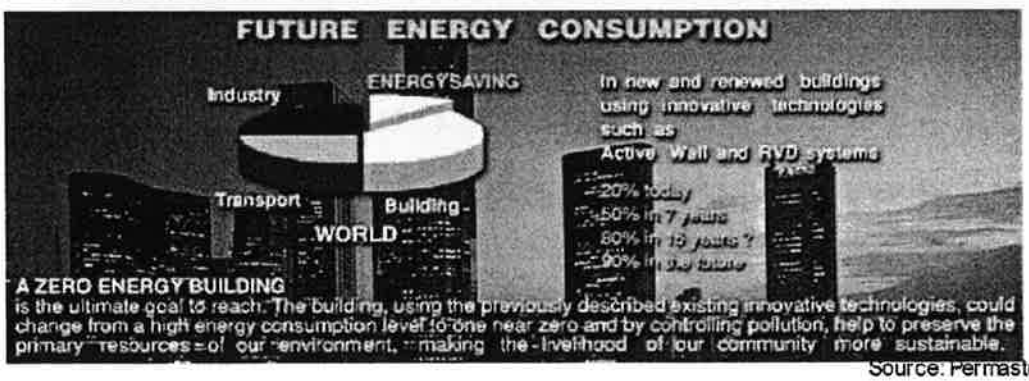
Estimated range of future electrical demand. Source:- EMA, 2010, pg 8

The current growth in global population and urbanization is creating intense pressure on our natural resource levels.



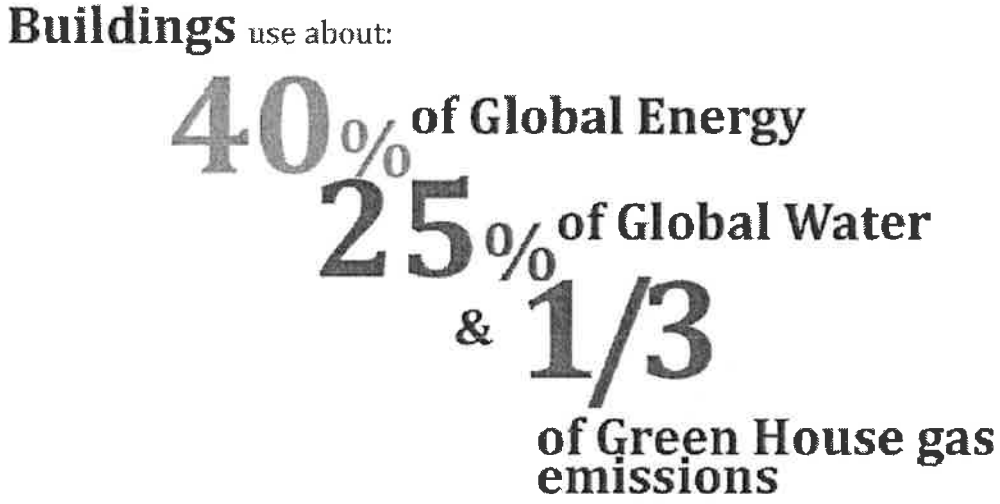
The Solution





Source: Permasteelisa, 2001

The Solution



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

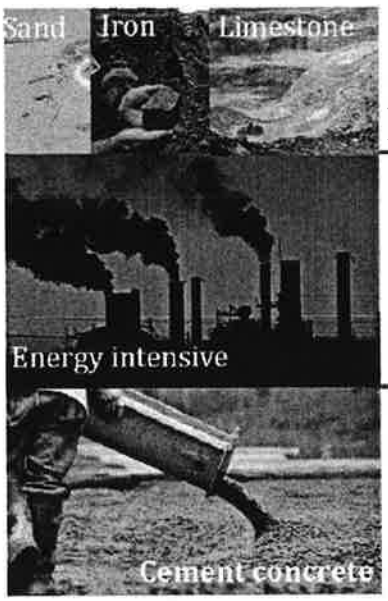
-China and India is unprecedented and estimated to represent half of the world's total market today. From a property owner perspective, resources, space utilisation, and long-term operational costs are increasingly factored in during planning. With buildings responsible for about half of all energy consumption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



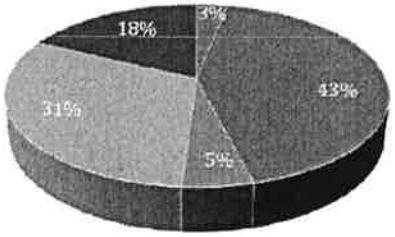
Embodied Energy is defined as the energy used during the entire life cycle of a product including the energy used for manufacturing, transporting, and disposing of the product.

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. 2006)

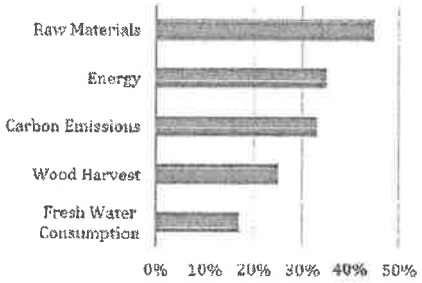
Electricity Consumption in Singapore (2005)



• Others ▪ Industry * Transport * Buildings ▪ Households

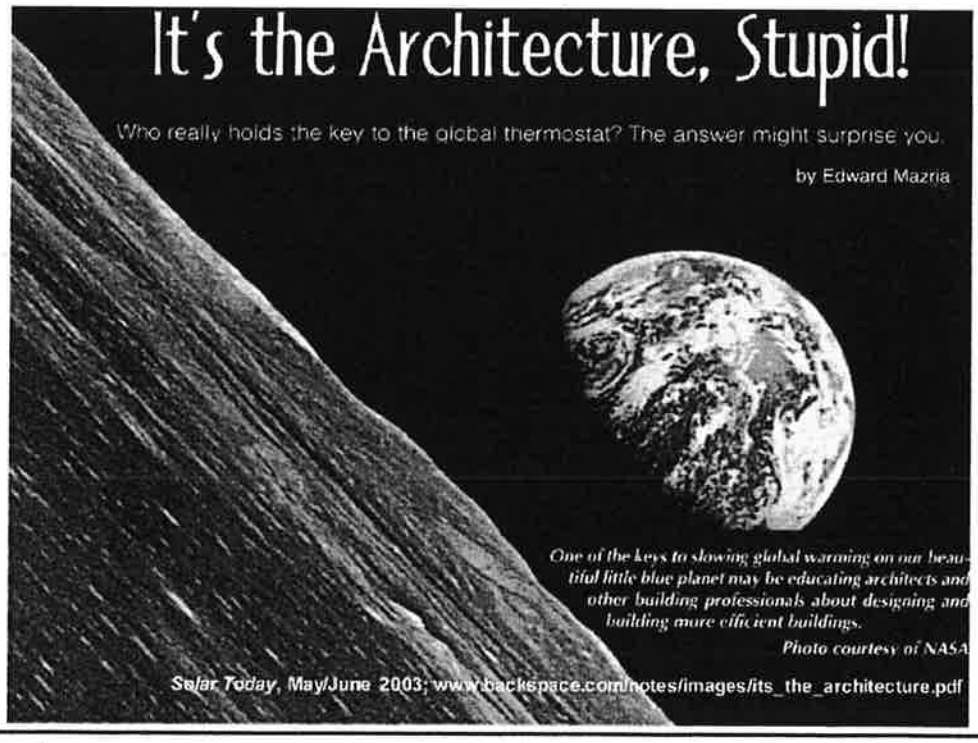
Global Building Consumption

(Kishnani, 2010)



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



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:

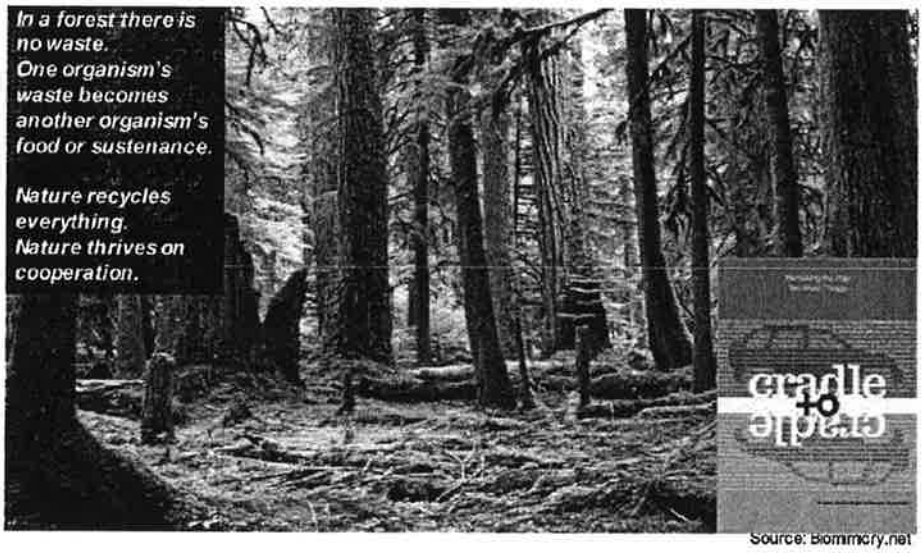
- *Responsible use of the worlds resources*
- *Working with the climate (do we need air conditioning, what's wrong with being warm???)*
- *Responsible material consumption.*

2. How we live well with each other

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

Learning from Mother Earth

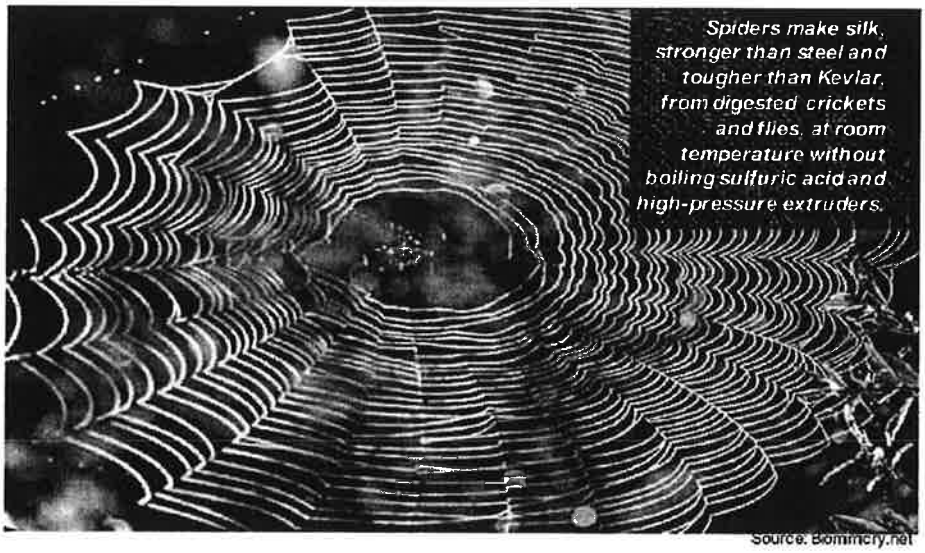
Recycling



Source: Biommicry.net

Learning from Mother Earth

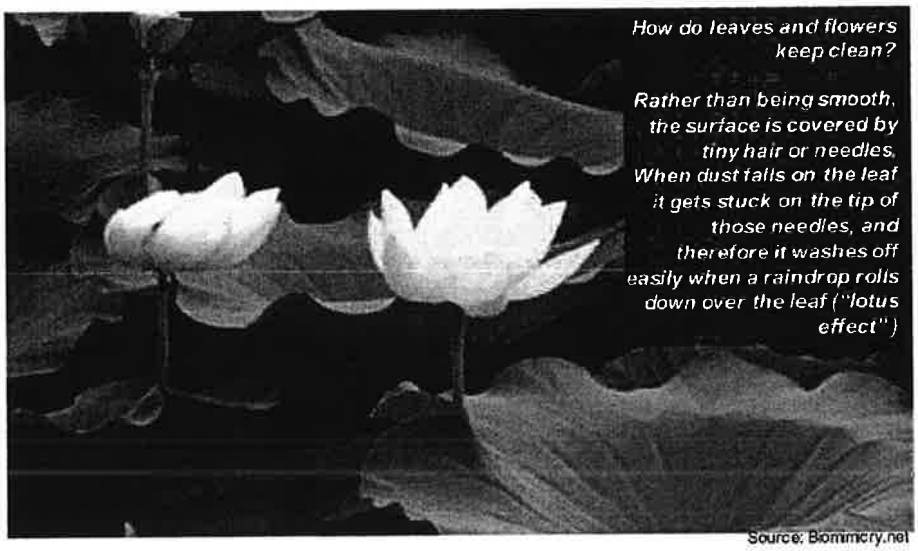
Harmful emissions



Source: Biommicry.net

Learning from Mother Earth

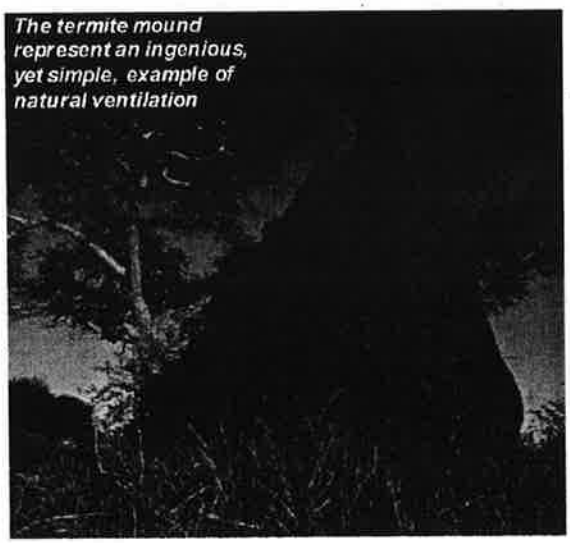
Water Efficiency

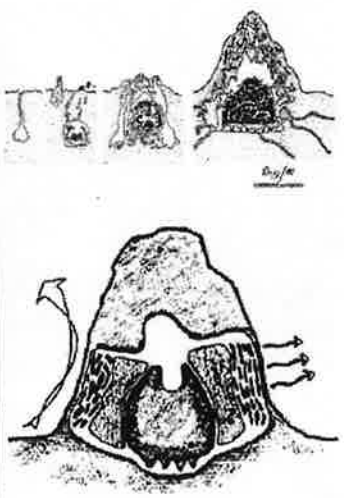


Source: Biommicry.net

Learning from Mother Earth

Natural ventilation





Source: Biommicry.net

Architecture is the Key .....

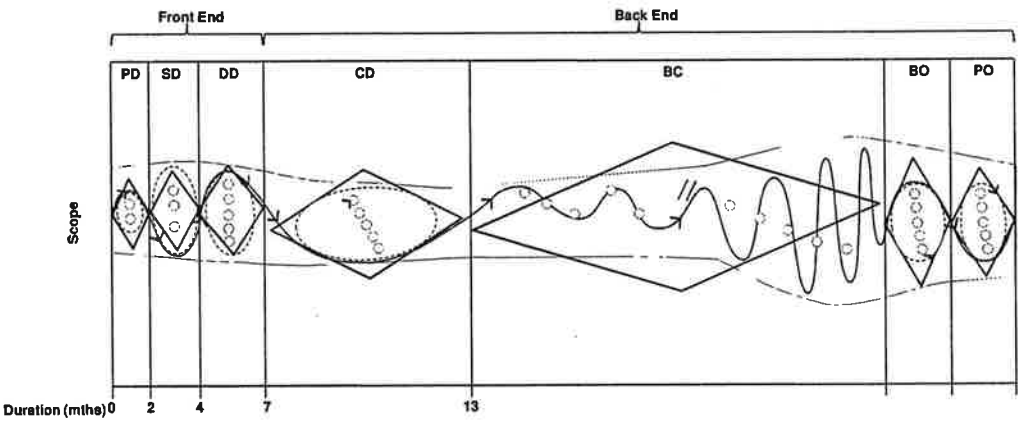
快速增长中。。。

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

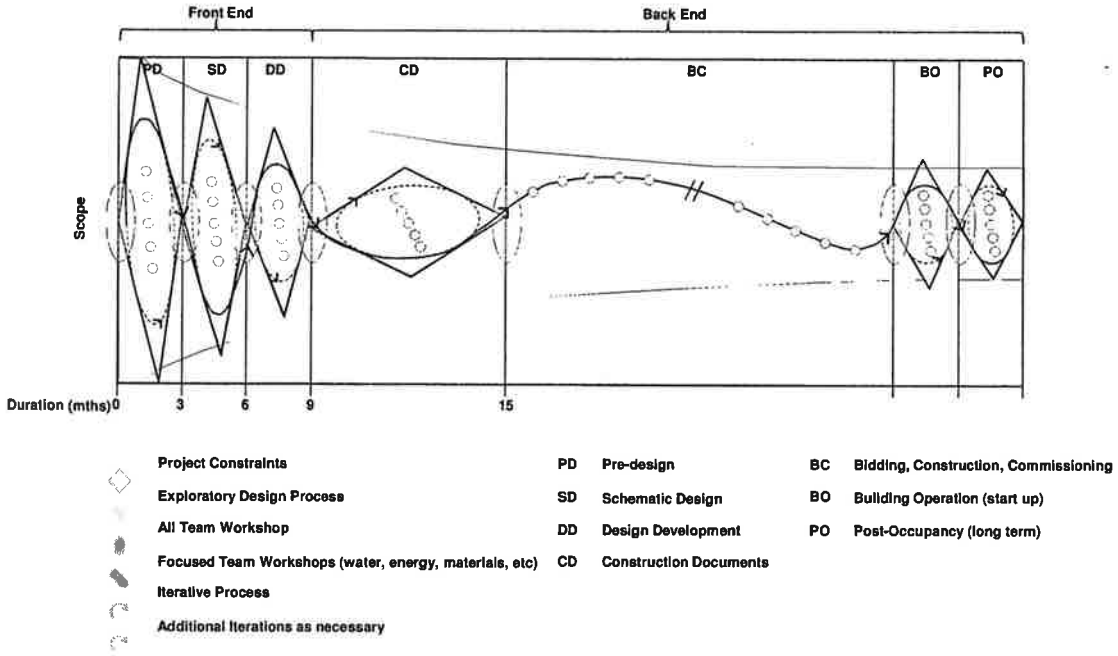
The Solution



◇	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 Frven Lim & Owen Wee , Surbana IGBC 2012

The Solution



Source Frven 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 ……..	

Part 2 : SUSTAINABLE DEVELOPMENT (ARCHITECTURE)

Why sustainable development ?

Sustainable Framework

Climate and adaption

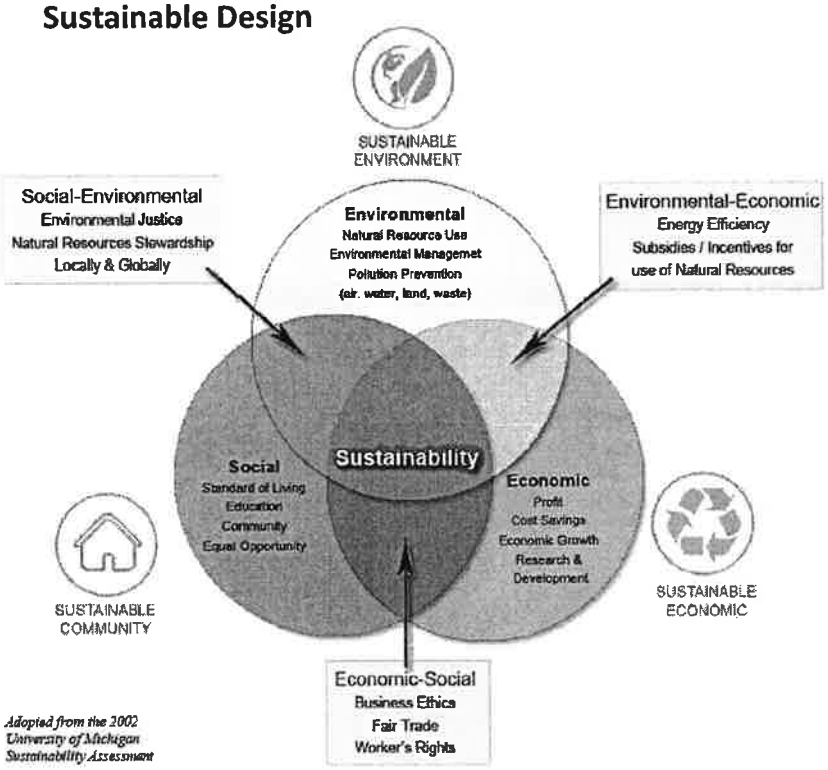
Thermal comfort

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

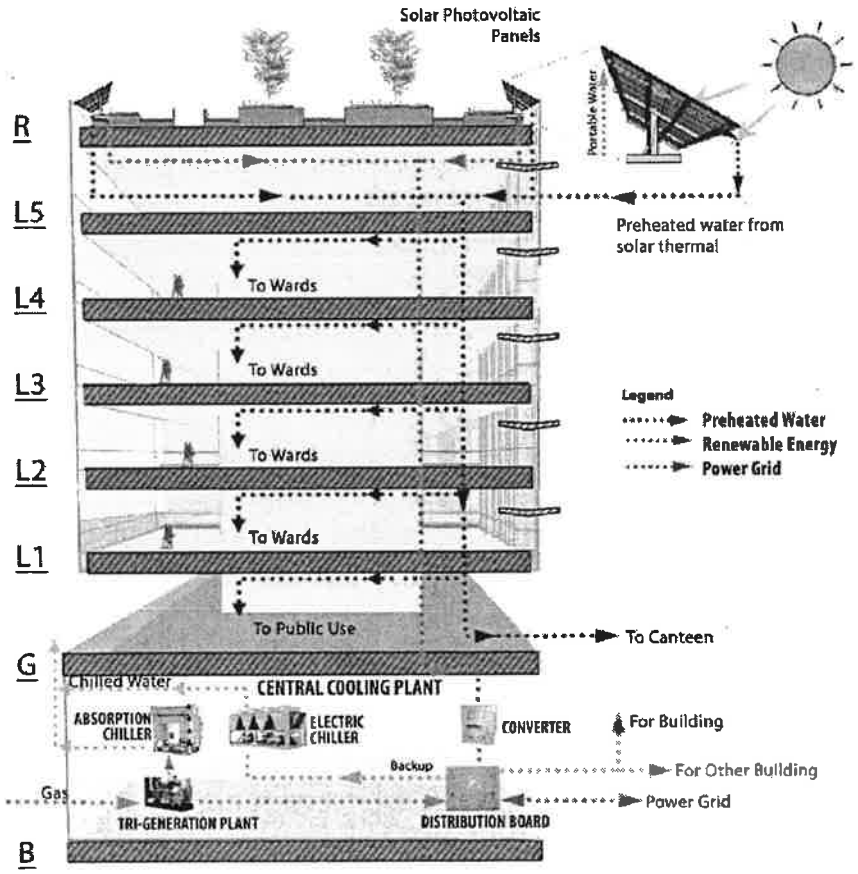


The Three trigger issues of Sustainability

Passive Design & Natural Ventilation, GMP 2015

Designing Sustainably

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



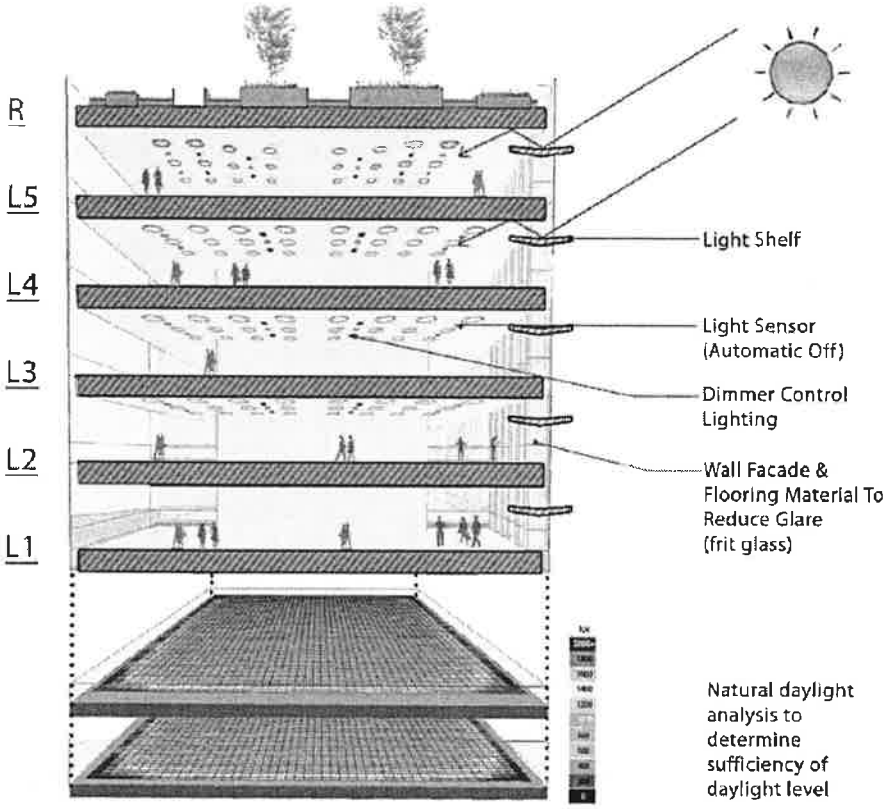
Passive Design & Natural Ventilation, GMP 2015

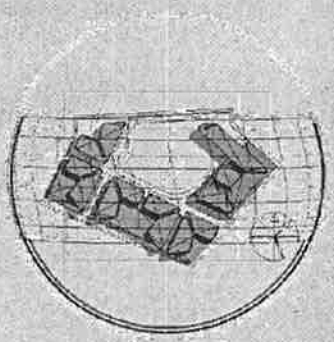
Sustainable Energy

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.

Maximizing Daylight

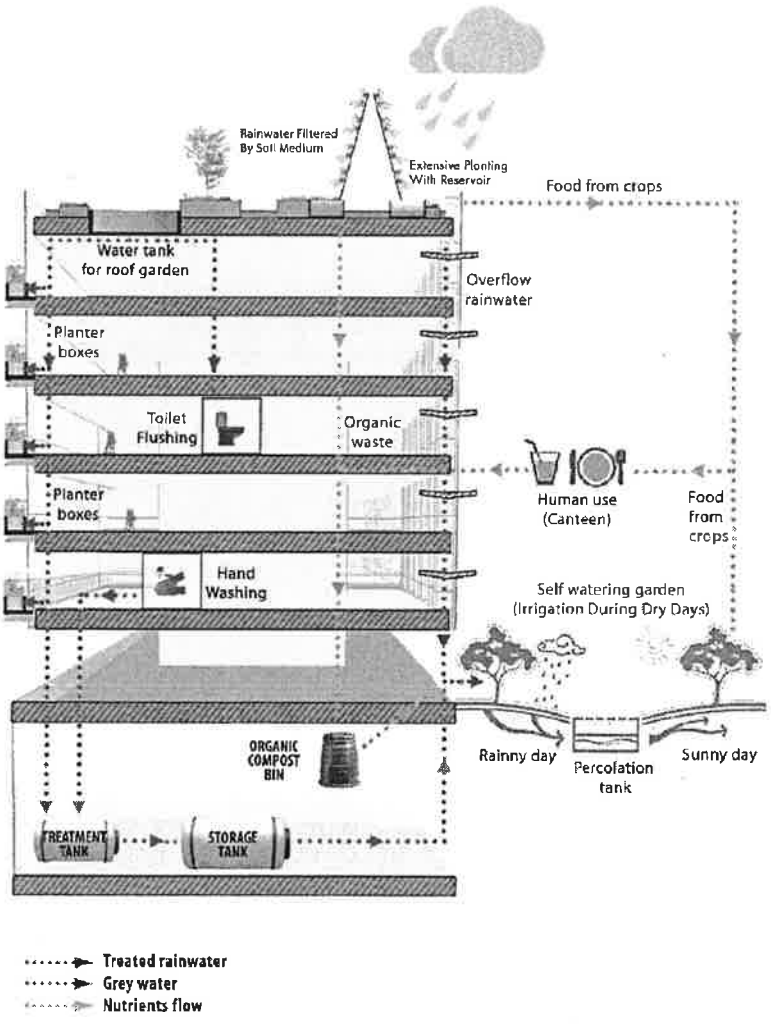
The key to daylight design is to bring in usable light while keep glare and heat away. Natural light improve human productivity and concentration of occupants as well as overall well being of patients. Integrated Façade design with light shelves and shading devices together will pre-construction daylight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

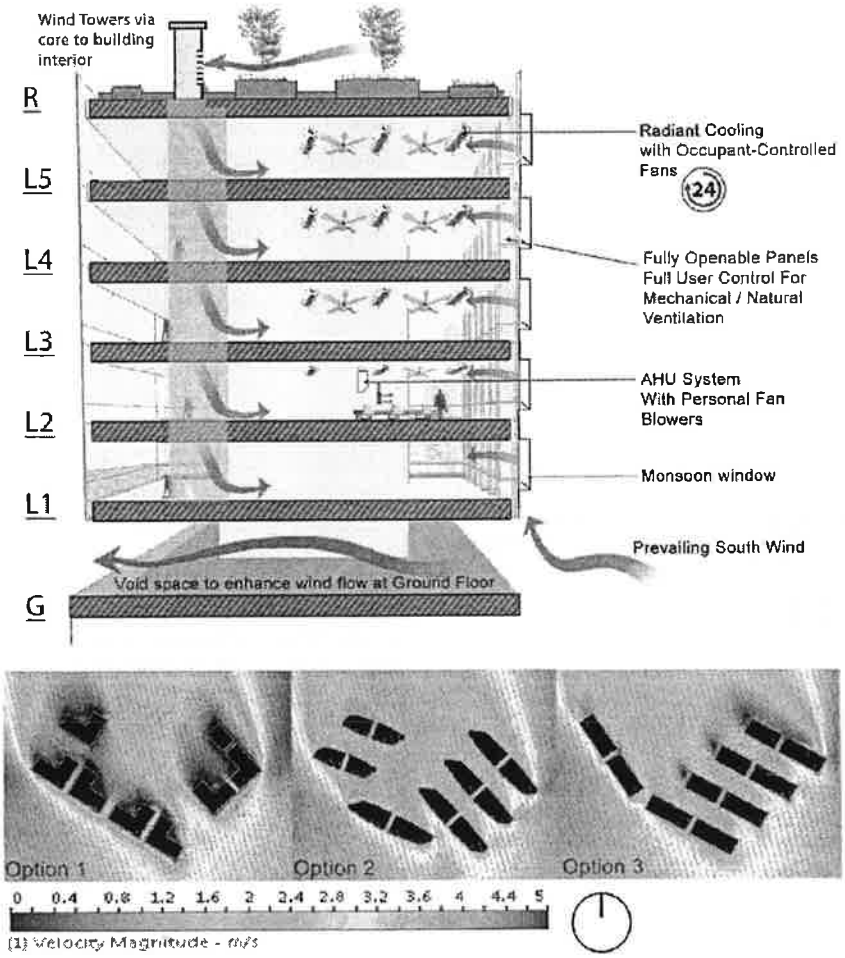




Sustainable Eco-system

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 healing and growing old in a healthy as well as a dignified way





Passive Design & Natural Ventilation, GMP 2015

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.

Passive Design & Natural Ventilation, GMP 2015

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

Passive Design & Natural Ventilation, GMP 2015

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

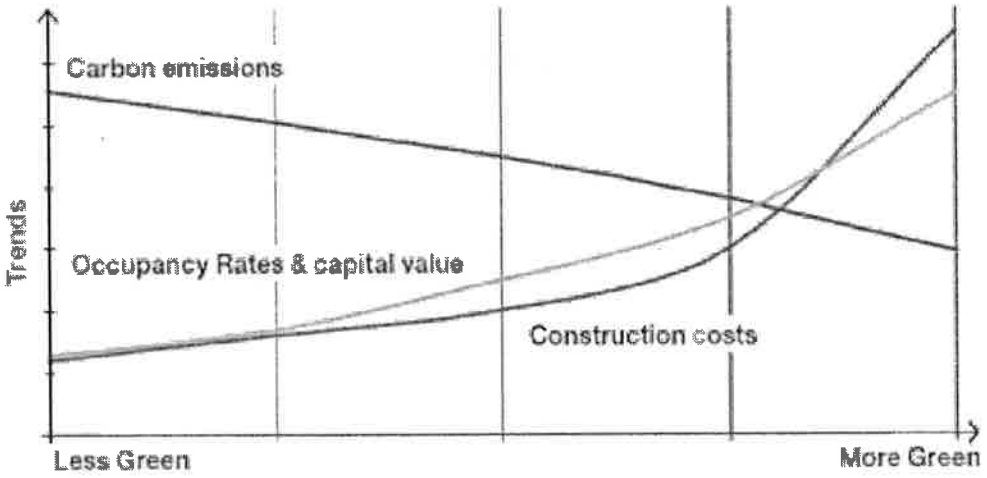
three-year study of some 1,300 properties, which Compared similar properties within a quarter-mile (400 meter) radius, LEED-certified buildings had 4.1% greater occupancy and $11.33 per sq.ft. (about 30%) higher rent and sold about 30% higher prices.

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

Passive Design & Natural Ventilation, GMP 2015

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

Passive Design & Natural Ventilation, GMP 2015

What does passive design cost?

Improve productivity and reduce health costs

Productivity improves 5%-10%

Medical absences 30% - 40%

Source:-Jerry Yudelson,2009

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

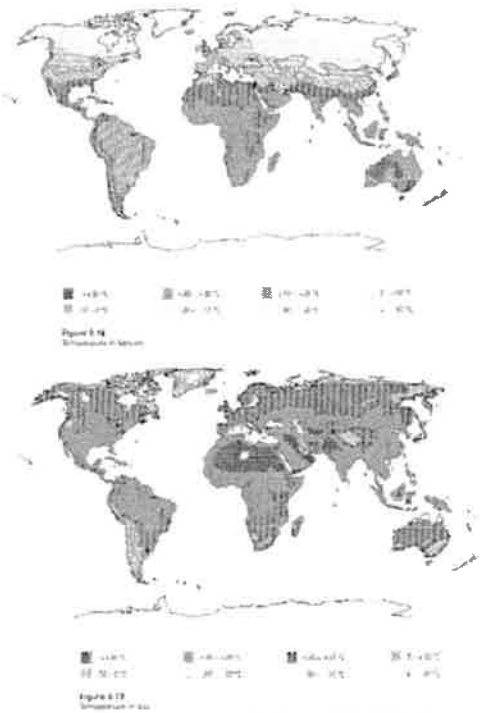
3 main things about the climate :-

Temperature & Humidity
Solar
Wind

Passive Design & Natural Ventilation, GMP 2015

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**).



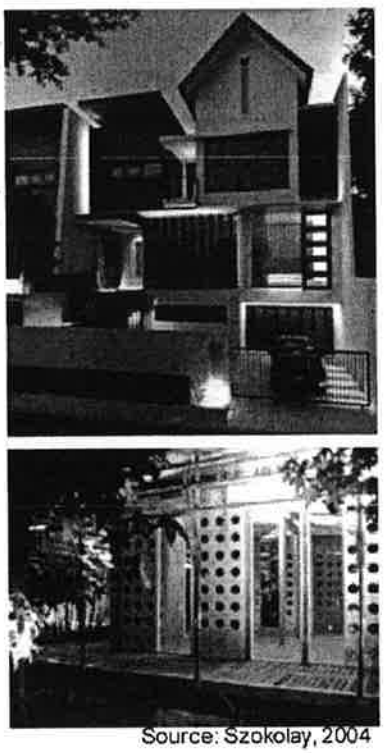
Source: http://www.ace.mmu.ac.uk

Passive Design & Natural Ventilation, GMP 2015

Climate Adaption : Climate

Warm humid Climates

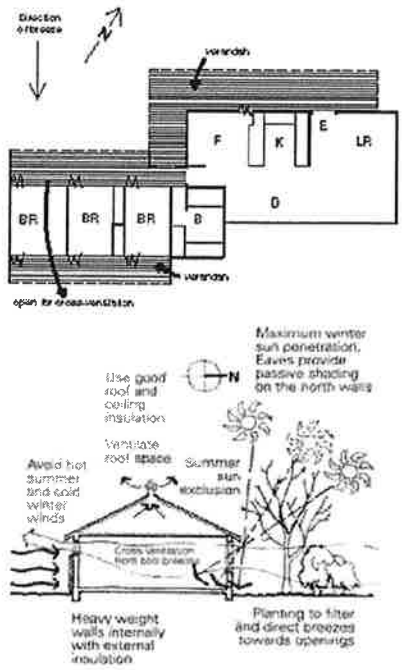
- It is possible to **influence the wind**, but not the solar incidence, so **solar orientation** should be a dominant aspect in the design;
- **North** and **south walls** can have **large openings**;
- **Rooms** could be arranged **in one row** in order to allow both **inlet** and **outlet openings** 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 :-





Passive Design & Natural Ventilation, GMP 2015

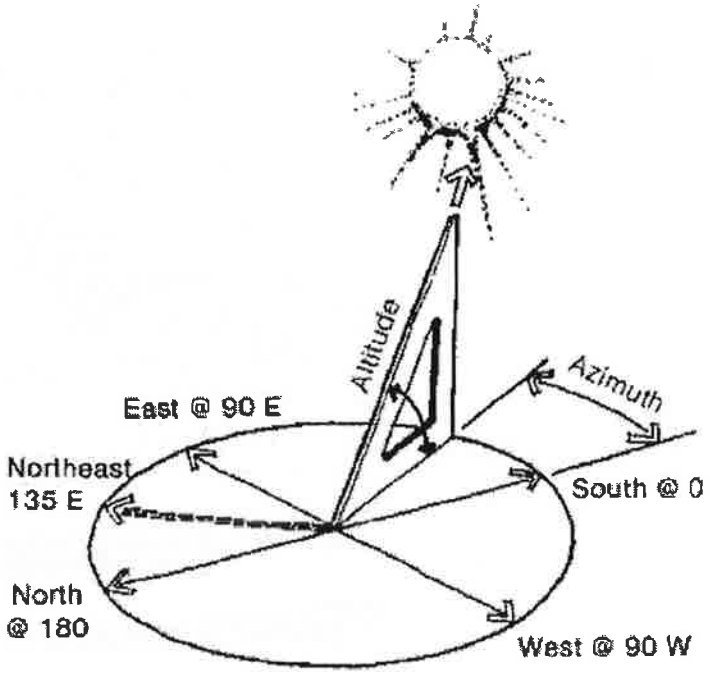
Climate Adaption : Climate





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

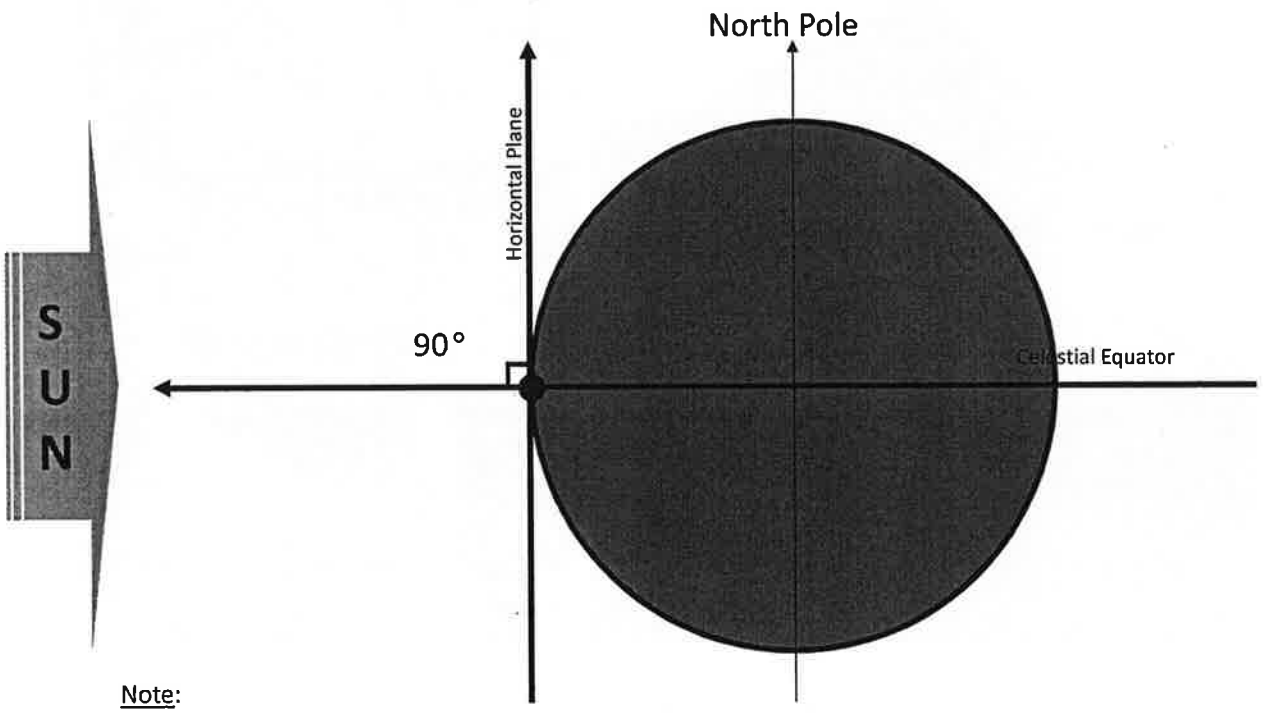
Climate Adaption : Solar



Source : Ben Lim University of Nottingham, 2009

Passive Design & Natural Ventilation, GMP 2015

Climate Adaption : Solar

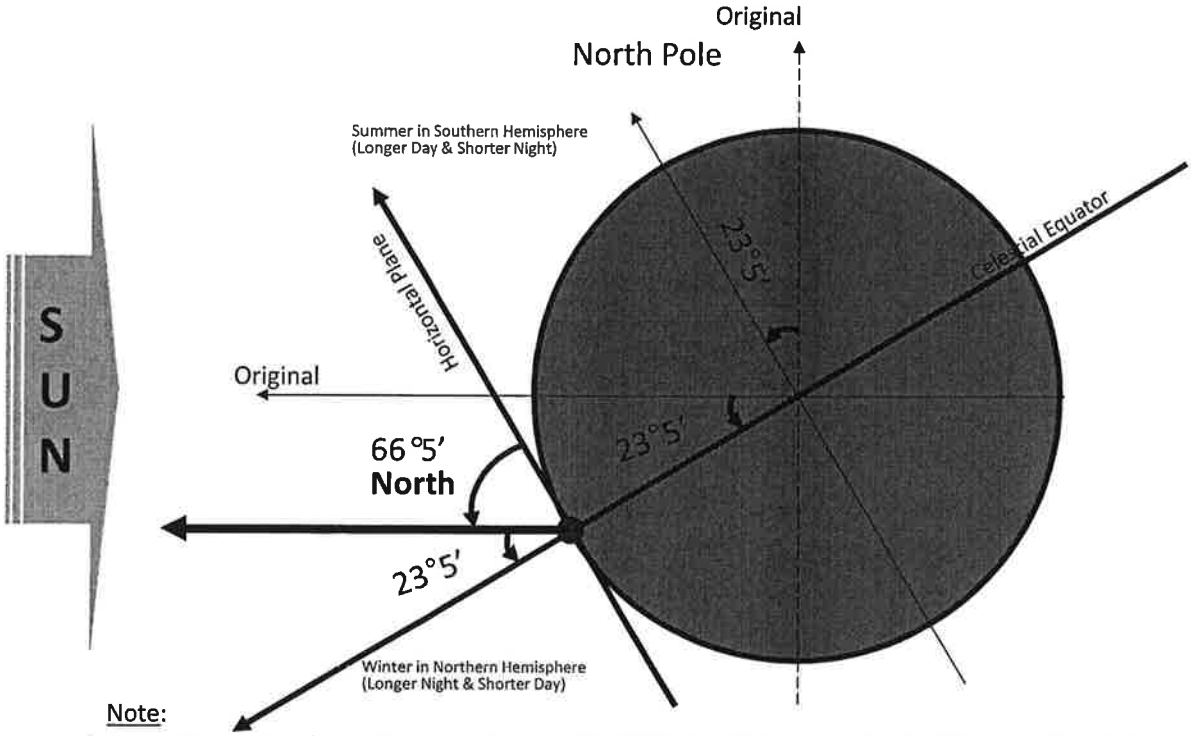


Note:

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



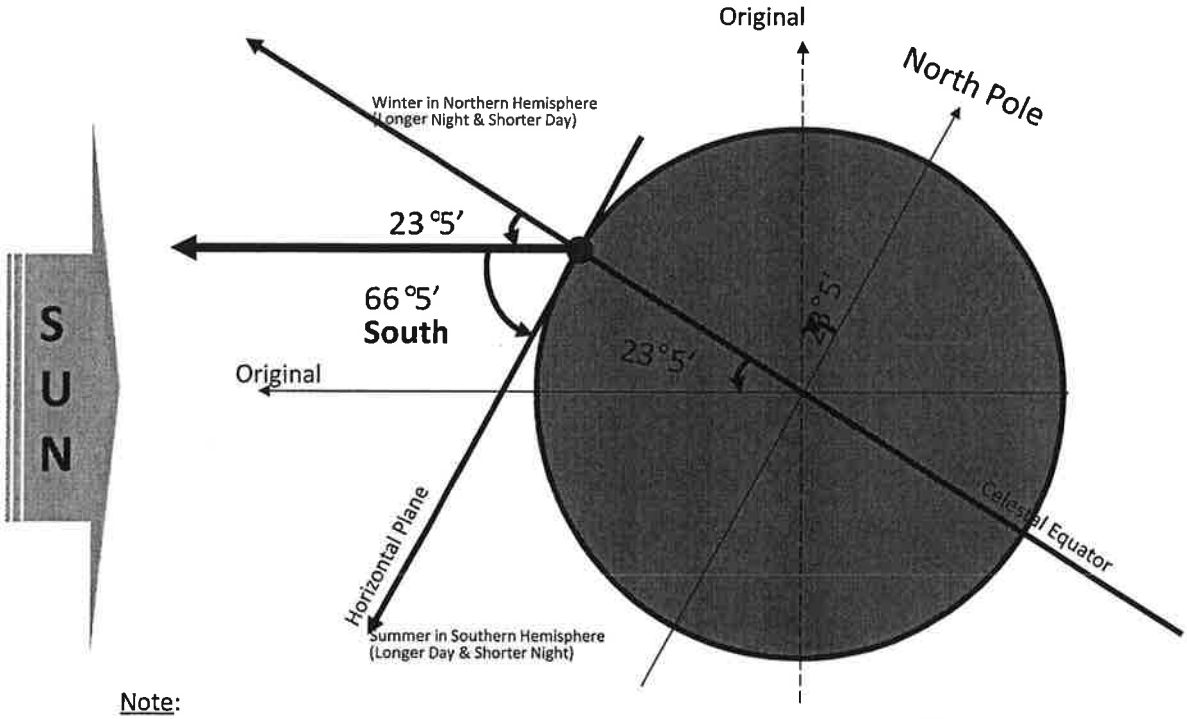
Note:

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 2015

Climate Adaption : Solar

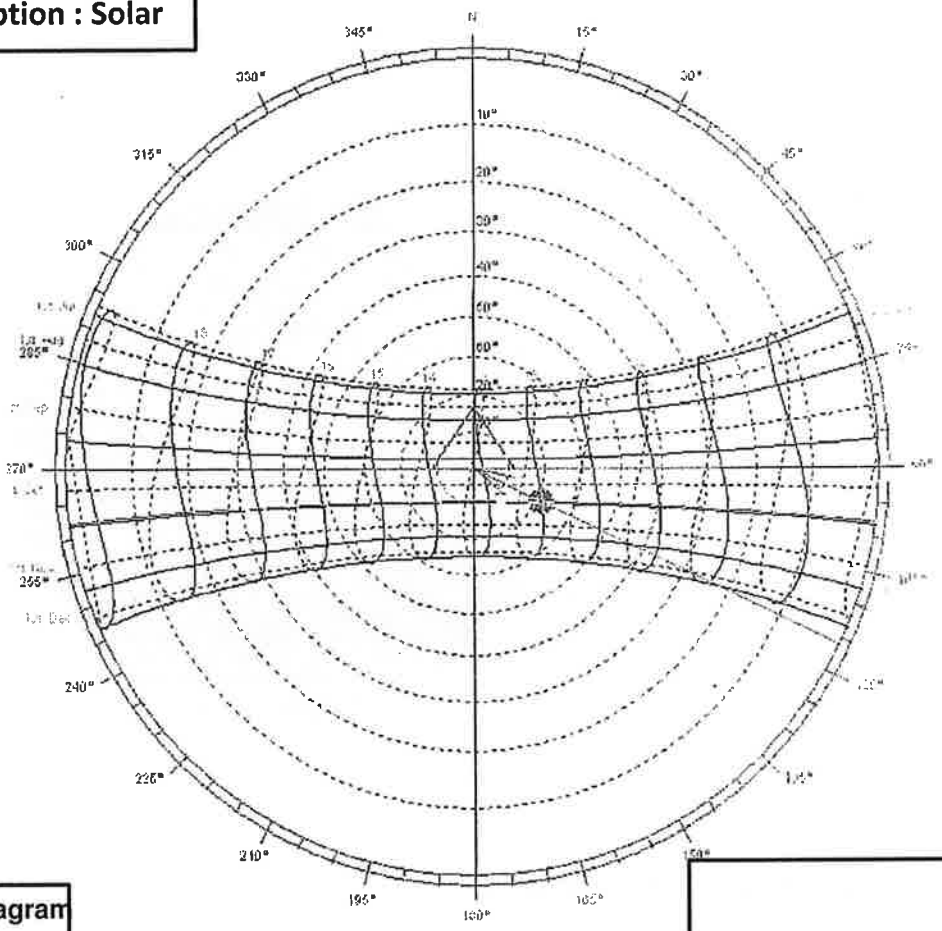


Note:

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)

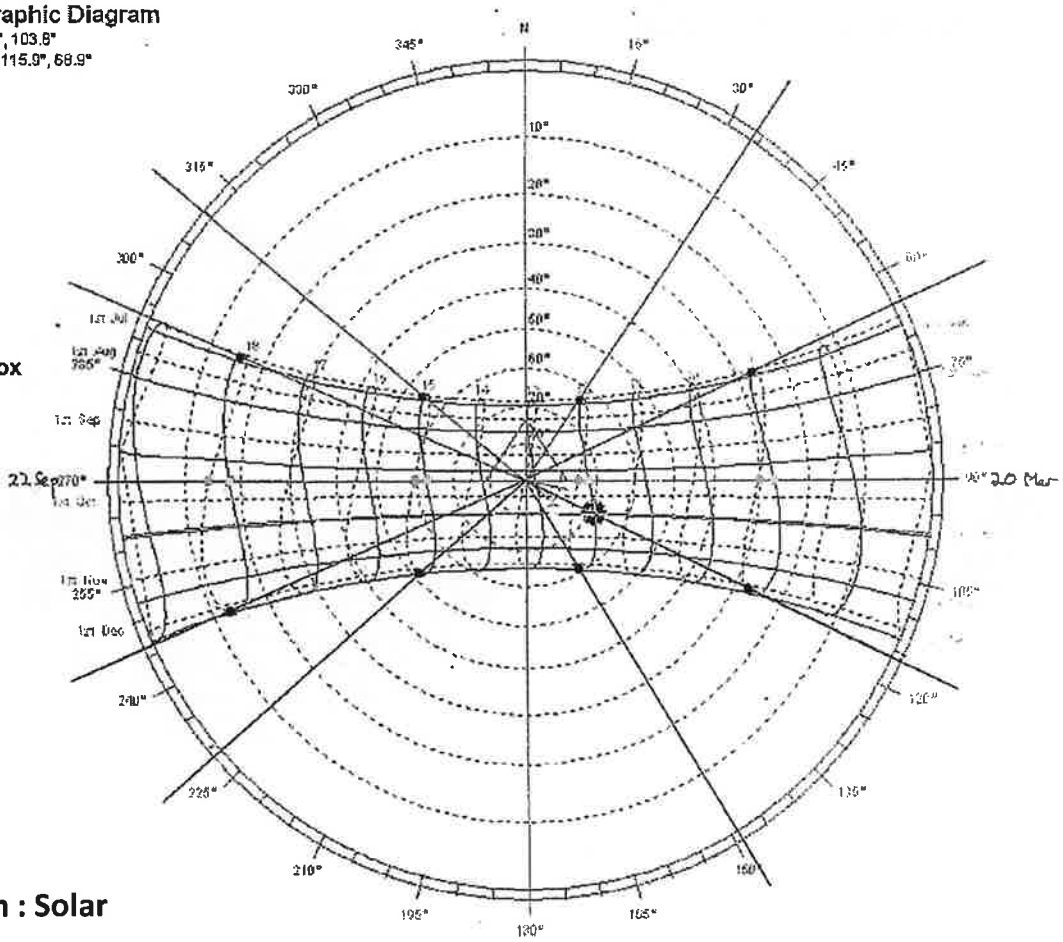
Climate Adaption : Solar

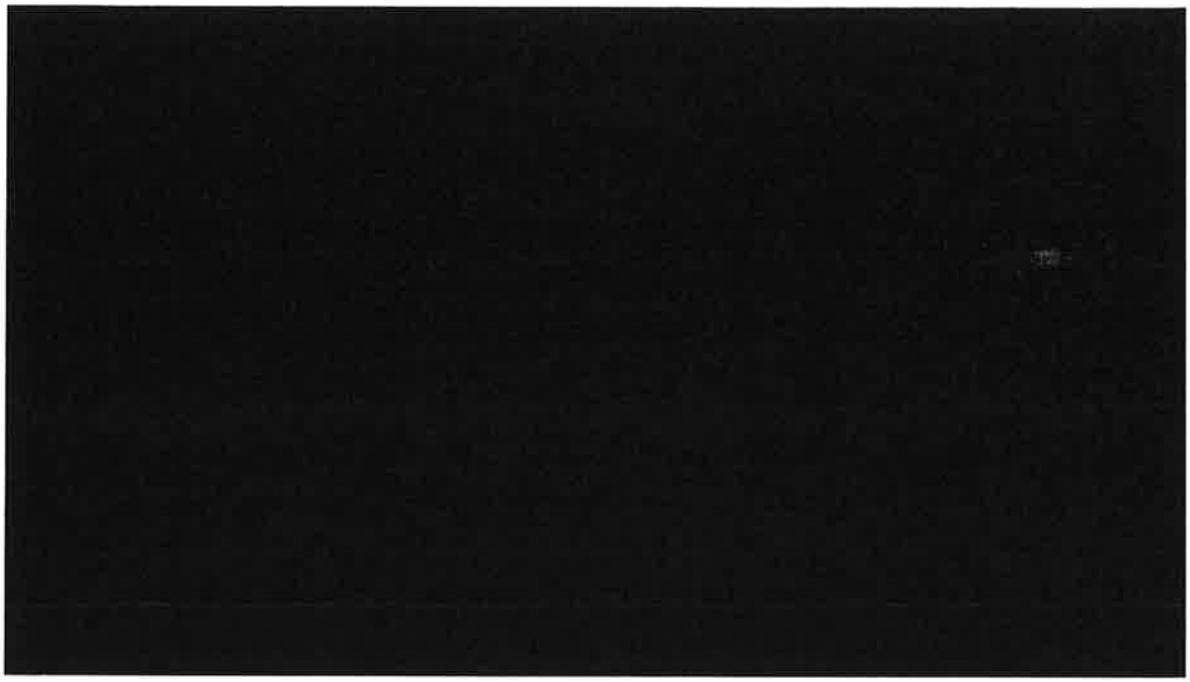


Stereographic Diagram

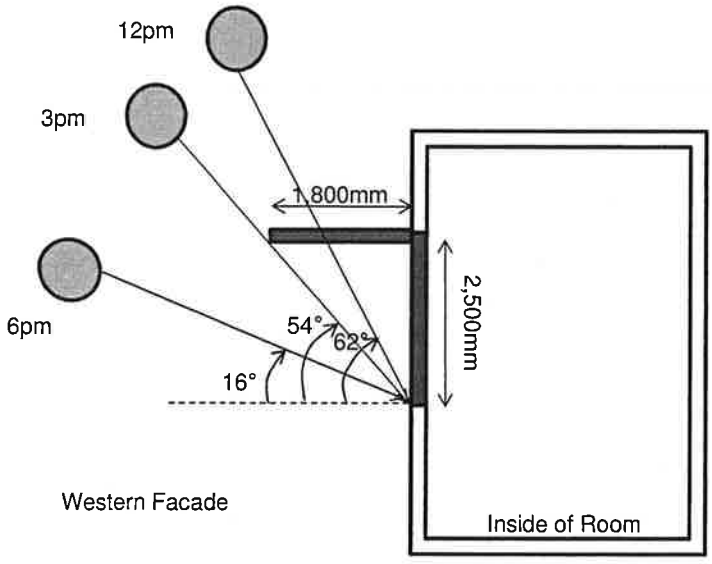
SINGAPORE
Stereographic Diagram
Location: 1.4°, 103.8°
Sun Position: 115.9°, 68.9°
HSA: 115.9°
VSA: 99.6°

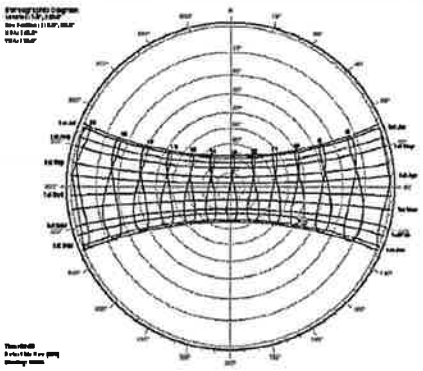
- Vernal Equinox
- Summer Solstice
- Autumnal Equinox
- Winter Solstice





Passive Design & Natural Ventilation, GMP 2015





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.

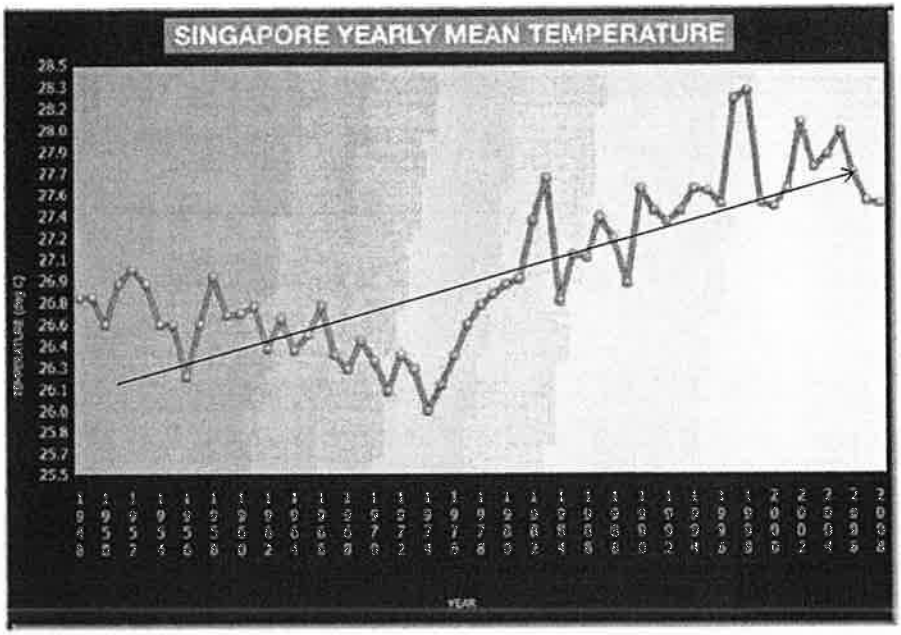
Horizontal shading is impractical on the western façade in Singapore.

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

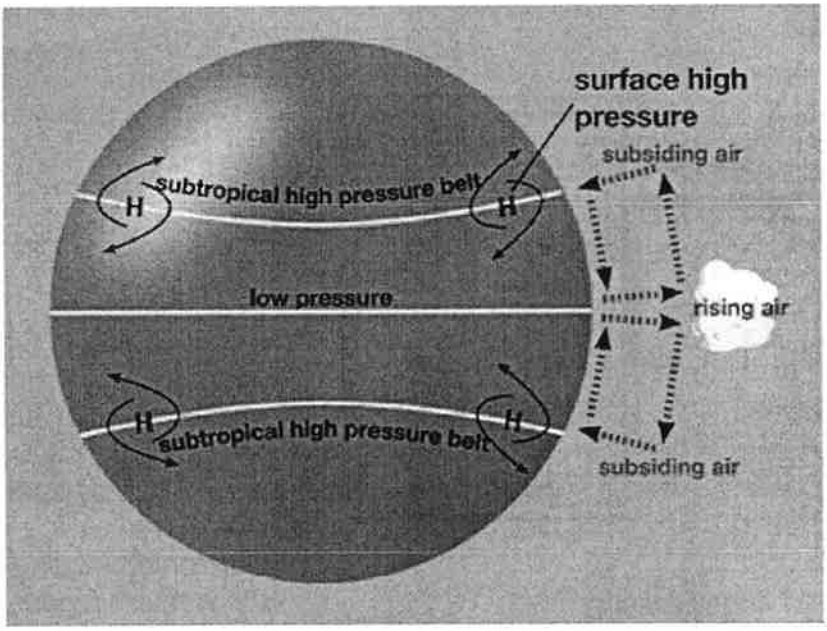
Climate Adaption : Singapore Climate



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

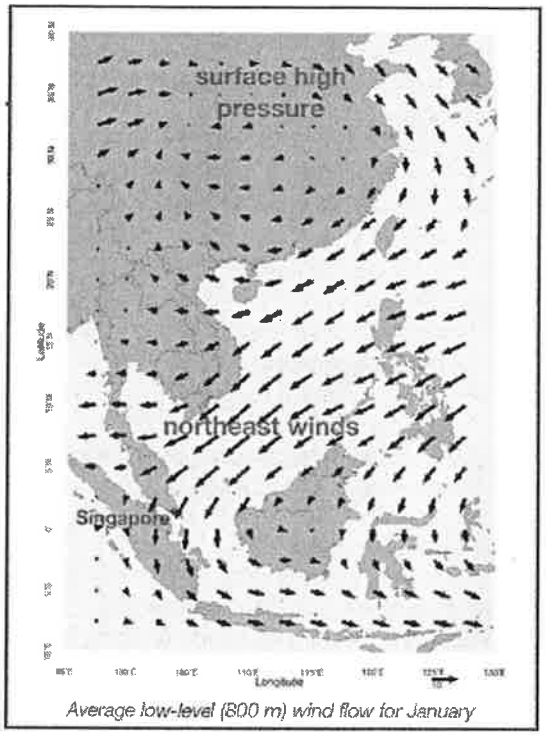
Climate Adaption : Wind



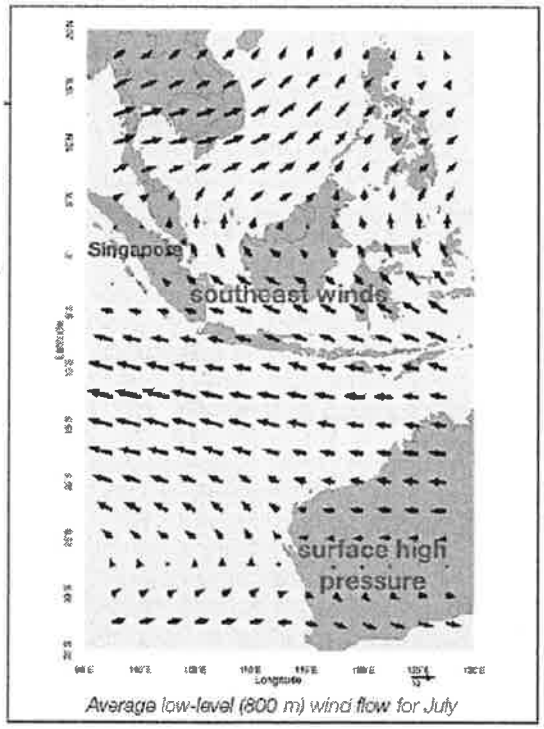
Average vertical circulations (Hadley cells) are shown on both sides of the equator. The downward arms are associated with the subtropical high pressure belts and fine weather.

Source : NEA , Weatherwise , 2009

Climate Adaption : Singapore Climate



Average low-level (800 m) wind flow for January



Average low-level (800 m) wind flow for July

Source : NEA , Weatherwise , 2009

Rainfall :- Monsoon seasons

Passive Design & Natural Ventilation, GMP 2015

Climate Adaption : Singapore Climate

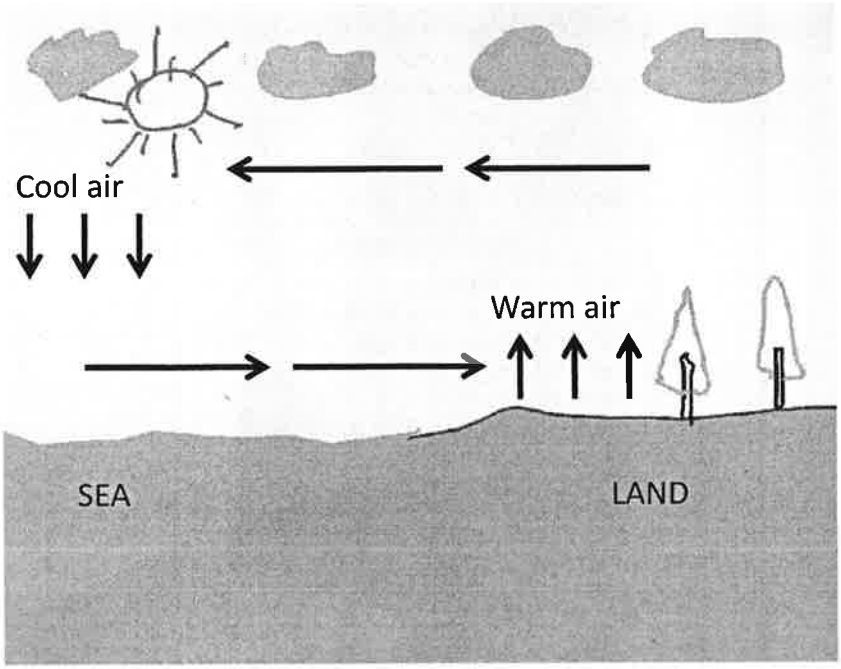
Rainfall

Month	Mean Total Raindays[4]	Occurrence of highest hourly mean[5] (hrs)	Monthly mean[6] (mm)
January	13.8	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	158.8
November	18.0	1500	262.4
December	18.5	1700	329.5

[4] Calculated by averaging the total number of raindays for each month for the 27 year period (1982 – 2008)
[5] Time of day for which the mean rainfall occurs for each month for the 27 year period (1982 – 2008) is the highest
[6] 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

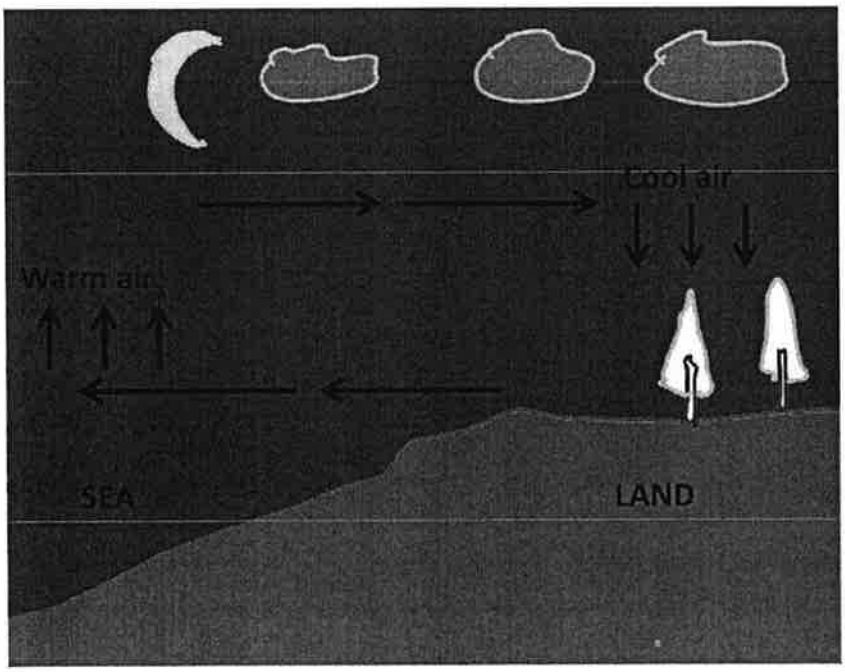


Sea wind (afternoon)

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

Passive Design & Natural Ventilation, GMP 2015

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

Passive Design & Natural Ventilation, GMP 2015

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

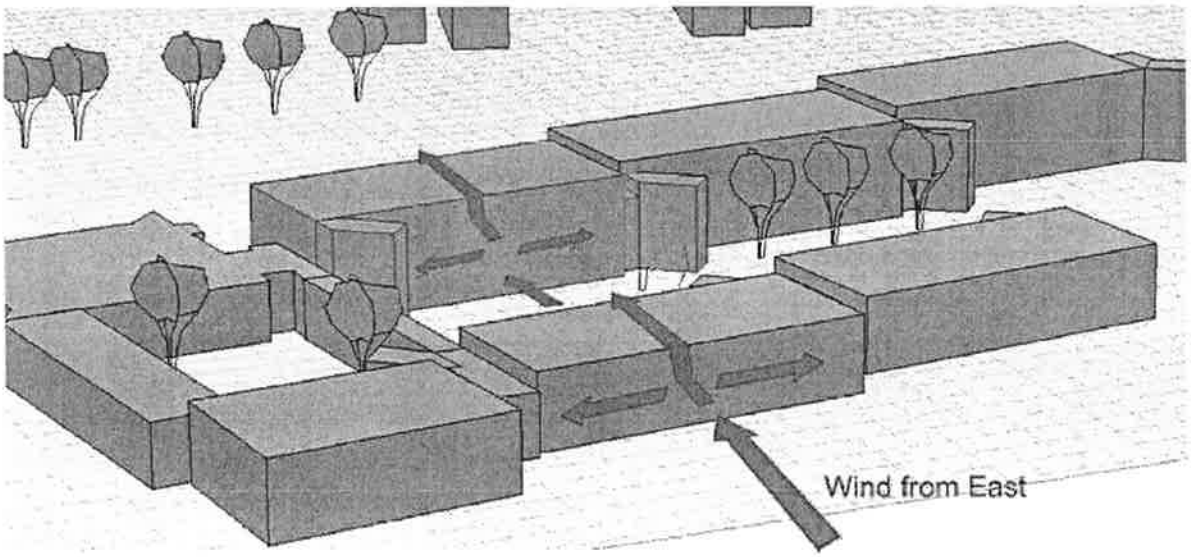
Month	General Weather Features
October	October is the beginning of the second Inter-monsoon Period of the year. It often marks the beginning of the rainy season which may last till January the following year. Monthly rainfall jumps more than 20 mm on the average from the September value. However, this 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

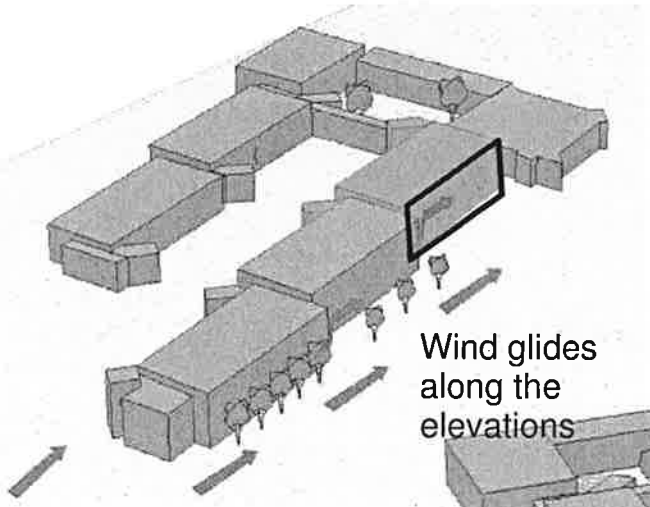
Passive Design & Natural Ventilation, GMP 2015

Design Strategies – Potential for ventilation



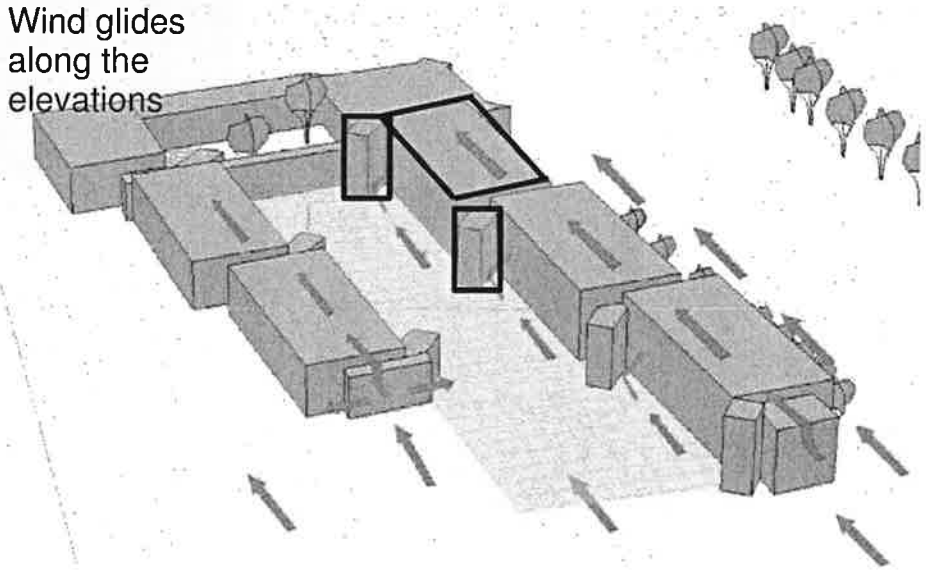
- The existing building prevents the wind from moving through and into the building

Design Strategies – North Wind



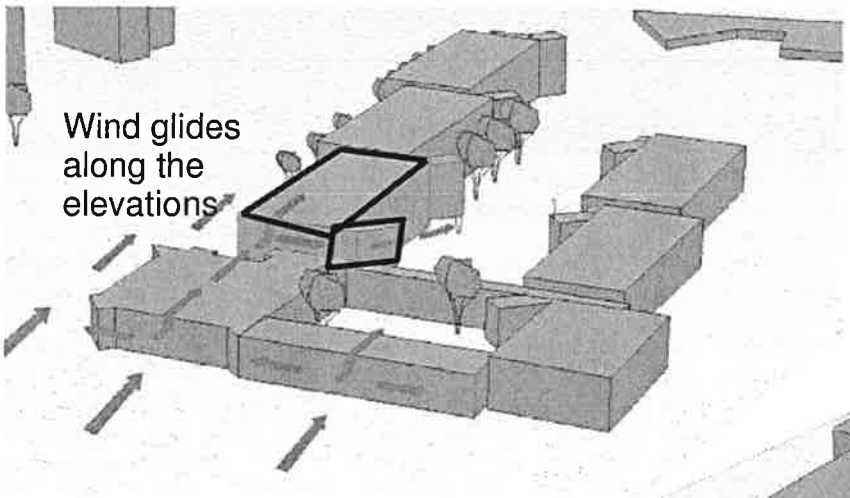
Wind Movement study

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

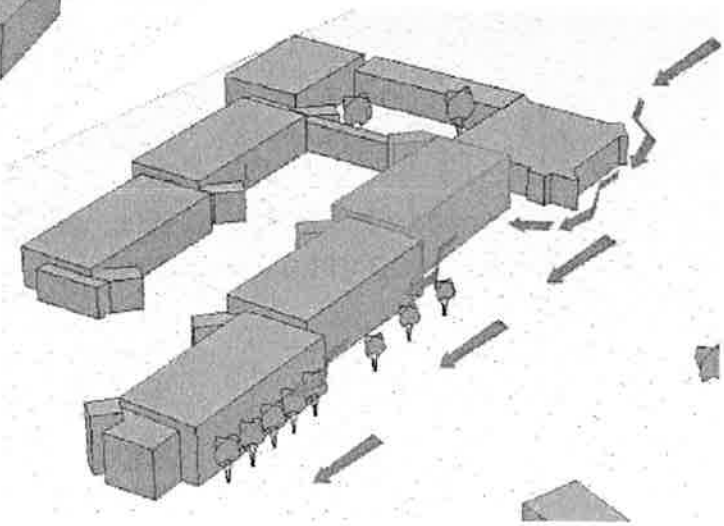


Passive Design & Natural Ventilation, GMP 2015

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

Thermal Comfort

Passive Design & Natural Ventilation, GMP 2015

Thermal Comfort : Factors of Thermal comfort in buildings

1) <u>Environmental Factors</u>

a) Dry bulb temperature

b) Mean Radiant Temperature

c) Relative Humidity

d) Air movement

2) <u>Physiological</u>

a) Metabolic Rate

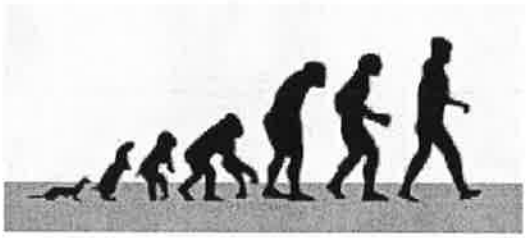
b) Clothing Level

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



Passive Design & Natural Ventilation, GMP 2015

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

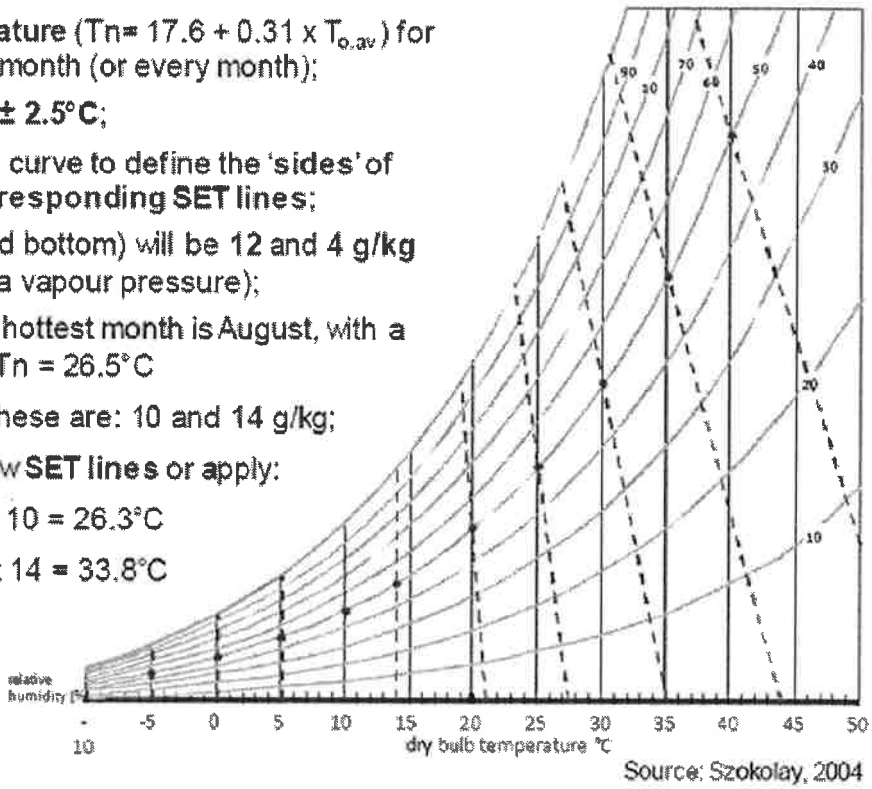
  T_L = 24°C; T_U = 29°C; AH for these are: 10 and 14 g/kg;

- For the **side boundaries** follow **SET lines** or apply:

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

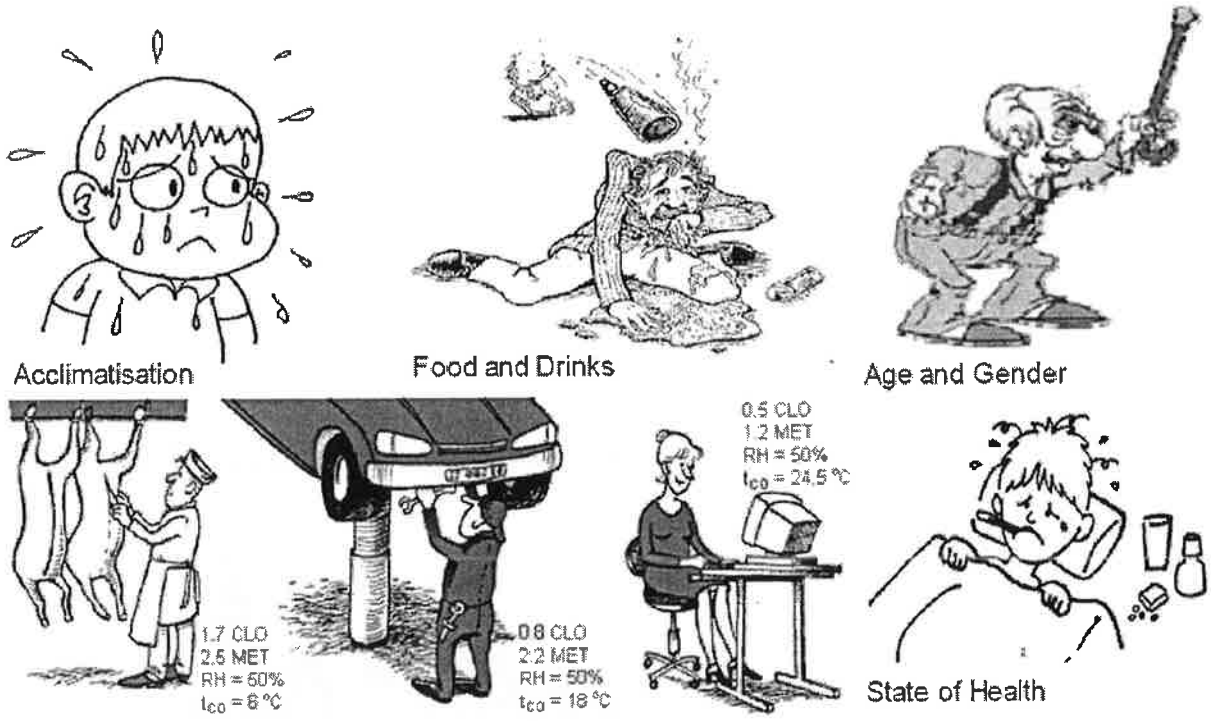
  for T_U = 29 + 0.023 (29 – 14) x 14 = 33.8°C

Source :Nottingham University , Sergio A



Source: Szokolay, 2004

Thermal Comfort : Factors of Thermal comfort in buildings



Passive Design & Natural Ventilation, GMP 2015

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

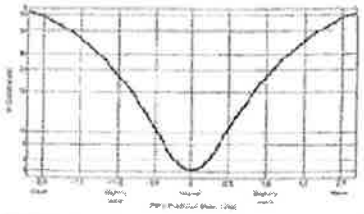
Adaptive Comfort

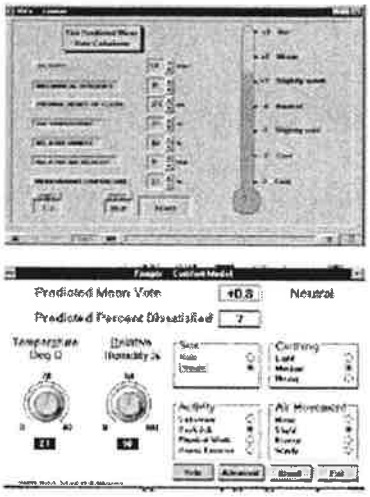
Adaptive comfort models add a little more human behaviour to the mix. They assume that, if changes occur in the thermal environment to produce discomfort, then people will generally change their behaviour and act in a way that will restore their comfort. Such actions could include taking off clothing, 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).

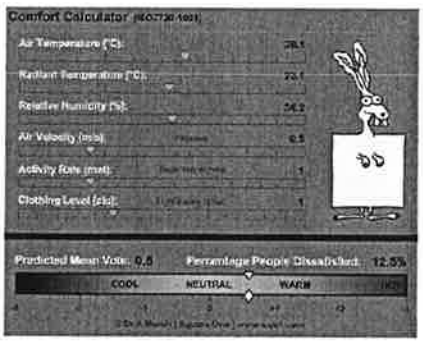




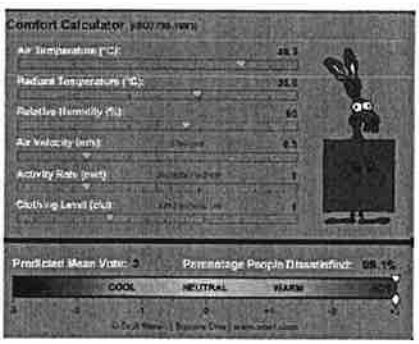
Source :Nottingham University , Sergio A
Image : Http://squ1.org

Passive Design & Natural Ventilation, GMP 2015

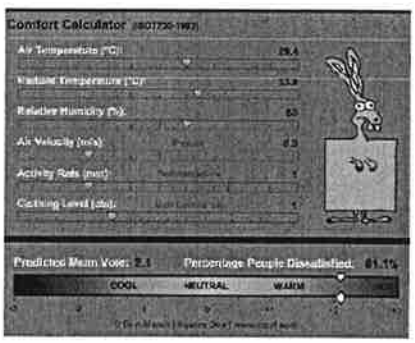
Thermal Comfort : Factors of Thermal comfort in buildings



PMV = 0.6



PMV = 3

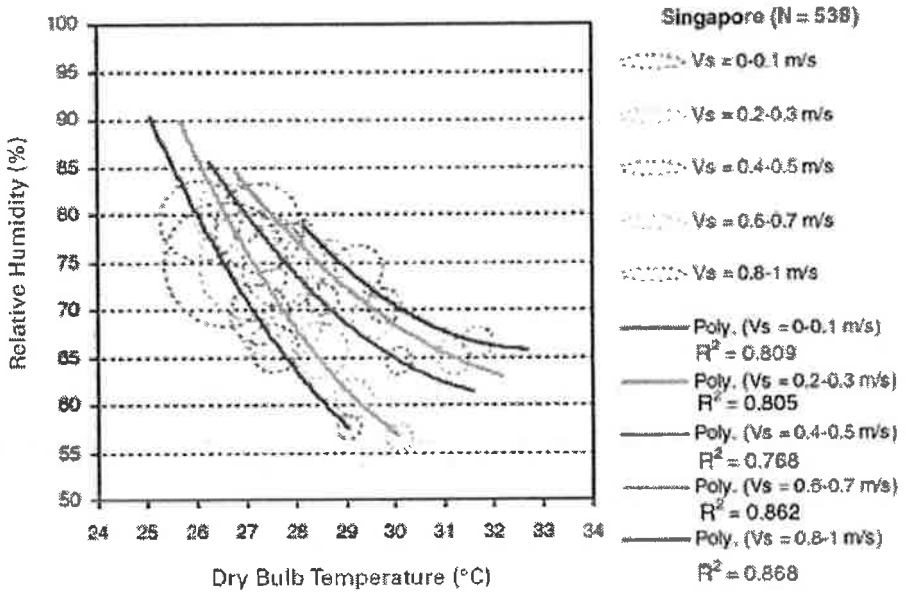


PMV = 2.1

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 Wong 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).

Passive Design & Natural Ventilation, GMP 2015

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

Part 3 : Principles of Natural Ventilation

Measuring Wind
Factors affecting wind movement

Passive Design & Natural Ventilation, GMP 2015



Air-conditioning liberated Design

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 big air-condition compressors outside one 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 34 deg C, night and day.

The maximum outdoor temperature recorded by the Meteorological Service for the past two days has been 33 deg C.

The 43-year-old said: "My 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 window to keep out the heat is the lesser of two 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 Cheow 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 Bishan-Toa Payoh North Education Centre to relocate the compressors. She said the HDB would "monitor the situation closely".



TERRENCE TAN

Heat is emitted by these four compressors up to Mr Tan Cheow Beng's second-floor bedroom.

Source : Straits times

Passive Design & Natural Ventilation, GMP 2015

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

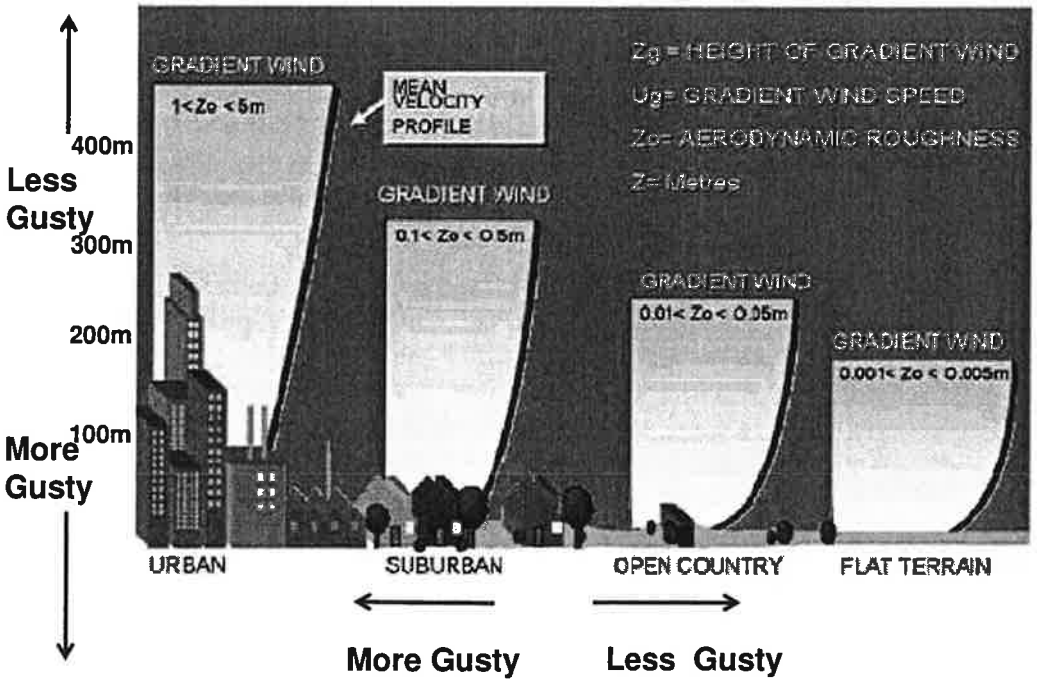
-Building Height and Form

-Orientation

Source : philp oldfield University of Nottingham, 2009

Passive Design & Natural Ventilation, GMP 2015

Wind and its surrounding

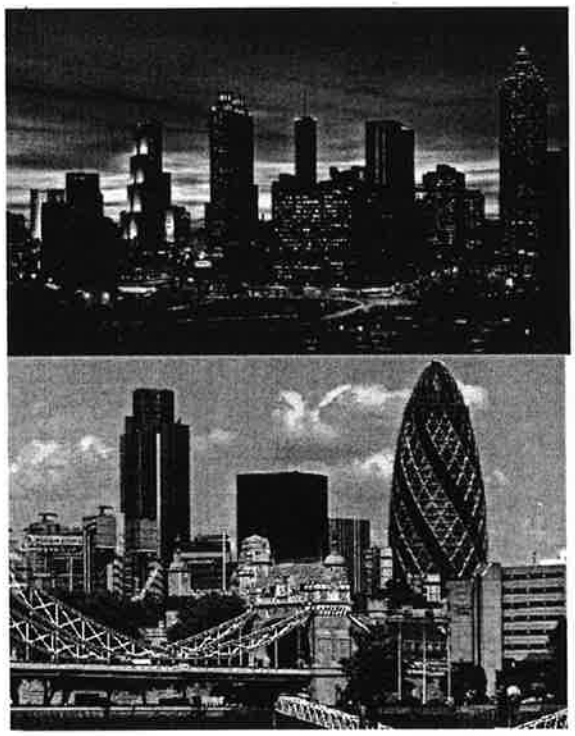


Beaufort Scale

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	



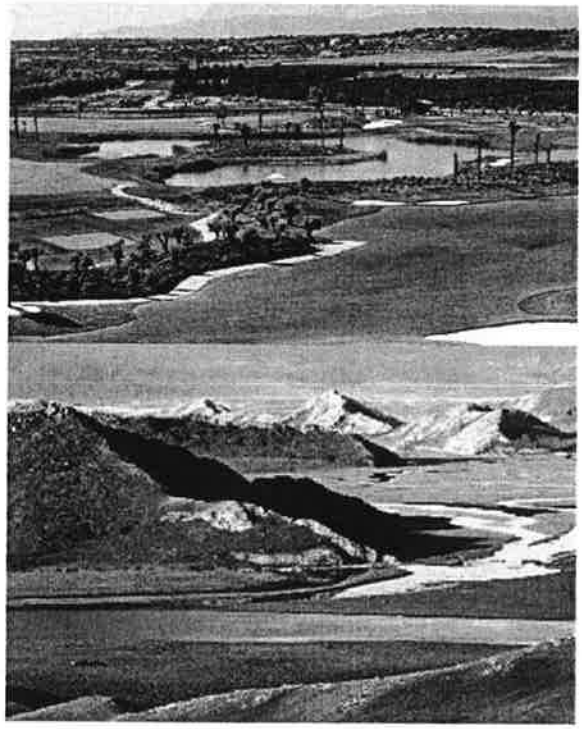
Developed in 1805 by Sir Francis Beaufort, U.K. Royal Navy



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.

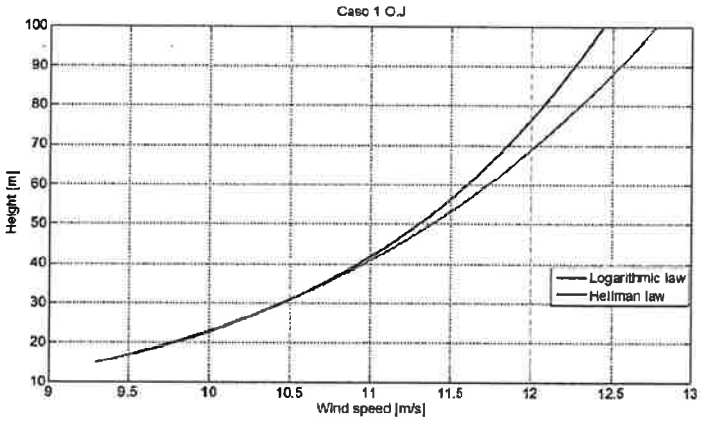


http://www.tour-beijing.com/china_silk_road_tours/bayanbulak_travel.php

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



www.intechopen.com1440

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.

Passive Design & Natural Ventilation, GMP 2015

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

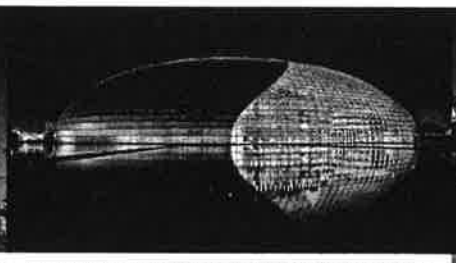
Speed of the wind at 60 m is 9.297 m/s

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



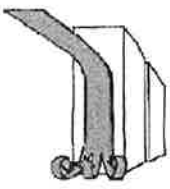
http://menswear223.rssing.com/chan-15643960/all_p49.html



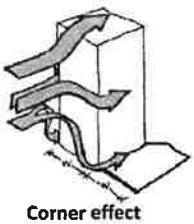
http://www.plbusinessgroup.com/50-most-innovative-buildings-of-last-decade/

Passive Design & Natural Ventilation, GMP 2015

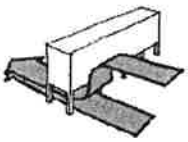
Factors that affect wind : Building form



Downwash



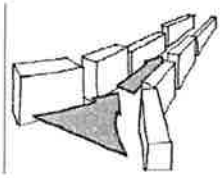
Corner effect



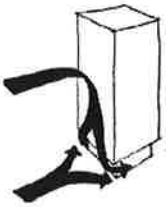
passageways



Low & high bldgs



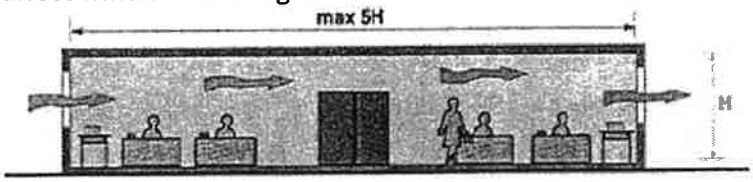
Channel effects



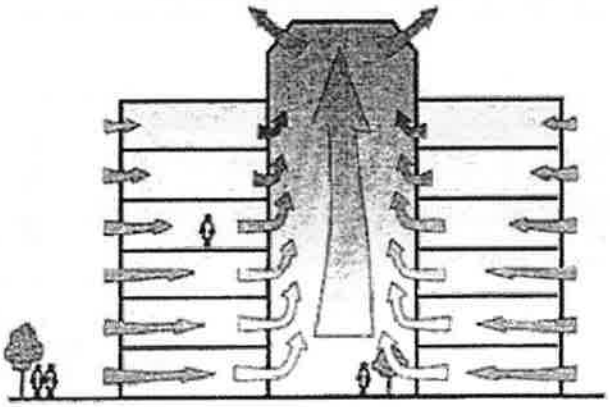
Building recesses

Source : philip oldfield University of Nottingham, 2009

Factors that affect wind : Building form



Cross ventilation



Stack ventilation

Passive Design & Natural Ventilation, GMP 2015

Factors that affect wind : Wind Sensitivity for building

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

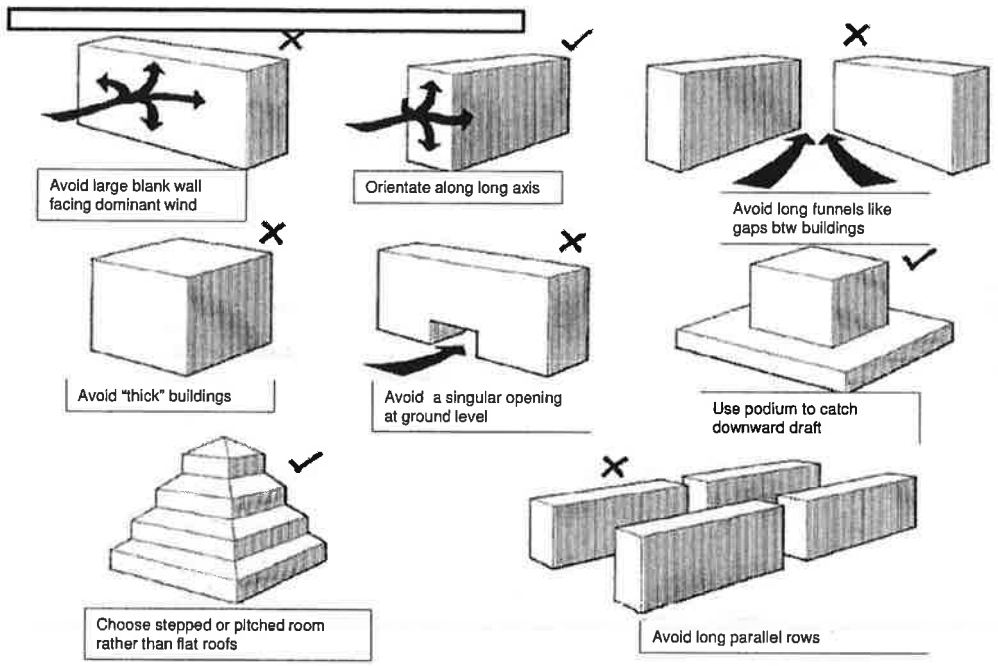
Wind environment is the wind flow experience by people and the influence it has on their activities

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.

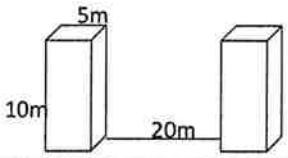
Source : philip oldfield University of Nottingham, 2009

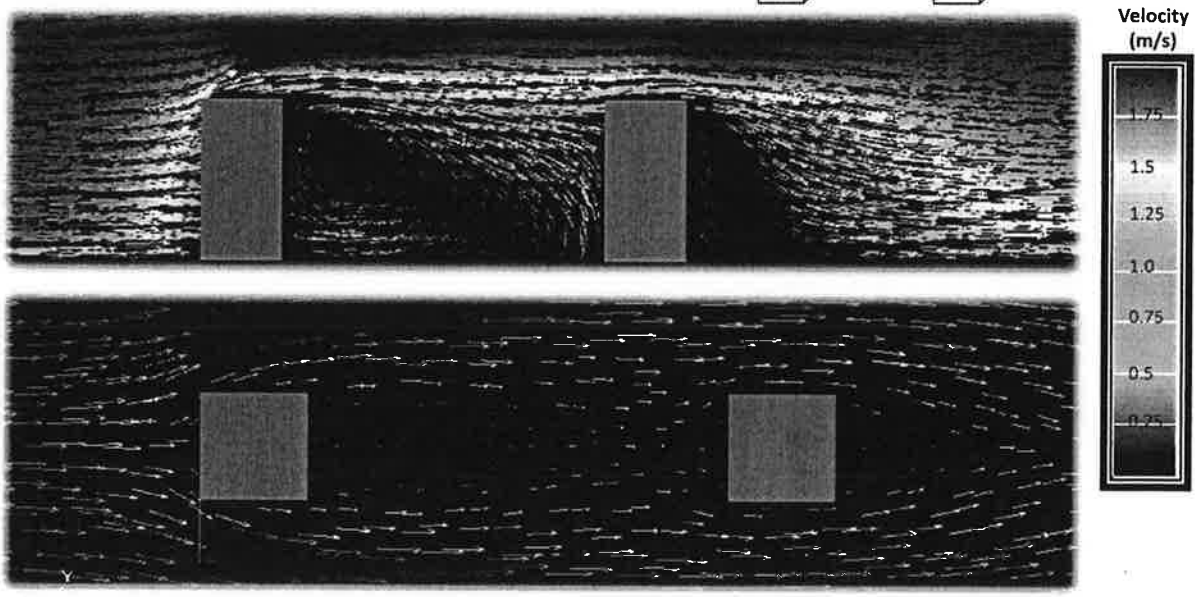
Passive Design & Natural Ventilation, GMP 2015

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.

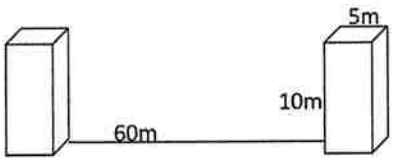


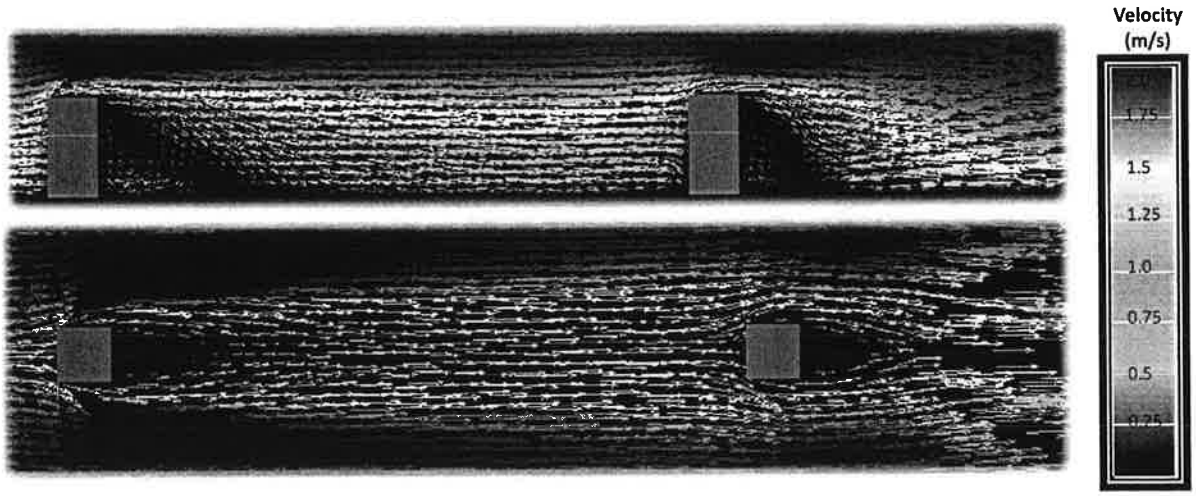


Source : Surbana ESD , 2010

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.



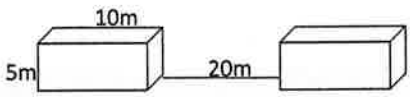


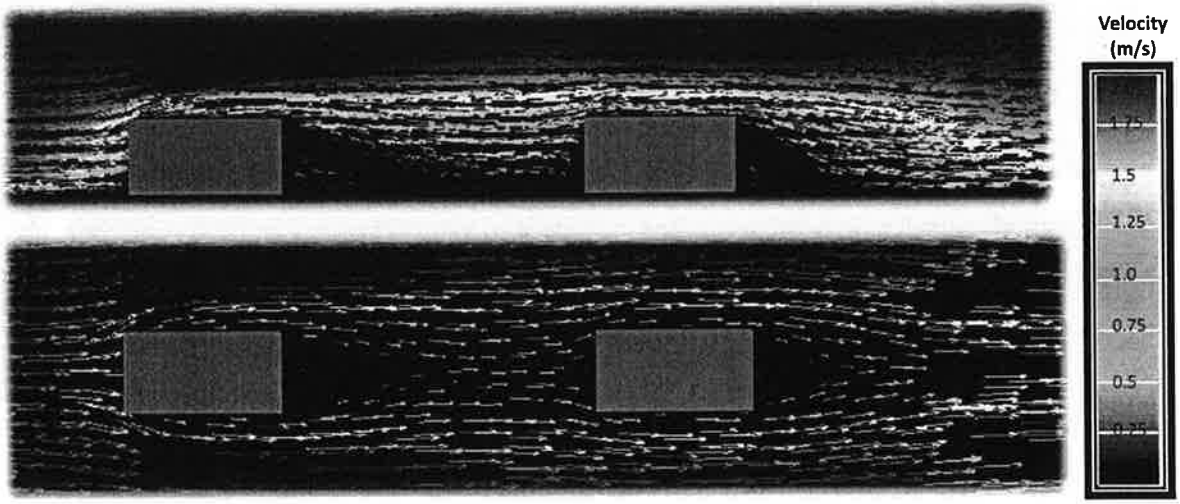
Source : Surbana ESD , 2010

Passive Design & Natural Ventilation, GMP 2015

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.



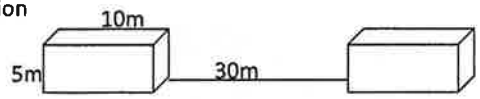


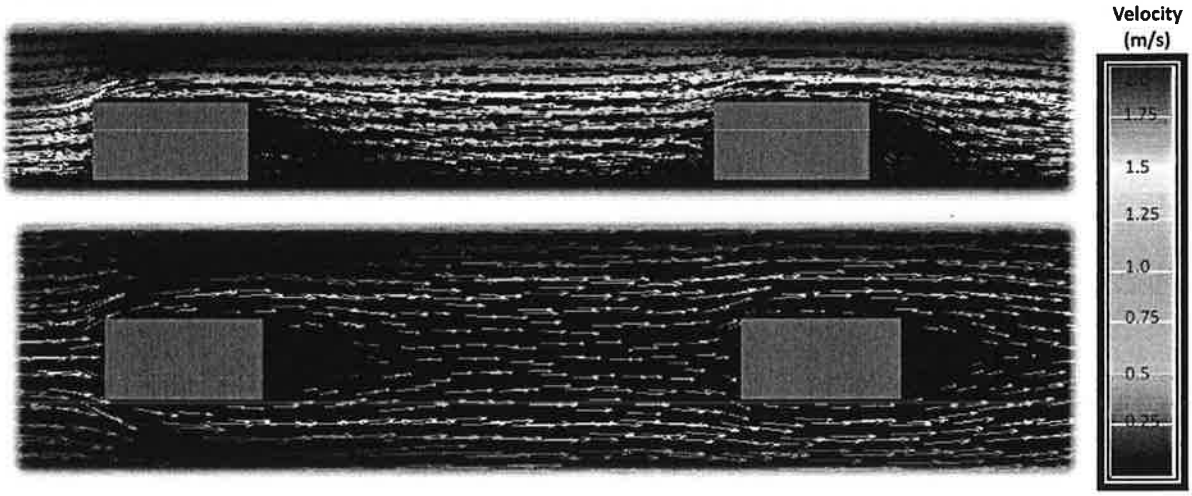
Source : Surbana ESD

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.

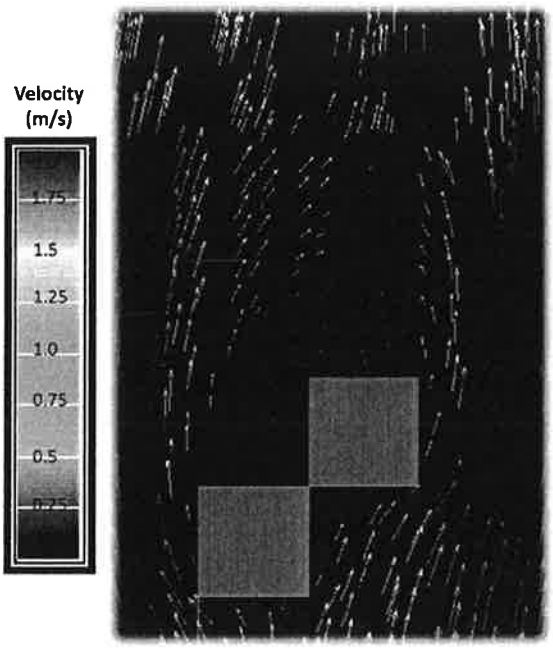




Source : Surbana ESD

Passive Design & Natural Ventilation, GMP 2015

Factors that affect wind : Building form

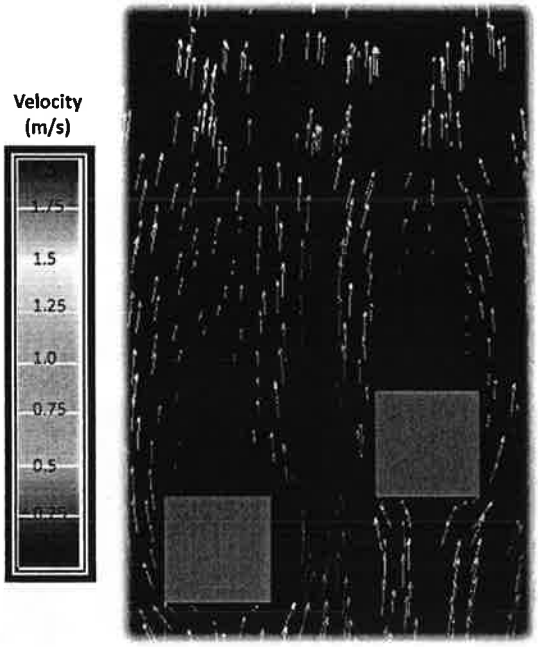


Stagnant effect

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



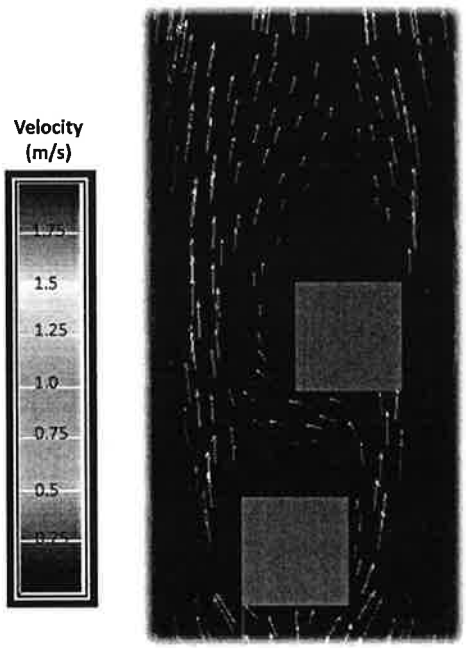
Canyon effect

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

Source : Surbana ESD

Passive Design & Natural Ventilation, GMP 2015

Factors that affect wind : Building form

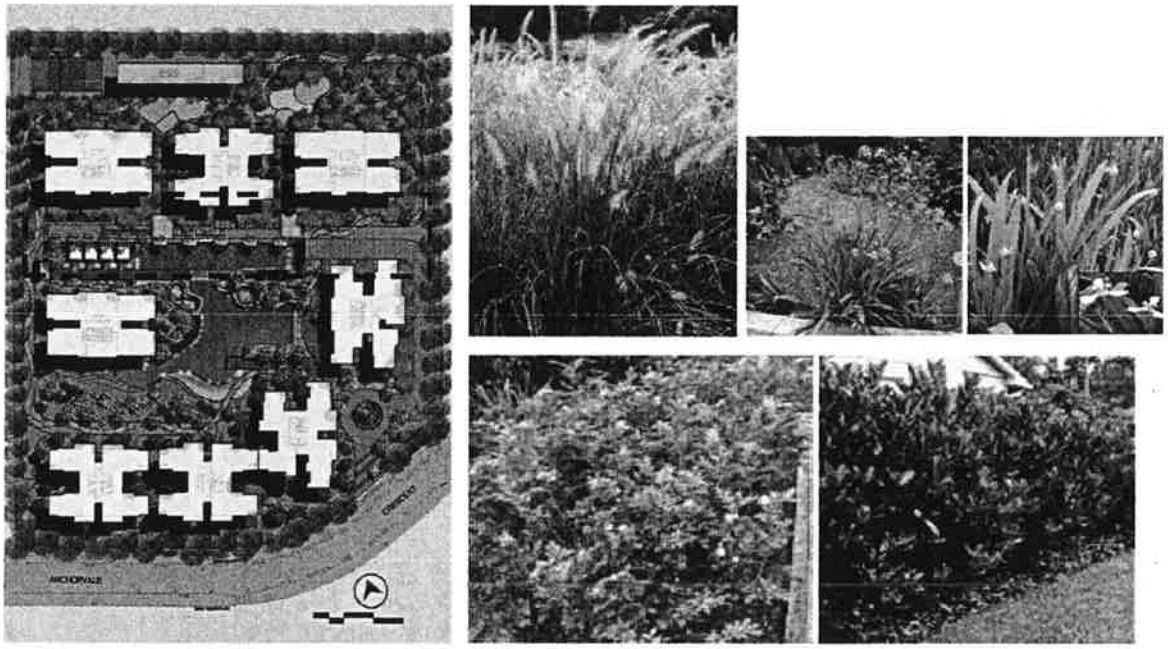


Scooping effect

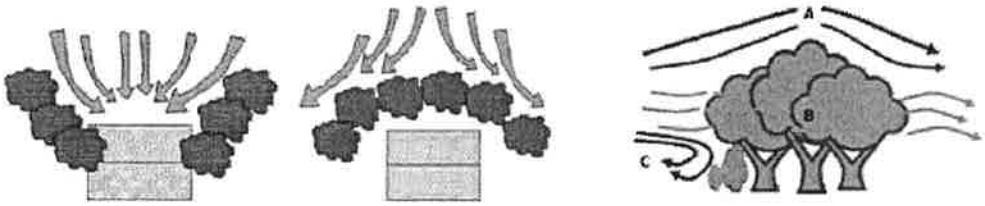
When buildings created a gap in between and behind each other, wind will be scooped into the gaps.

Source : Surbana ESD

Vegetation influence air movement



Vegetation influence air movement



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

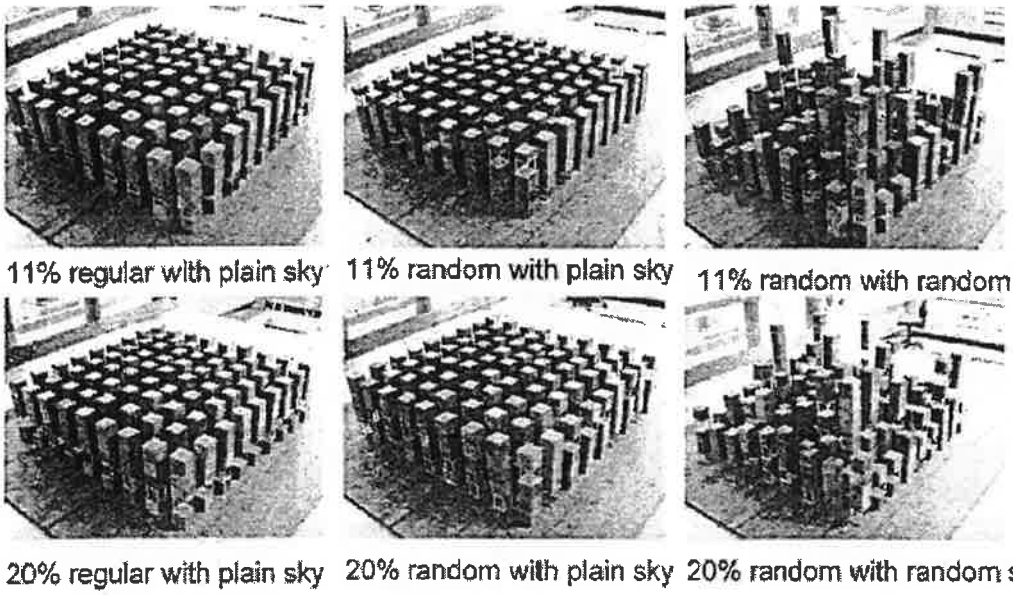
Angle	Wall	No trees				Trees				Effect Trees	
		WT	*CFD*	*diff.*	R^2	*WT*	*CFD*	*diff.*	R^2	*WT*	*CFD*
90°	*A*	19.7	23.0	17%	0.78	32.7	35.0	7%	0.77	66%	52%
90°	*B*	5.34	8.03	50%	0.95	2.69	7.10	169%	0.83	-50%	-12%
45°	*A*	18.4	14.3	-22%	0.81	31.0	21.6	-30%	0.75	69%	51%
45°	*B*	3.70	2.41	-35%	0.80	5.27	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

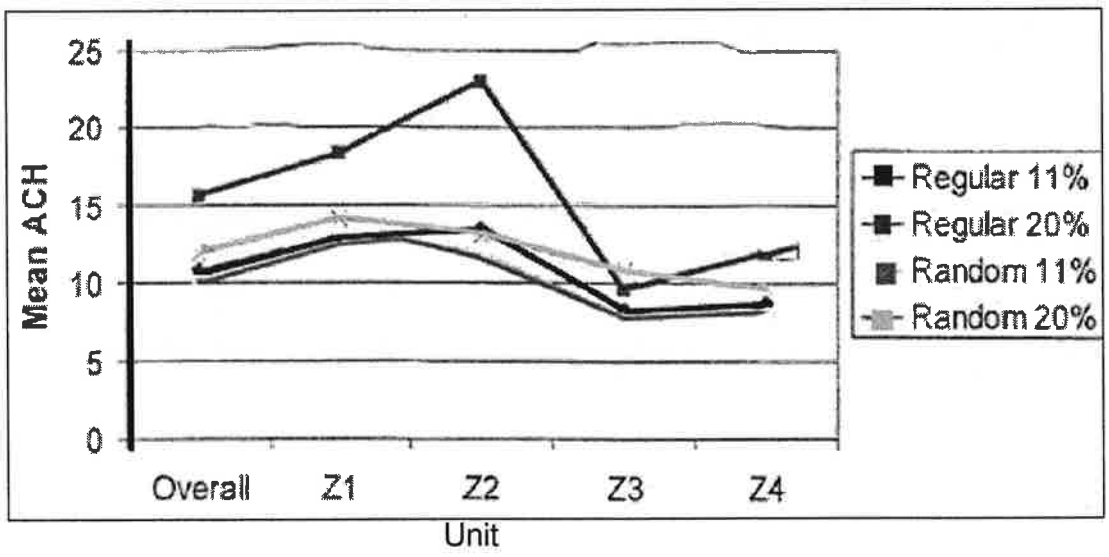


Source : Prof Wong Nyuk Hien, NUS , Studies on Urban design and their implied performance on ventilation , joint research Chinese Hong Kong University and NUS

Passive Design & Natural Ventilation, GMP 2015

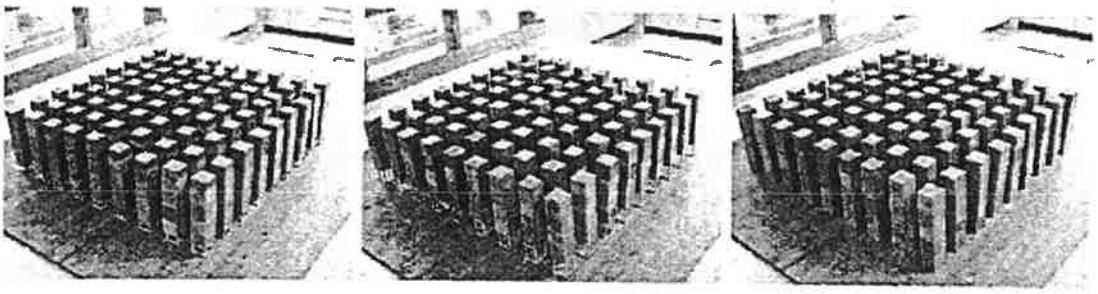
Factors that affect wind : Permeability

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



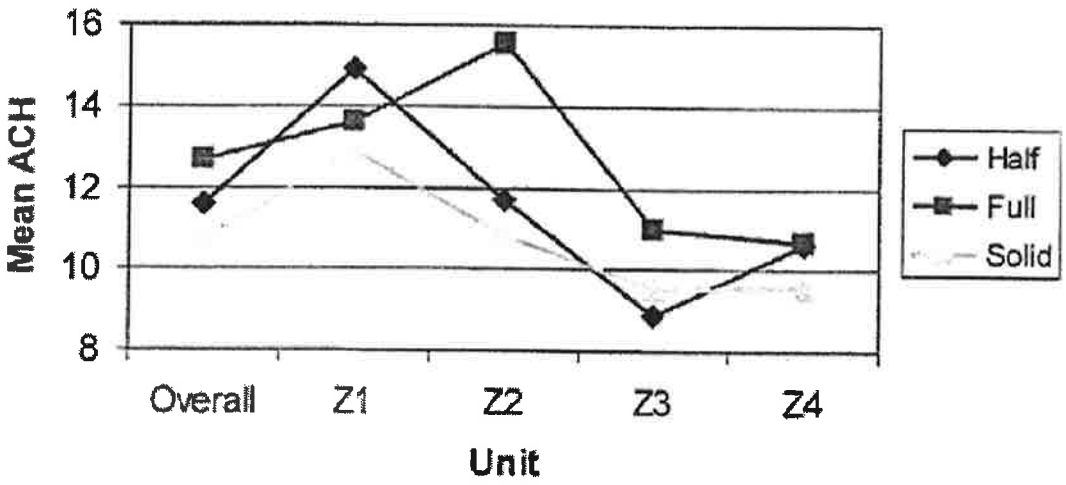
Half Voids | Full Voids | No Void

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

Passive Design & Natural Ventilation, GMP 2015

Factors that affect wind : Permeability

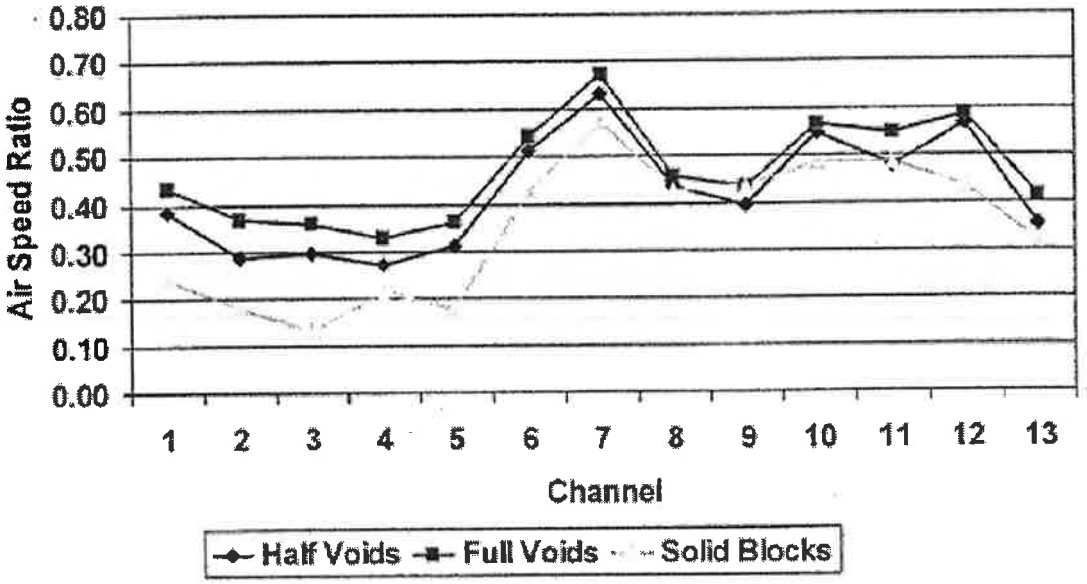
Plain Skyline under different type of voids (air change)



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 : Permeability

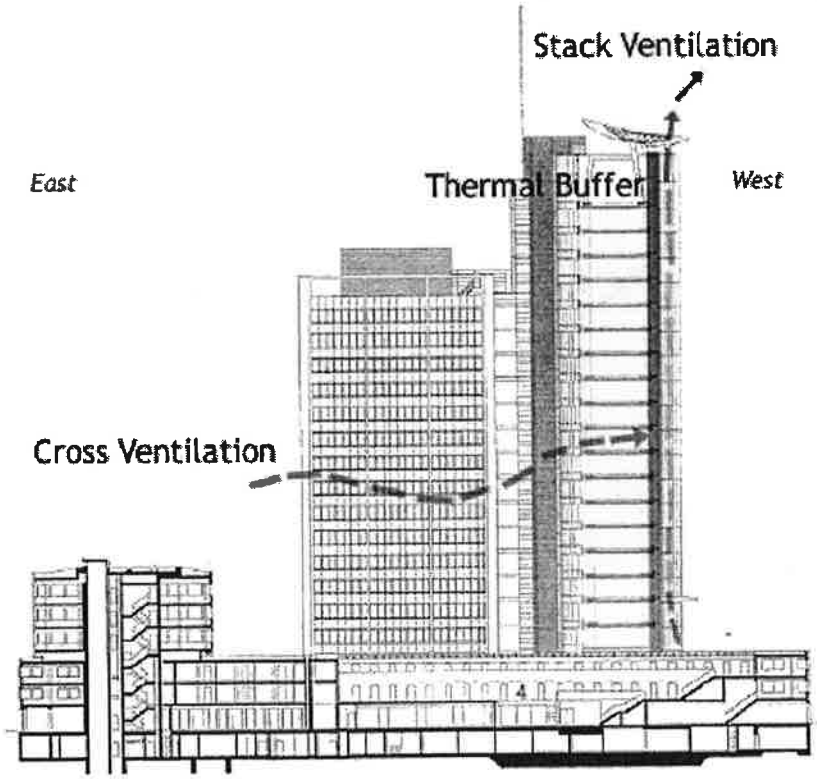
Air speed ratio for different types of voids



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

GSW Headquarters

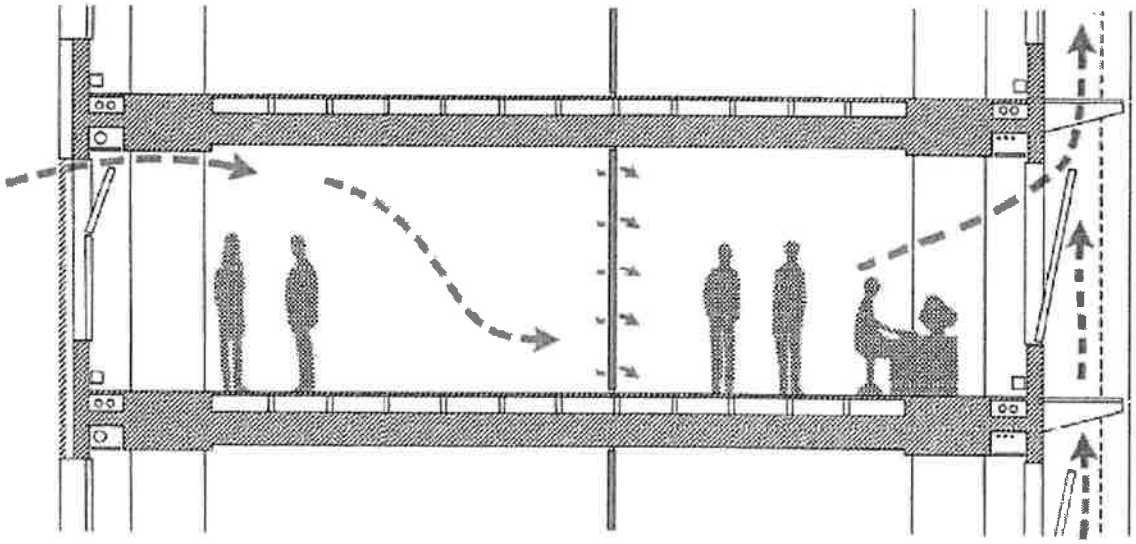
Berlin, 1999, Sauerbruch & Hutton



Factors that affect wind : Internal Layout

GSW Headquarters

Berlin, 1999, Sauerbruch & Hutton



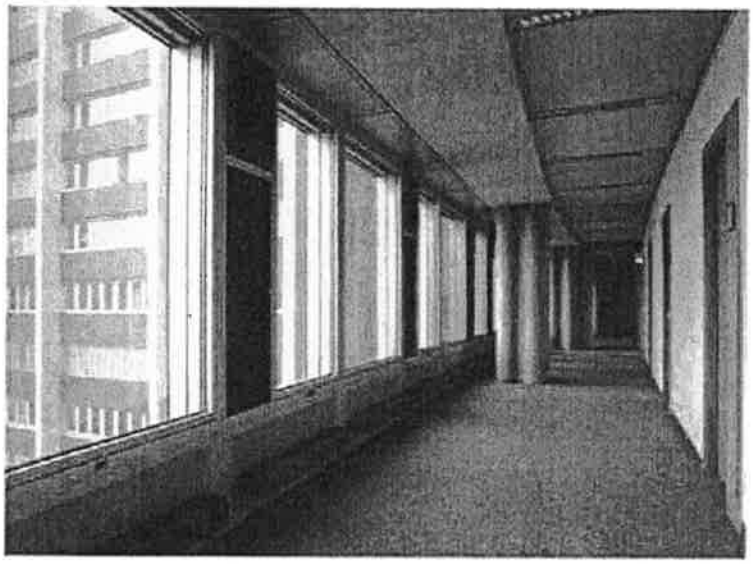
querlüftung - kombi/ost

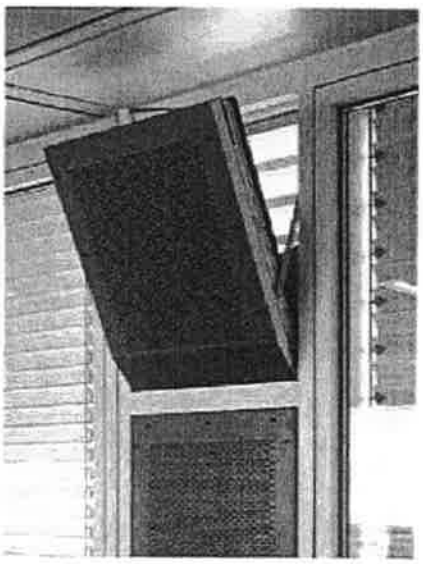
cross ventilation - combi/east

Passive Design & Natural Ventilation, GMP 2015

GSW Headquarters

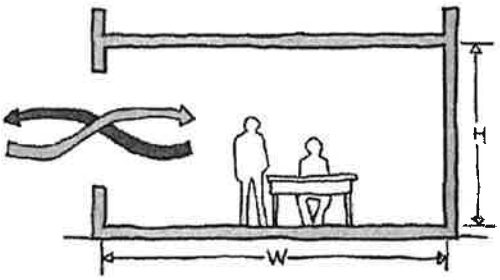
Berlin, 1999, Sauerbruch & Hutton



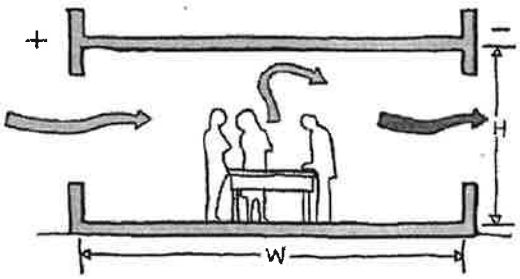


Eastern facade showing fresh air intake

Factors that affect wind : Internal Layout



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



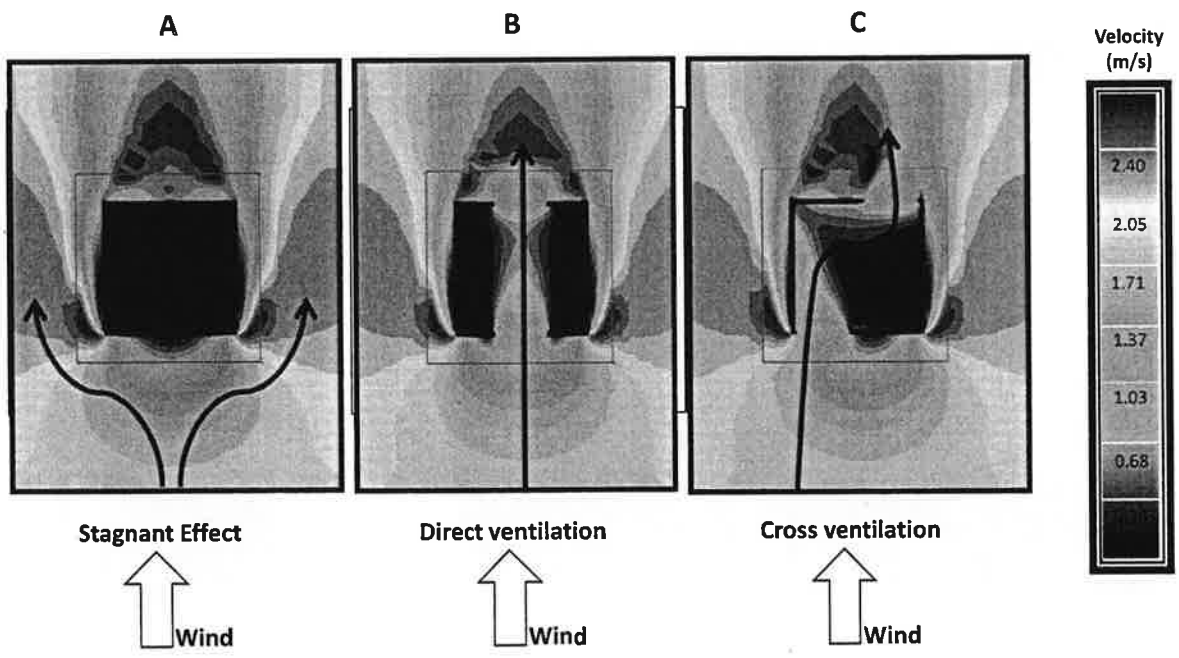
Cross ventilation (openings on opposite or 2 sides) maximum room width (w) = 5 x height (h), limit to 12-14m maximum

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

Source, SGBC 2013

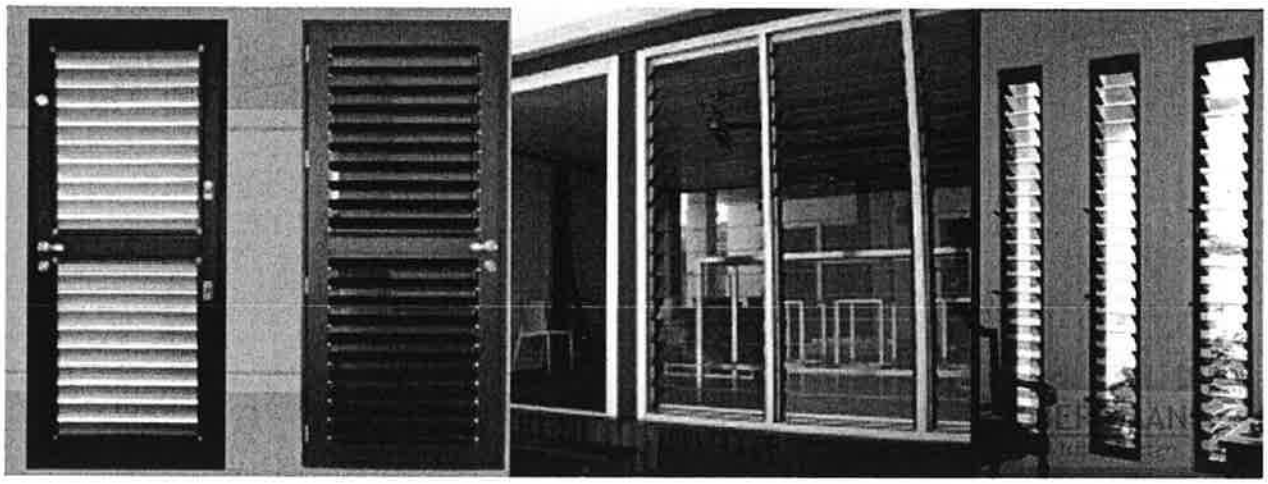
Passive Design & Natural Ventilation, GMP 2015

Factors that affect wind : Internal Layout



Source : Surbana ESD

Factors that affect wind : Internal Layout



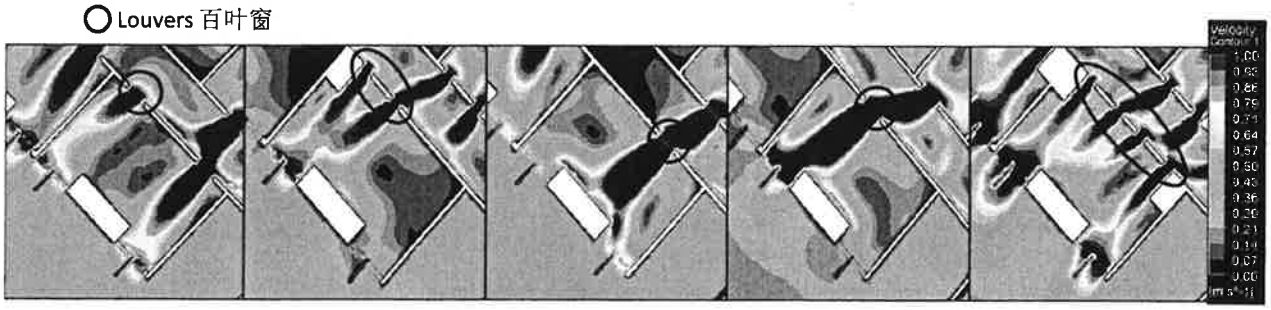
www.lufata.com960 www.pinterest.com236 gzmingqi.en.alibaba.com

Factors that affect wind : Internal Layout

NATURAL VENTILATION ANALYSIS

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

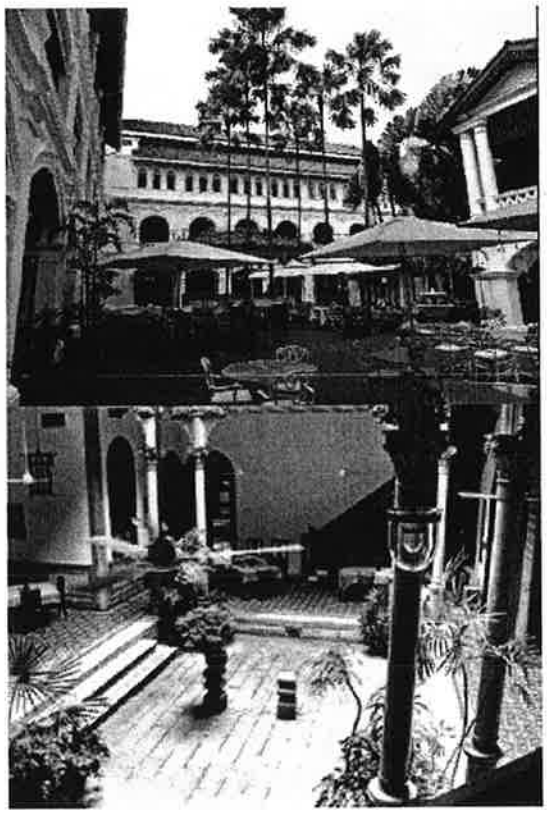
○ Louvers 百叶窗



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

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,

Passive Design & Natural Ventilation, GMP 2015

Courtyard strategy

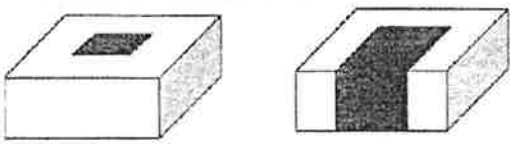
A. CLOSE COURTYARD

- 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 :fie-nut.com

Courtyard strategy

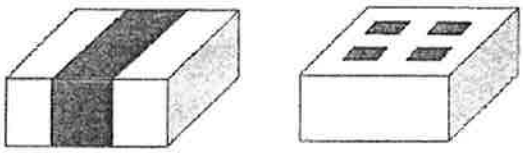
C. LINEAR COURTYARD

- 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 : Ferladi, H., Wong
Image Source : toberlinandbeyond.weebly.com1100

Passive Design & Natural Ventilation, GMP 2015

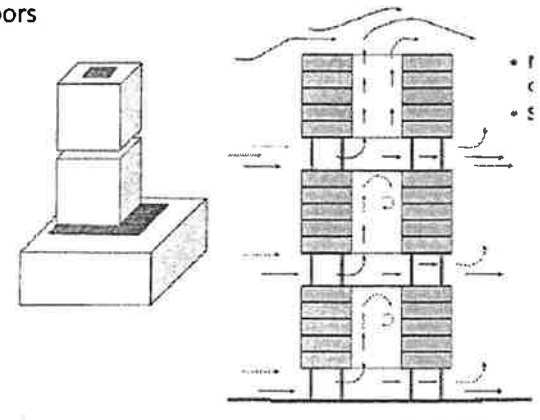
Courtyard strategy

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://memolition.com/



Source : Ferladi, H., Wong

Optimizing Wind in the Tropics

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

-Courtyard can be considered in a city environment which is usually of a higher density

-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)

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

- Providing fresh air for the benefit of the occupants health
- Reducing the concentration of room pollutions
- Contributing to thermal comfort of the building users
- Accelerating passive building mass cooling
- Reducing indirectly building energy consumption

Passive Design & Natural Ventilation, GMP 2015

Purpose of Natural Ventilation

Some considerations for the use of natural ventilation :

- Does it require tight control for the room air conditioning ? (such as: clean room, ICU dll)
- Does it need fresh air ? (such as: kitchen, sport hall, factory etc)
- 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 etc)
- Are there any other considerations due to energy constrains ? (regarding the cost and availability of the electric power)

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.



Passive Design & Natural Ventilation, GMP 2015

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

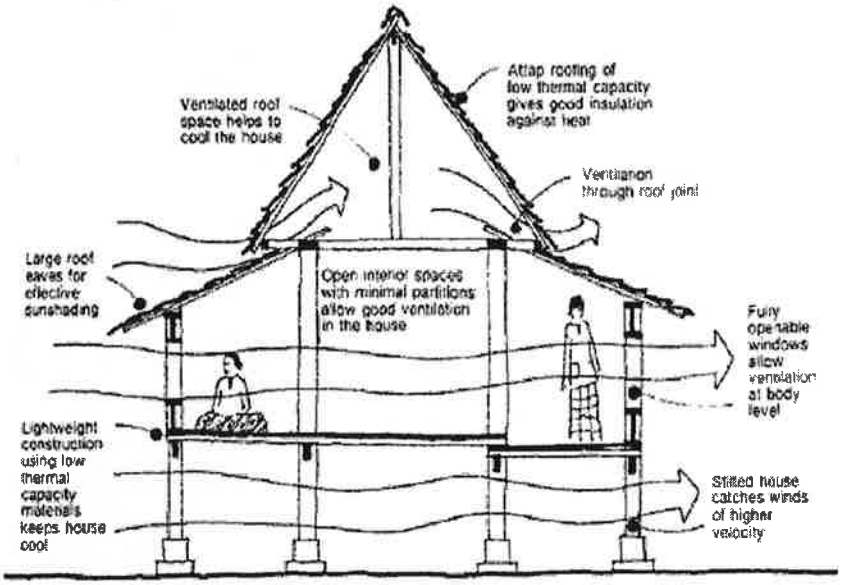
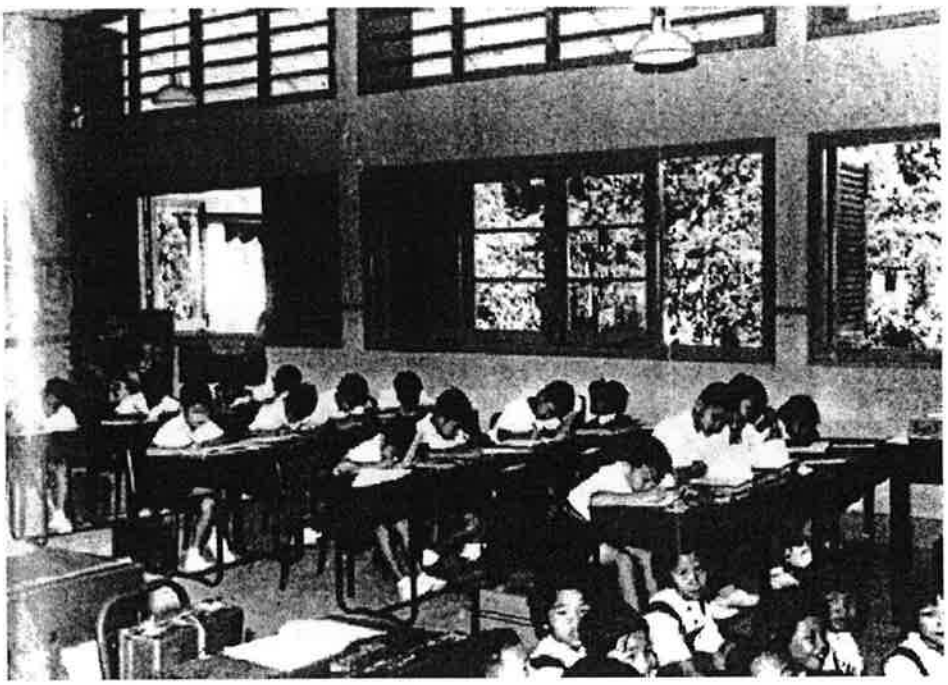


Fig-3.3 Climatic design of a Malay house
Source : Lim Jee Yuan; 1991, pg 80

Passive Design & Natural Ventilation, GMP 2015

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 2015

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

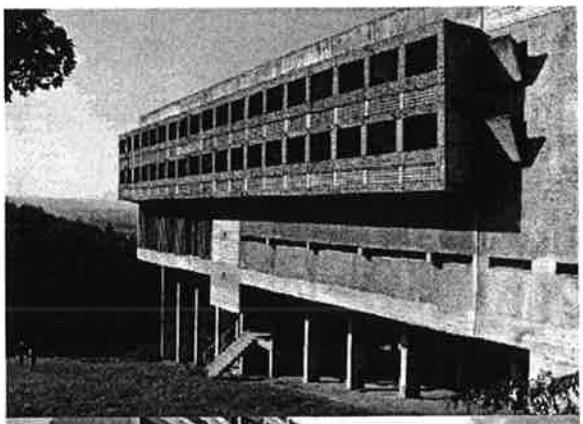


www.world-traveler.eu432

Case studies

Adapting Modern Architecture to the Tropics

Sainte Marie de La Tourette



Le Corbusier's Berlin Unité

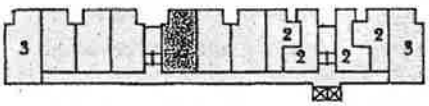


en.wikipedia.org2560

Passive Design & Natural Ventilation, GMP 2015

Case studies

Adapting Modern Architecture to the Tropics



Source:- "thin" 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/

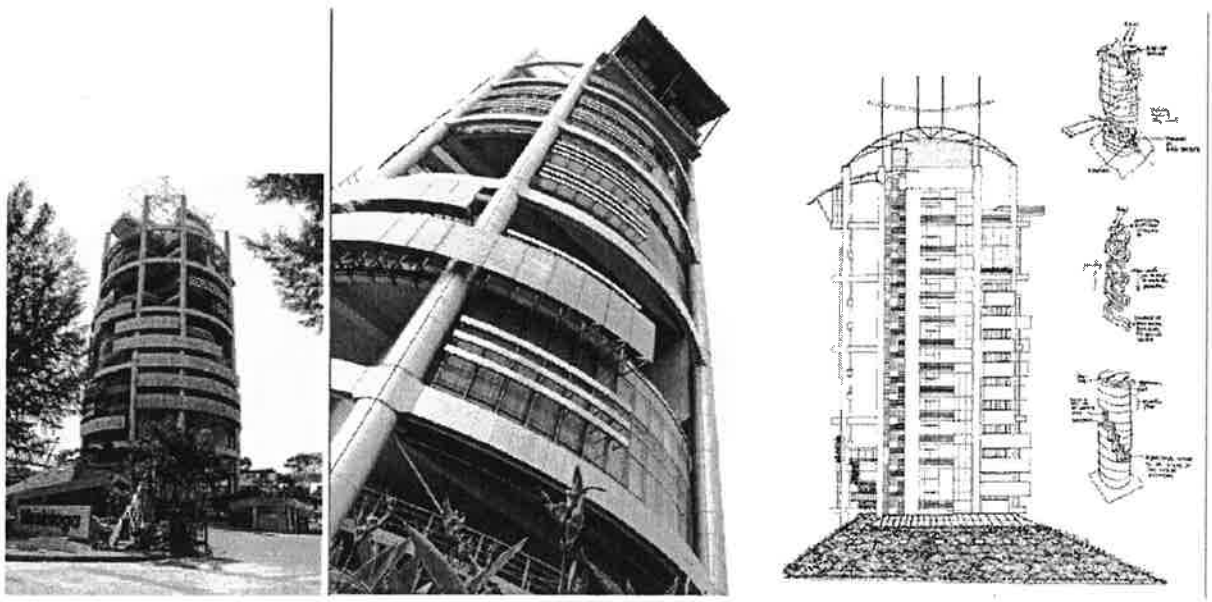
Case studies

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
			

Passive Design & Natural Ventilation, GMP 2015

Case studies

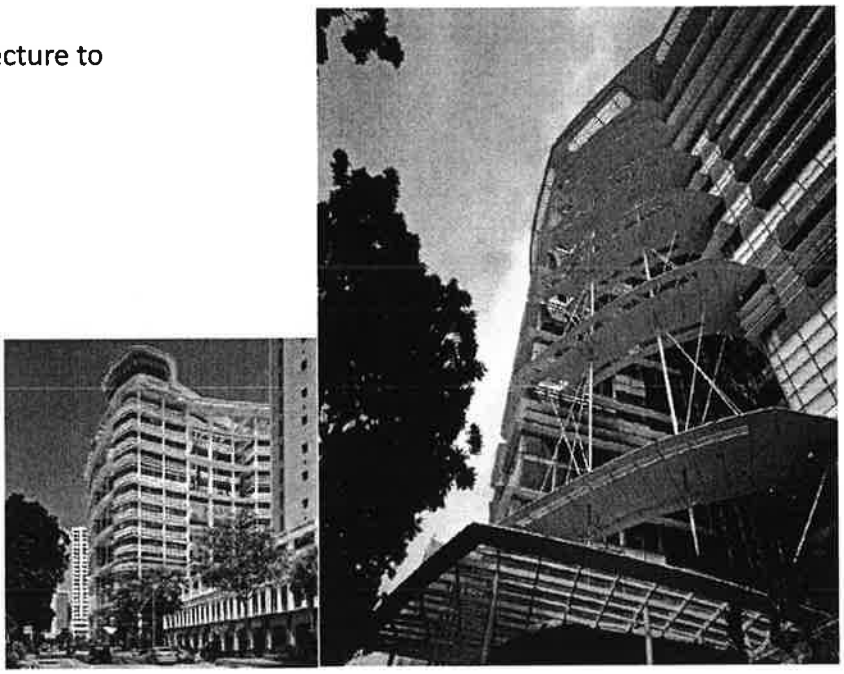
Adapting Modern Architecture to the Tropics



Kenneth Yeang ,Menara Mesin, KL

Case studies

Adapting Modern Architecture to the Tropics



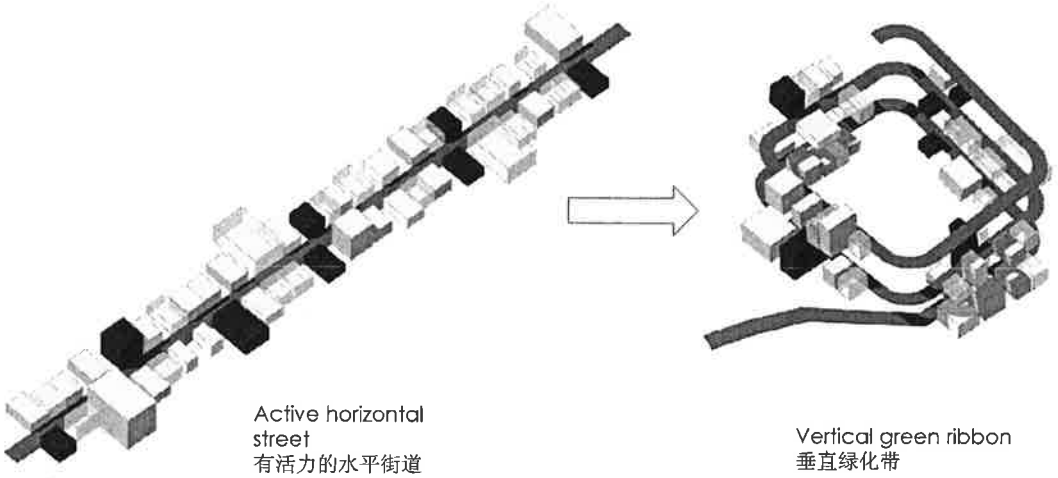
Kenneth Yeang , Singapore National Library

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

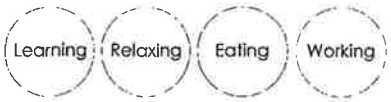
Vertical green ribbon
垂直绿化带

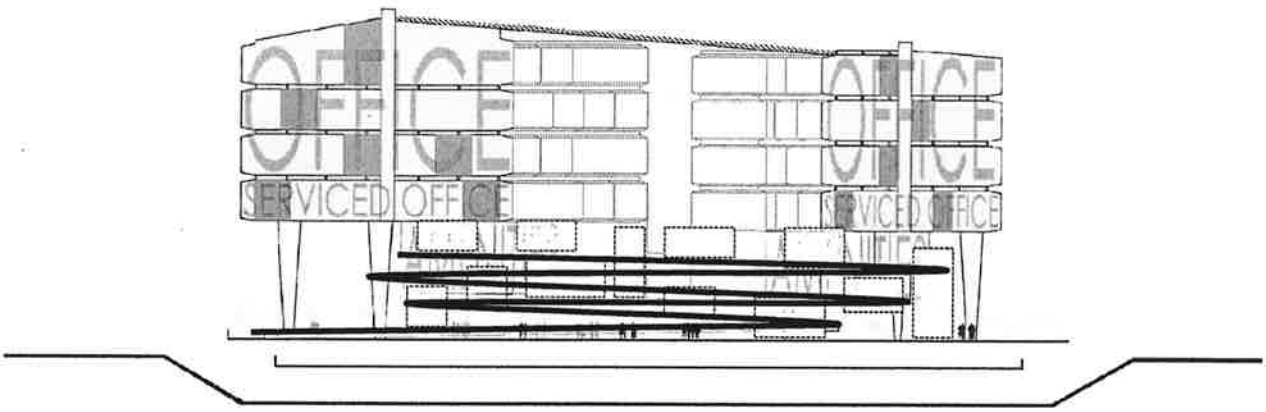
Green Ribbon Gallery
垂直绿化带

Surbana

Passive Design & Natural Ventilation, GMP 2015

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

Surbana

Case studies



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

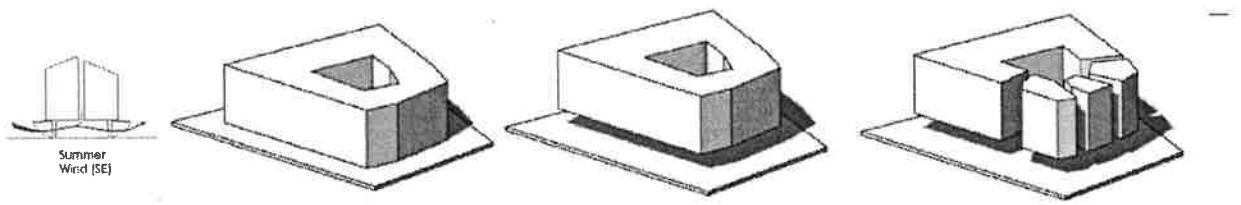
Building Form Analysis
建筑形体设计分析

Surbana

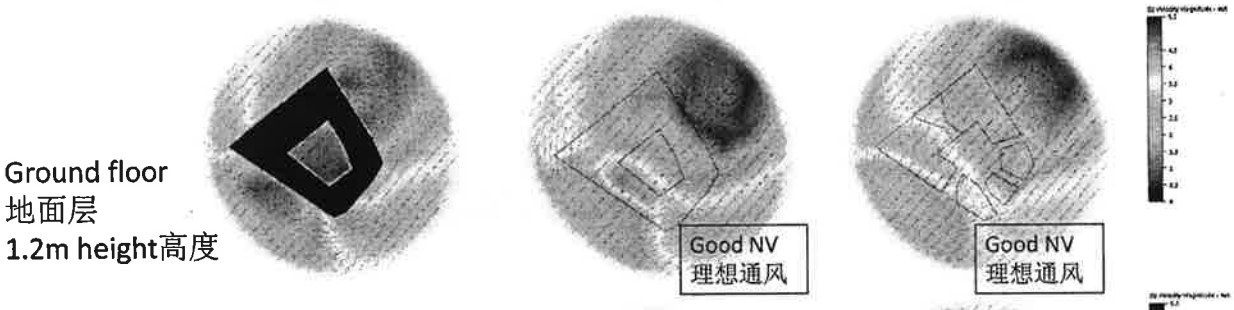
Passive Design & Natural Ventilation, GMP 2015

N

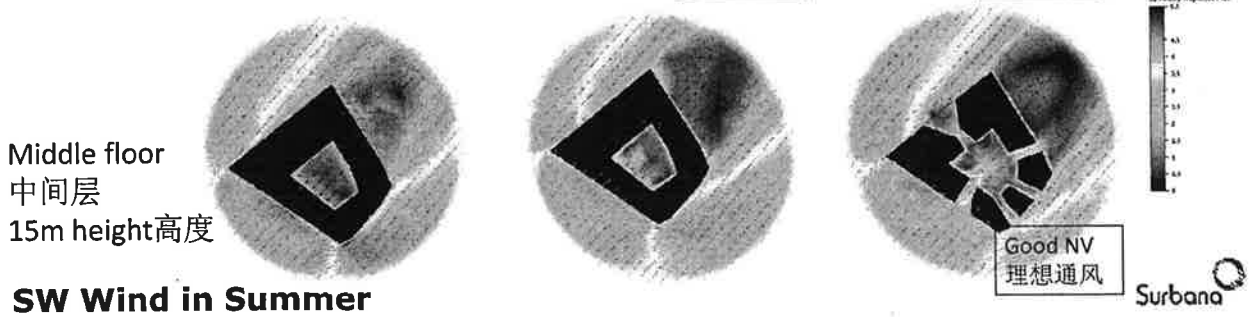
Case studies



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



Ground floor
地面层
1.2m height高度

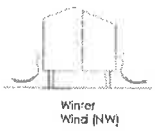


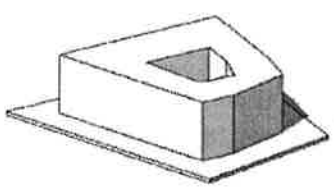
Middle floor
中间层
15m height高度

SW Wind in Summer

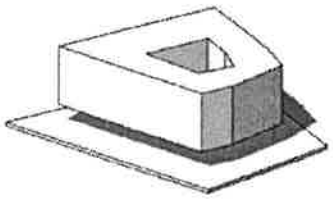
Surbana

Case studies

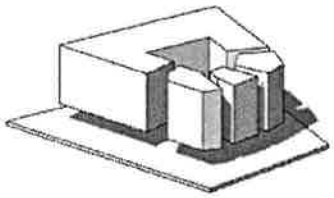




1. Void only中空



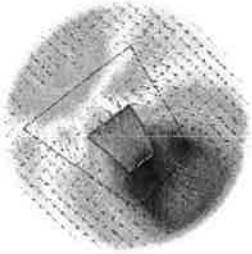
2. Lift up抬高

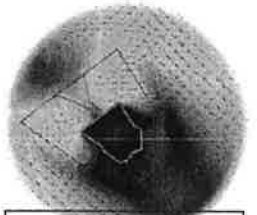


3. Split拆开

Ground floor
地面层
1.2m height高度



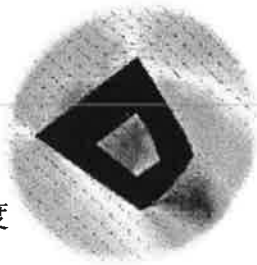




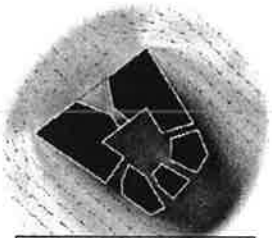
Better heat retention
理想保温功能



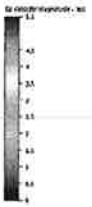
Middle floor
中间层
15m height高度







Better heat retention
理想保温功能



NW Wind in Winter

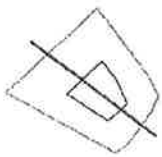
Surbana

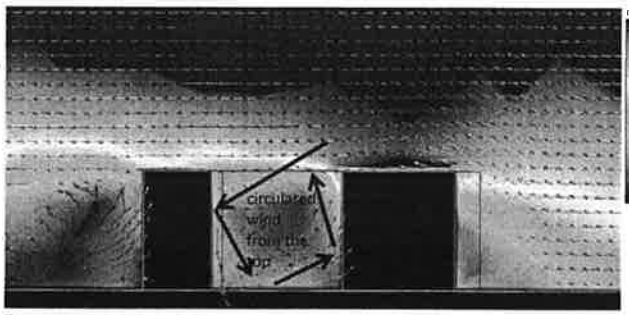
NW Wind in Winter

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

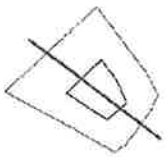
Courtyard gets circulated wind from the top

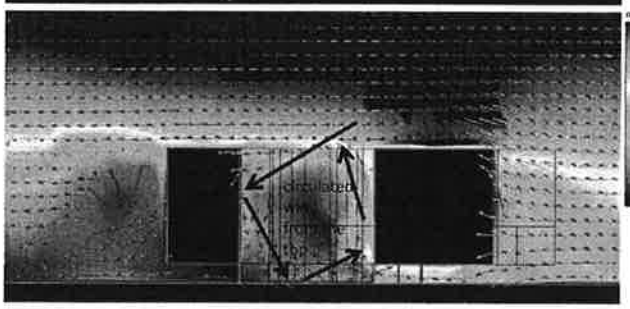




2. Lift up抬高

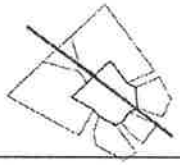
Courtyard gets circulated wind from the top



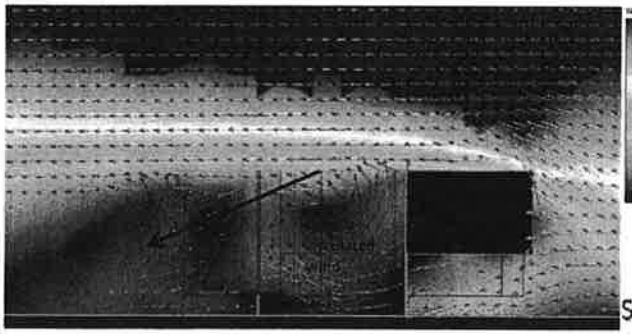


3. Split拆开

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



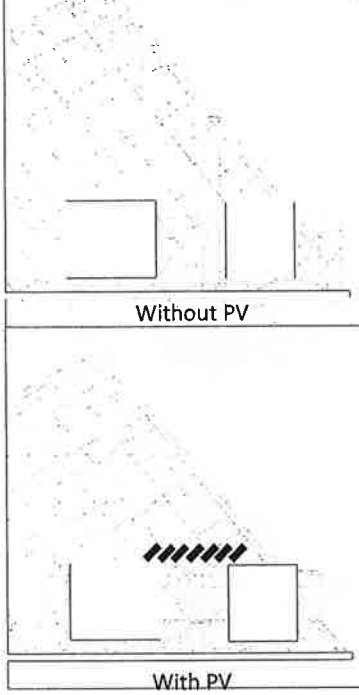
Surbana

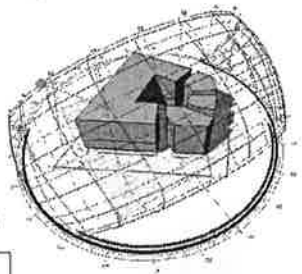
Solar Analysis in Summer夏季日照分析

Case studies

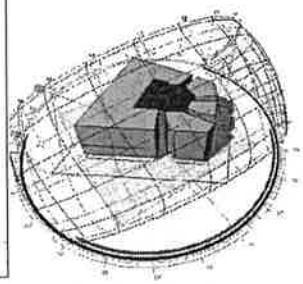
Objectives: to reduce heat gain for court yard

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





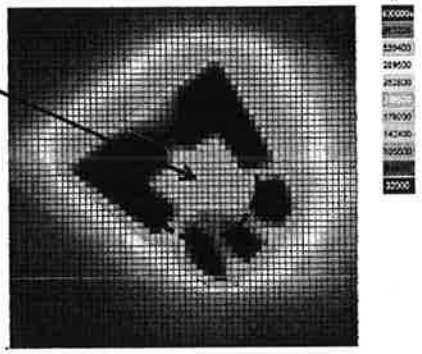
Summer Solstice



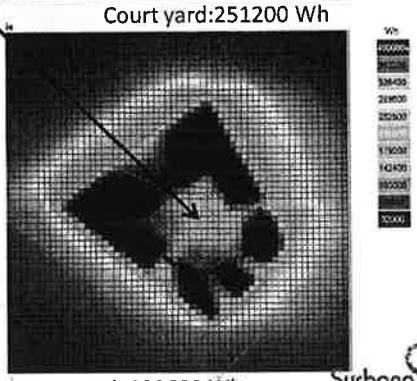
Summer Solstice

Hot area 热点

Reduce **55%** solar heat gain with solar panel



Court yard:251200 Wh

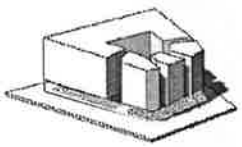


Court yard: 139600 Wh

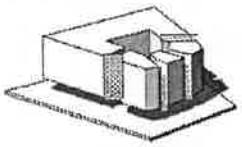
Surbana

Case studies

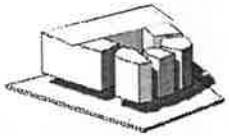
Conclusion 结论



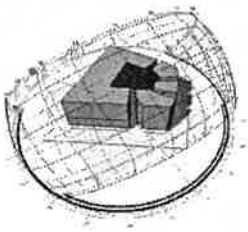
1. Lift up, open space at ground floor
   向上提起，一层作为开放空间



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



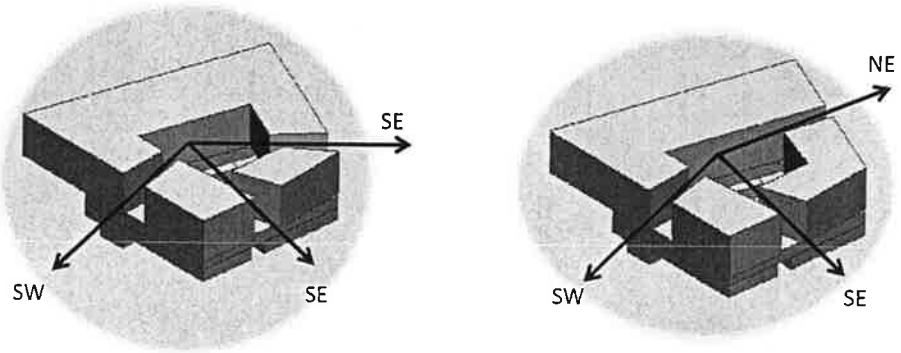
3. Close at NW to prevent cold wind in Winter
   冬季封闭防止冷风进入



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

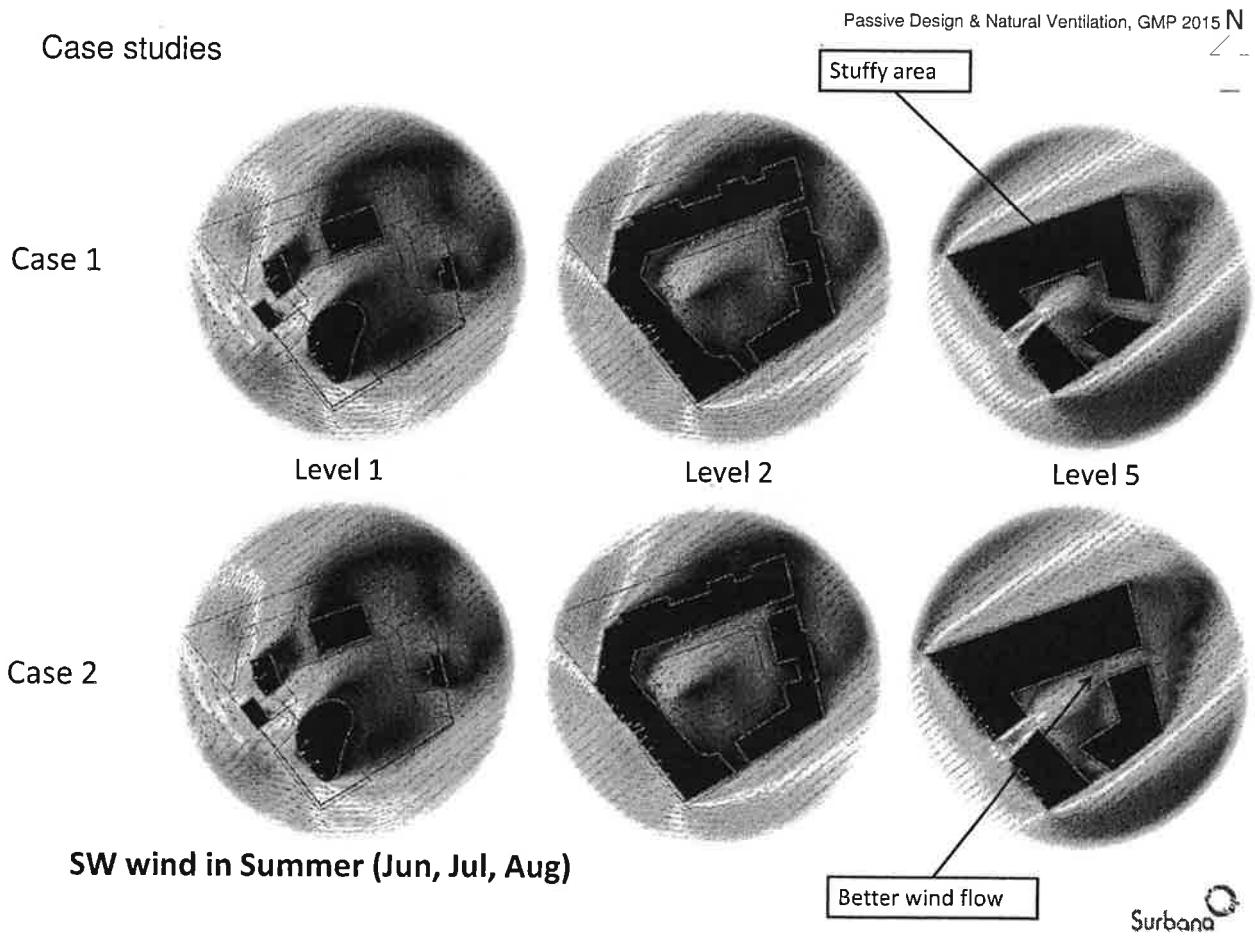
Compare to

Case 2
方案二

Natural Ventilation Analysis
自然通风设计分析

Surbana

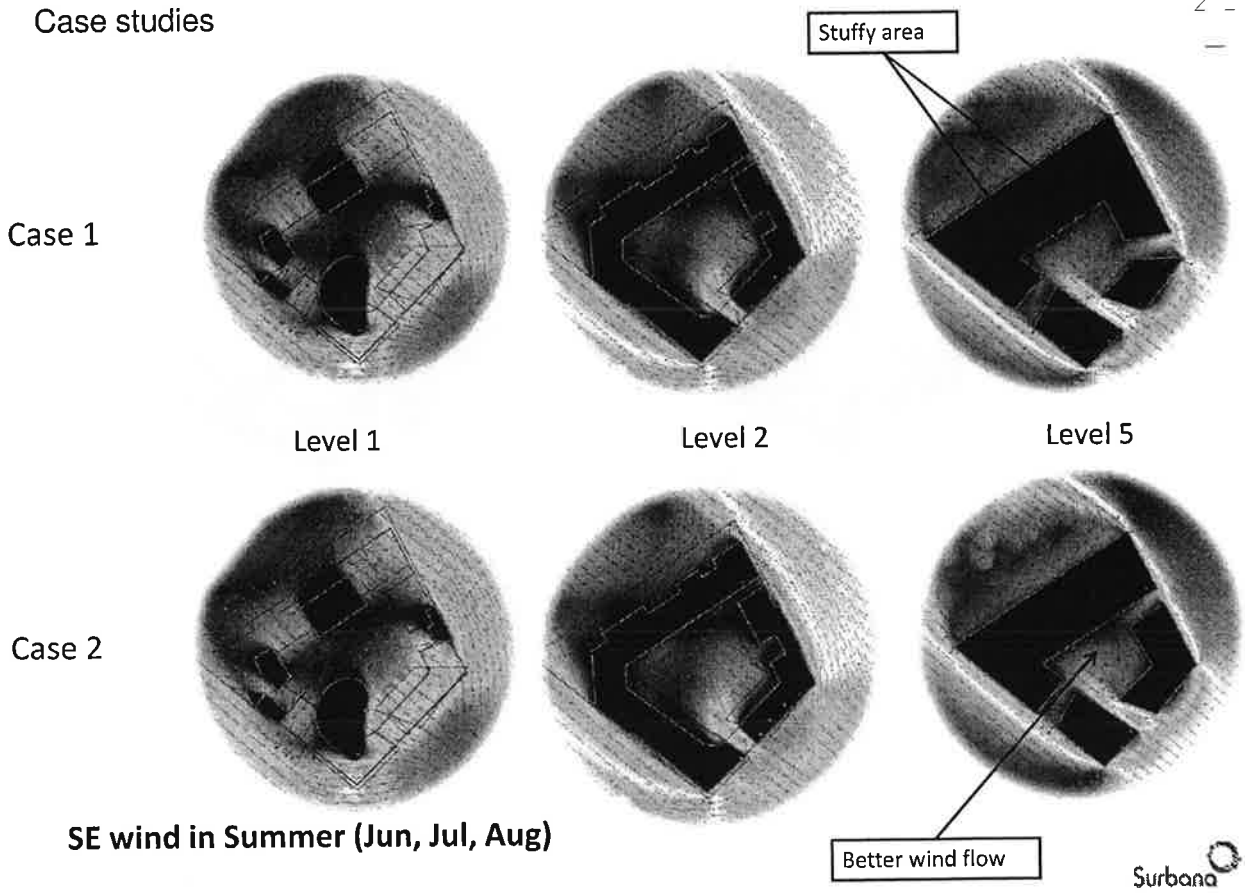
Case studies



SW wind in Summer (Jun, Jul, Aug)

Surbana

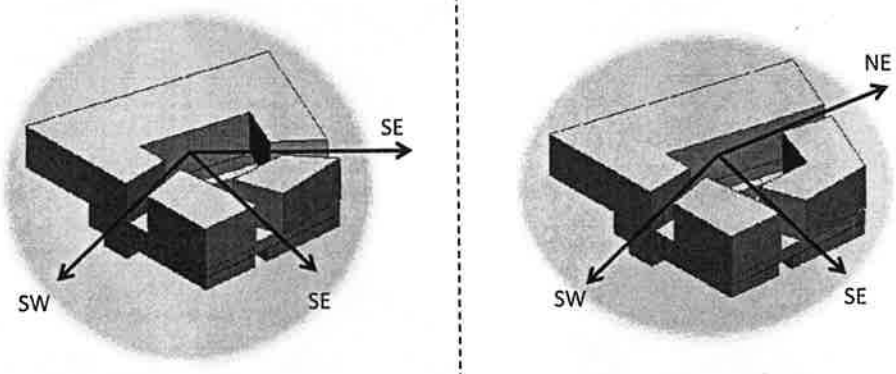
Case studies



Case studies

Conclusion 结论

Case 2 achieves better Natural Ventilation in Summer than Case 1



Case 1
方案一

Case 2
方案二

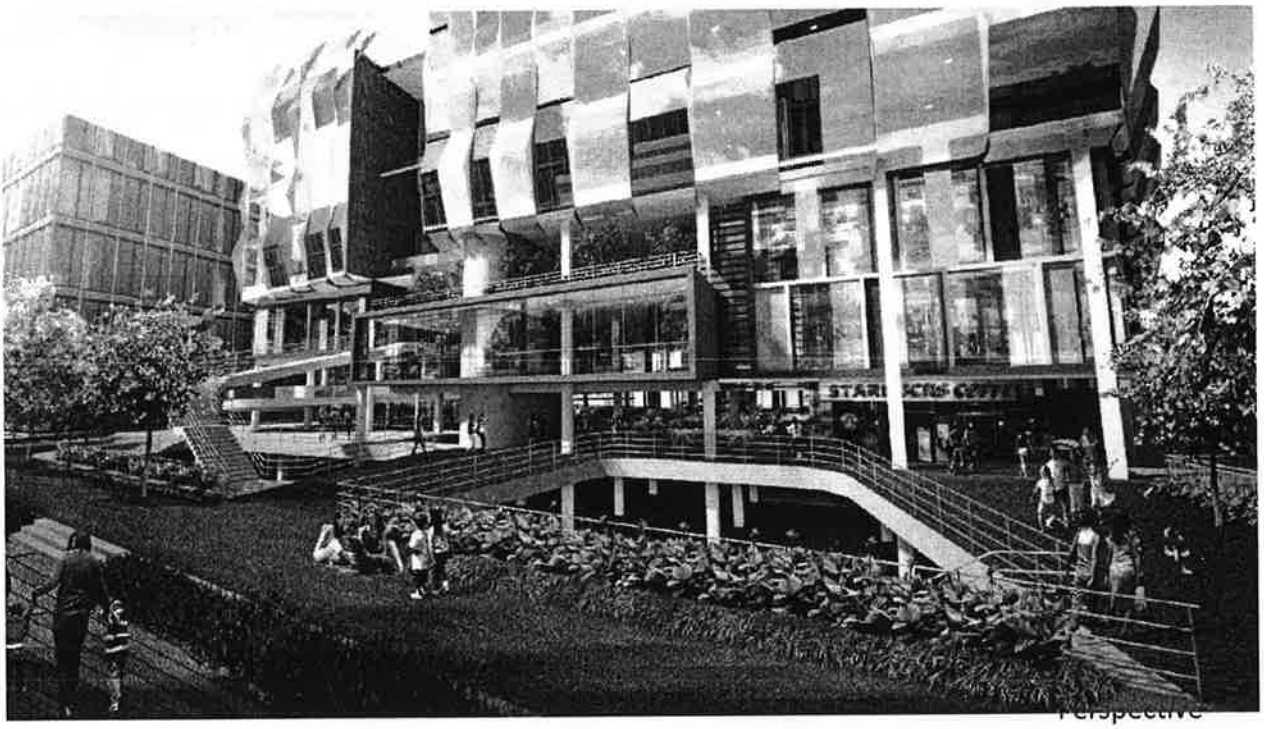
Natural Ventilation Analysis
自然通风设计分析

Surbana



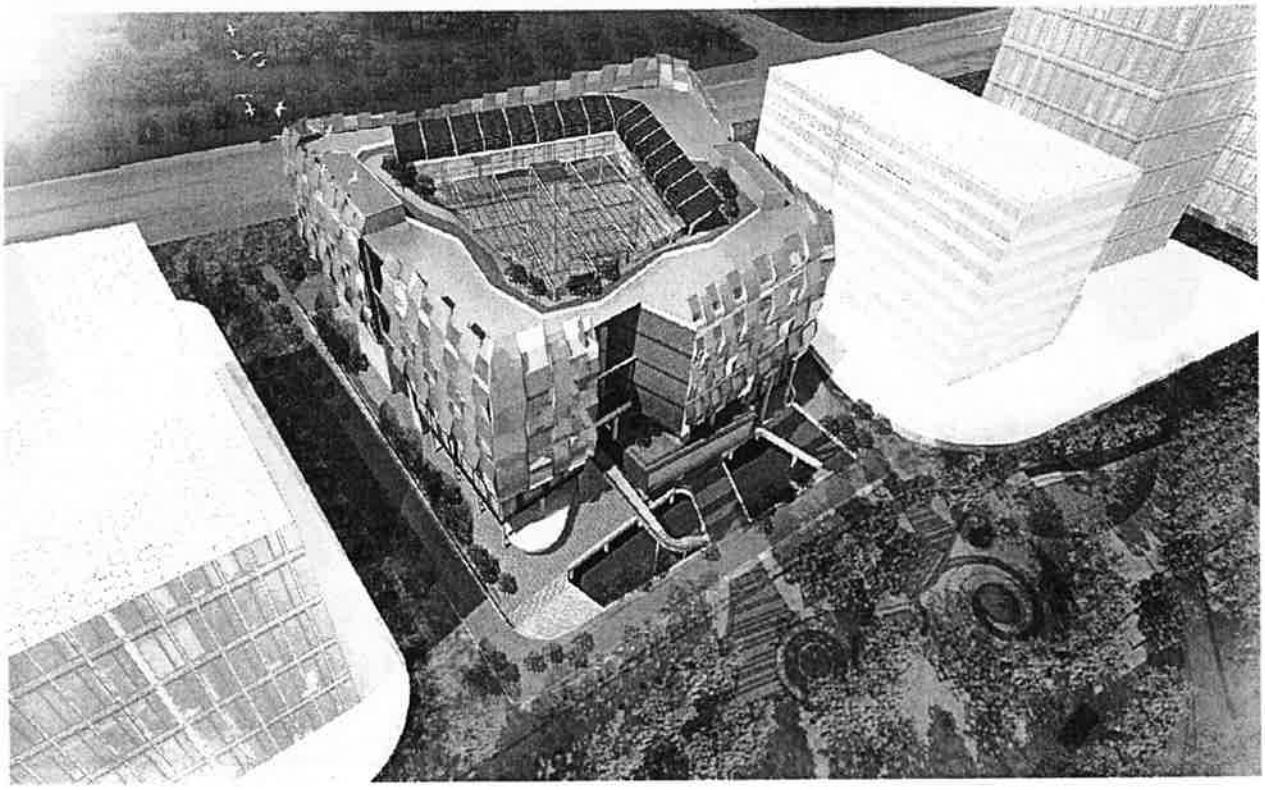
Passive Design & Natural Ventilation, GMP 2015





Perspective
透视意向

Passive Design & Natural Ventilation, GMP 2015



DESIGNING FOR NATURAL VENTILATION

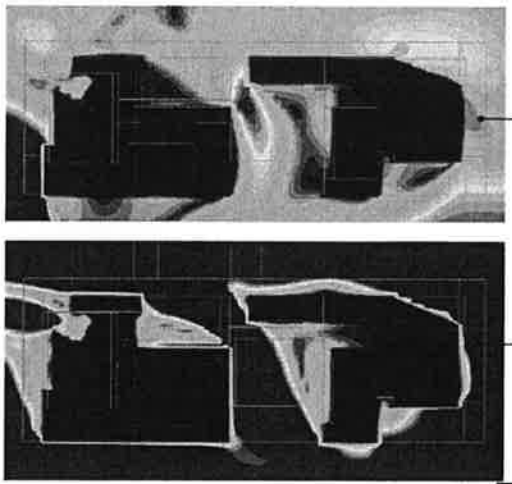
Case 2

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.

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

Level 1 Workshop



Wind Velocity taken at 1.2m above the floor

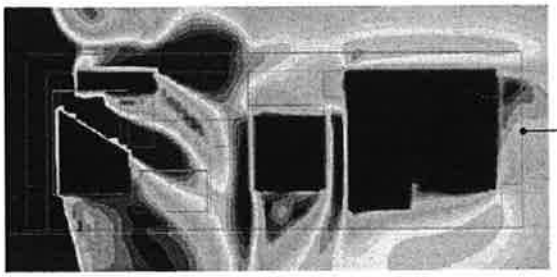
Surbana

Case studies

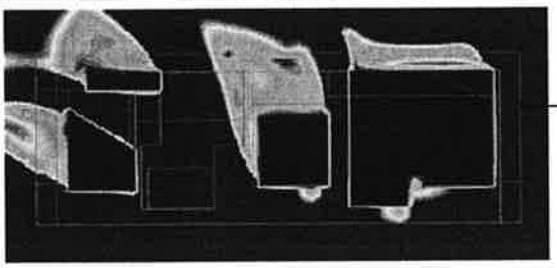
Findings
The Wind Analysis Contour Diagram reveals that Level 2 receives adequate natural ventilation most of the time.

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

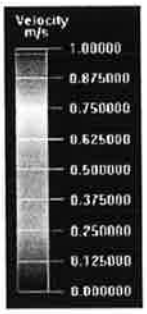
Level 2 Social Plaza



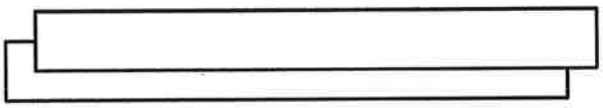
Velocity Contour with Prevailing Wind Direction from NORTH
Wind Velocity at 2.0m/s



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



Wind Velocity taken at 1.2m above the floor



Surbana

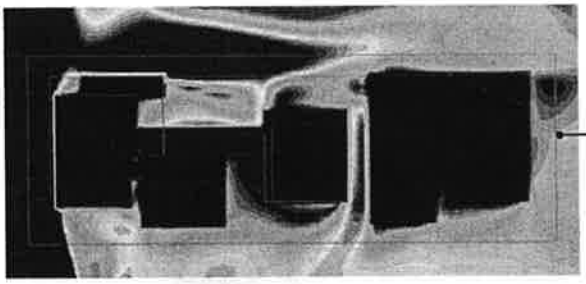
Passive Design & Natural Ventilation, GMP 2015

Case studies

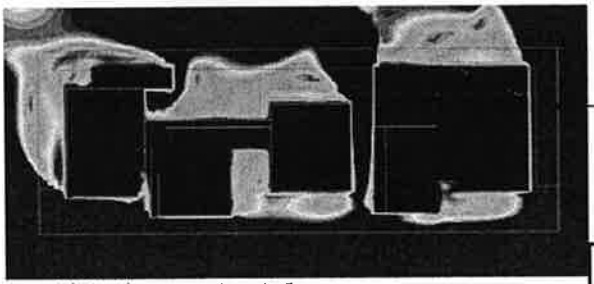
Findings
The Wind Analysis Contour Diagram reveals that generally Level 5 receives adequate natural ventilation.

Recommendation
Additional ventilating solutions may not be necessary.

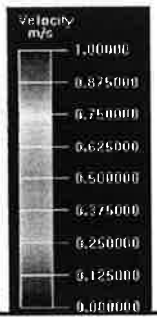
Level 5



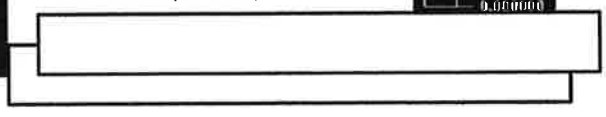
Velocity Contour with Prevailing Wind Direction from NORTH
Wind Velocity at 2.0m/s



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



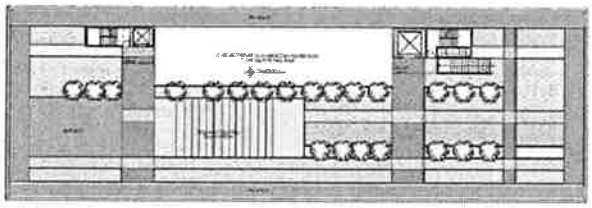
Wind Velocity taken at 1.2m above the floor



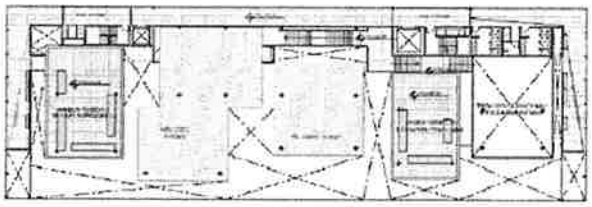
Surbana

Case studies

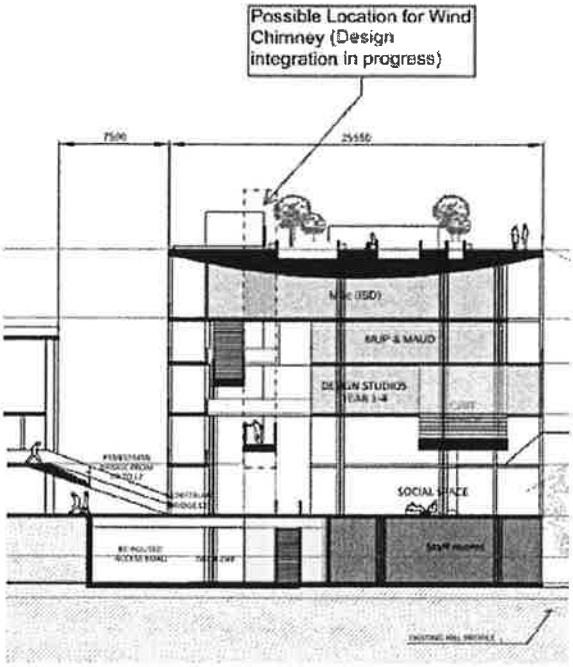
Proposal
Wind Chimney is proposed to help ventilate spaces that are unable to receive much natural ventilation.



ROOF
(FFL 49.55 M)



LEVEL 6
(FFL 44.65 M)

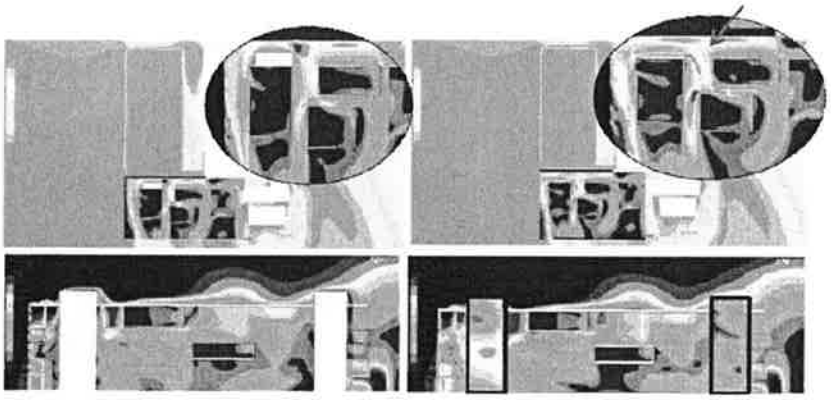


Surbana

Passive Design & Natural Ventilation, GMP 2015

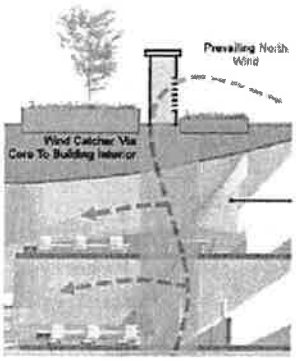
Case studies

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



Surbana

DESIGNING FOR NATURAL VENTILATION

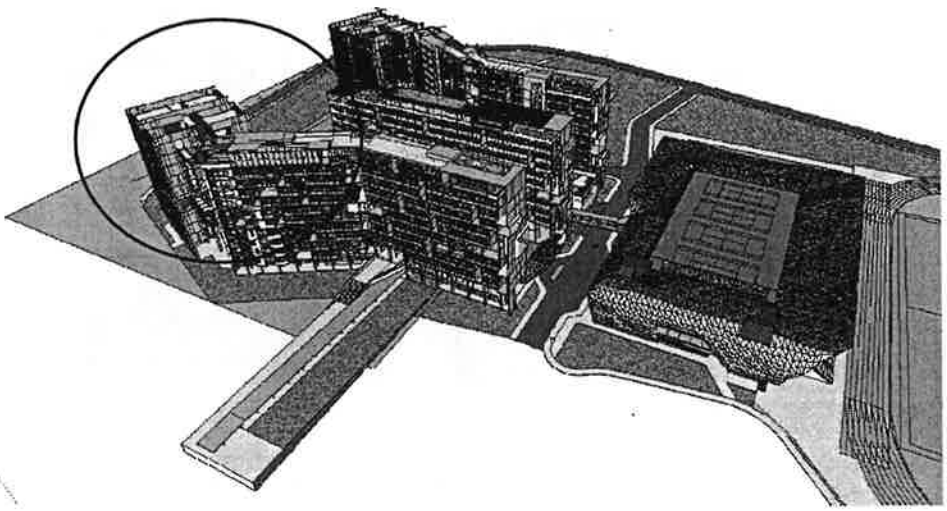
Case 3

Case studies

Building Layout Design

Air movement principle – wind scoop





Surbana LOOK Architects

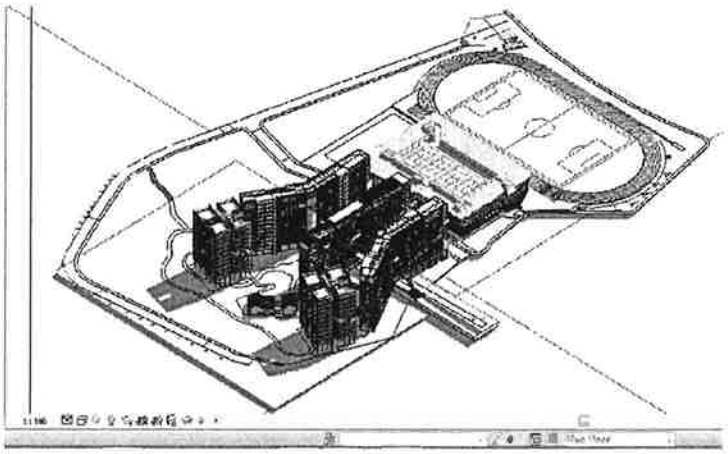
Case studies

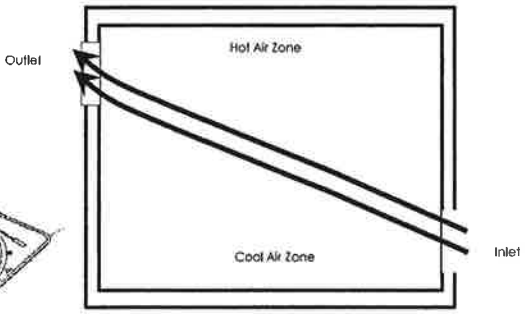
Thermal Comfort Design

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





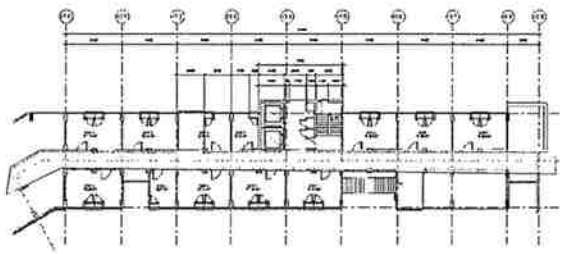
Surbana LOOK Architects

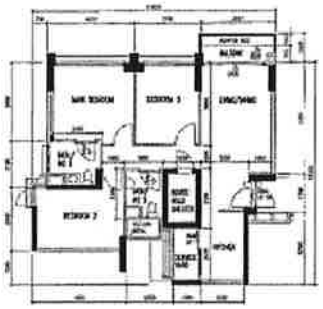
Case studies

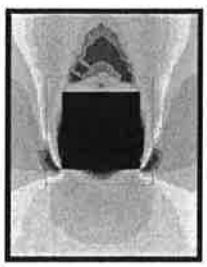
The Challenge

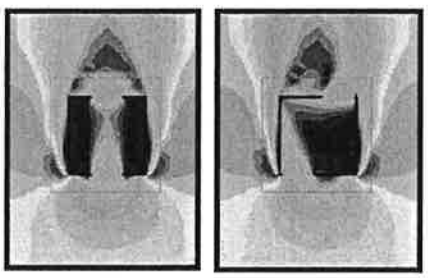
No cross ventilation for the hostel unit

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





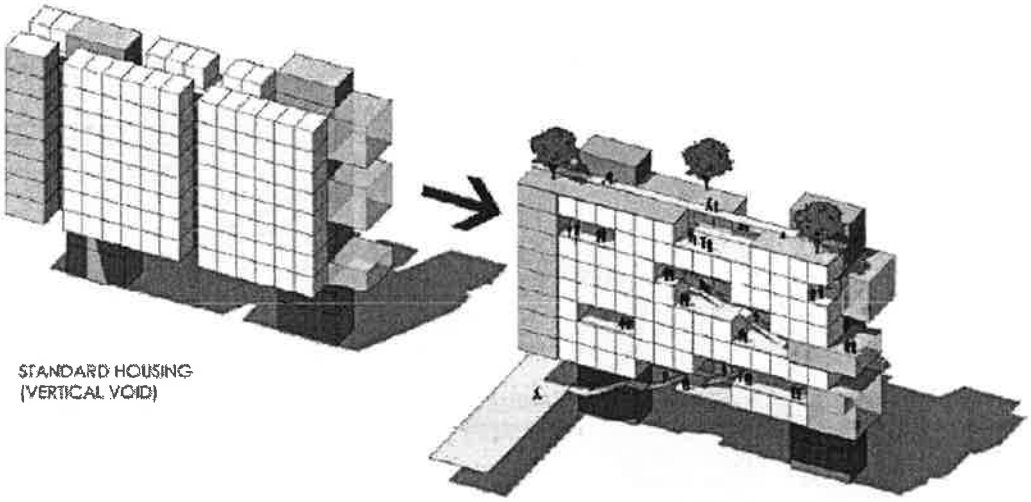




Surbana LOOK Architects

Case studies

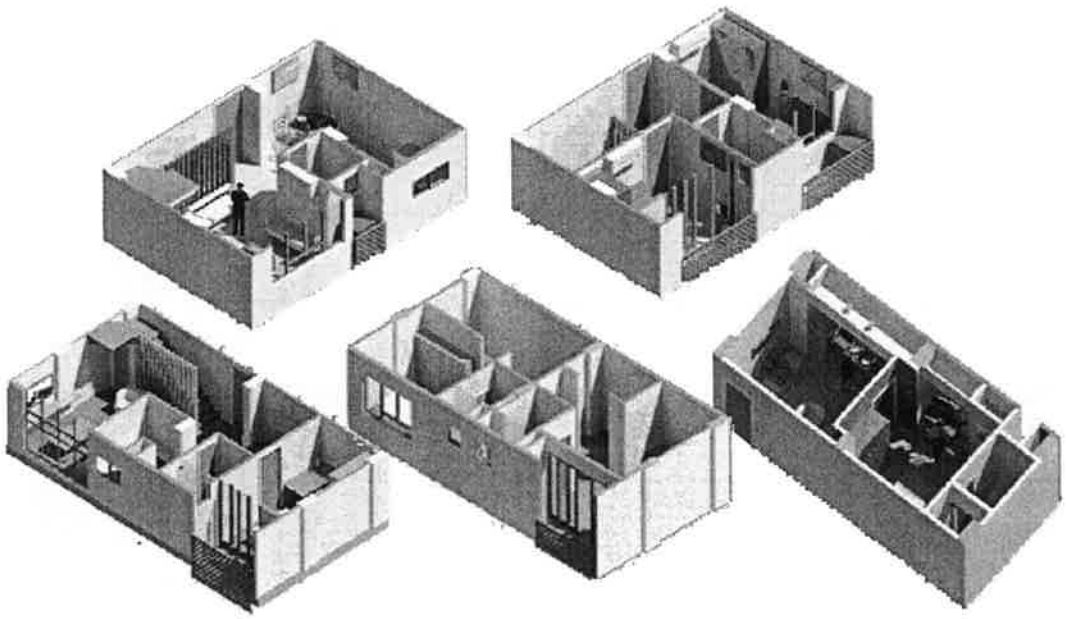
Create Openings



Surbana LOOK Architects

Case studies

Unit Design



Surbana LOOK Architects

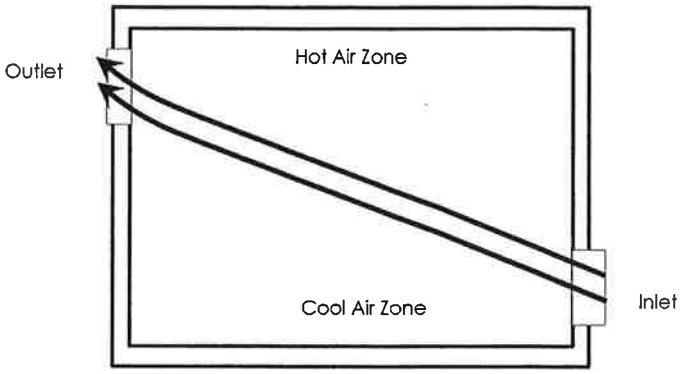
Case studies

Step One – Allow more air in

Air movement principle – wind scoop



Air movement principle – hot air rises

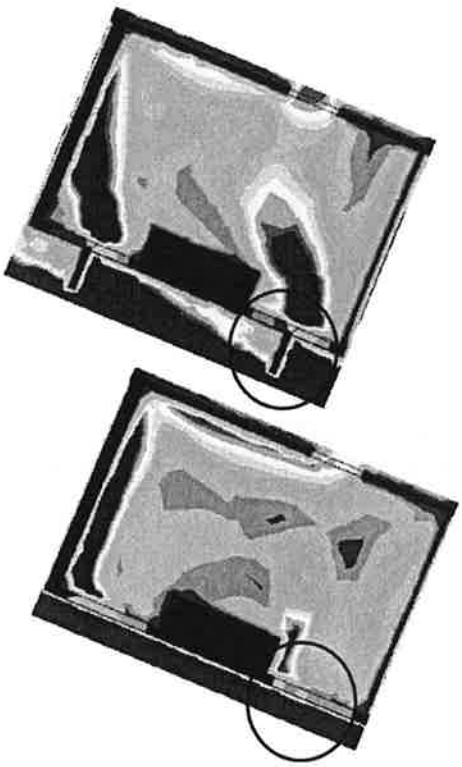


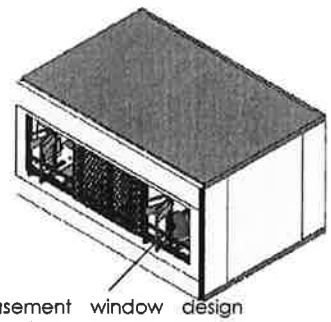
Surbana

Passive Design & Natural Ventilation, GMP 2015

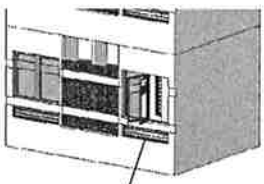
Case studies

Step One – Allow more air in

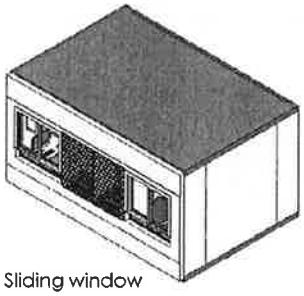




Casement window design as "Wind scoop" to increase the wind speed intake into the units



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



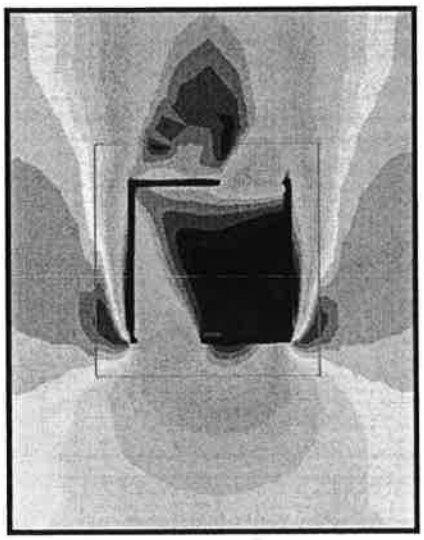
Sliding window

Surbana

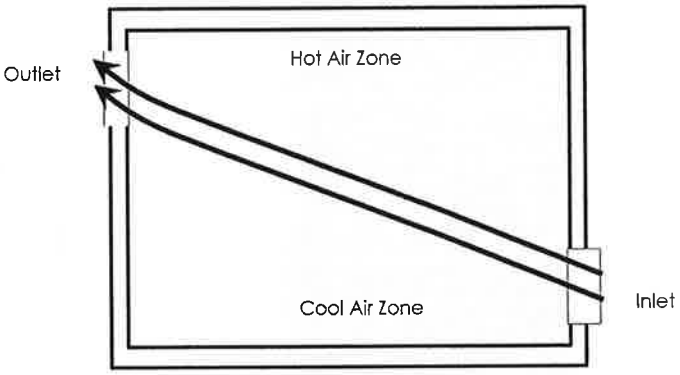
Case studies

Step Two – Allow more air out

Air movement principle – cross ventilation



Air movement principle – hot air rises

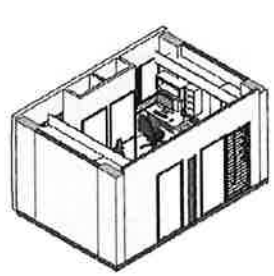


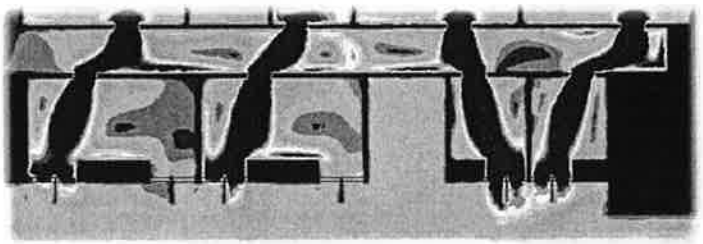
Surbana

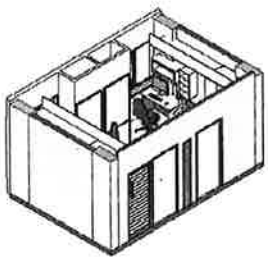
Case studies

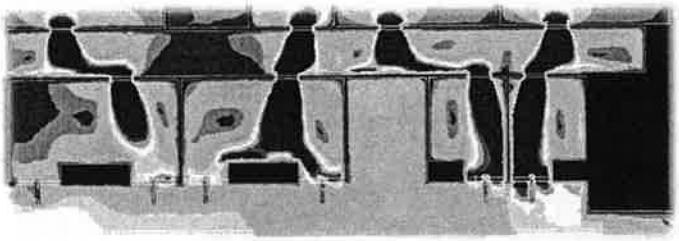
Step Two – Allow more air out

Casement window and 1 large louver on either side of UGD







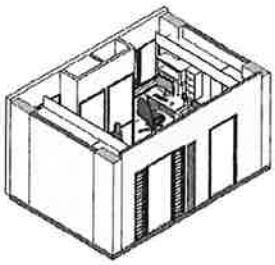


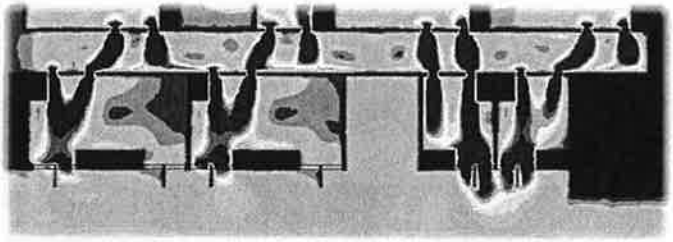


Surbana

Case studies

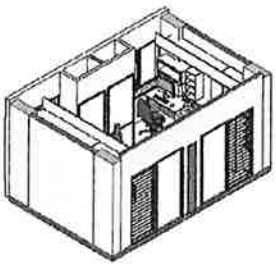
Step Two – Allow more air out

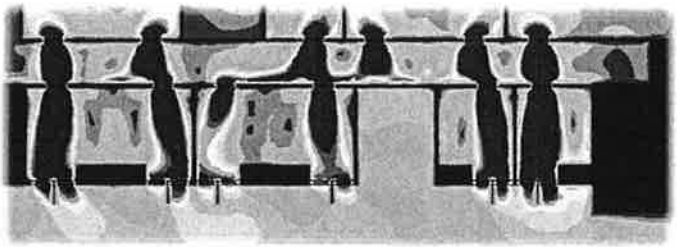






Casement window and 1 set of small double louvers on door





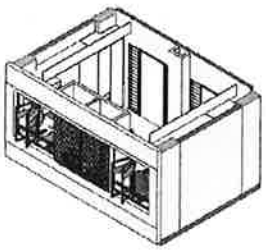
Casement window and One set of large louver by each door. Total 2 sets of louvers

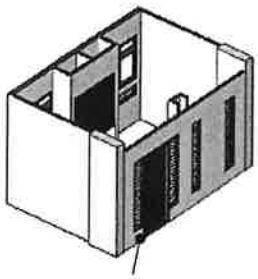
Surbana

Passive Design & Natural Ventilation, GMP 2015

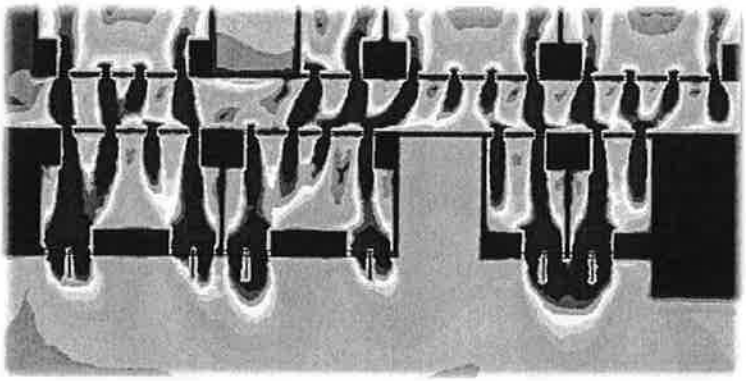
Case studies

Step Two – Allow more air out





Louver design to allow cross ventilation





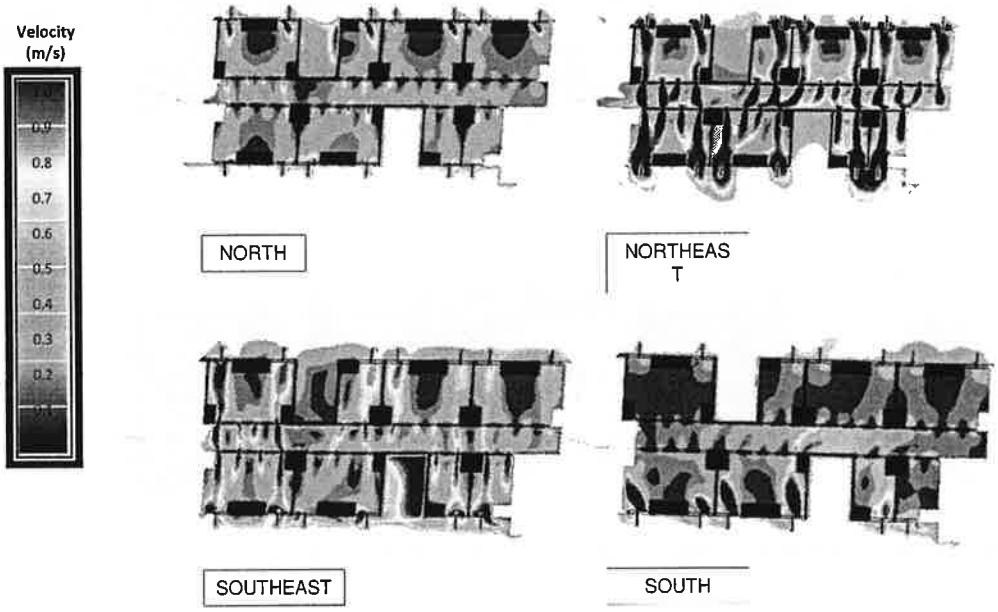
Casement window with 2 sets of double louvers

Surbana

Case studies

Natural Ventilation Results

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



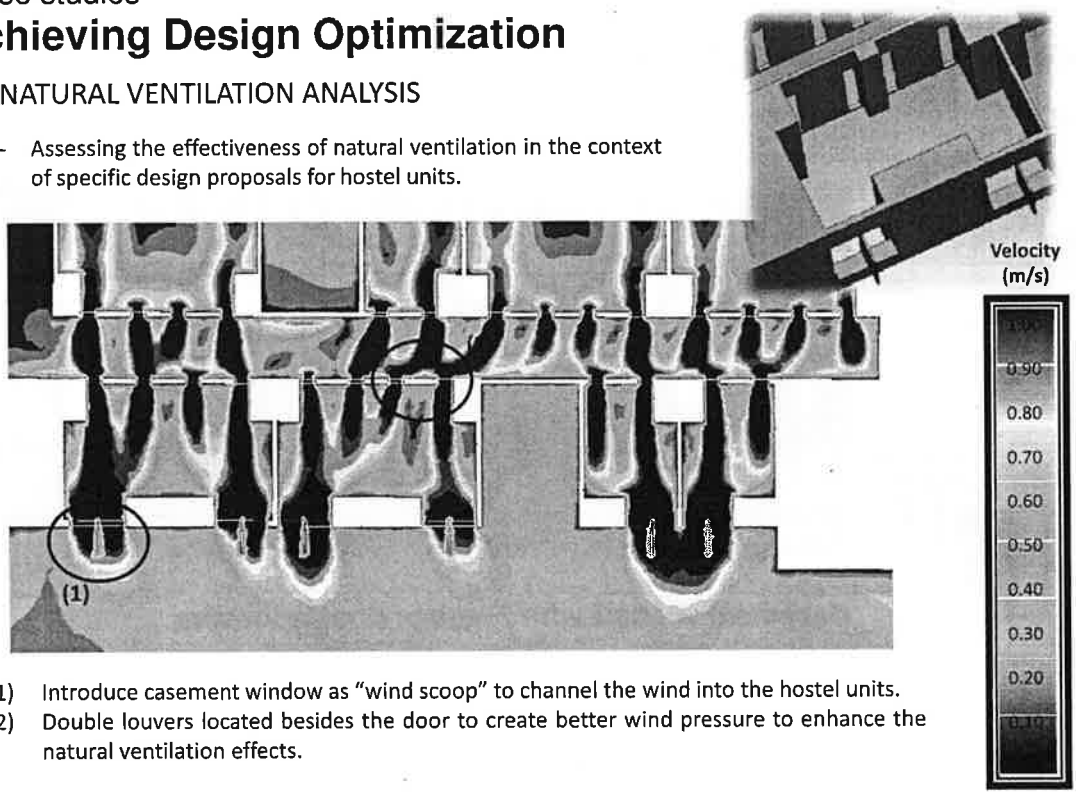
Surbana

Case studies

Achieving Design Optimization

NATURAL VENTILATION ANALYSIS

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

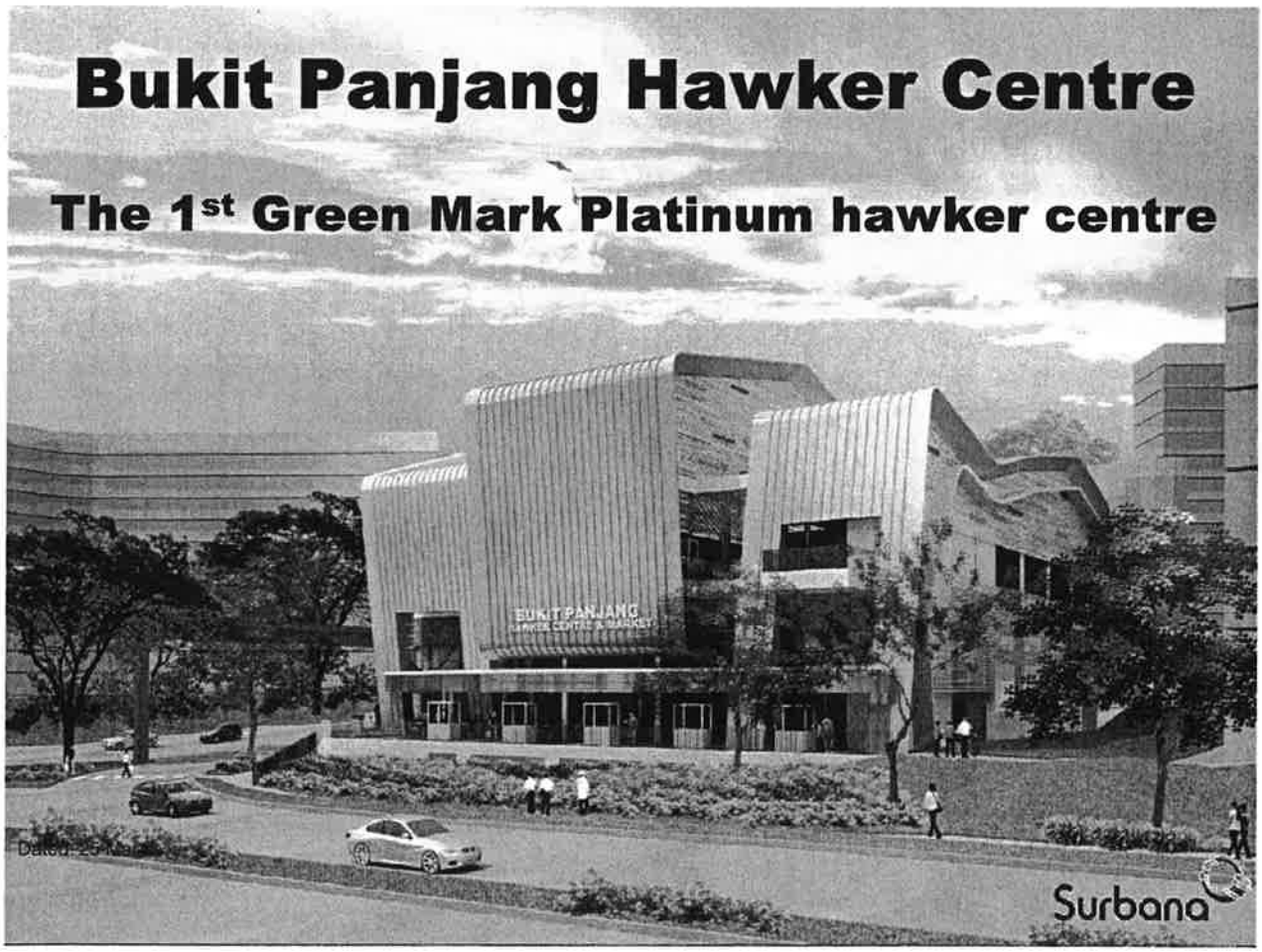


1) Introduce casement window as "wind scoop" to channel the wind into the hostel units.
2) Double louvers located besides the door to create better wind pressure to enhance the natural ventilation effects.

Surbana

DESIGNING FOR NATURAL VENTILATION

Case 4



Case studies

1st Storey Plan

28 nos. of Cooked Food Stalls

Comprising of :

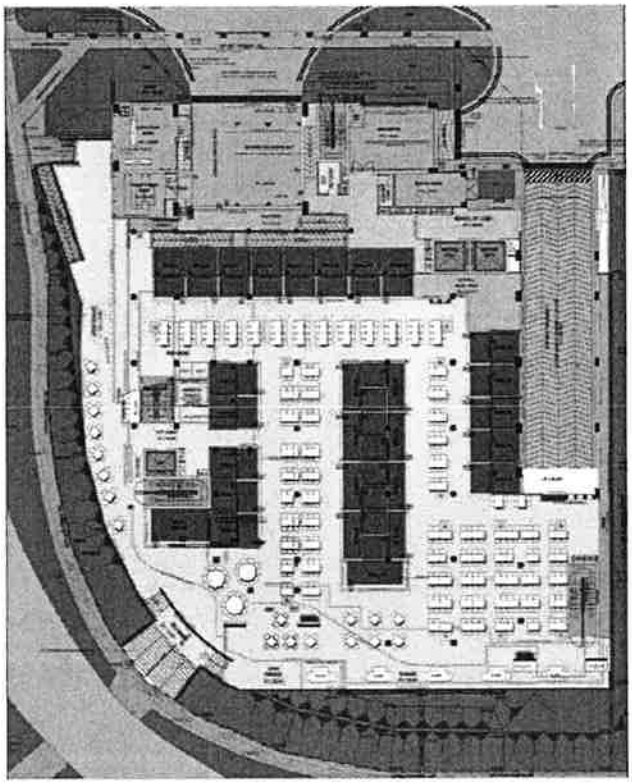
- 1 Drink Stall
- 1 Zi Char Stall
- 26 nos of cooked food Stalls

Seatings Capacity

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

Seating ratio to Stalls = 1:18.1

(Requirement = 1 :18)



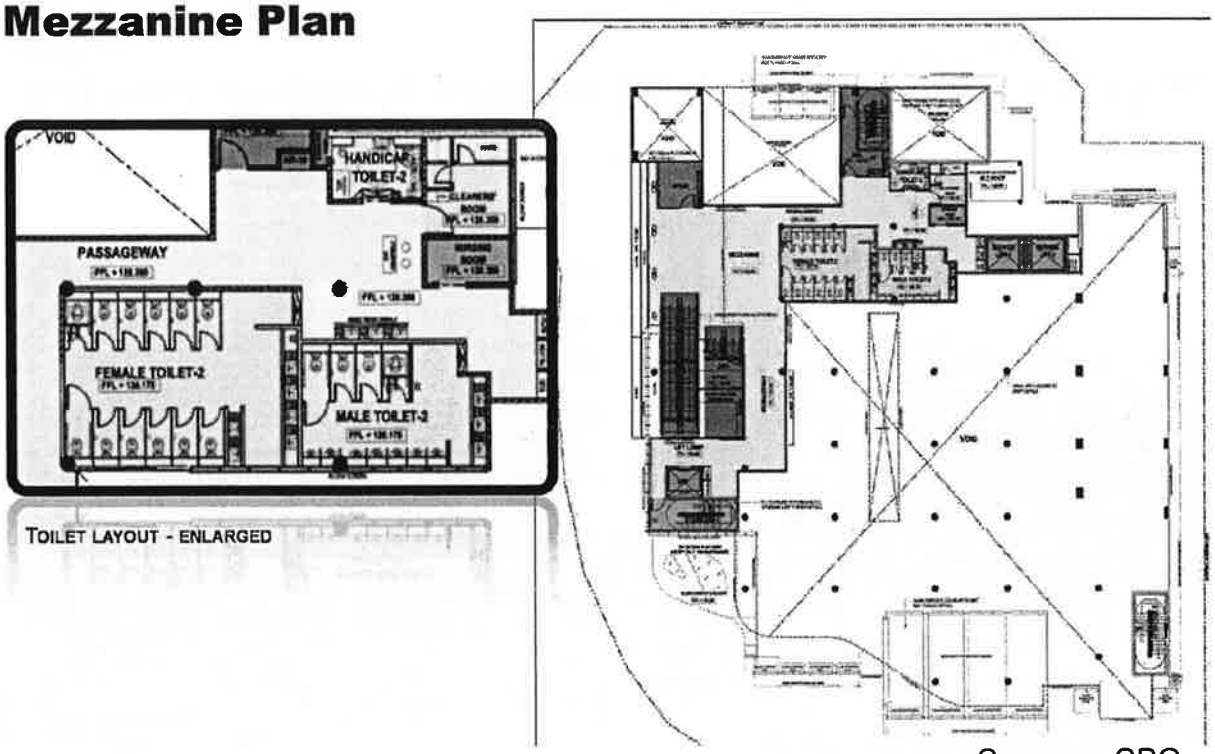
Source :- CPG

Surbana

Passive Design & Natural Ventilation, GMP 2015

Case studies

Mezzanine Plan



Source :- CPG

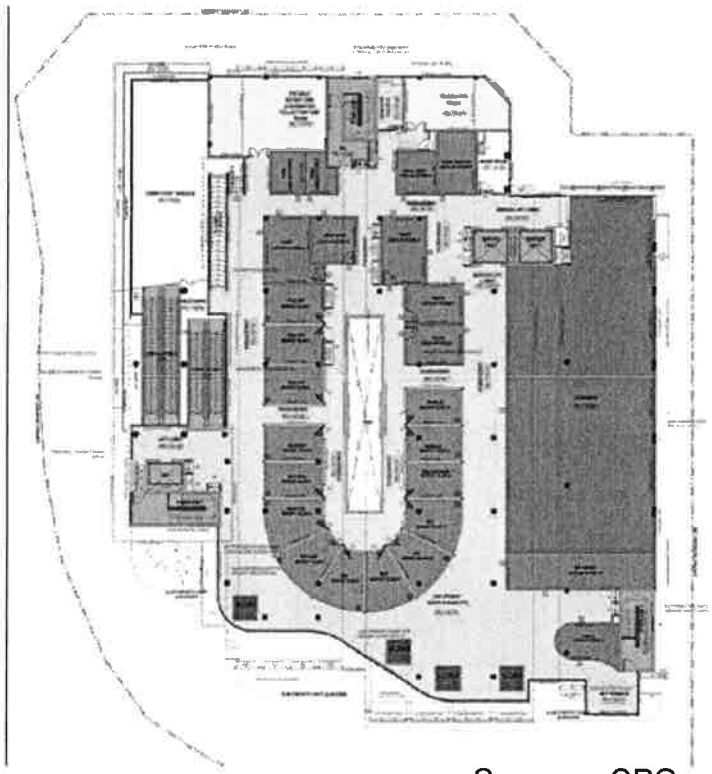
Surbana

Case studies

2nd Storey Plan

Comprising of :

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



Source :- CPG

Surbana

Passive Design & Natural Ventilation, GMP 2015

Case studies

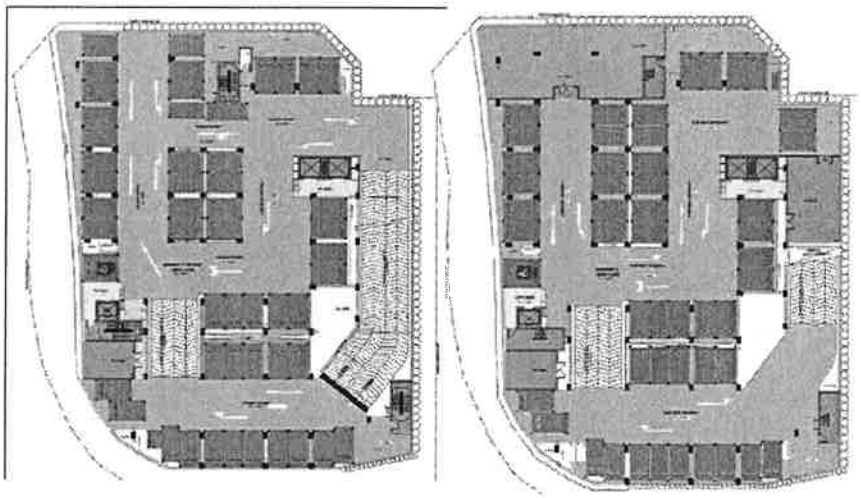
Basement 1 & 2

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

Surbana

Case studies





Bird-Eye Perspectives

Isometric Perspectives

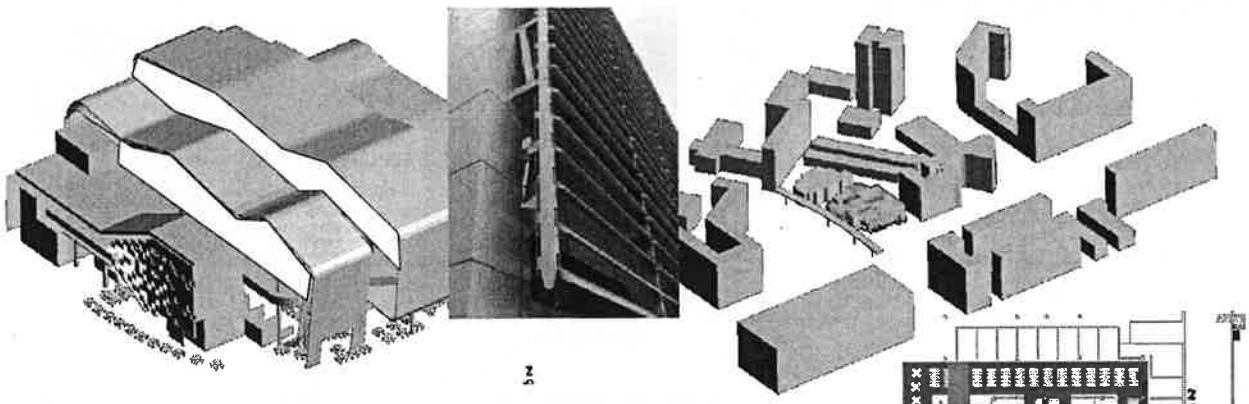




Source :- CPG

Passive Design & Natural Ventilation, GMP 2015

Case studies

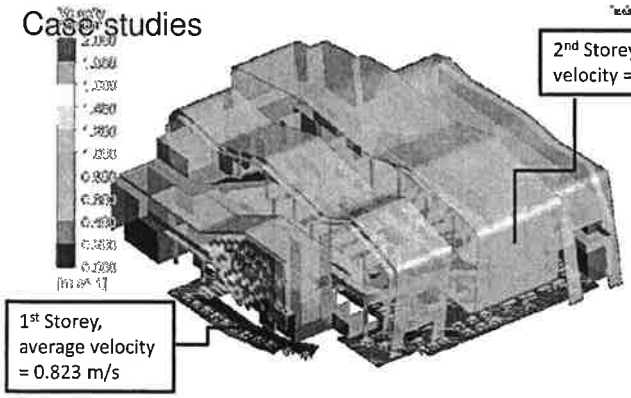


Unit type	Area (m^2)	North-Vavg (m/s)	South-Vavg (m/s)
1st Storey	852.521	0.237	0.352
2nd Storey	316.282	0.225	0.207

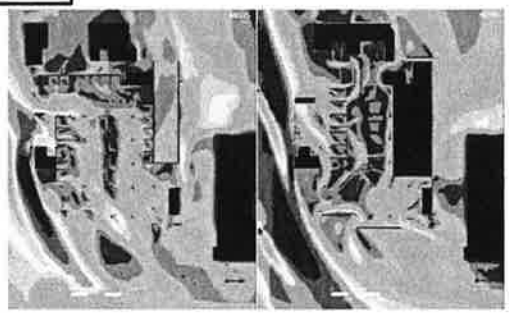
Unit type	Area (m^2)	North-Vavg (m/s)	South-Vavg
1st Storey	852.521	0.603	0.672
2nd Storey	316.282	0.346	0.219

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

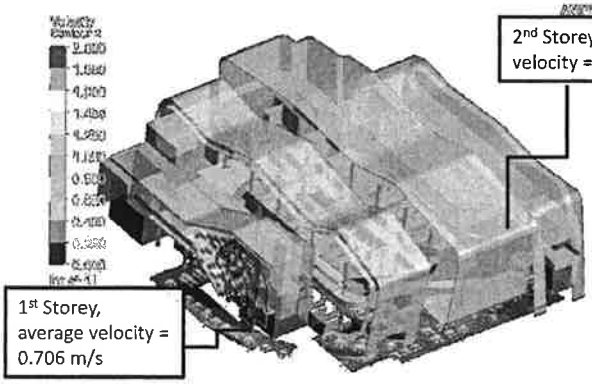
Case studies



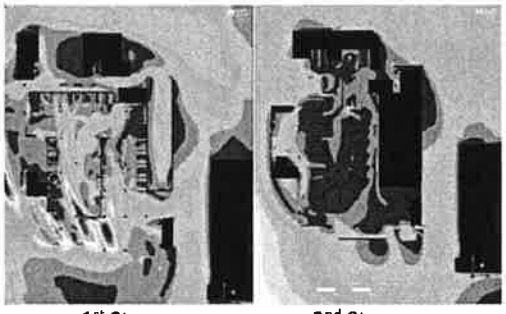
North



1st Storey 2nd Storey



South



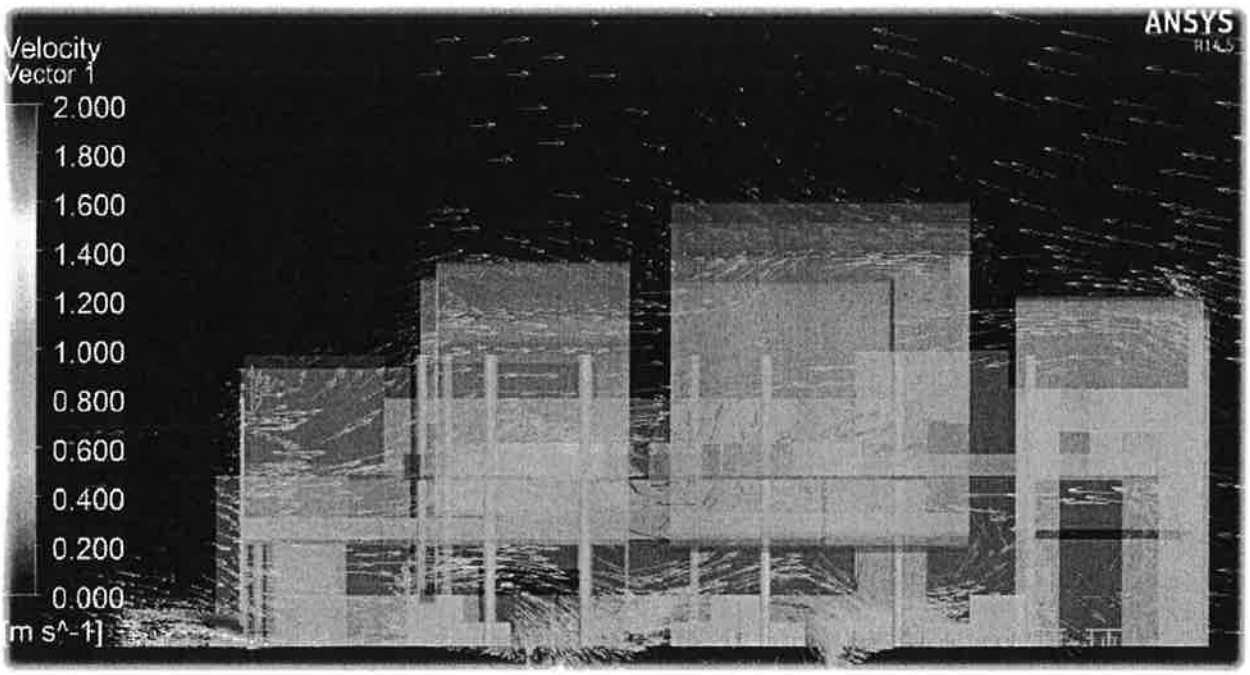
1st Storey 2nd Storey

Surbana

Passive Design & Natural Ventilation, GMP 2015

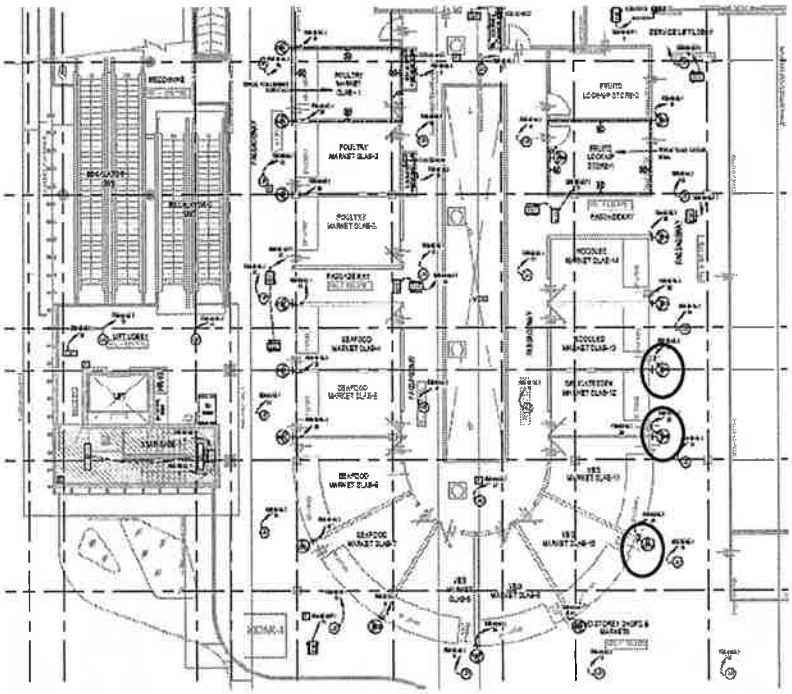
Case studies

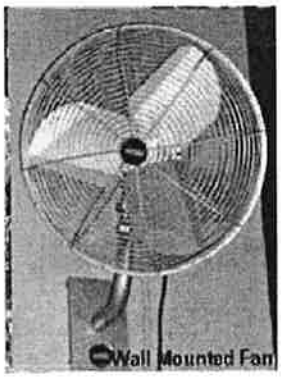
Natural ventilation through adjustable louvers North



Surbana

Case studies





Assisted Ventilation

Surbana

DESIGNING FOR NATURAL VENTILATION

Case 5

Case studies

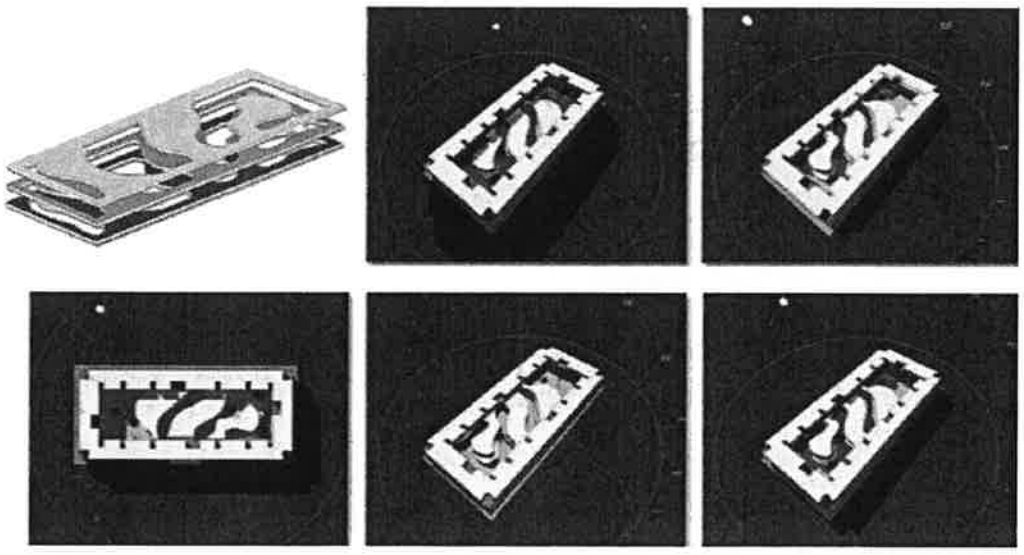
Academy of Fashion
Jaipur

Manit Rastogi , Morphogenesis

Passive Design & Natural Ventilation, GMP 2015

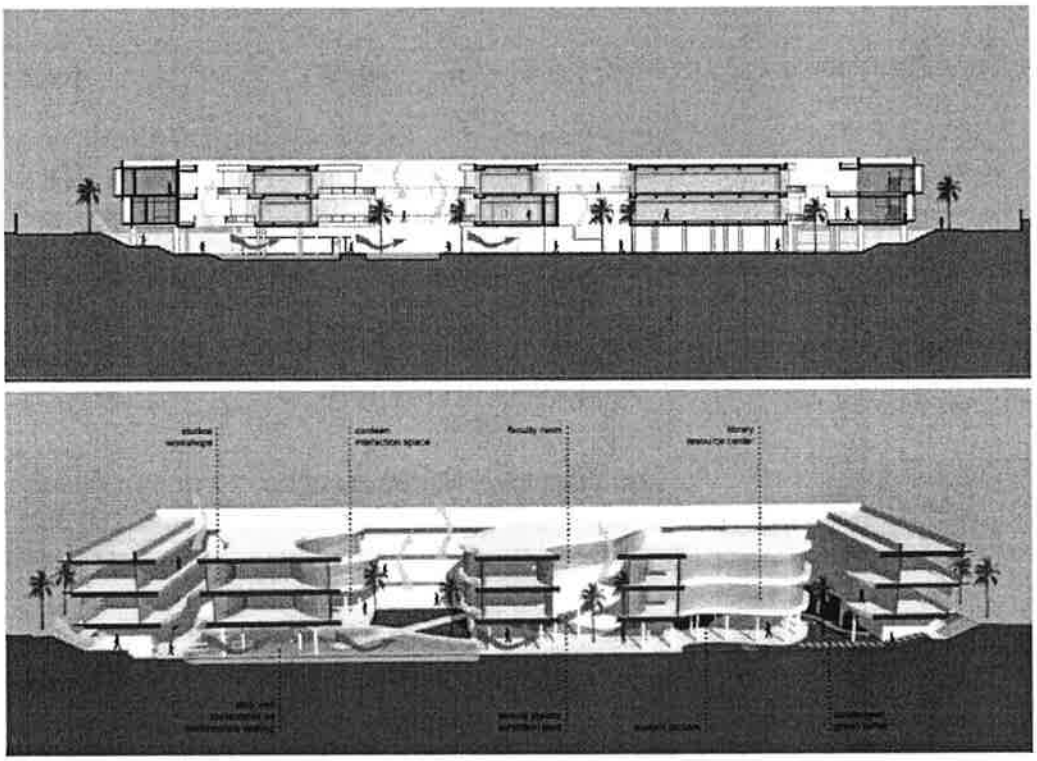
Case studies

Shadow Studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

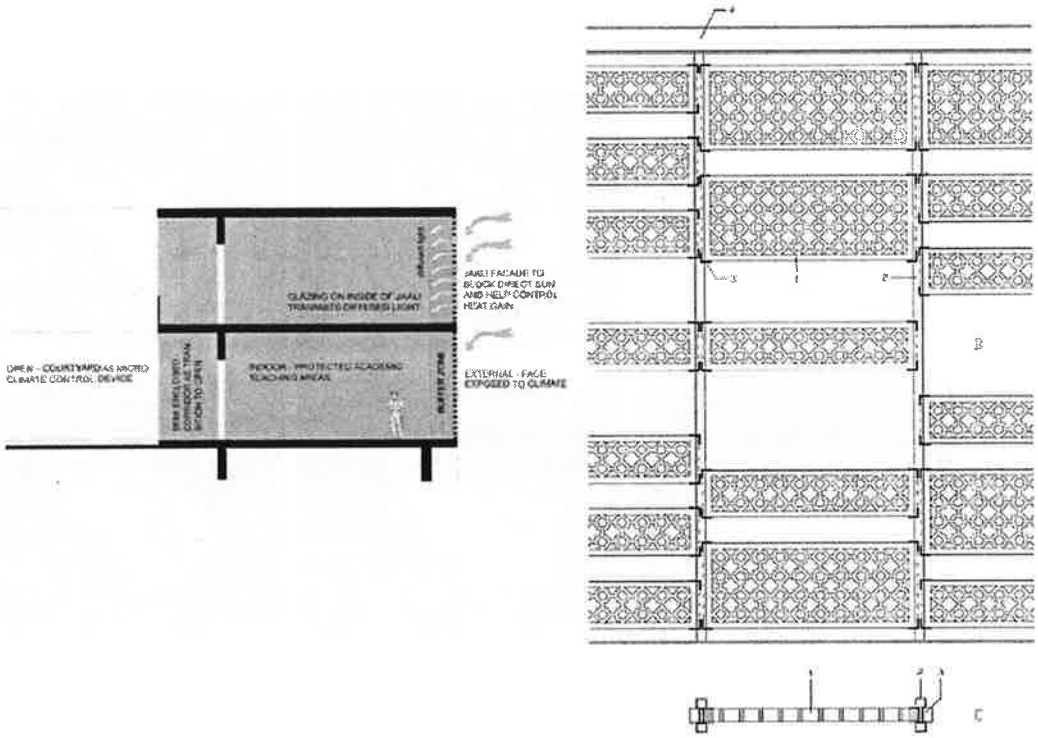
Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

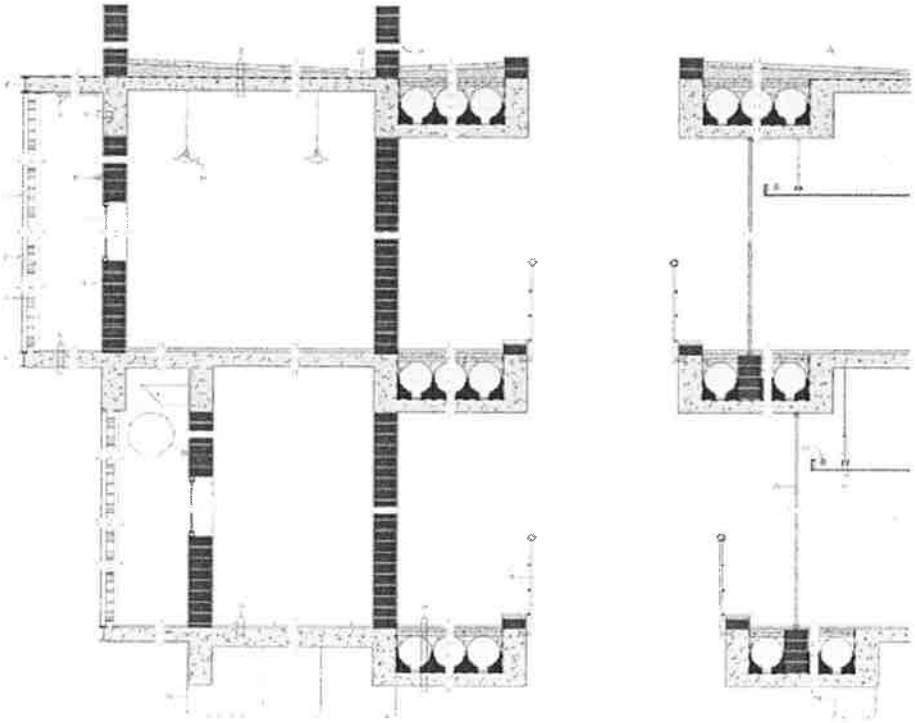
Passive Design & Natural Ventilation, GMP 2015

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

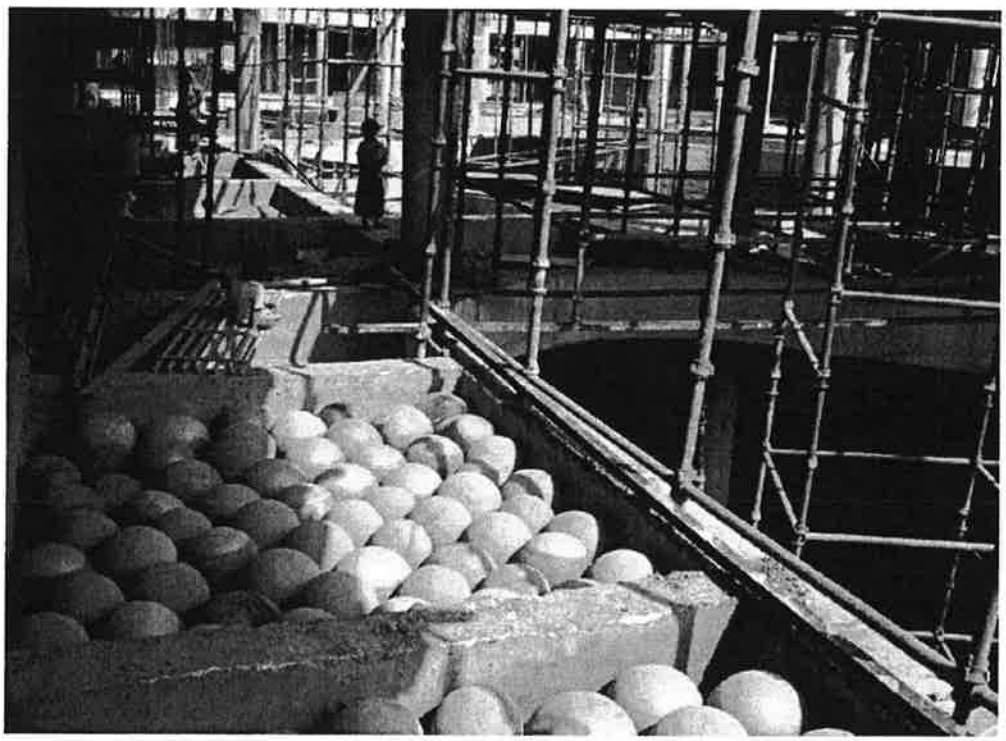
Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

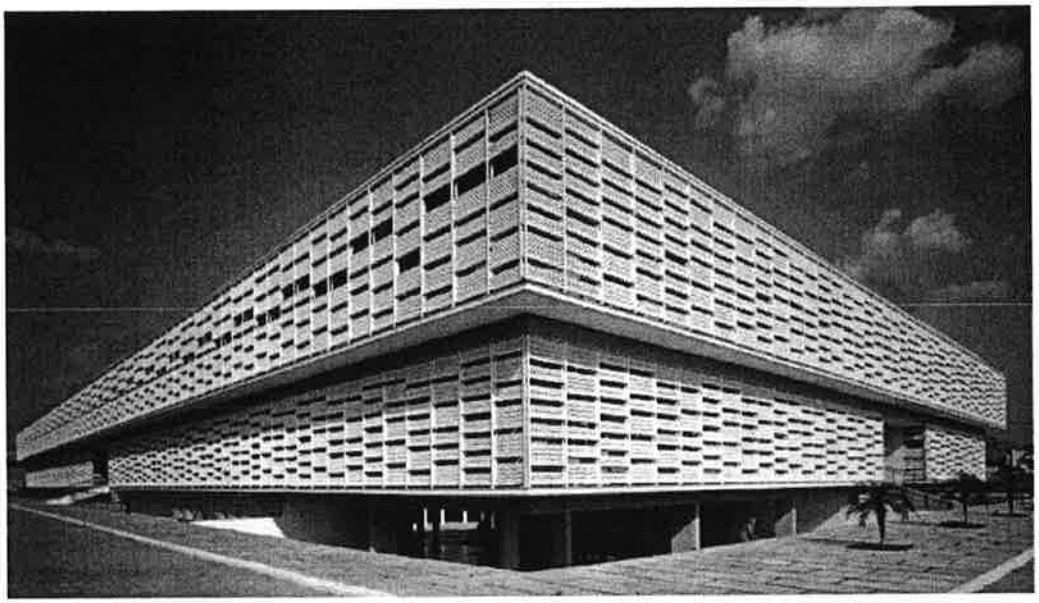
Passive Design & Natural Ventilation, GMP 2015

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

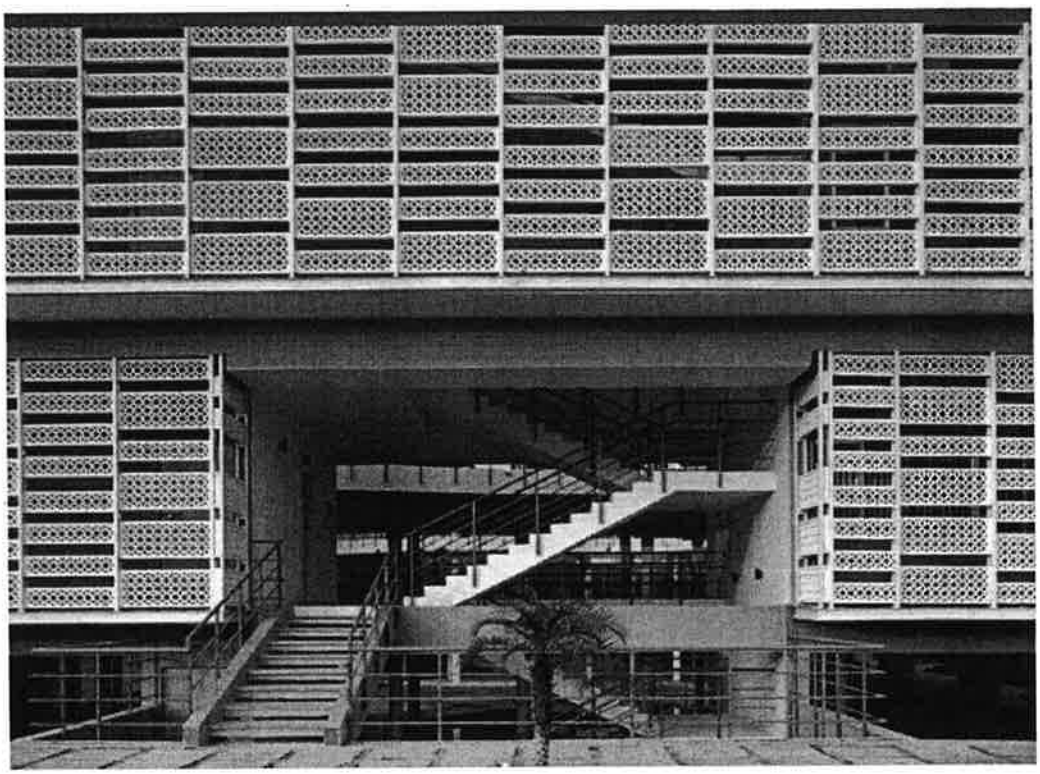
Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

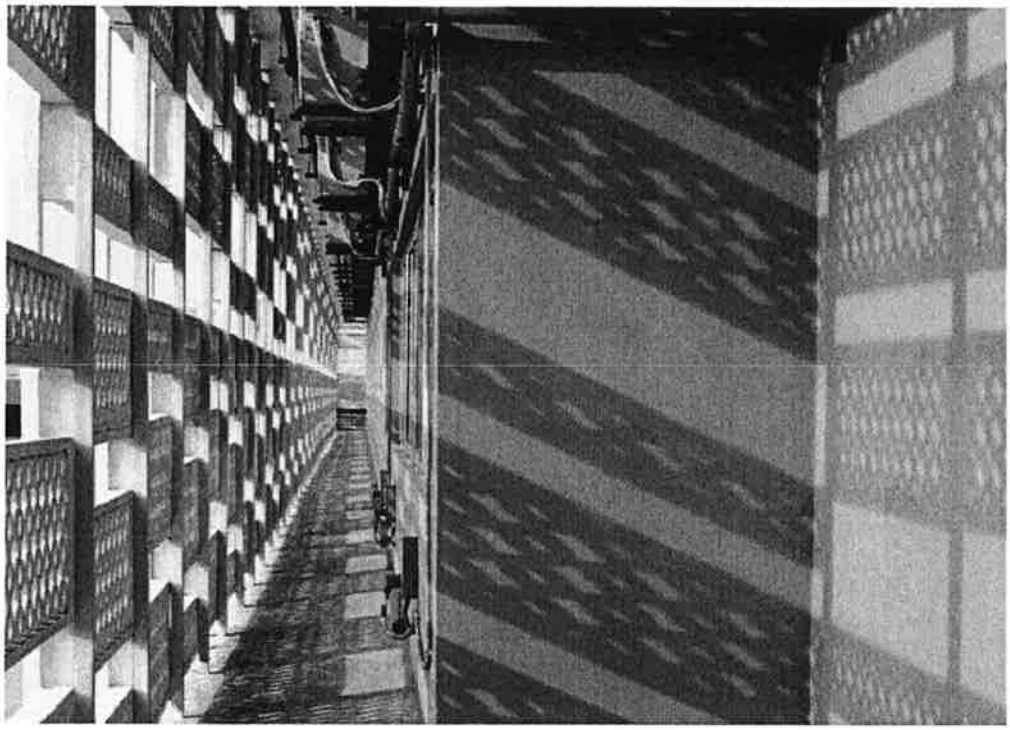
Passive Design & Natural Ventilation, GMP 2015

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Passive Design & Natural Ventilation, GMP 2015

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Passive Design & Natural Ventilation, GMP 2015

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Passive Design & Natural Ventilation, GMP 2015

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

DESIGNING FOR NATURAL VENTILATION

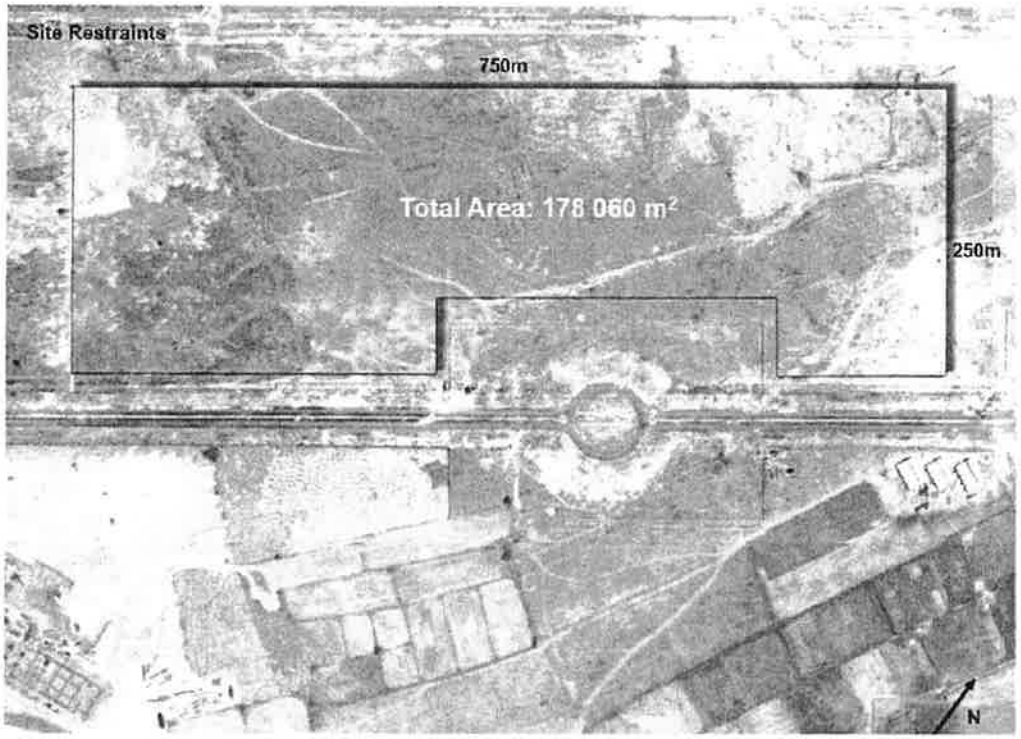
Case 6

Case studies

Educomp University
Greater Noida

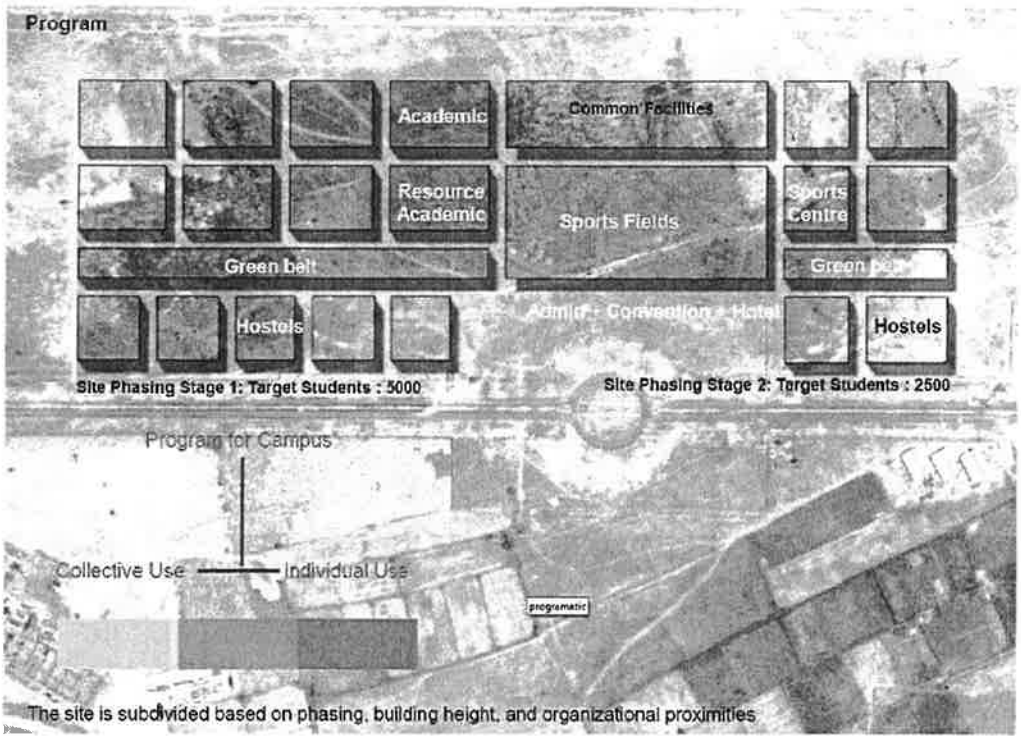
Manit Rastogi , Morphogenesis

Case studies



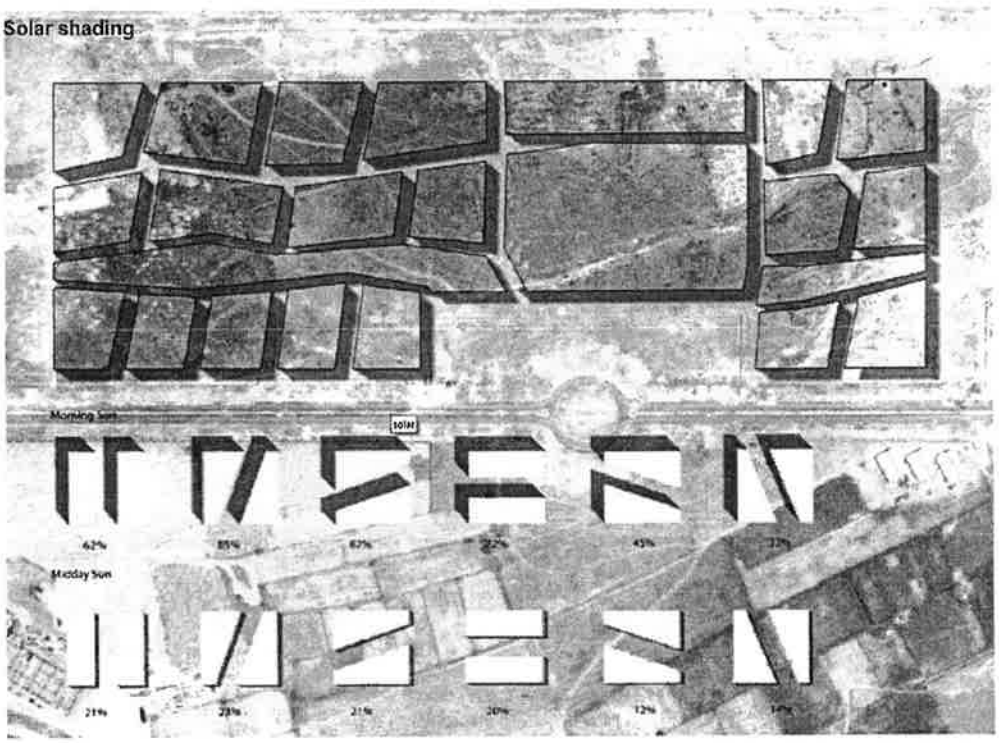
Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



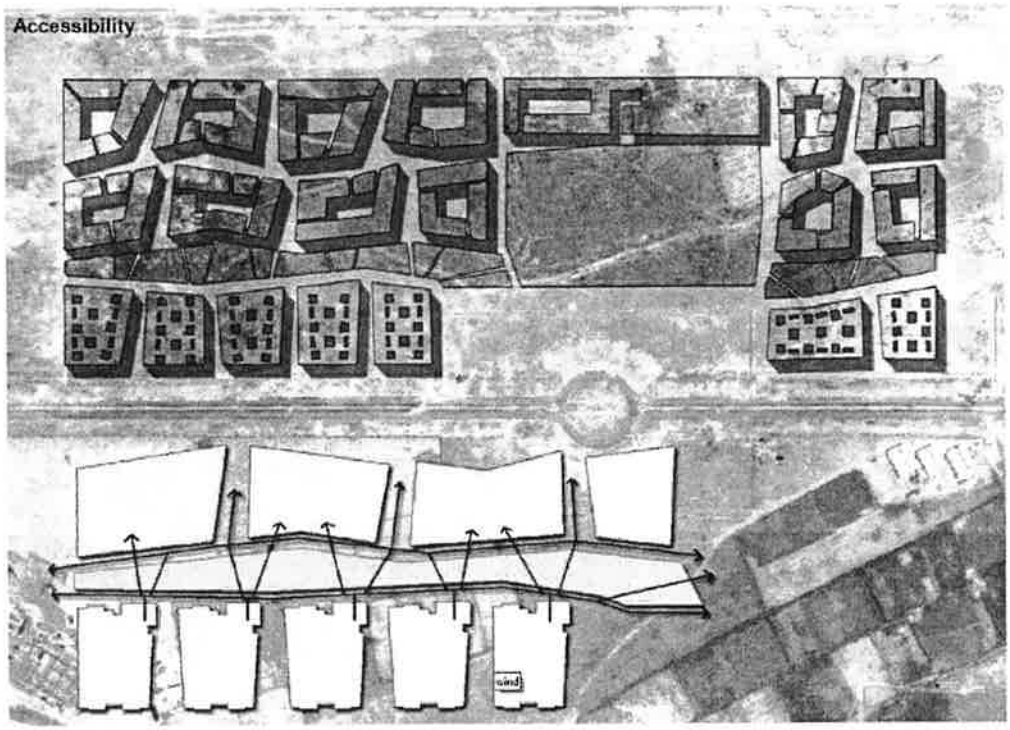
Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



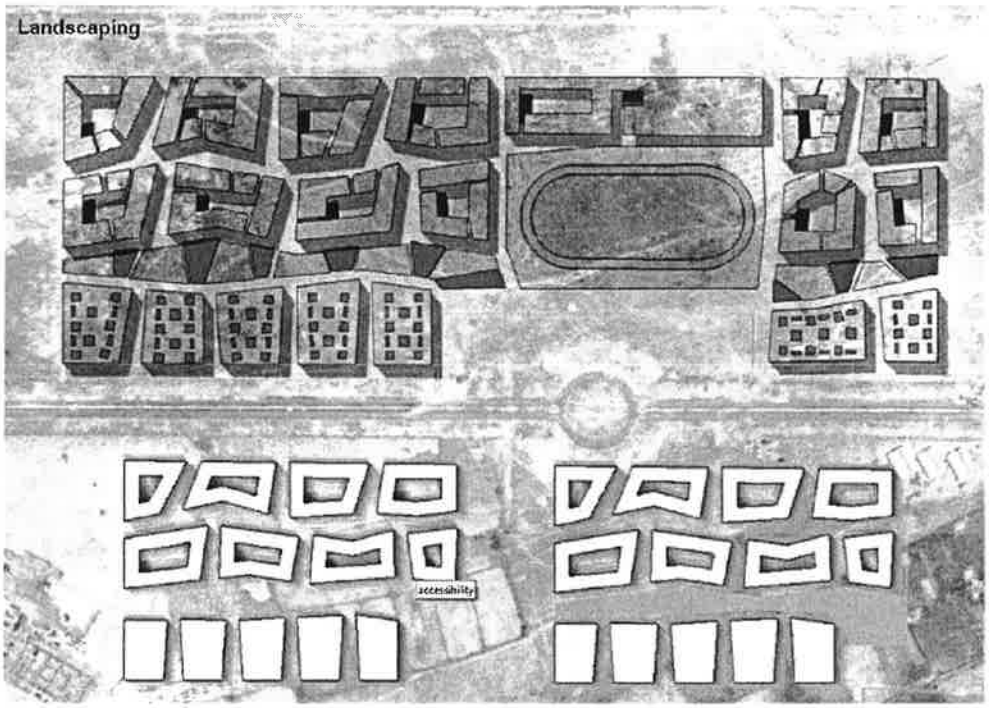
Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies

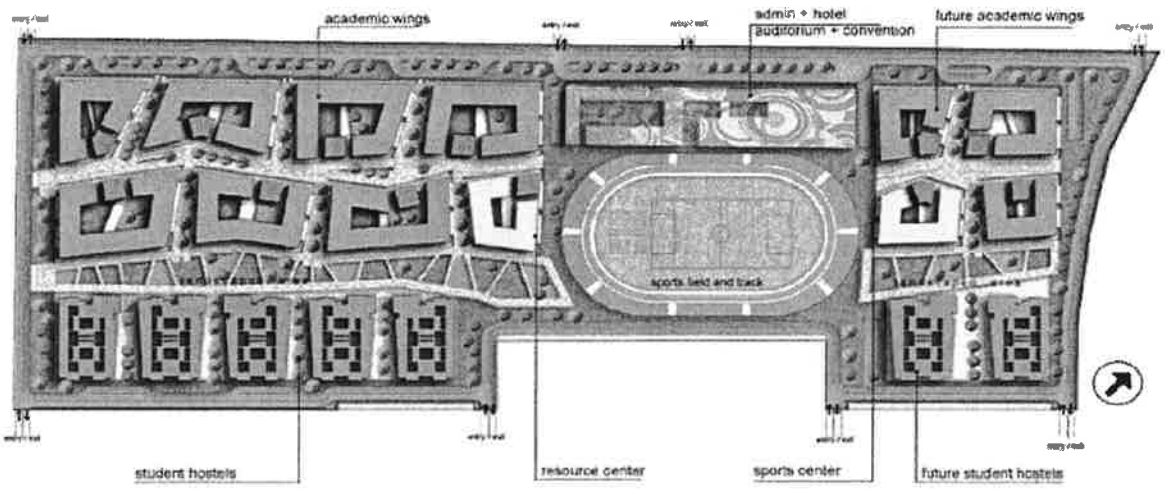


Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Passive Design & Natural Ventilation, GMP 2015

Case studies

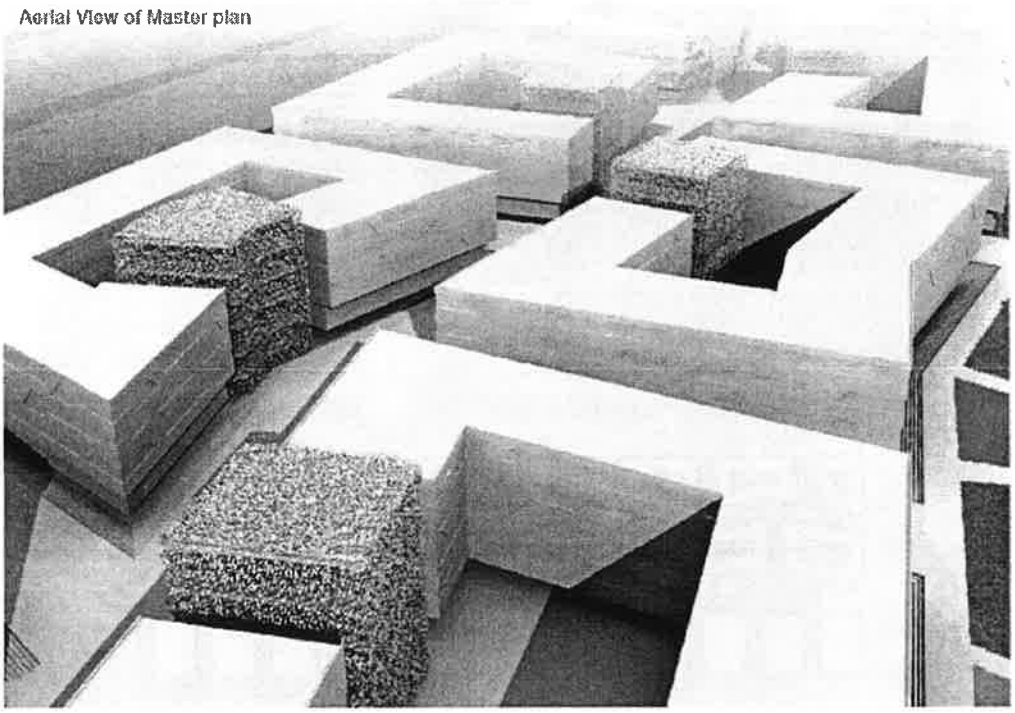
Master Plan



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies

Aerial View of Master plan

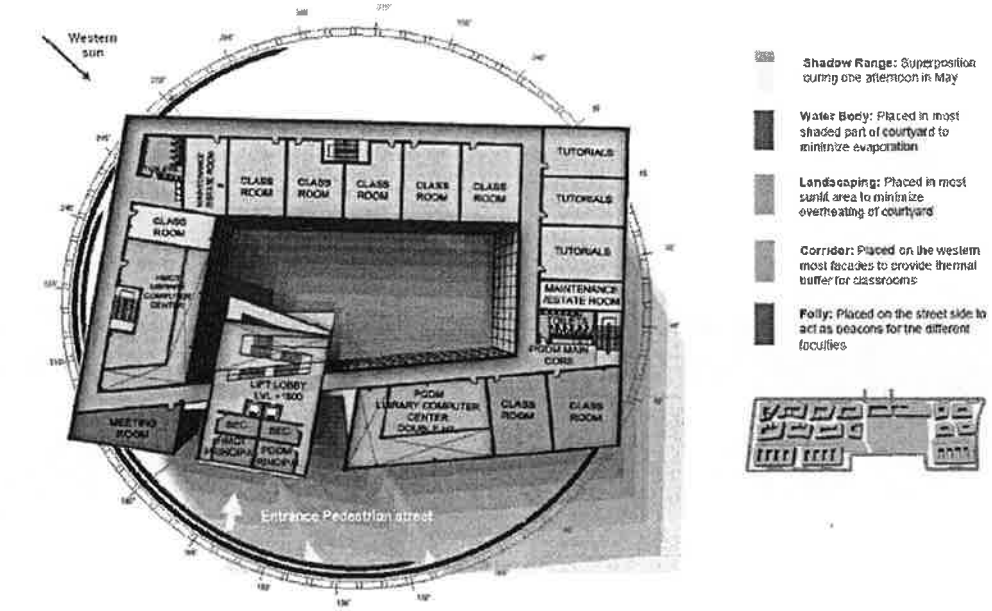


Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Passive Design & Natural Ventilation, GMP 2015

Case studies

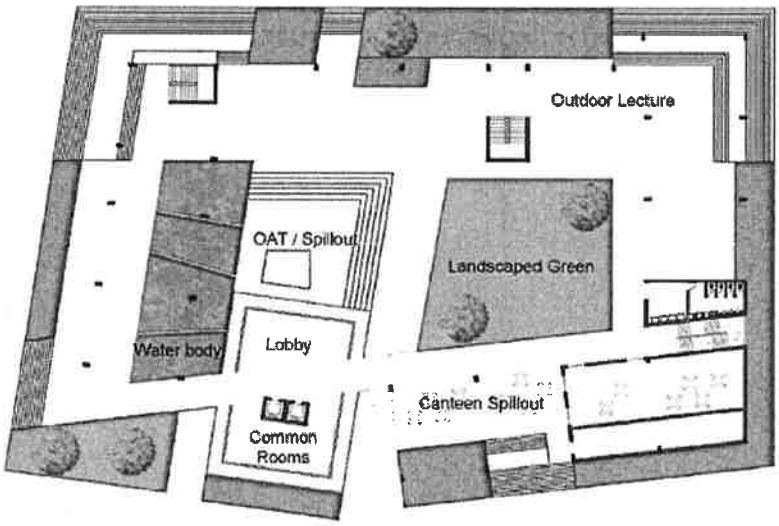
Functional layout of Block



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies

Academic Building 1: Plan at Underbelly Level -2500

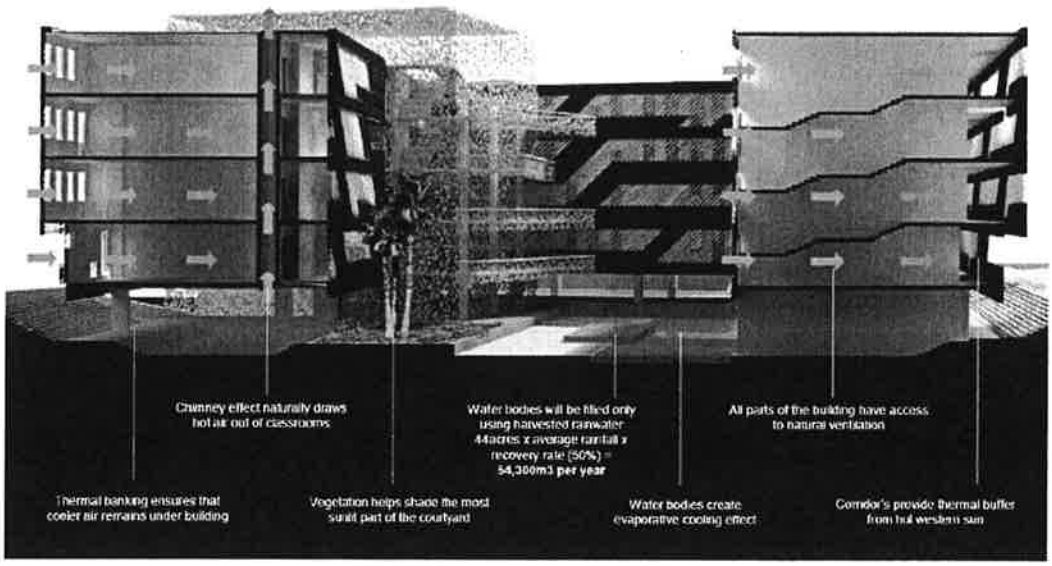


Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Passive Design & Natural Ventilation, GMP 2015

Case studies

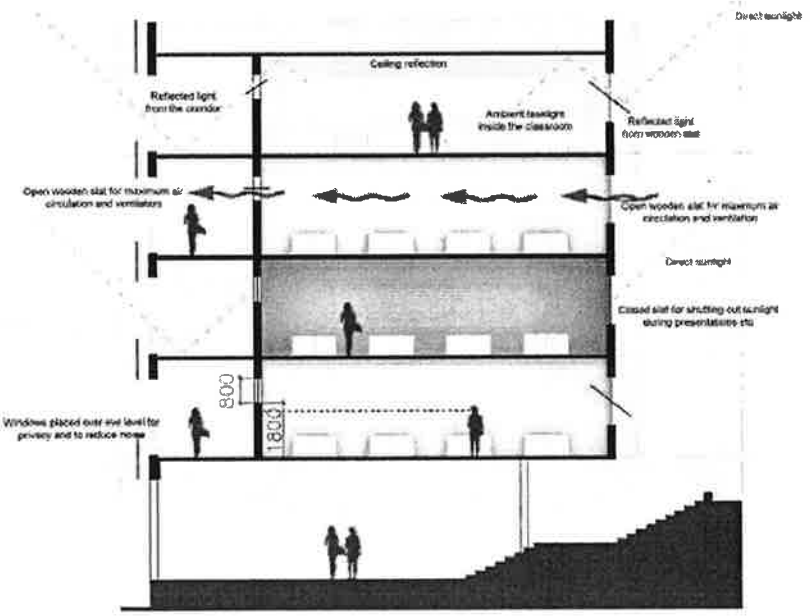
Functional section of Block/Underbelly



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies

Classroom - section

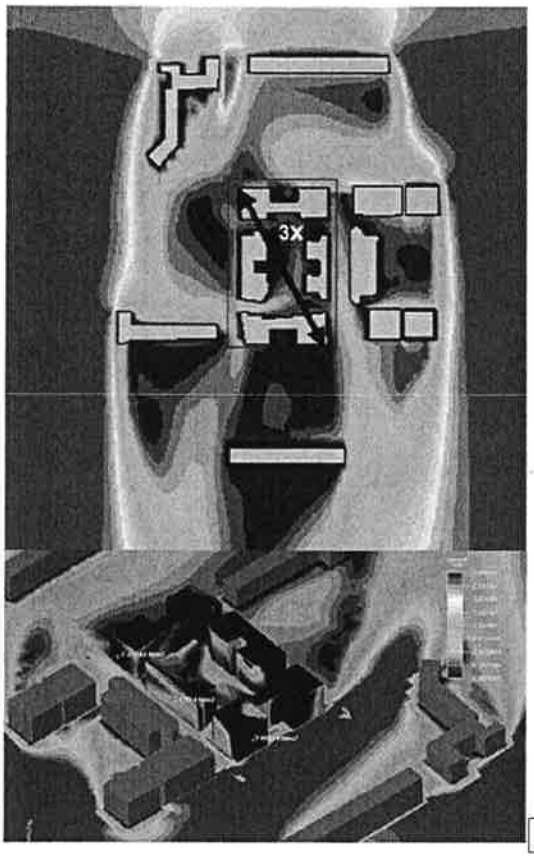


Source : Manit Rastogi , Morphogenesis , 2009 IGBC

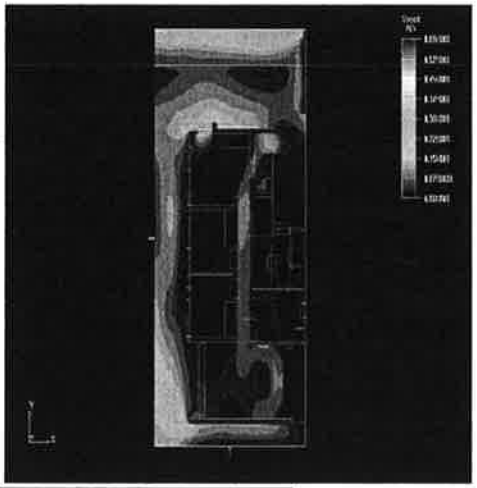
DESIGNING FOR NATURAL VENTILATION

Case 7

Case studies



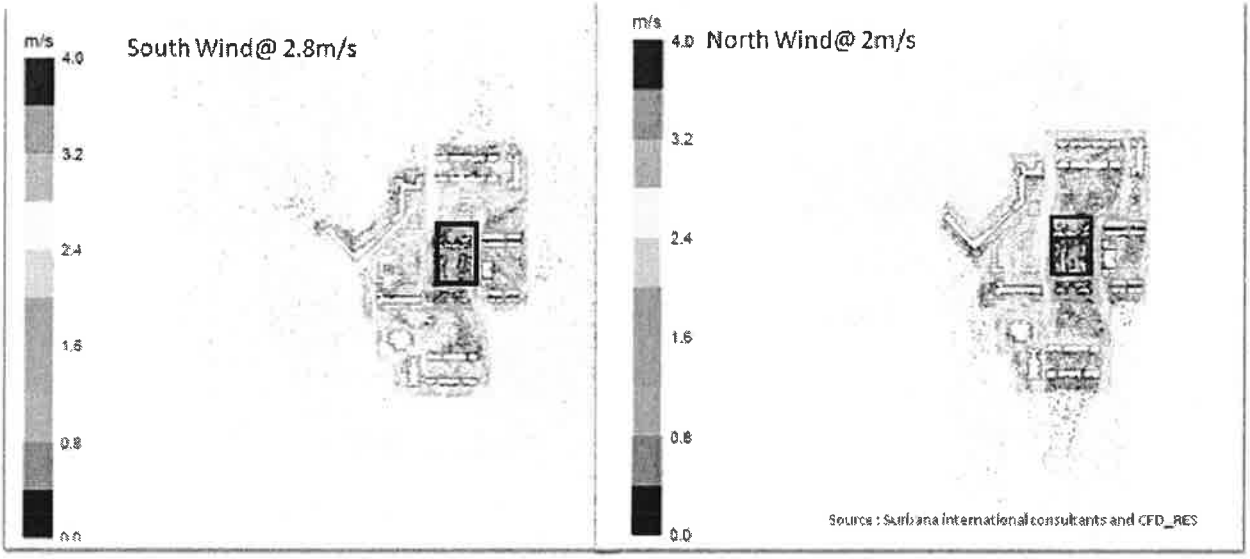
The project is a high density residential condo with full facilities targeted at mid market. The target was to achieve the highest Green Mark Platinum Award.



Source : Surbana International Consultants

Passive Design & Natural Ventilation, GMP 2015

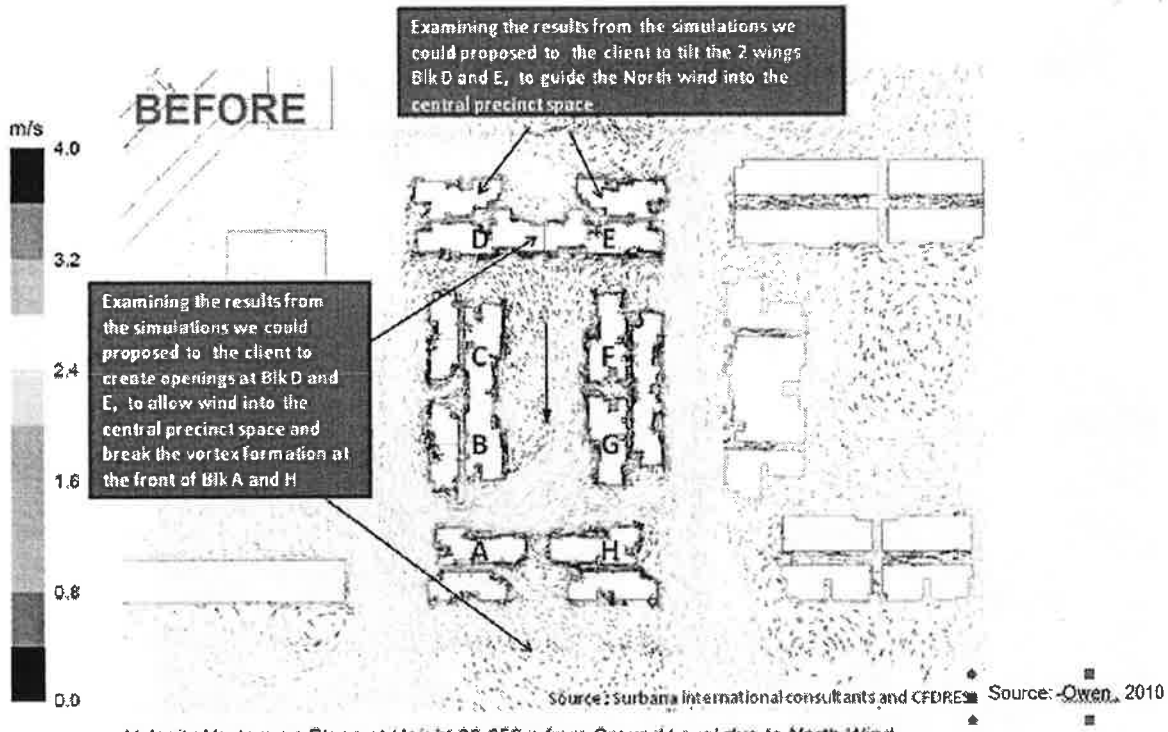
Case studies



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

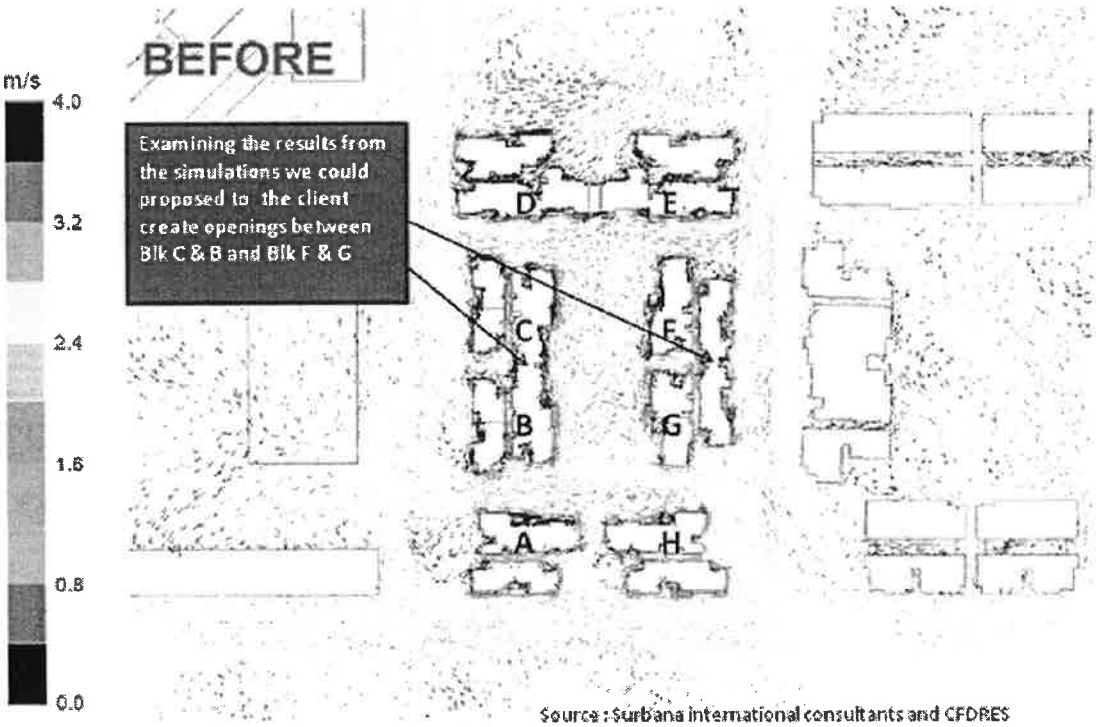


Source: -Owen., 2010

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

Passive Design & Natural Ventilation, GMP 2015

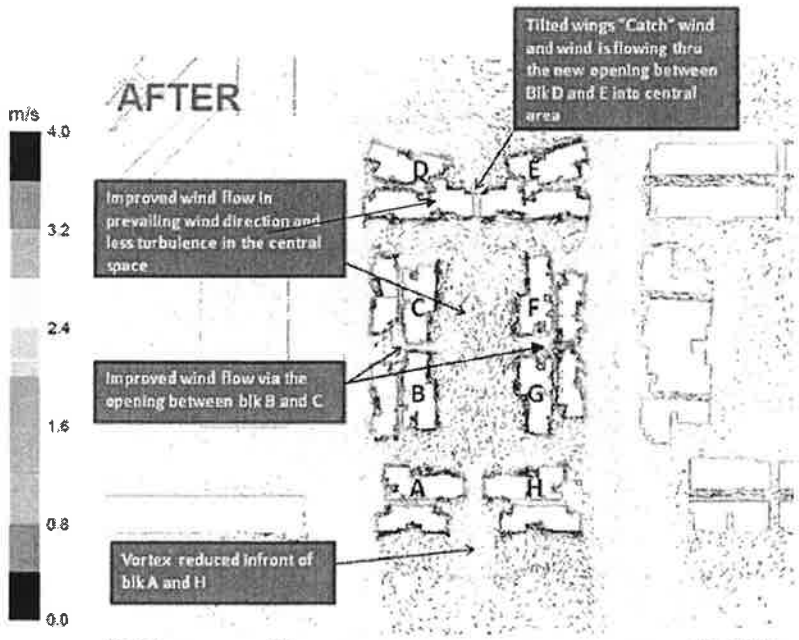
Case studies



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 and H



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

Source : Surbana international consultants and CFDRES 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

Passive Design & Natural Ventilation, GMP 2015

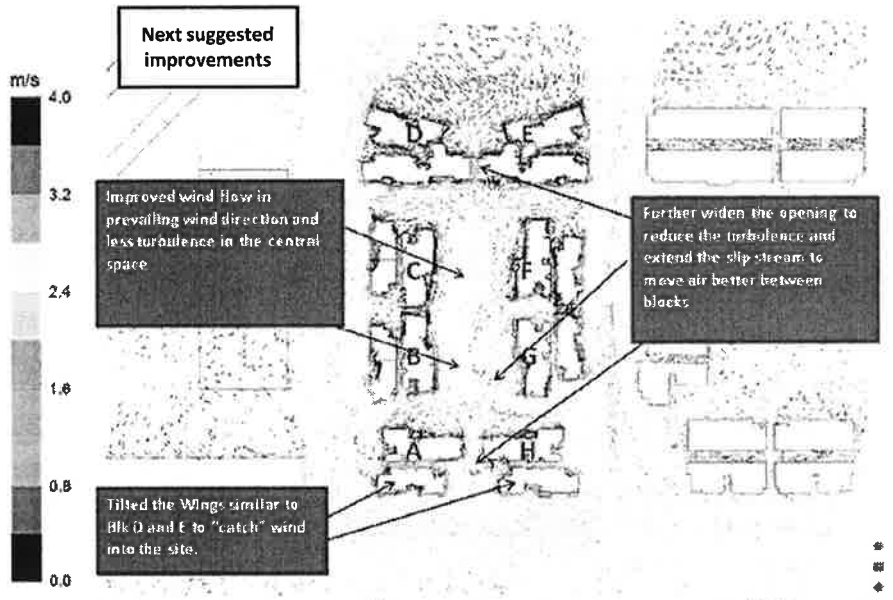
Case studies

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

THE END

BCA GMP Core Module Design and Simulation for Natural Ventilation

Dr. Poh Hee Joo (pohhj@ihpc.a-star.edu.sg)
Institute of High Performance Computing, A*STAR

Feb 2016

Course Objectives

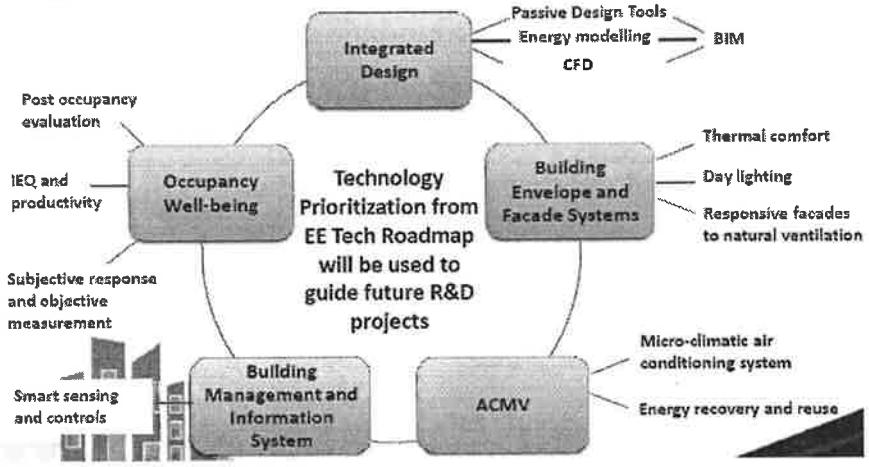
- To equip GMP students with the basic skill set of applying CFD tool to building ventilation design
  - Creating 3D geometry model
  - Meshing the computational model
  - Applying appropriate boundary condition and physical assumption
  - Analyzing CFD result for design and improvement
  - Establishing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

Day 2 (9:00am – 10:30am)

- Introduction to CFD in Green Building Application
  - CFD as a potential tool to optimize and deliver green buildings, and case studies
  - Natural ventilation requirements in Green Mark criteria
  - Strategies At Estate Level To Achieve Good Natural Ventilation

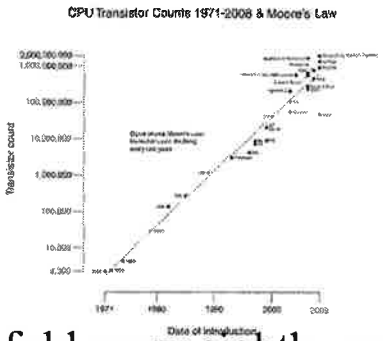
Building Energy Efficiency Technology Road Mapping

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)



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

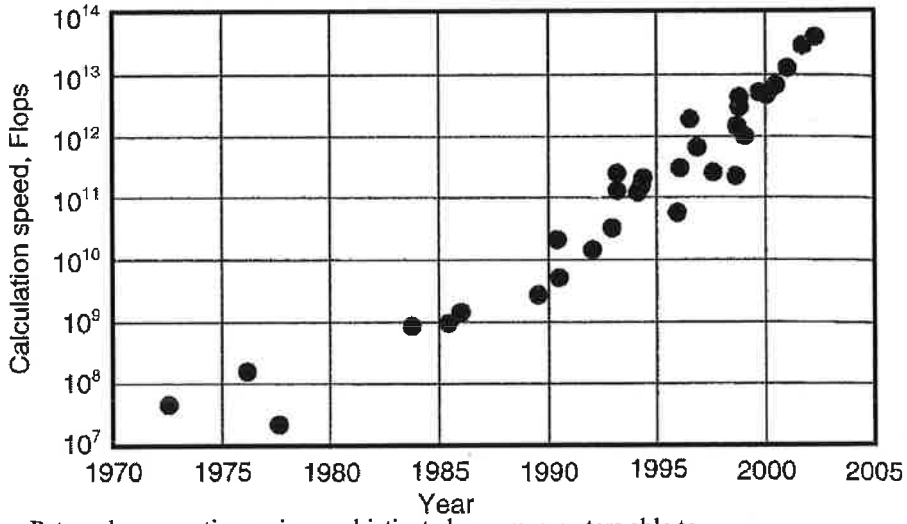
*Teraflops: One trillion floating point operations per second

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

Mathematical Modeling

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

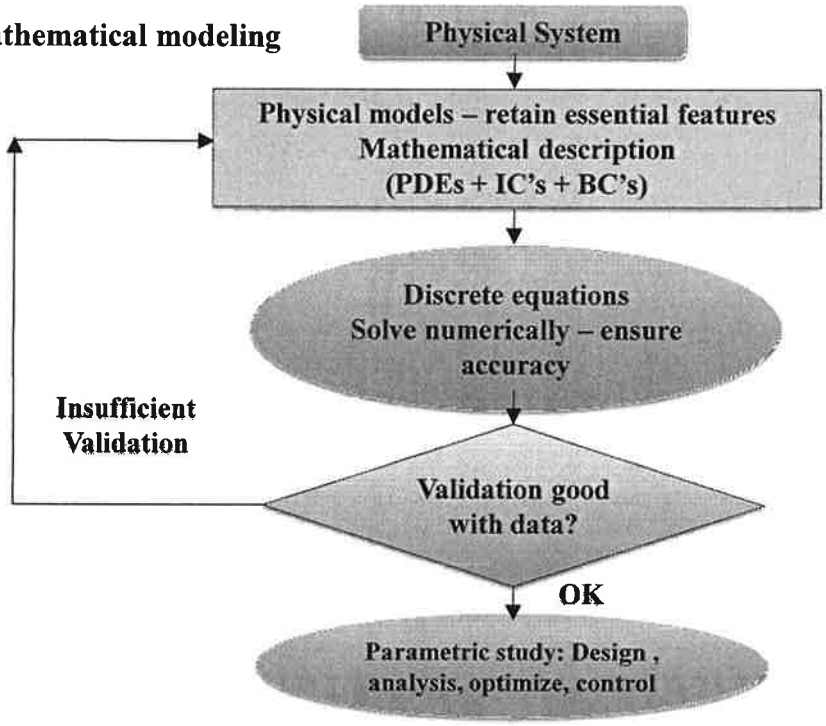
A representation of the essential aspects of a real system (or a system to be constructed) which presents knowledge of that system in a usable form. Should lead to innovative engineering design concepts

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

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

Introduction

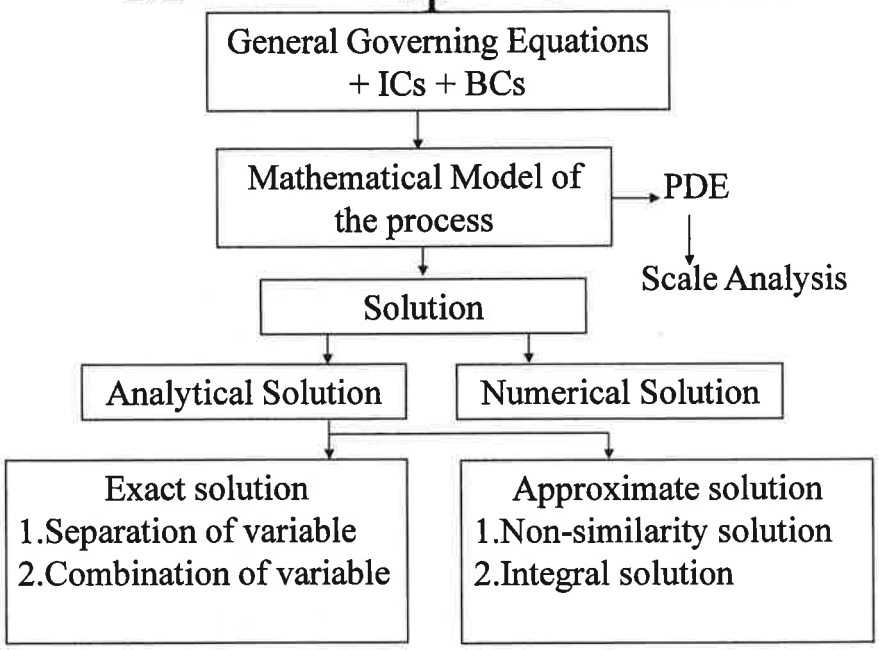
Outline of mathematical modeling



Modeling Momentum, Heat and Mass Transport Problems

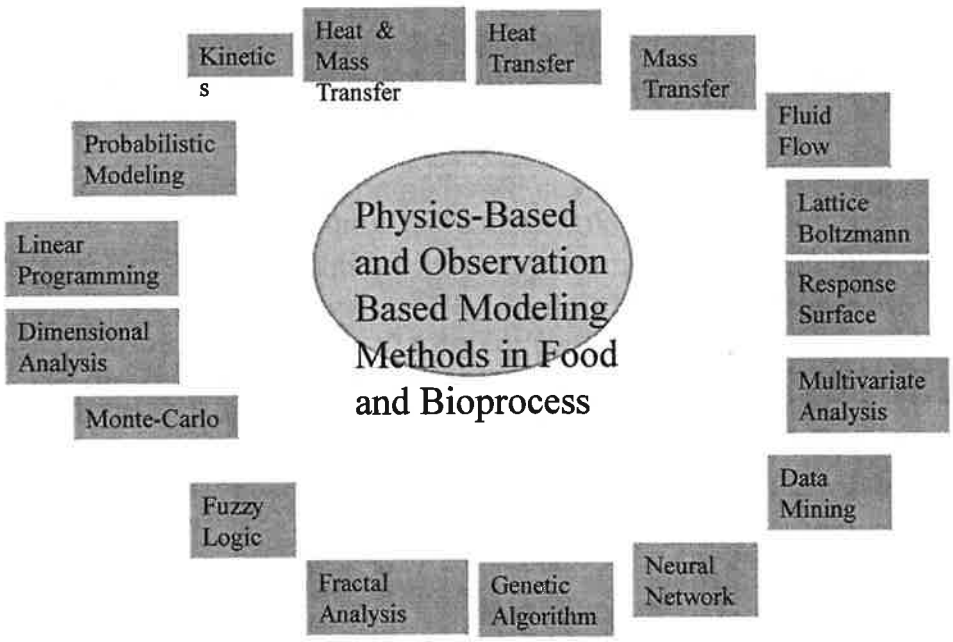
- Complete set of governing equations as well as initial and boundary conditions needed in the modeling in momentum, heat and mass transfer processes are in general too complex to solve analytically and even numerically
- Necessary to simplify them based on physical understanding of the problem leading 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

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

Application of CFD Codes in Building Design

The applications of CFD in buildings may be grouped under the following headings:

- 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

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

Introduction

- Fundamental equations (PDEs) in air flow prediction
  - Continuity equation
  - 3 Momentum equations (x, y, z)
  - Energy equation
  - Species equation
  - 2 Turbulence equations (κ, ε) – Eddy viscosity model

Introduction

- Room is divided into 90 x 90 x 90 cells
- 8 differential equations would give 5.8 million equations (and same number of unknown variables) at each grid points
- Typically 3000 iterations is required for converged solutions.
- 17 billion grid point calculations are made for airflow prediction

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:*
  - Limited to zones close to the openings
- *Cross-Ventilation:*
  - Two or more openings on opposite walls - covers a larger zone than the single-sided openings
- *Stack Ventilation:*
  - Buoyancy-driven gives larger flows
- *Wind Catchers:*
  - Wind and buoyancy driven - effective in warm and temperate climates
- *Solar-Induced Ventilation:*
  - Using the sun to heat building elements to increase buoyancy - more effective in warm climates

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

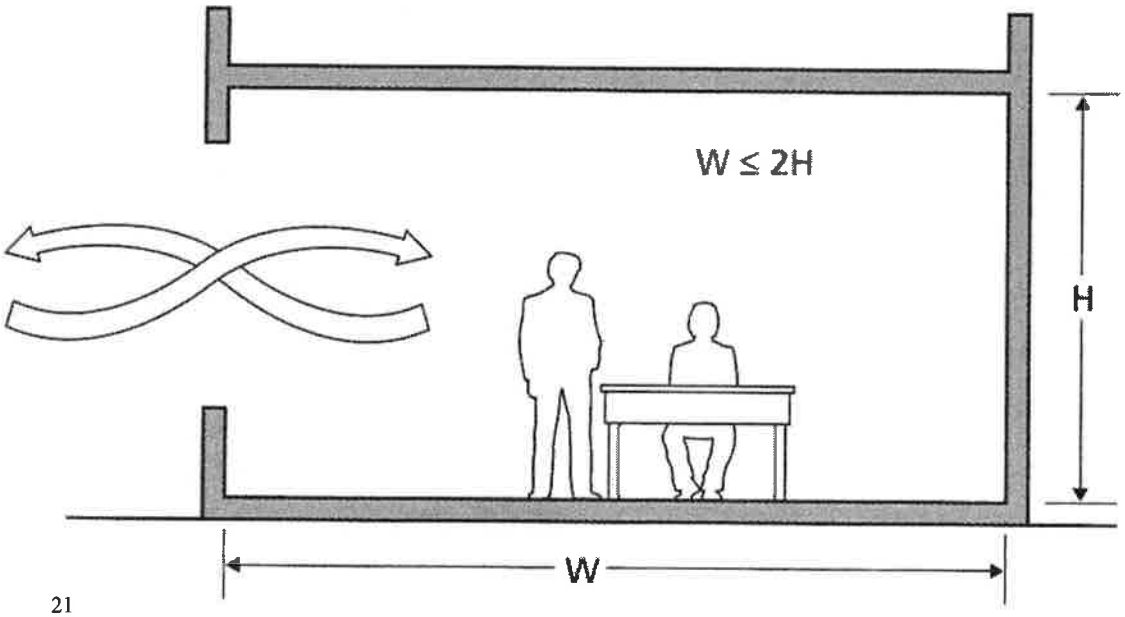
Factors Influencing the airflow through openings

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

Indoor Environmental considerations

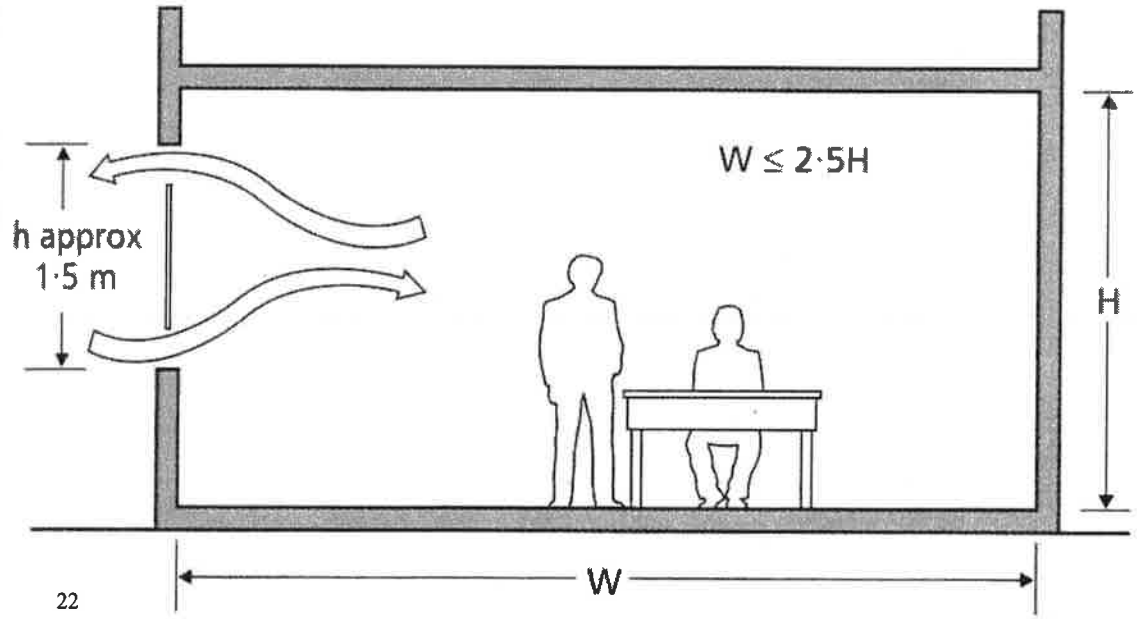
- **Thermal comfort**
- **Indoor air quality**

Single-sided Ventilation:
Single opening



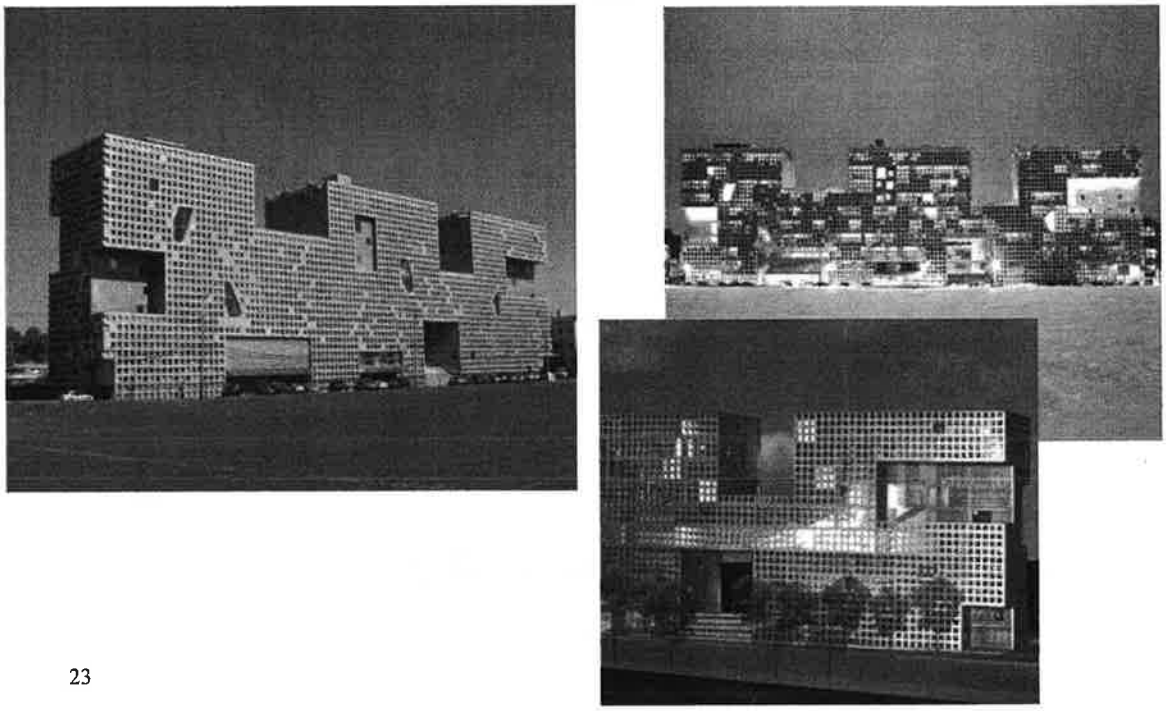
21

Single-sided Ventilation:
Double openings

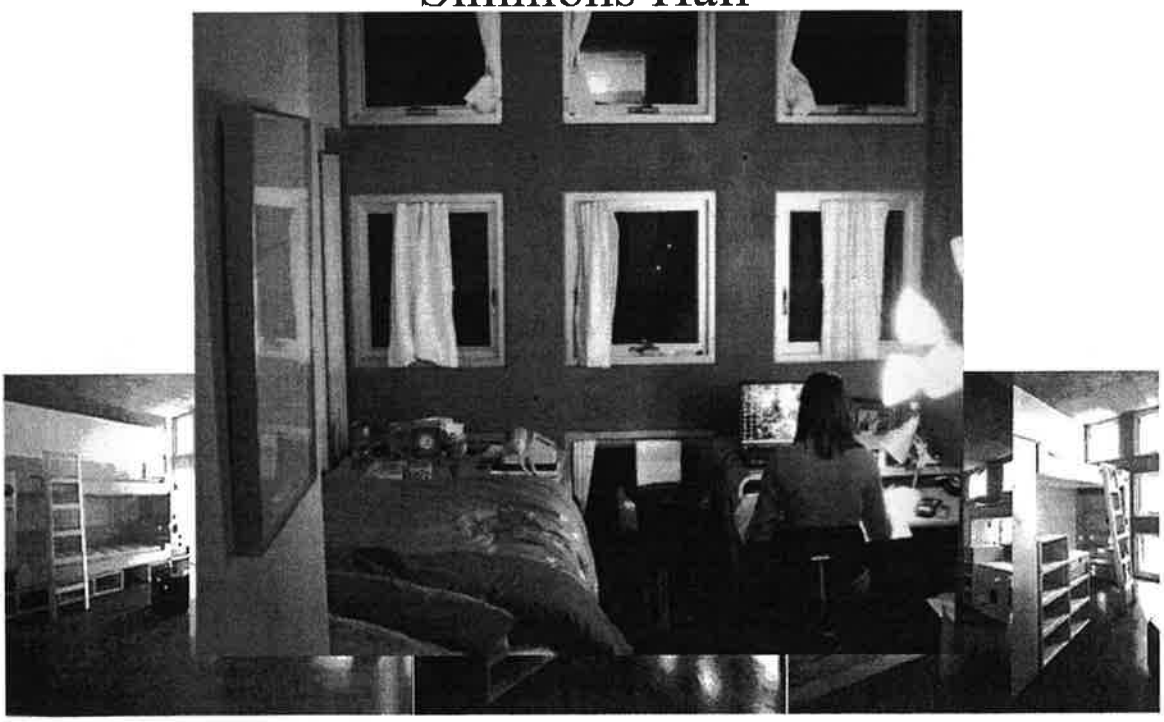


22

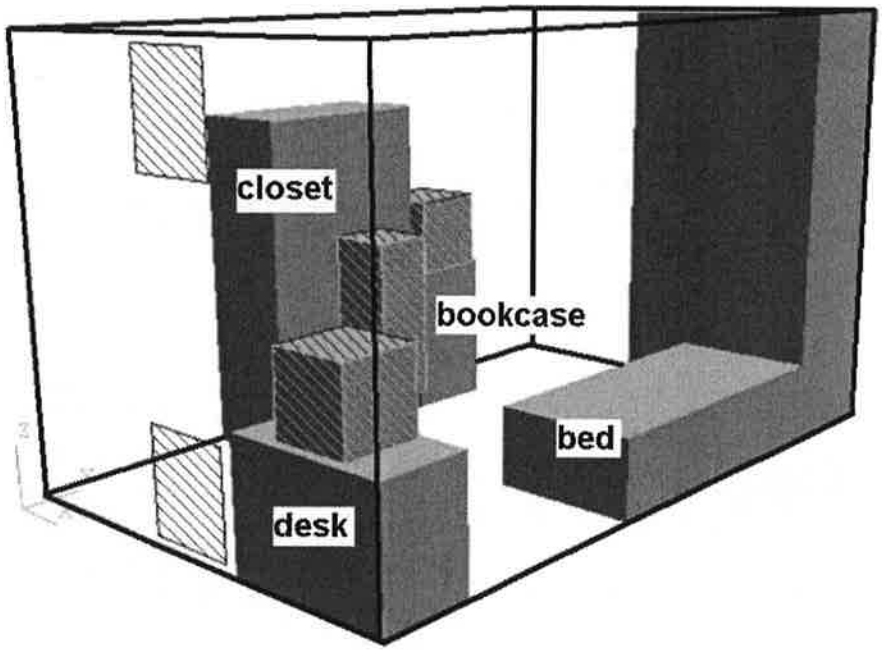
Single-sided Ventilation:
Simmons Hall



Single-sided Ventilation:
Simmons Hall

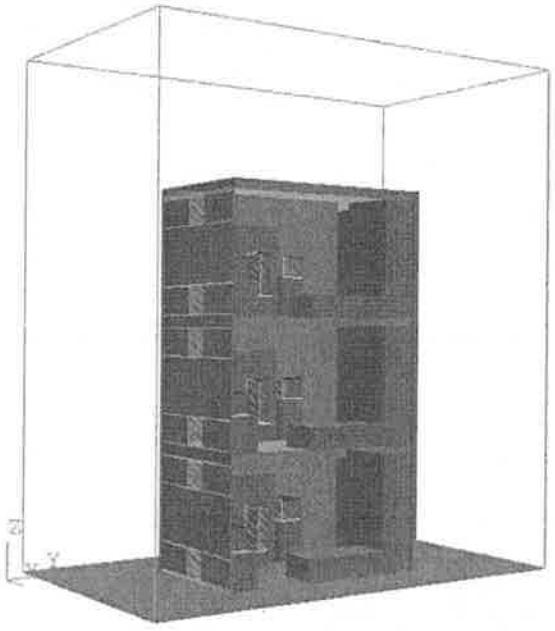


Single-sided Ventilation:
Room model



25

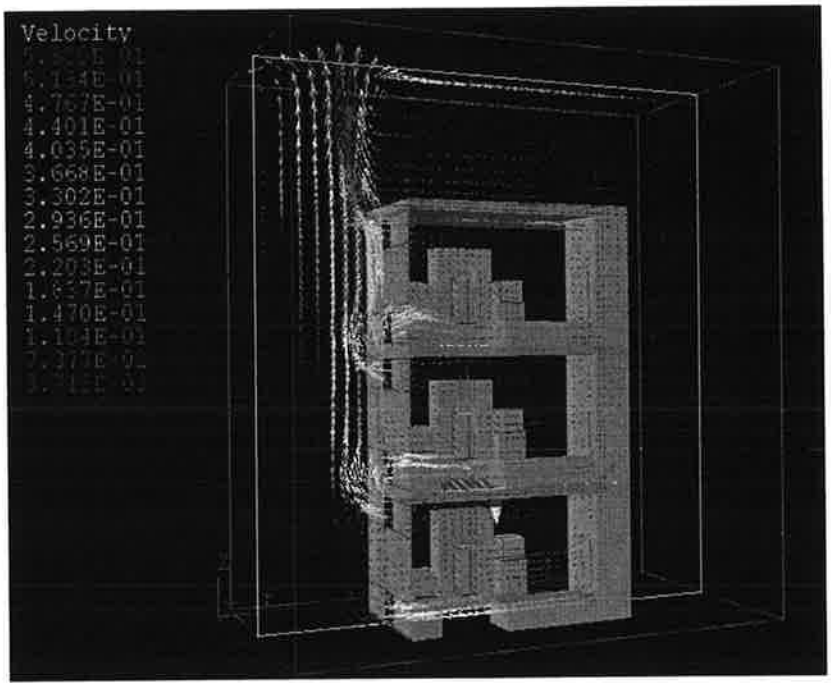
Single-sided Ventilation:
Room model



26

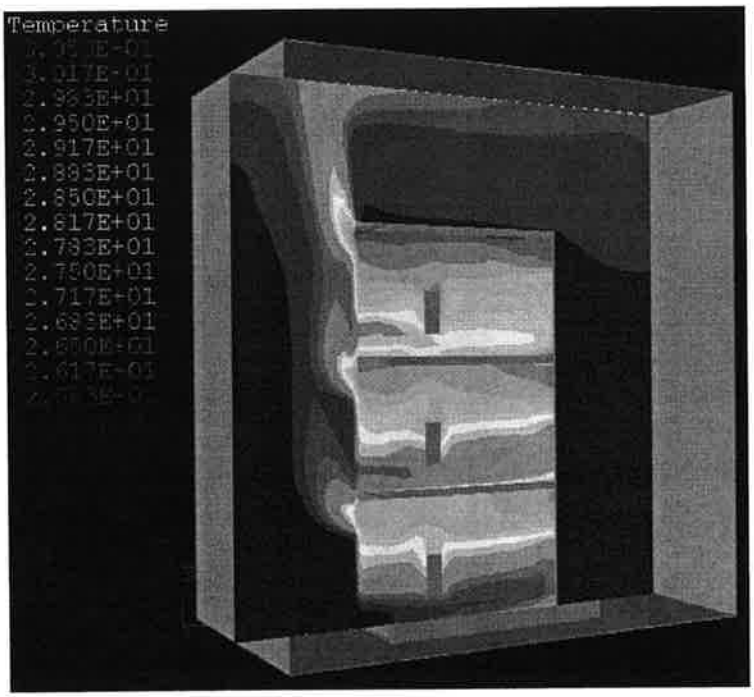
Single-sided Ventilation:

Room air velocity distribution (stack only)



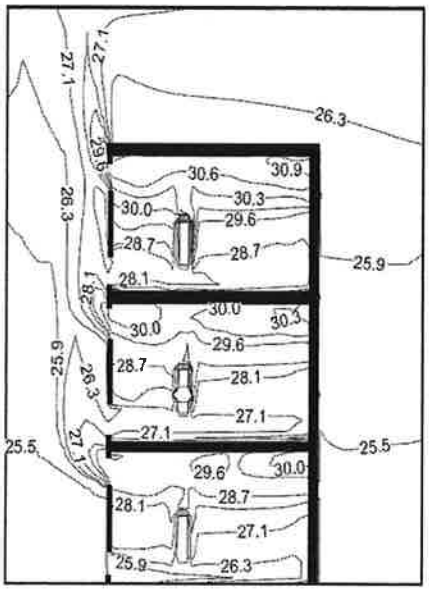
Single-sided Ventilation:

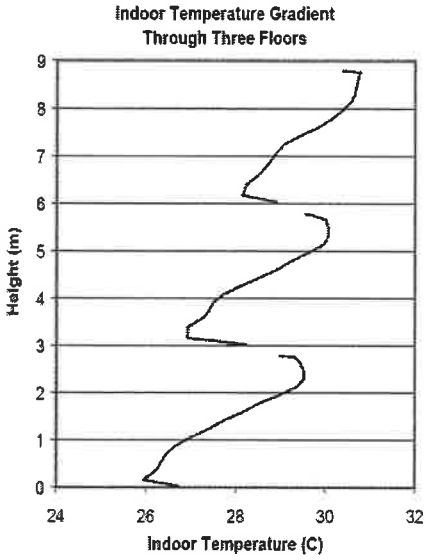
Room air temperature distribution (stack only)



Single-sided Ventilation:

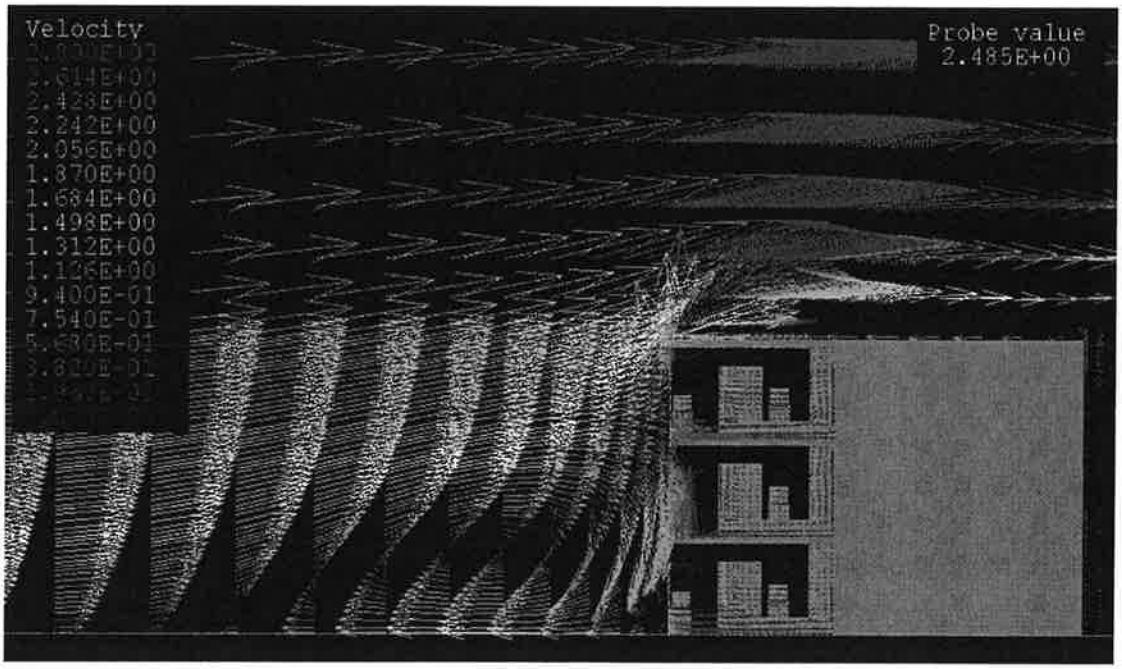
Room air temperature distribution (stack only)





Single-sided Ventilation:

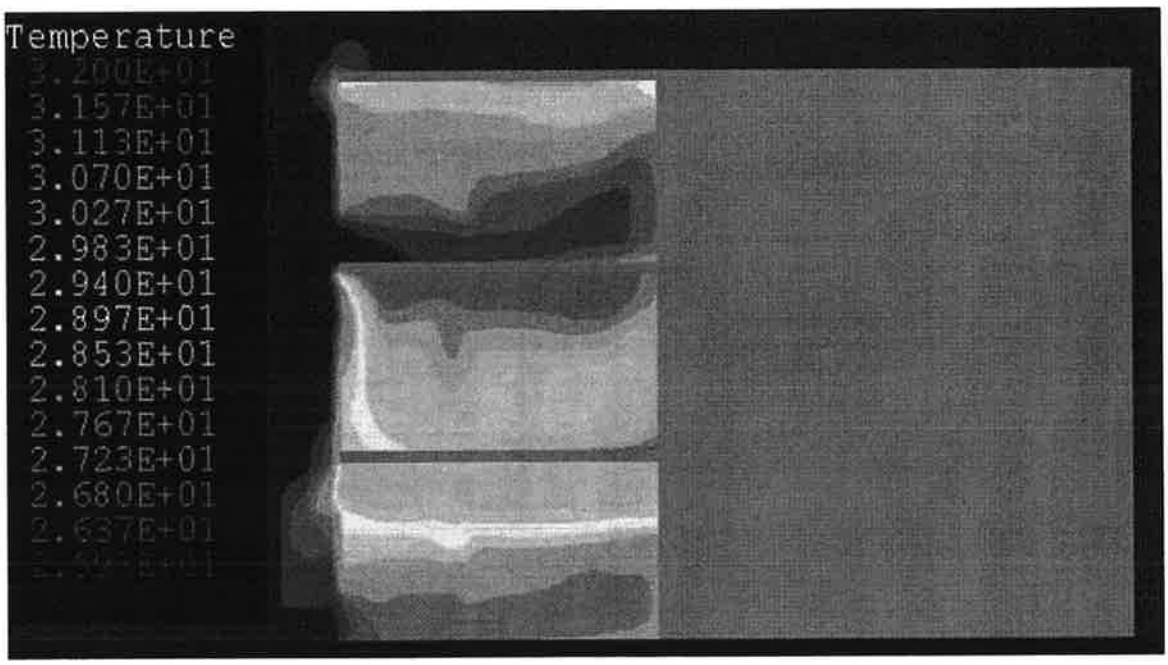
Room air velocity distribution
(combined wind and stack effect)



Single-sided Ventilation:

Room air temperature distribution

(combined wind and stack effect)



32

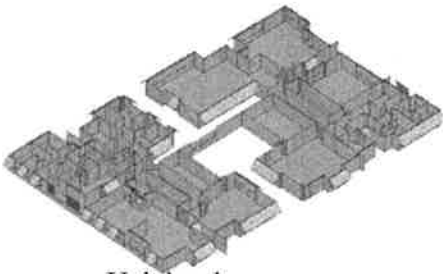
BCA GM – CFD Methodology

- To ensure computational domain length over development length – minimum 3 times
- Info on surrounding buildings
- Mesh size – to comply with GM criteria
- Clear geometry description (2D & 3D). Window schedule
- Unit level selection - proper tabulation of pressure differential data from estate level simulation result
- Turbulence model used? Atmospheric Boundary Layer implementation?
- Results converged? Show residual plot & time history of monitor point

BCA Green Mark Projects



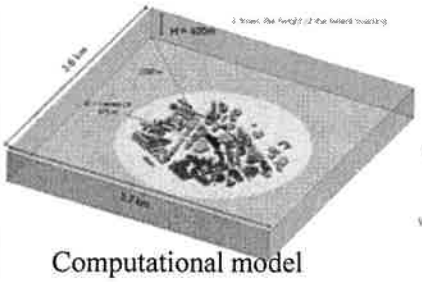
Computational model



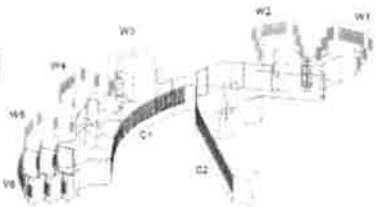
Unit level geometry



CFD result



Computational model



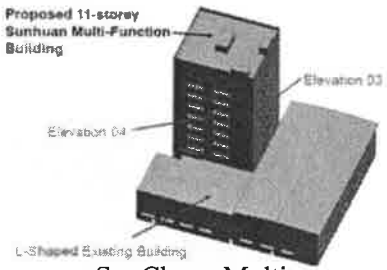
Unit level geometry



CFD result

33

BCA Green Mark Assessment for Natural Ventilation Simulation



SunChuan Multi-Function Buildings



NTU North Hill Residential hall



Singapore Polytechnics Sport Arena



HDB Water Cascadia

34

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

Strategies At Estate Level To Achieve Good Natural Ventilation

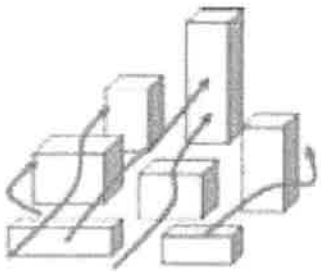
1. Create flow paths.
   - Link open spaces
   - Create open plazas at road junctions
   - Low-rise structures along routes of prevailing wind
   - Greater road widths to increase overall permeability



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

Strategies At Estate Level To Achieve Good Natural Ventilation

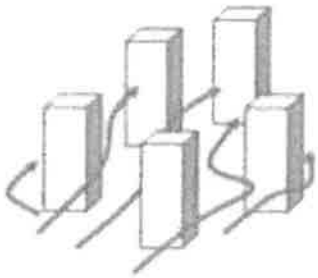
2. Arrange buildings according to ascending heights.
   - Lower rise buildings in front and towards direction of prevailing wind.



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

Strategies At Estate Level To Achieve Good Natural Ventilation

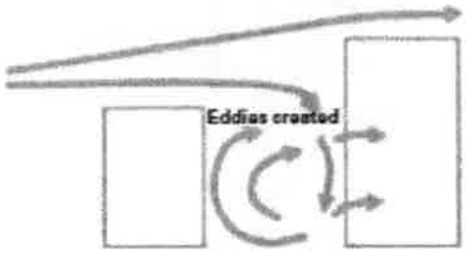
3. Stagger buildings.
   - Blocks behind should be able to receive the wind penetrating through the gaps between the blocks in the front row.



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

Strategies At Estate Level To Achieve Good Natural Ventilation

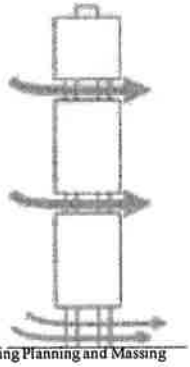
4. Create downwash wind.
   - Building design should consider capturing the downwash wind to reach the street level. Eddies may form in the canyon and this may assist in providing air movement within the canyon as well as into the buildings.



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

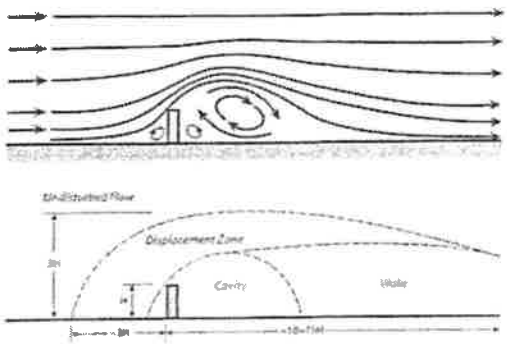
Urban Canopy Flow

- 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

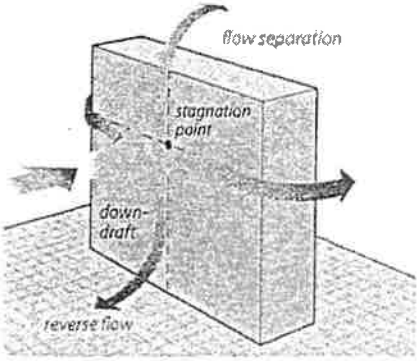
Reference : Urban microclimate : designing the spaces between buildings

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)

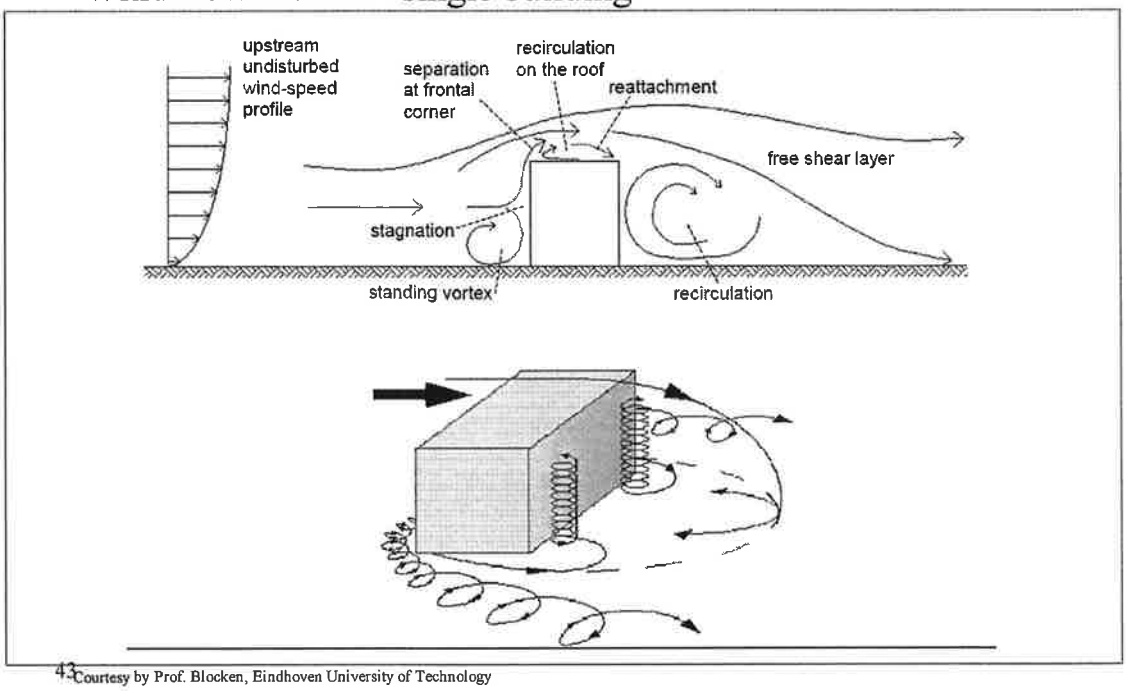


Wind flow pattern around tall slab building

Reference : Urban microclimate : designing the spaces between buildings

Urban Canopy Flow

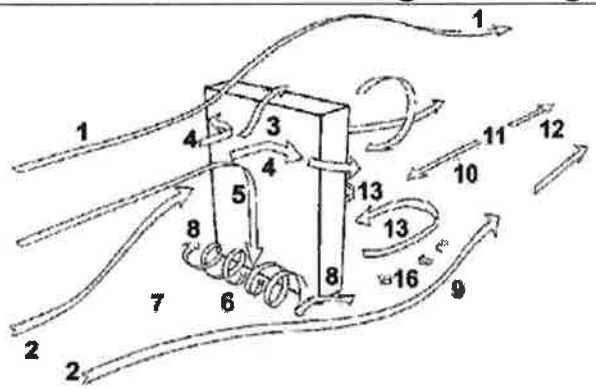
- Wind flow around a single building



43 Courtesy by Prof. Blocken, Eindhoven University of Technology

Urban Canopy Flow

- Wind flow around a single building

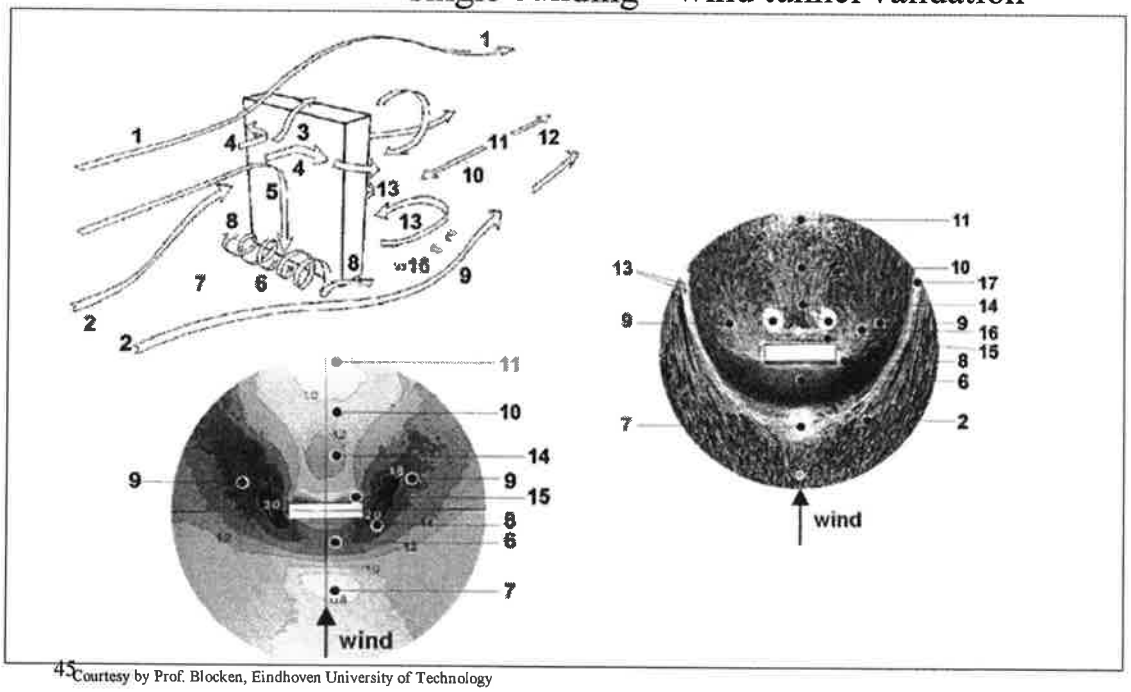


1. Flow over building
2. Oncoming flow
3. Flow from stagnation point over building
4. Flow from stagnation point around vertical building edges
5. Downflow from stagnation point
6. Standing vortex, base vortex or horseshoe vortex
7. Stagnation flow in front of building near ground level
8. Corner streams (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

44 Courtesy by Prof. Blocken, Eindhoven University of Technology

Urban Canopy Flow

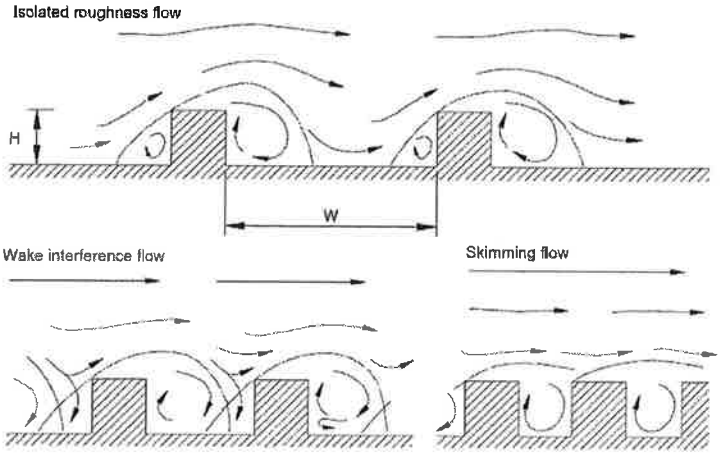
- Wind flow around a single building – wind tunnel validation



45 Courtesy by Prof. Blocken, Eindhoven University of Technology

Urban Canopy Flow

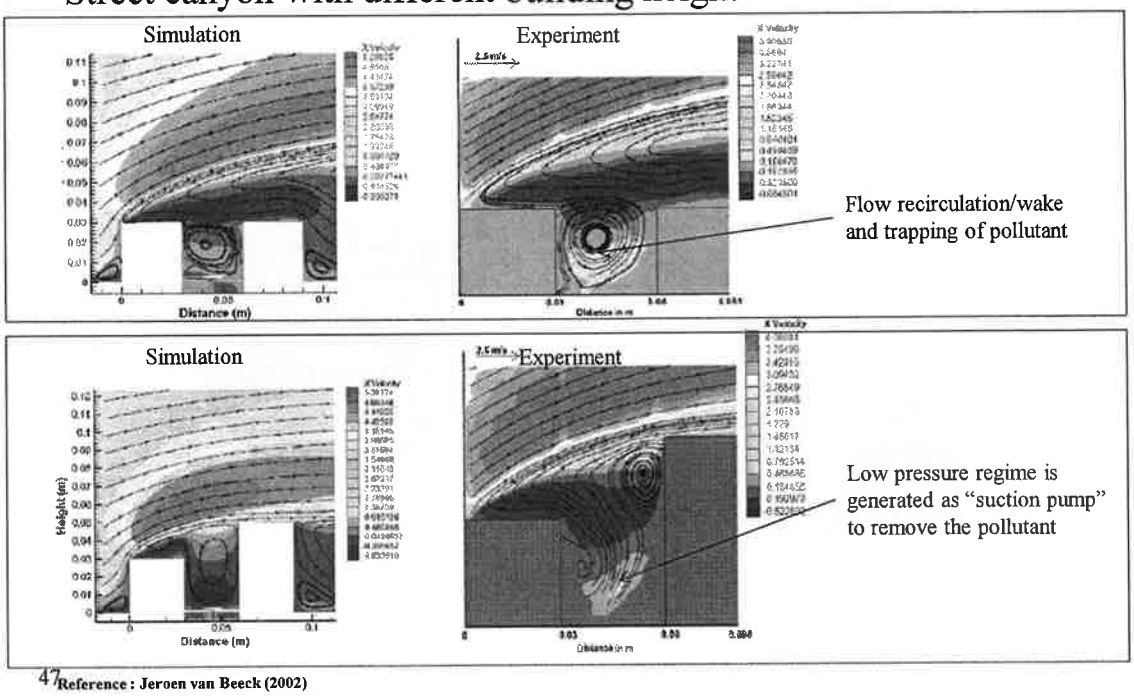
- Wind in the urban canopy – wind flow regimes
  1. Isolated Roughness Flow (IRF) → H/W<0.3
  2. Wake Interference Flow (IRF) → 0.3<H/W<0.7
  3. Skimming Flow (SF) → H/W>0.7



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

Urban Canopy Flow

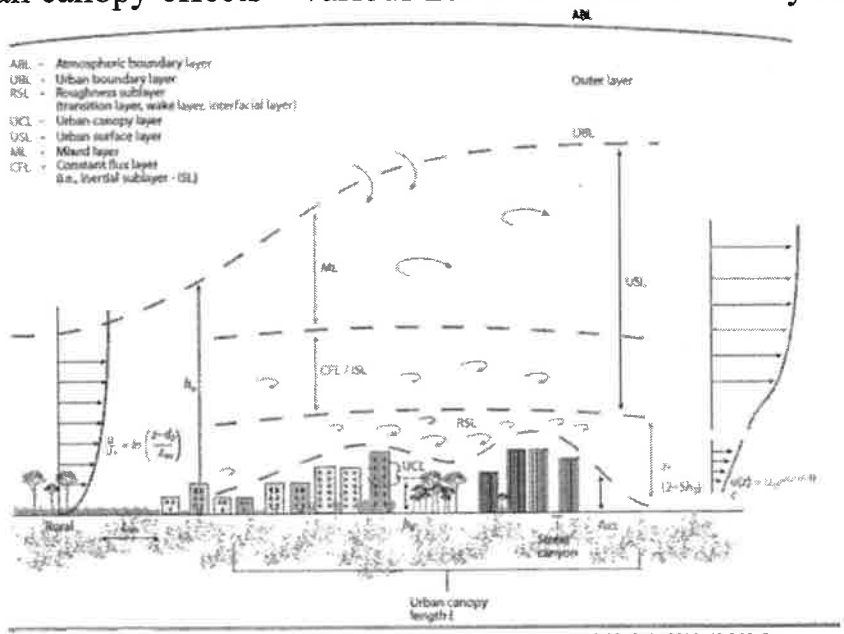
- Street canyon with different building height



47 Reference : Jeroen van Beeck (2002)

Urban Canopy Flow

- Urban canopy effects – various zones of urban boundary layer



48 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)

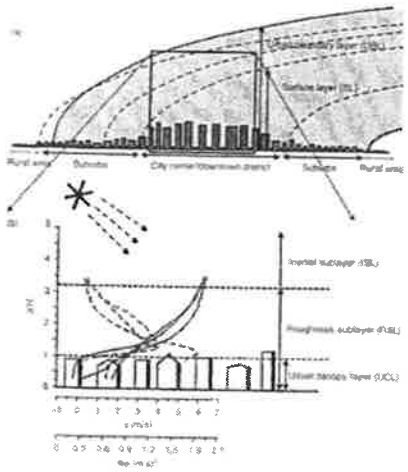


Fig 9.1 – Metropolitan sustainability

49 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)

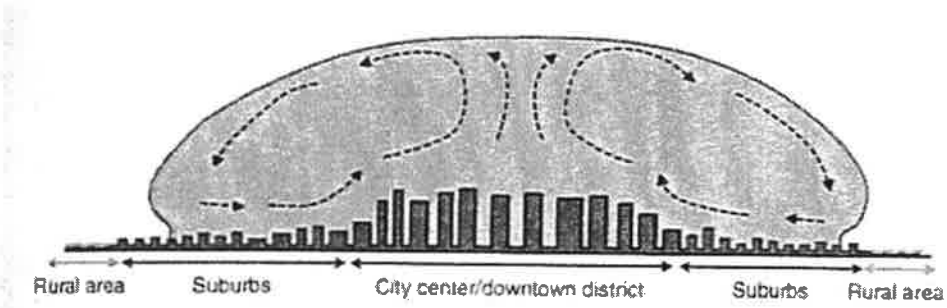
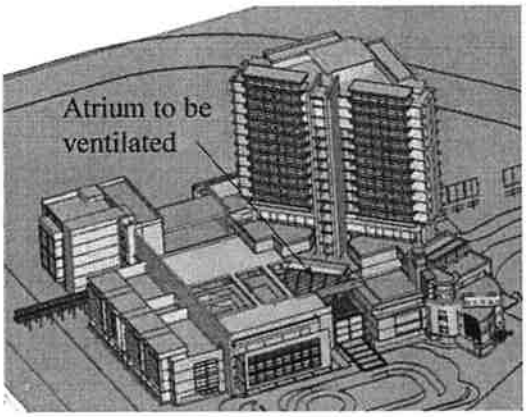


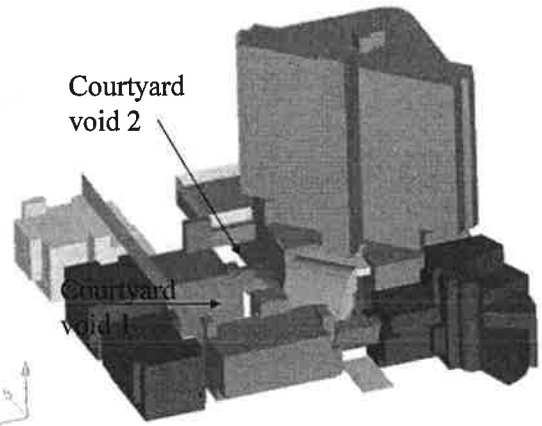
Fig 9.2 – Metropolitan sustainability

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

Architectural Strategy For Public Podium Spaces Ventilation

- To reduce energy cost, only air-conditioning is planned for working spaces which necessitate full environmental control and to adopt naturally ventilation strategies where possible, in transition spaces, e.g. corridors and lobbies.





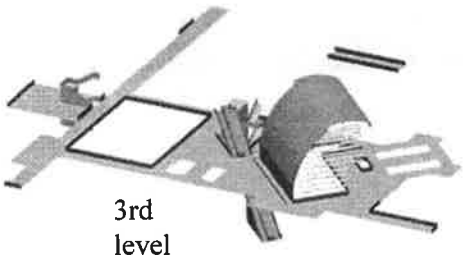
Artist Impression Drawing

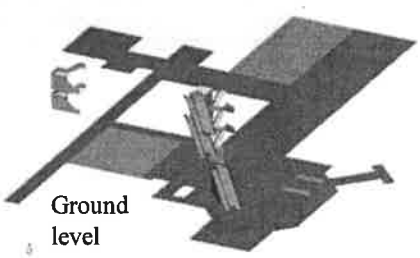
Simplified CFD Model

*Alex Lee, Joseph Lim Ee Man, **Hee Joo Poh**, George Xu Xiang Guo, Kurichi Kumar. Andrew Meier, Shane Amaratunga, CFD simulation in the architectural Strategy for the ventilation of public podium spaces in a local hospital building, CIB World Building Congress, 2 – 7 May 2004, Toronto, Canada*

Architectural Strategy For Public Podium Spaces Ventilation

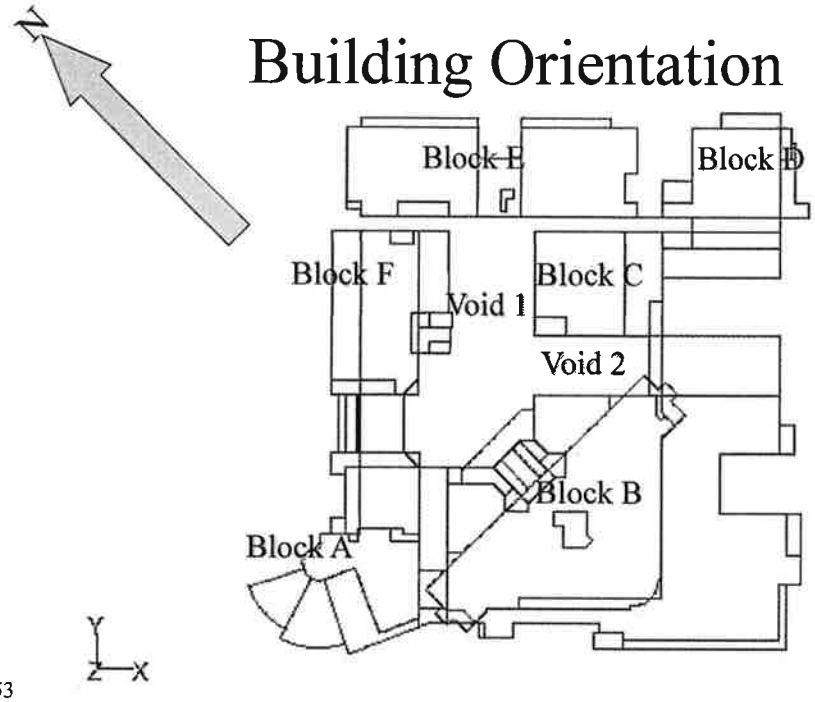
- Creation of wind path through deep podium space in the building are affected by the following factors in the physical form of building:
  - Positions and shapes of neighbouring buildings
  - Depth of building space between inlet and outlet
  - Shape and positions of building parts adjacent to inlet and outlet of wind openings.
  - Size and extent of wind openings
  - Restriction to wind flow in wind passage
  - Alignment of building openings to prevalent wind direction



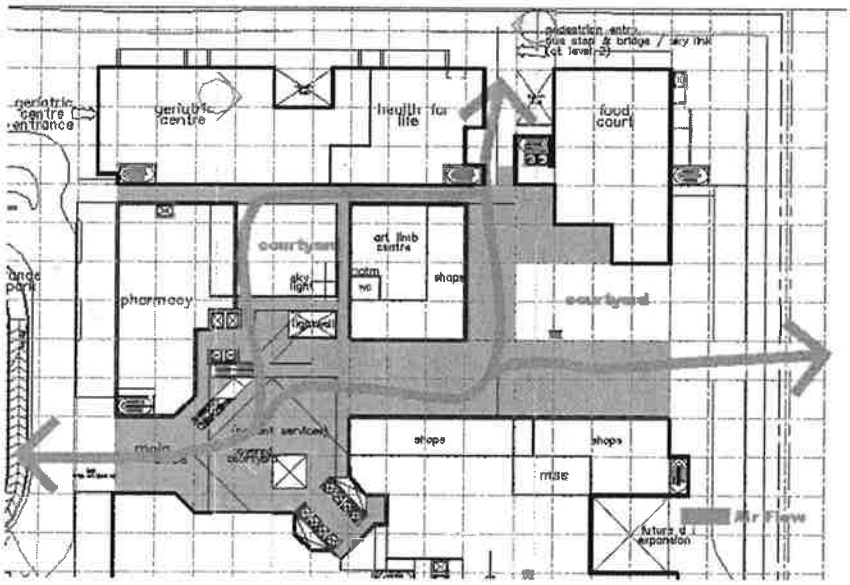


Architectural Strategy For Public Podium Spaces Ventilation

Building Orientation



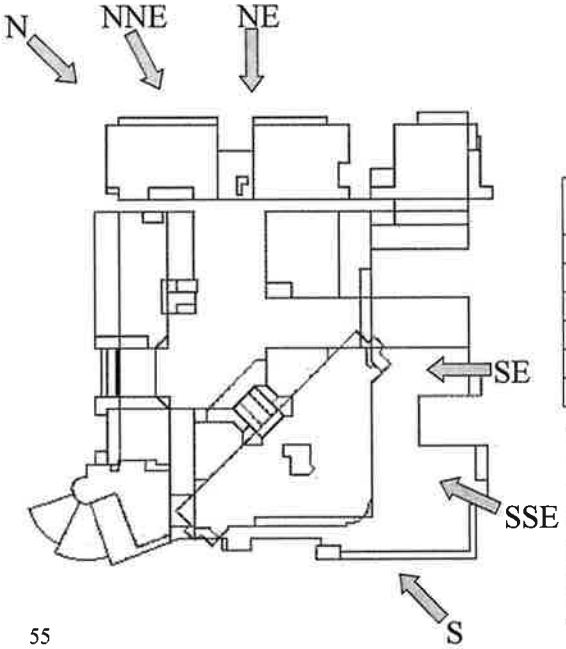
Architectural Strategy For Public Podium Spaces Ventilation

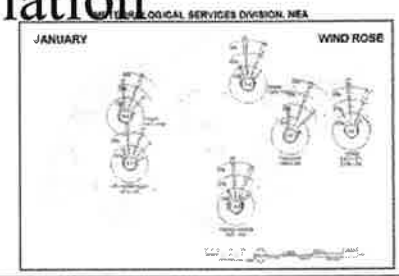


Desired Wind Path Direction

Architectural Strategy For Public Podium Spaces Ventilation [55]

Various Cases of CFD Simulation



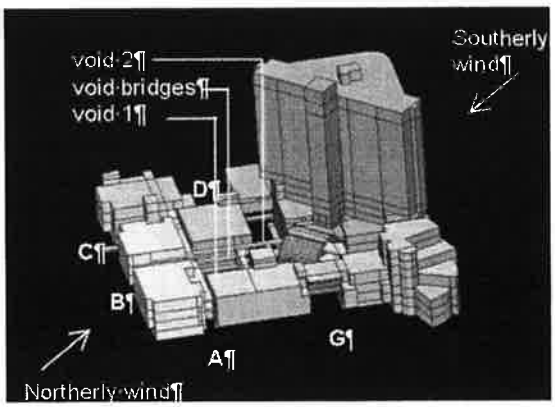


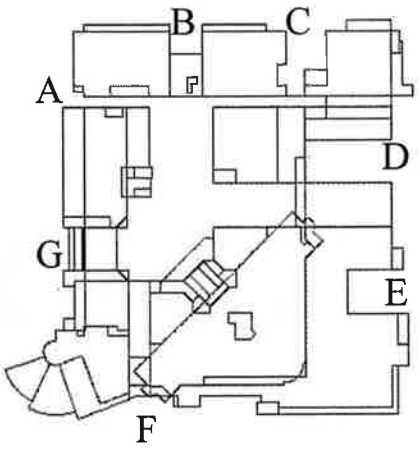
Month	% frequency of mean hourly wind from various direction				
	N	NNE	NE	Total	Calm
Dec	26.2	18.9	6.4	51.5	14.5
Jan	32.7	33.2	6.7	72.6	6.9
Feb	27.4	34.9	11.2	73.5	8.8
Mar	13.8	25.2	15.5	54.5	17.0
Average	25.0	28.1	10.0	63.0	11.8

Month	% frequency of mean hourly wind from various direction				
	SE	SSE	S	Total	Calm
June	10.0	11.7	18	39.7	22.4
July	14.0	15.7	18.7	48.4	18.8
Aug	14.2	15.2	19.5	48.9	17.2
Average	12.7	14.2	18.7	45.7	19.5

Architectural Strategy For Public Podium Spaces Ventilation [56]

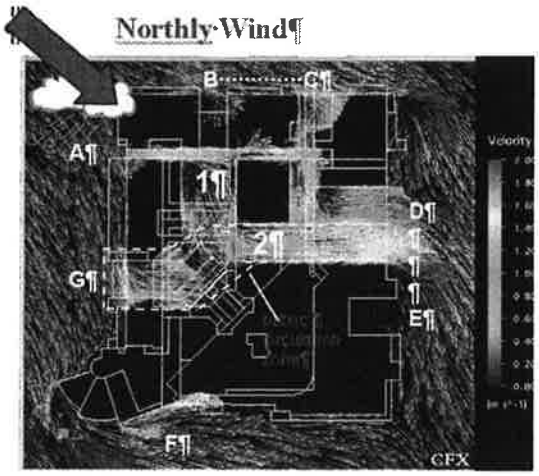
Geometrical Model



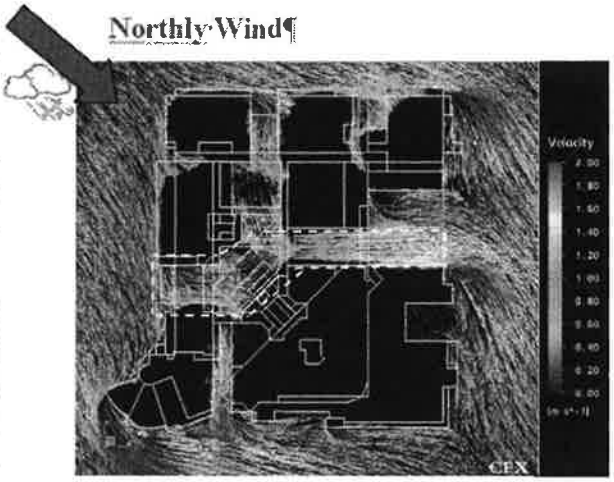


Different Openings (Inlet or Outlet) for Wind Passage

Architectural Strategy For Public Podium Spaces Ventilation



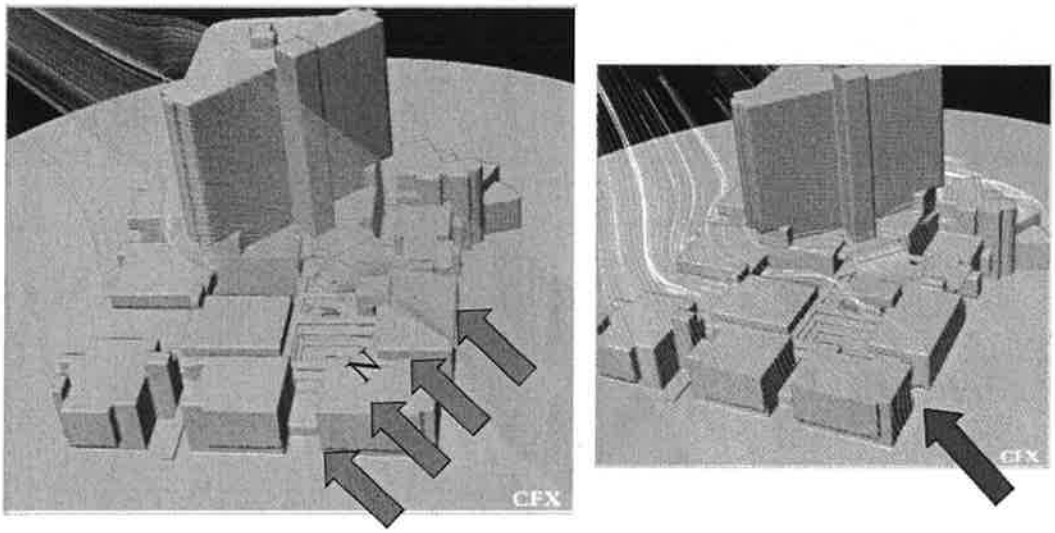
Velocity contour at 1.5m above ground



Velocity contour at 5m above ground

- G,A,B & C are wind opening inlets
- High wind velocity at opening 2 is attributed to the downwash of wind from the flat face of tower

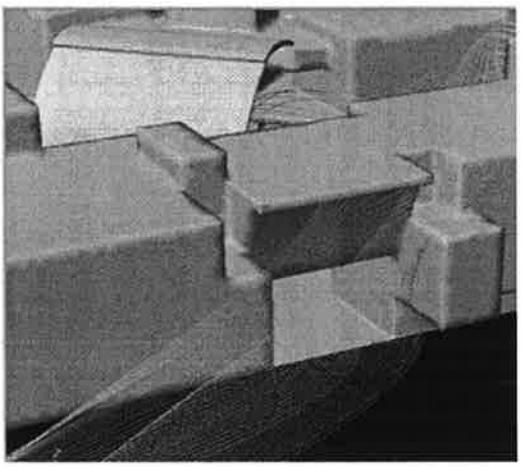
Architectural Strategy For Public Podium Spaces Ventilation

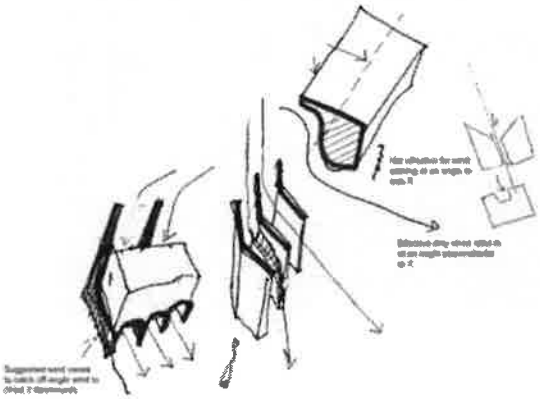


Streamline flowing across building envelop

- The downwash benefits spaces after void 2 at the eastern portions of buildings, but does not benefit the atrium space

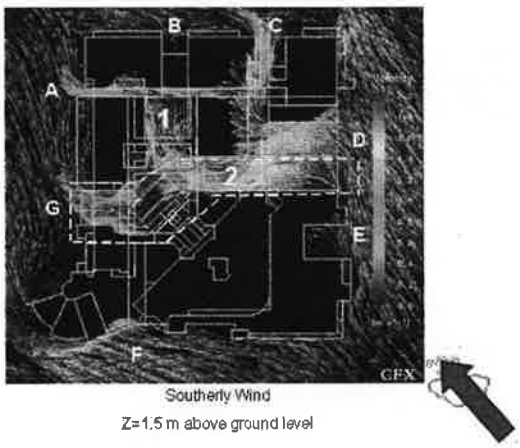
Architectural Strategy For Public Podium Spaces Ventilation



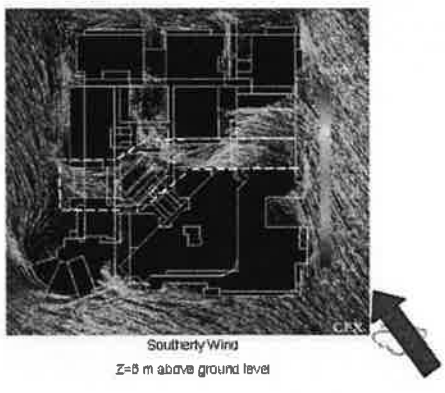


- To draw ambient air into podium through opening G, the scoop could be replaced by vertical guided vanes

Architectural Strategy For Public Podium Spaces Ventilation



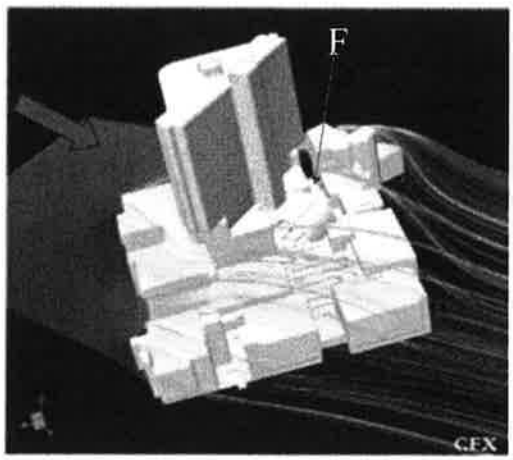
Velocity contour at 1.5m above ground



Velocity contour at 5m above ground

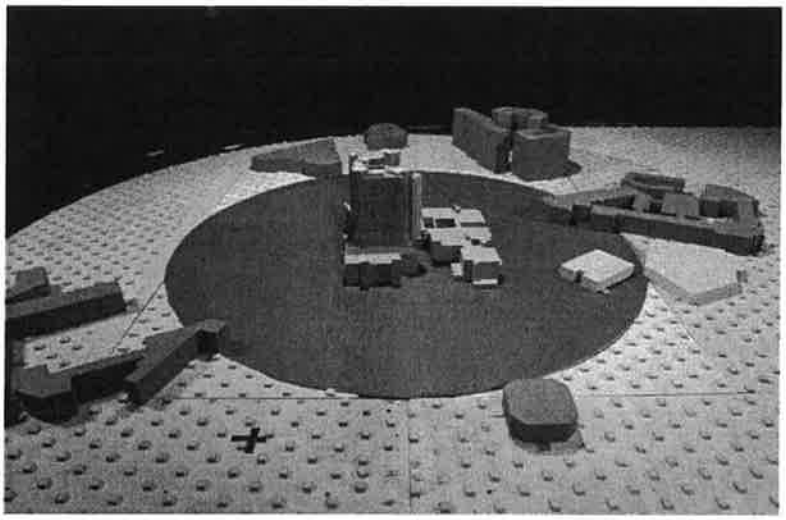
- opening D is a major inlet drawing in significant wind flow from the southerly winds at velocities above 1.8 m/s but exiting at a diminished velocity of 0.8 m/s at outlets G, A, C.

Architectural Strategy For Public Podium Spaces Ventilation



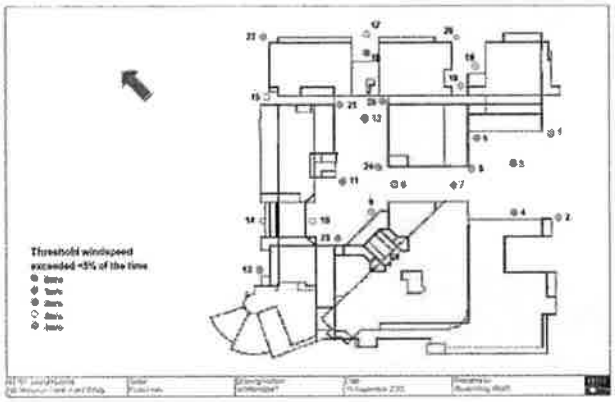
- For opening F to be effective as an inlet, the built-up volume should be removed to form a wind passage leading directly to the atrium

Architectural Strategy For Public Podium Spaces Ventilation

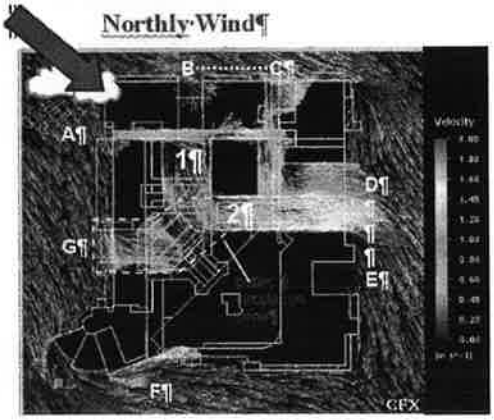


Wind tunnel model (Courtesy of BMT Fluid Mechanics Limited)

Architectural Strategy For Public Podium Spaces Ventilation [63]



Experimental data (Courtesy of BMT Fluid Mechanics Limited)

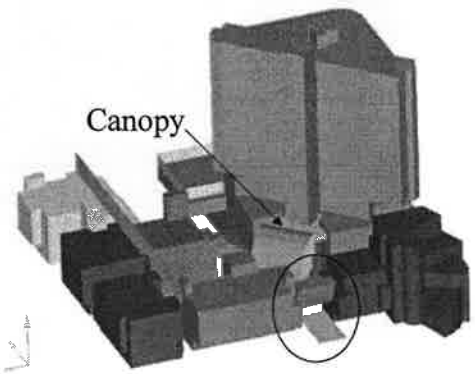


Simulation results

Architectural Strategy For Public Podium Spaces Ventilation [64]

Concluding Remarks

- The primary openings in the building podium façade are not aligned to prevalent wind directions;
- The wind flow is not encouraged through the building podium because there are no connected pathways to minimize obstructions to flow;
- Canopy could be built higher to scoop the downwash effect for ventilating corridor space



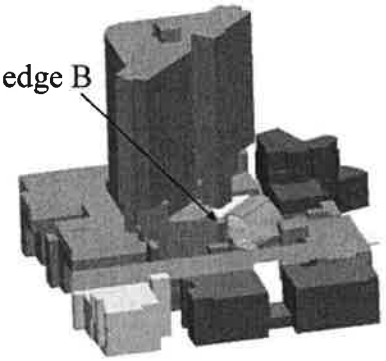
Podium facade

Architectural Strategy For Public Podium Spaces Ventilation

Concluding Remarks

- Multi-level bridges surrounding the atrium obstruct wind flow by dissipating wind energy from ground level up to the full height of the atrium; and

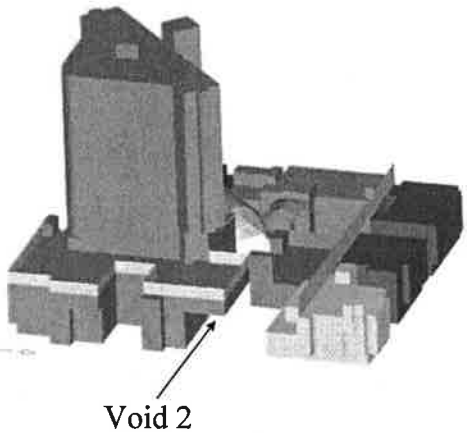
- Chambering of building edge B would help to catch ambient air flow into atrium area - during mid year wind condition



Architectural Strategy For Public Podium Spaces Ventilation

Concluding Remarks

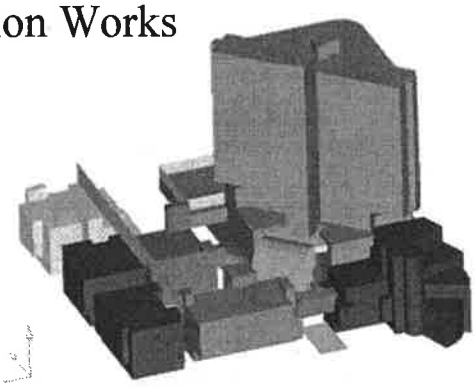
- Downdrafts flowing from Northerly wind to the tower face through the void 2 do not benefit the atrium space which needs to be ventilated most



Architectural Strategy For Public Podium Spaces Ventilation

Potential Extension Works

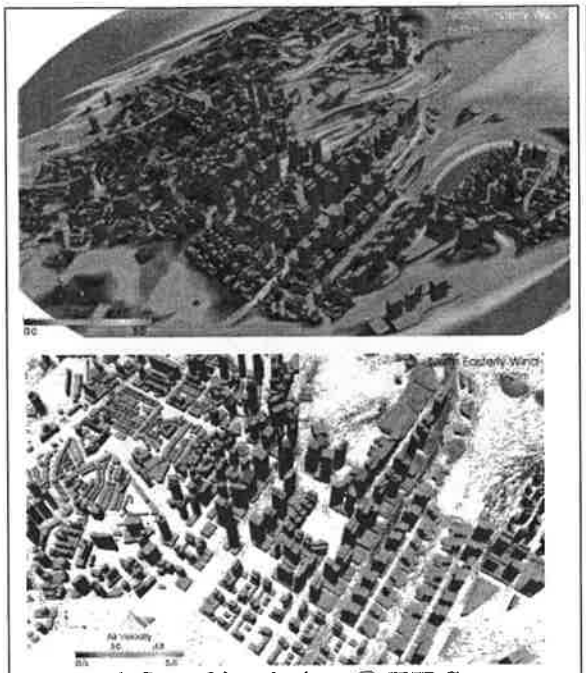
- More parametric studies on the factors affecting wind path through atrium space of the big buildings should be carried out in order to from the database for architect and building designers for further references



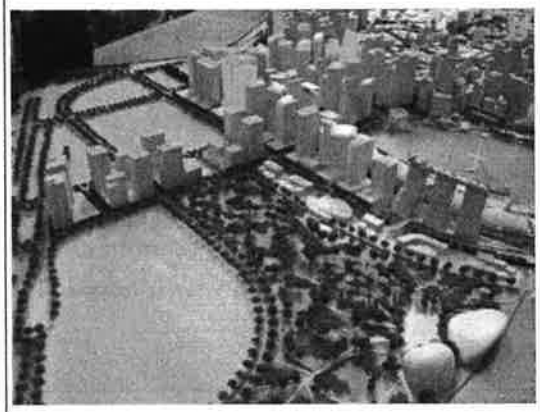
1) Relative position and opening size between inlet & outlet
2) Restriction of wind flow in the wind passage
3) Alignment of building opening to prevalent wind direction
4) Depth of atrium space between inlet and outlet
5) Size and shape of building adjacent of inlet & outlet
6) Positions and shapes of neighbouring blocks

CFD model demonstration

Urban Air Flow Simulation



Airflow Simulation @ IHPC



Architecture Model @ URA

CFD model demonstration

Urban Air Flow Simulation

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**)

+

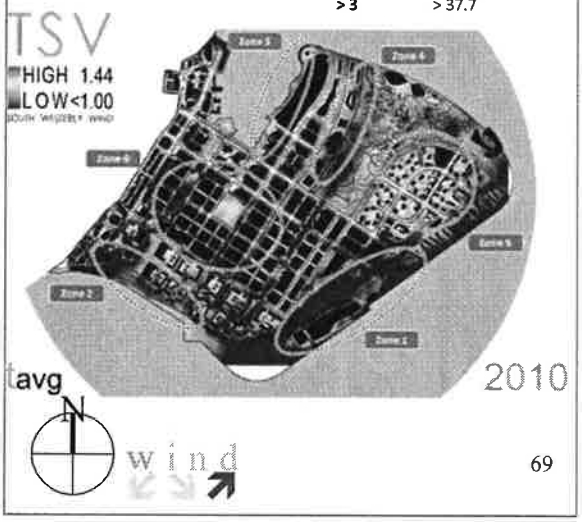


Wind Map (**CFD**)

69

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

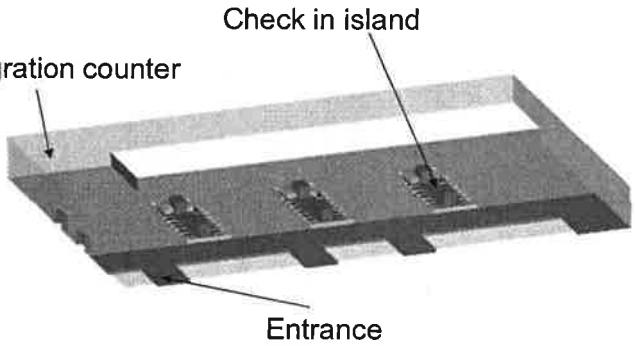


69

CFD model demonstration

Indoor Environmental Quality at Changi Terminal 3



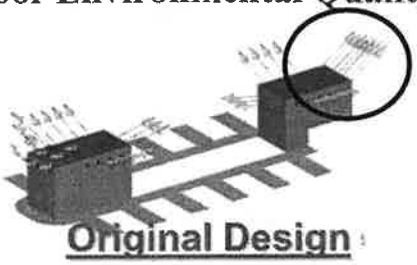


Objective:

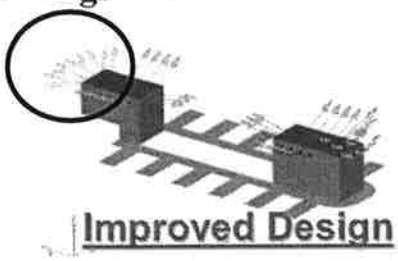
To provide a performance-based simulation for the ACMV design in Changi Airport Terminal 3

CFD model demonstration

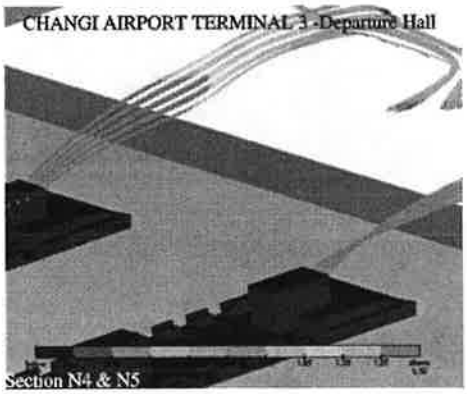
Indoor Environmental Quality at Changi Terminal 3

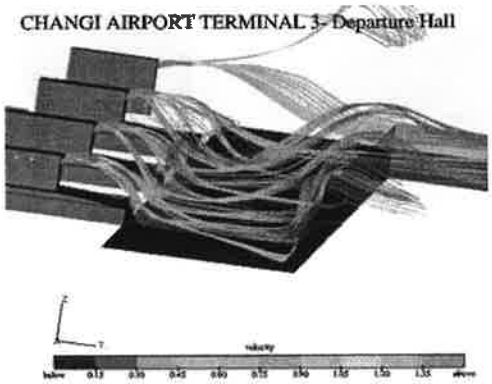


Original Design



Improved Design

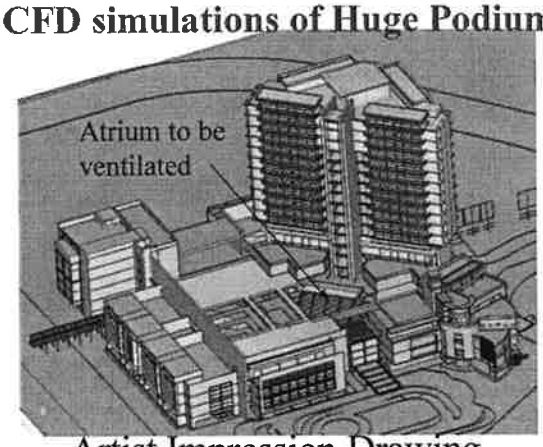




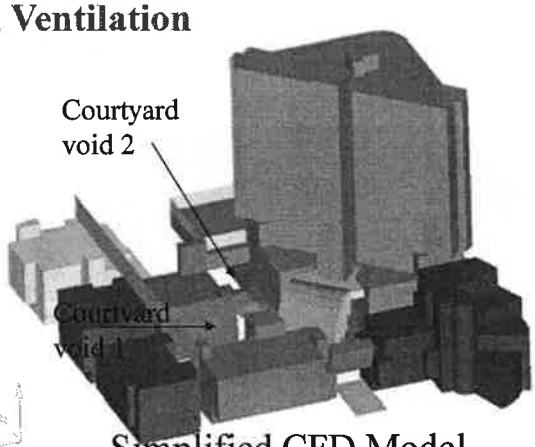
Flow from Jet Nozzle – to optimize throw angle

CFD model demonstration

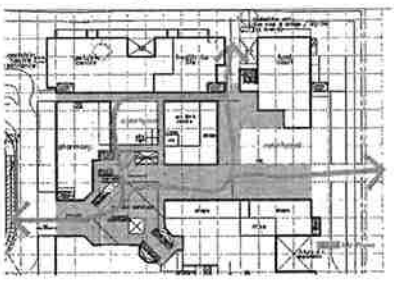
CFD simulations of Huge Podium Ventilation



Artist Impression Drawing



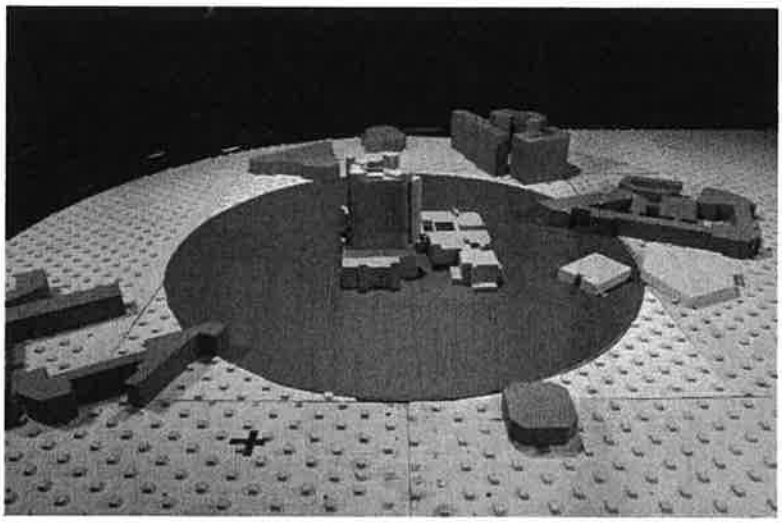
Simplified CFD Model



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

CFD model demonstration

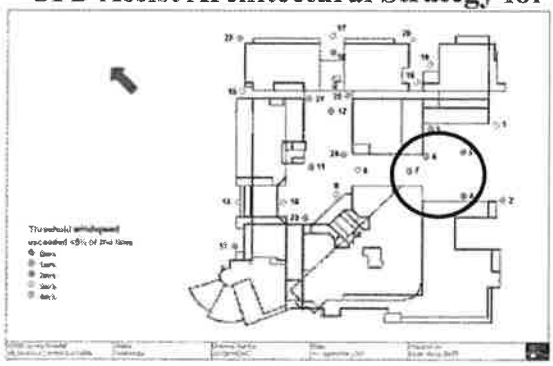
Wind Tunnel tests to validate CFD Simulations

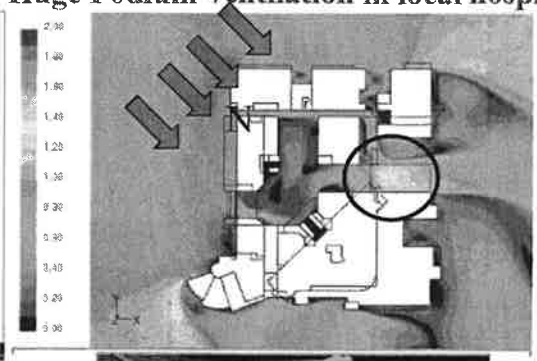


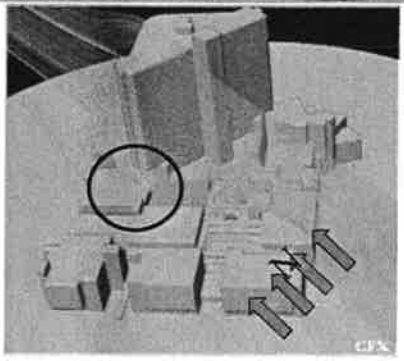
Wind tunnel model (Courtesy of BMT Fluid Mechanics Limited)

CFD model demonstration

CFD Assist Architectural Strategy for Huge Podium Ventilation in local hospital





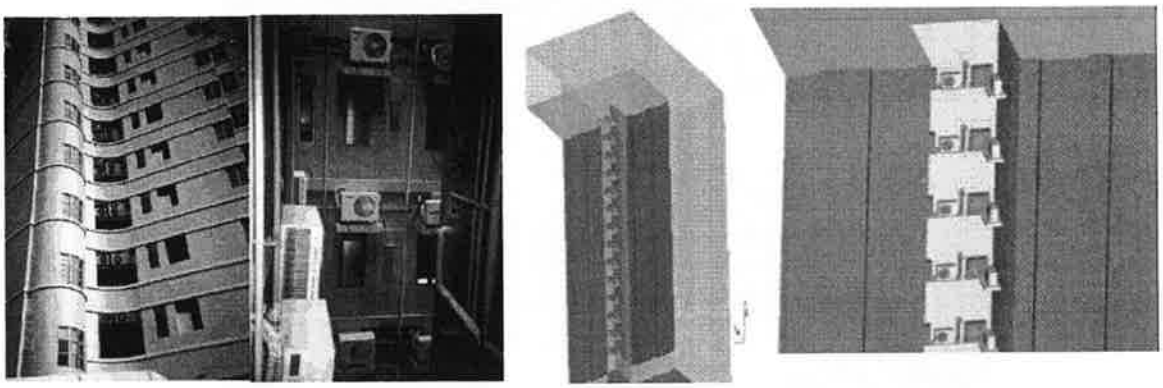


Comparison between Wind Tunnel & CFD simulation results

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

CFD model demonstration

Hot air rejection from condensers into air well

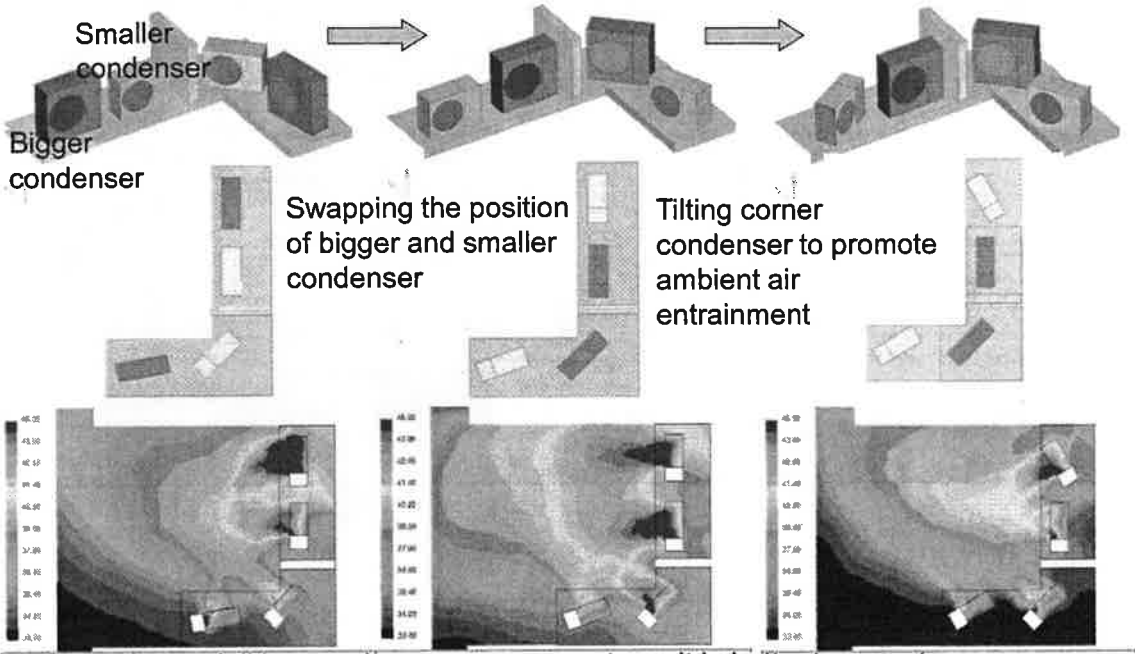


Heat accumulated at the top part of recess space in condominium and deteriorate condenser performance

Objective:

To derive optimum design for condensers arrangement in air well

CFD model demonstration

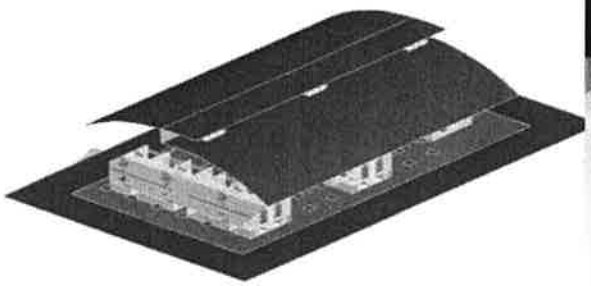


Swapping and tilting condenser arrangement result in better temperature reduction in air well

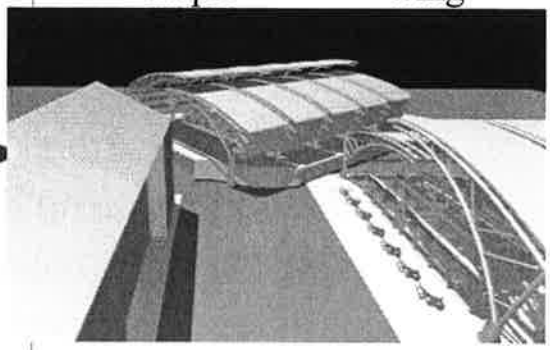
CFD model demonstration

CFD Simulations for the Jet Fan Ventilation System at Food Centre at Bedok Neighbourhood 6

CFD Model

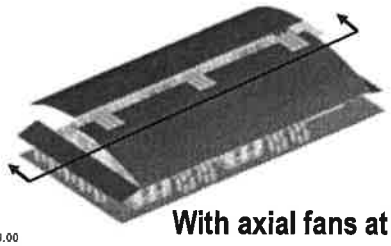


Artist Impression Drawing

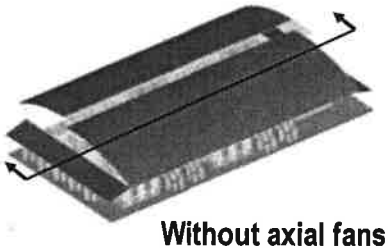


Objective:
To verify the performance of the ductless hood exhaust ventilation system at hawker centre in Bedok Neighbourhood 6

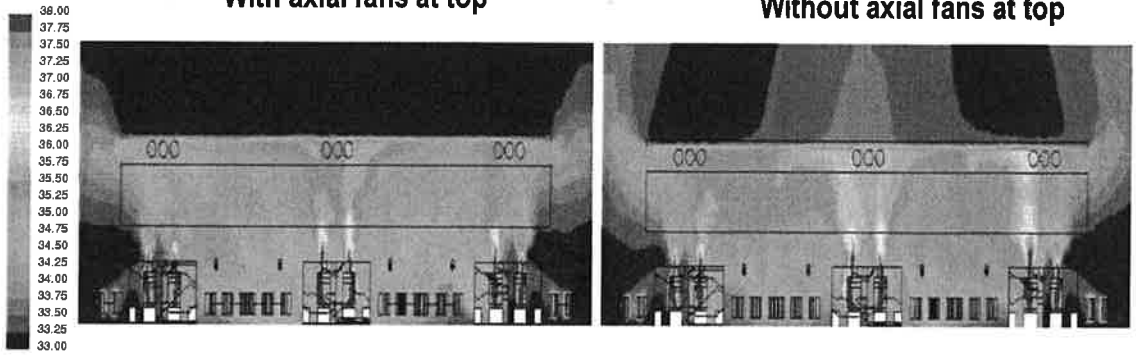
CFD model demonstration



With axial fans at top



Without axial fans at top



Even without the roof axial fans, the hot air can still rise up and escape from the roof opening gap effectively

Day 2 (11:00am – 12:30pm)

- Introduction to CFD in Green Building Application
  - BCA guidelines for running a ventilation simulation
  - Challenges and Opportunities in using CFD as a tool for design optimization
  - 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

Proposal Preparation for CFD Ventilation Project

- Problem Statement
  - In order to achieve BCA Green Mark Platinum certification (GM Version xx) for this new project, ventilation simulation modelling and analysis are to be carried out to ensure good natural ventilation with minimum weighted average wind velocity of 0.6 m/s (satisfying criteria) within functional space and unit. Additional requirement raised by BCA assessor (if any) for the new CFD guideline on the building premise such as air change per hour or PMV/PPD will also be addressed. CFD modelling tool is employed to study the natural ventilation design on the new project.

Proposal Preparation for CFD Ventilation Project

- Project Background
  - Context Relationship and Artist Impression of the project
  - Concept Diagram of Natural Ventilation Design
  - Sectional cut to illustrate the Natural Ventilation Design

- Project Objectives
  - The objective for this study is employ CFD modelling tool to identify the most effective building layout and design to ensure good natural ventilation for the new project; in accordance to BCA's Green Mark Version xx requirements

Proposal Preparation for CFD Ventilation Project

- Proposed Methodology
  - 3D steady state airflow CFD analysis for the new facility, subjected to the natural ventilation requirement and methodology of functional space simulation as stipulated BCA's Green Mark Version xx.
- Simulation Domain
  - To include the new facility, as well as the surrounding blocks within 3 times of development length as stipulated in BCA's Green Mark Version 4.1. Topographical terrain to be included, if necessary. Approximately 1km x 1km x 500m of simulation domain and an approximate of five (5) millions tetrahedral cells for the CFD analysis will be generated (by complying to the GM requirement of 0.5 to 1.0m near the buildings & ground level and 10m at the far field boundary)

Proposal Preparation for CFD Ventilation Project

- Proposed work scopes
  - **First Review of Simulation (Baseline Design Cases):** to study any recommendations needed in order to achieve the natural ventilation criteria in accordance to BCA's Green Mark Version xx requirements. A maximum number of three (3) modified geometries, and six (6) simulation case studies will be performed. Each case is subjected to two (2) prevailing wind conditions. The project duration is one (1) working month.
  - **Simulation for the Green Mark submission to BCA (Improved Design Cases)**: A maximum number of three (3) modified geometries, and six (6) simulation case studies will be performed. Each case is subjected to two (2) prevailing wind conditions. The project duration is one (1) working months.

Proposal Preparation for CFD Ventilation Project

- Input from client/developer
  - Detailed drawing of new facility (preferably in AutoCAD or BIM-REVIT format),
  - ETTV values and Shading Coefficient (SC)of Roof Materials – to determine heat flux transferred from roof level to indoor environment
  - Wind condition around the new NTU Sport Hall (if any); otherwise this will be taken from Singapore Metrological Station in BCA Green Mark methodology
  - Any relevant measurement data or practical observation available, and
  - Feedbacks and inputs (discussions) during the simulation process
- Input from ESD Consultant
  - CFD expertise,
  - CFD software tools – to specify
  - Computational facilities

Proposal Preparation for CFD Ventilation Project

- Deliverable
  - 3D steady-state CFD airflow simulation and analysis for First Review of Simulation (6 cases)
  - 3D steady-state CFD airflow simulation and analysis for Simulation of the Green Mark submission to BCA (6 cases).
  - Presentation and report
- Project Duration
  - Typically 2 months

Proposal Preparation for CFD Ventilation Project

- Project Schedule

Weekly Project Task	1	2	3	4	5	6	7	8
1. Receive drawing from client and start the construction of 3D Geometry model (with necessary clean up) and meshing. Domain will include 3 times development length. To include topographical terrain, if necessary								
1. CFD Simulation for First Review (Baseline Design Cases) - A maximum number of three (3) modified geometries, and six (6) simulation case studies will be performed								
1. Meeting with BCA Green Mark Officer to seek comment and input on Baseline Design Cases in order to proceed to the Improved Design Cases								
1. CFD Simulation for the Green Mark submission to BCA (Improved Design Cases) - A maximum number of three (3) modified geometries, and six (6) simulation case studies will be performed								
1. Final Report								

Why Use CFD for Natural Ventilation?

- Lack of confidence in pressure coefficient data for wind-driven flow calculations
- Lack of knowledge of effective opening area (i.e. C_d)
- Difficulty in differentiating between air supply and extract openings (dependent on wind direction, turbulence, temperature difference, etc.)
- Calculations are needed for different wind directions but classical methods cannot accurately predict the effect of wind direction
- Usually room air movement predictions are needed in addition to air flow calculation
- The interaction between mechanical and natural (hybrid) systems is difficult to predict by standard calculation methods

CFD Modelling Techniques for NV

There are two approaches –

Internal flow simulation:

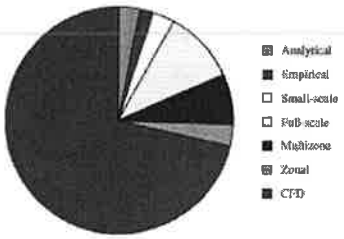
This requires boundary conditions at inlets & outlets which may be obtained from wind pressure data for the openings or CFD simulations for the whole site.

Coupled Internal and external flow simulation:

This requires climatic data, wind profiles and sheltering of surrounding structures.

Ventilation Performance Assessment Method: Review & Recent Application

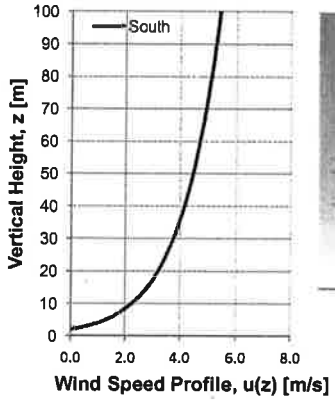
1. Analytical models – minimal contribution
2. Empirical models – minimal contribution
3. Reduced-scale experiments - generate data to validate numerical models
4. Full-scale experiments - generate data to validate numerical models
5. Multizone models - main tool for predicting ventilation performance 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

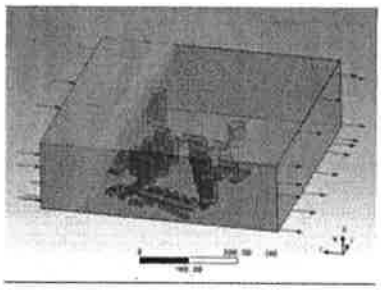


Qingyan Chen, Building and Environment, 44 (2009) 848–858

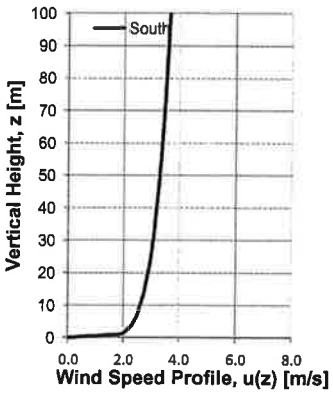
Natural Ventilation - Wind Speed

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





$$u(z) = u_{ref}\left(\frac{z}{z_0}\right)^{\alpha}$$

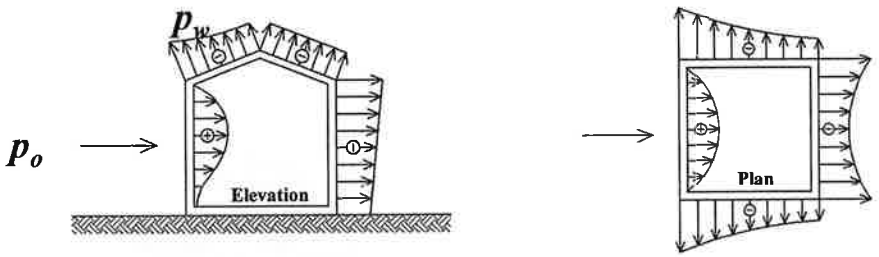


Natural Ventilation - Wind Pressure Coefficient

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

91

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.**

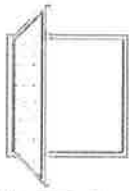
The effective area (A_{eff}) is given by: $A_{eff} = C_d A$ (7)

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.

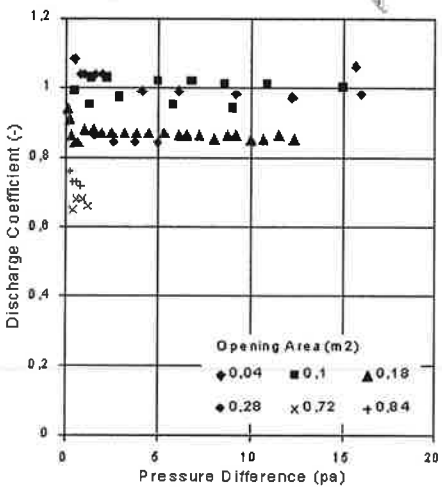
Data on C_d for natural ventilation openings is limited particularly its variation with wind direction. C_d is one of the main difficulties in manual calculation procedures and in network flow models for natural ventilation.

92

Flow through an Opening / …

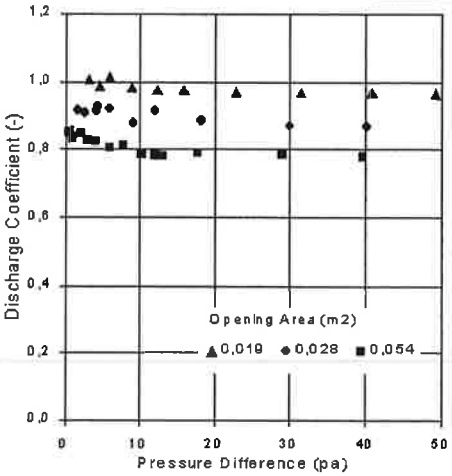


Discharge coefficient for side hung window



Discharge coefficient For bottom hung window

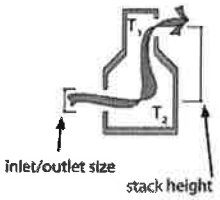


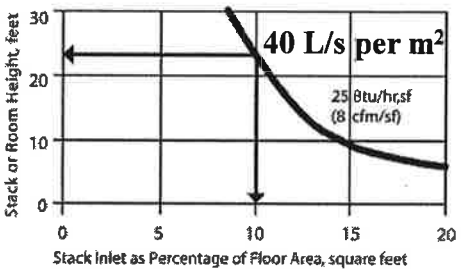


(Heiselberg *et al*, 2001)

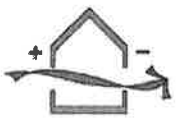
Simplified Flow Calculation for NV

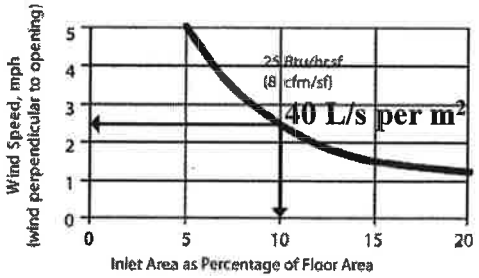
Stack





Cross-Flow



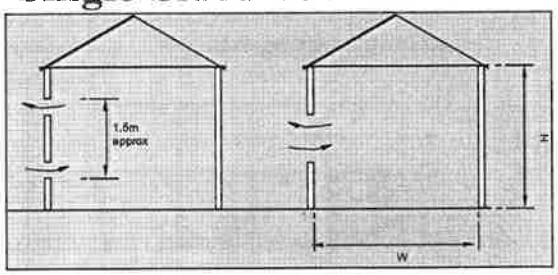


1 cfm/ft² = 5 L/s per m² 1 mph = 0.45 m/s

(Brown & Huang, 2006)

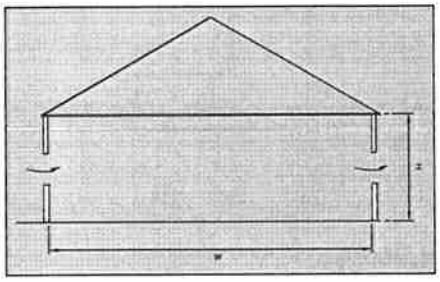
Natural Ventilation - Room openings

Single-Sided Ventilation



The depth should be < 2.5 H

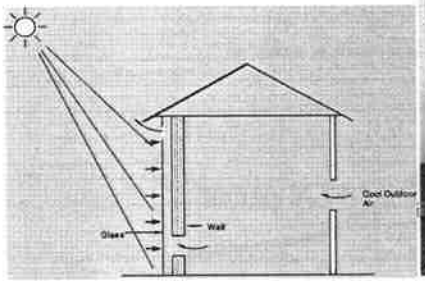
Cross-Ventilation



The depth should be < 5 H

95

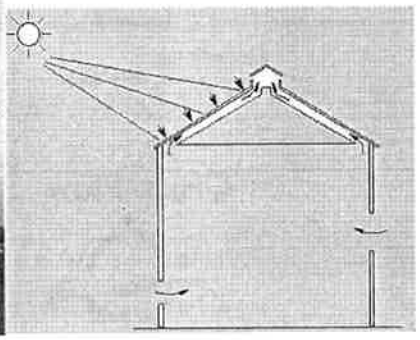
Natural Ventilation - Solar-Induced Ventilation



Solar Wall (Trombe Wall Ventilator)



Solar Chimney



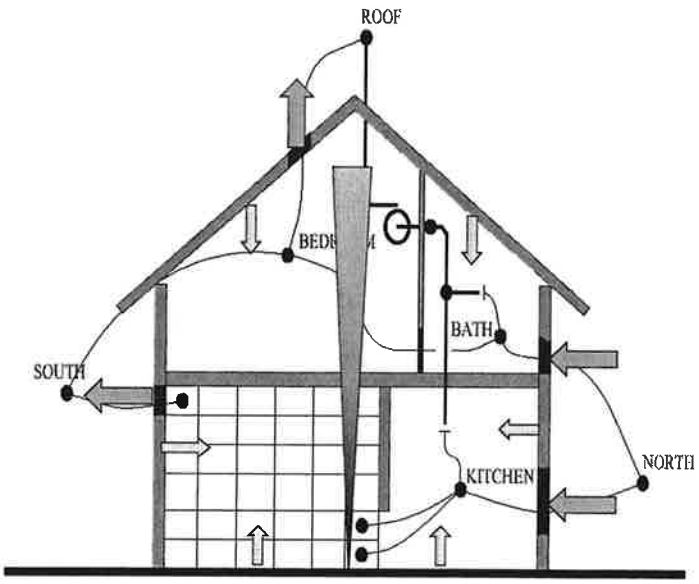
Solar Roof Ventilator

Information from ES and CFD

	ES	CFD
Weather and solar impact	Yes	No
Enclosure thermal behaviors	Yes	No
HVAC system capacity	Yes	No
Energy consumption	Yes	No
Thermal comfort (T, V, R, Turb)	No	Yes
Indoor air quality (C)	No	Yes
HVAC system distribution	No	Yes
Natural ventilation	No	Yes

97

Complementarities of ES and CFD: ES



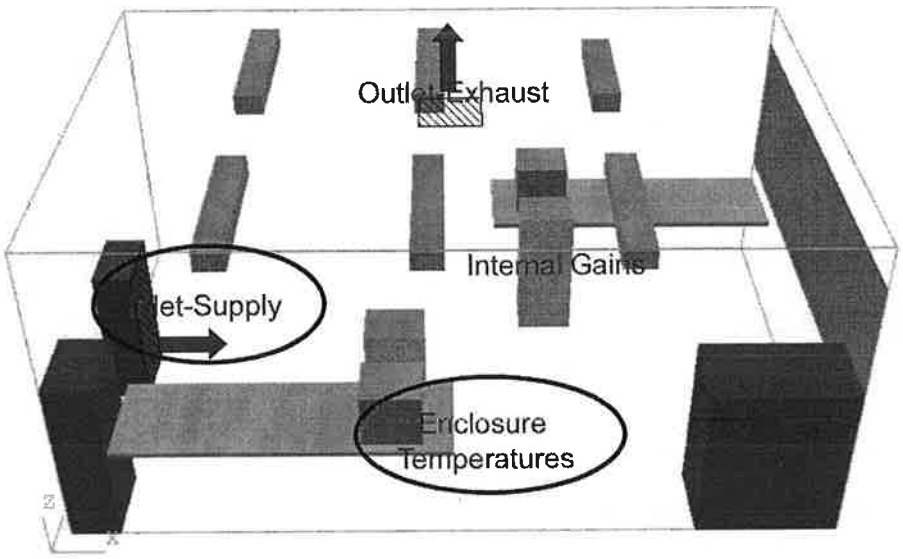
Ventilation Energy Loss
(up to 50% total energy demand)
Natural ventilation

Convective Heat Transfer

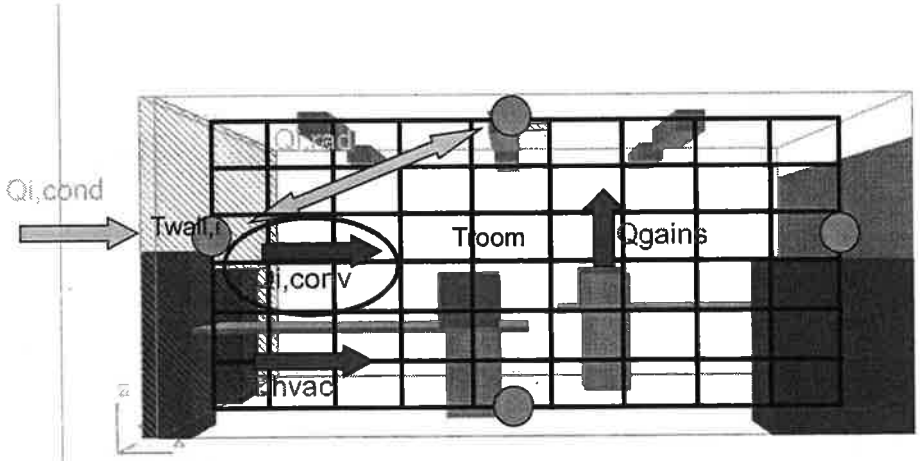
Temperature Stratification

98

Complementarities of ES and CFD: CFD

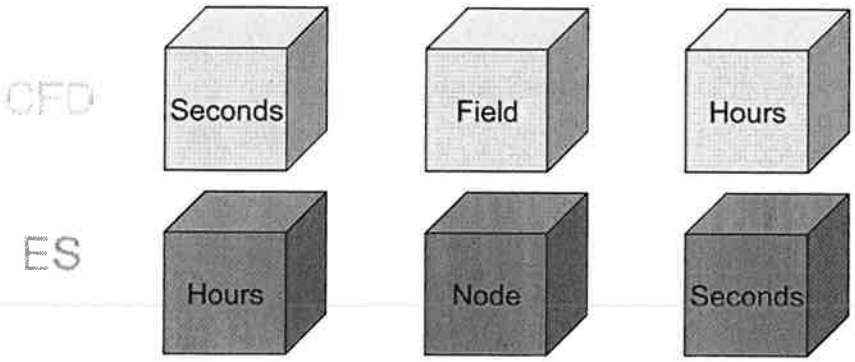


Integration of ES and CFD



- ❑
- ❑

Disparities between ES and CFD



Proposed CFD software

General Purposed CFD software

1. FLUENT (http://www.fluent.com/)
2. NUMECA (http://www.numeca.com/)
3. ANSYS-CFX (http://www.ansys.com/products/cfx.asp)
4. STAR-CD (http://www.cd-adapco.com/)
5. CFD-ACE+ (http://www.cfdrc.com/index.html)
6. FLOW3D (http://www.flow3d.com/)
7. CFDesign (http://www.cfdesign.com/)
8. FLUIDYN (http://www.fluidyn.com/)
9. ACUSIM (http://www.acusim.com/)
10. COMSOL (http://www.comsol.com/)
11. UrbaWiind (http://www.meteodyn.com)

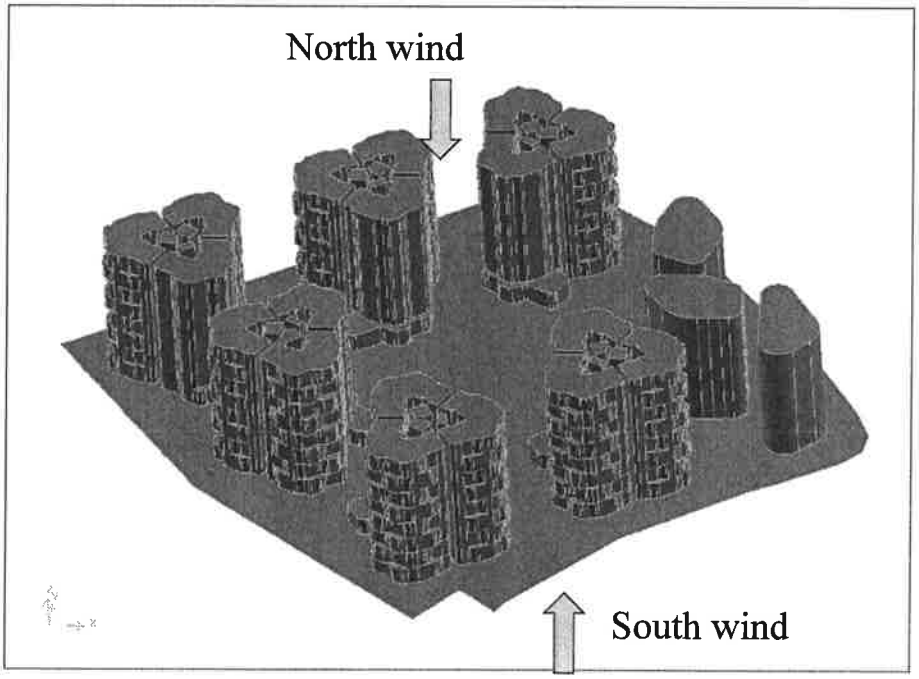
CFD Ventilation software

- VORTEX
- AIRPAK
- FLOTHERM

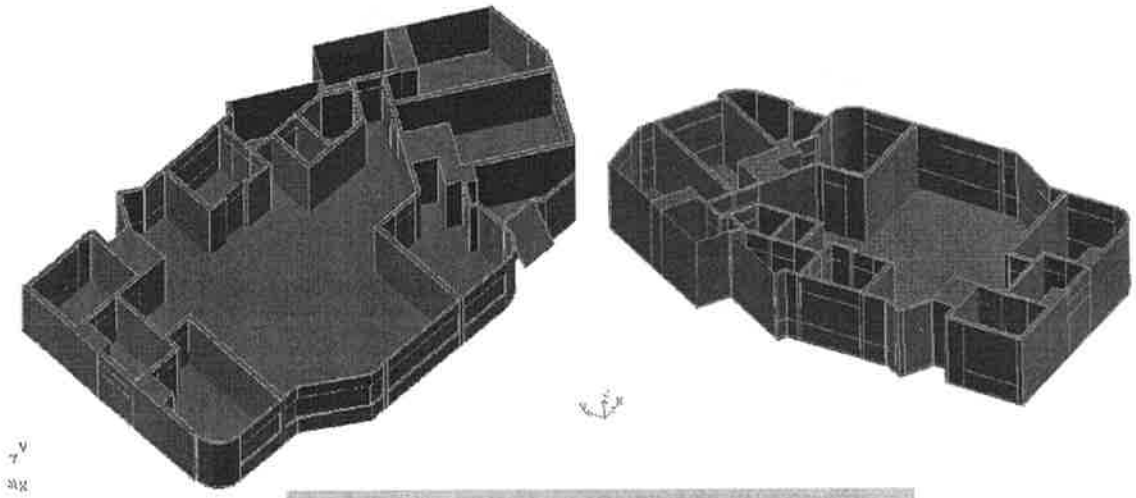
CFD Fire software

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

CFD simulation for building ventilation design – Example A

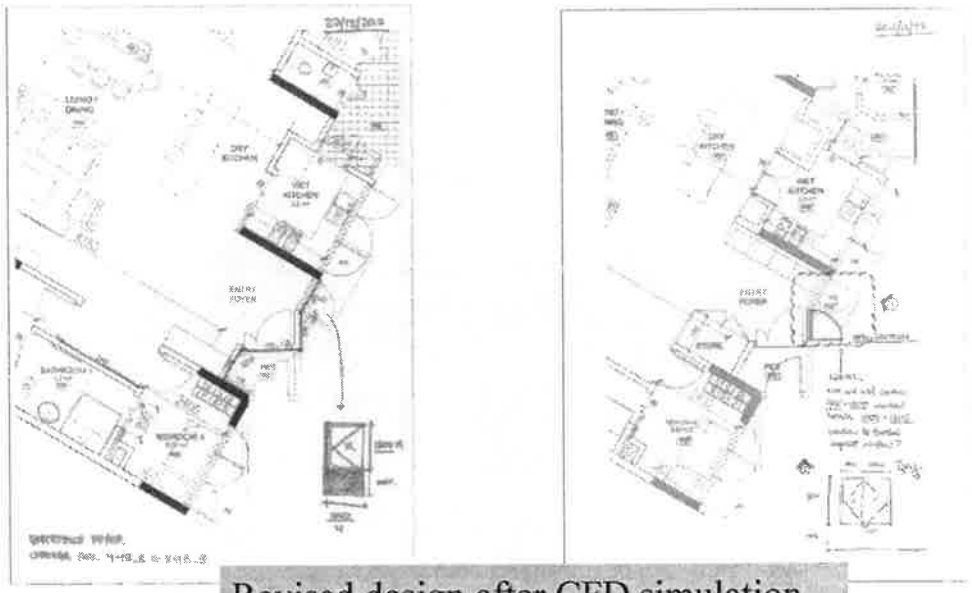


CFD simulation for building ventilation design – Example A



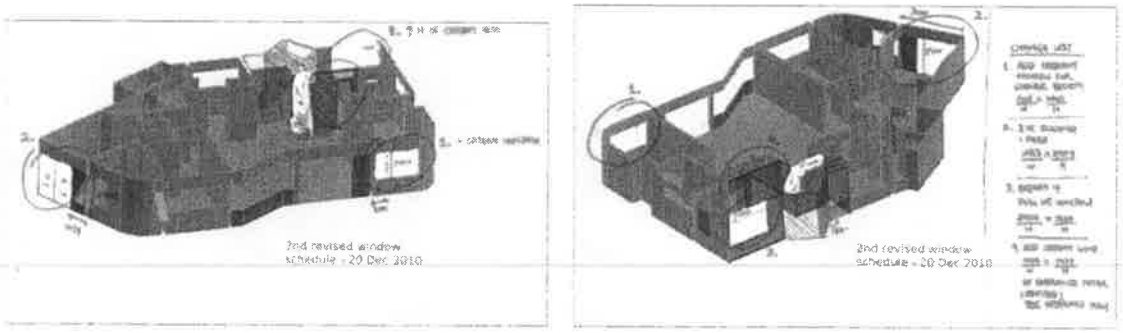
Original design after CFD simulation

CFD simulation for building ventilation design – Example A



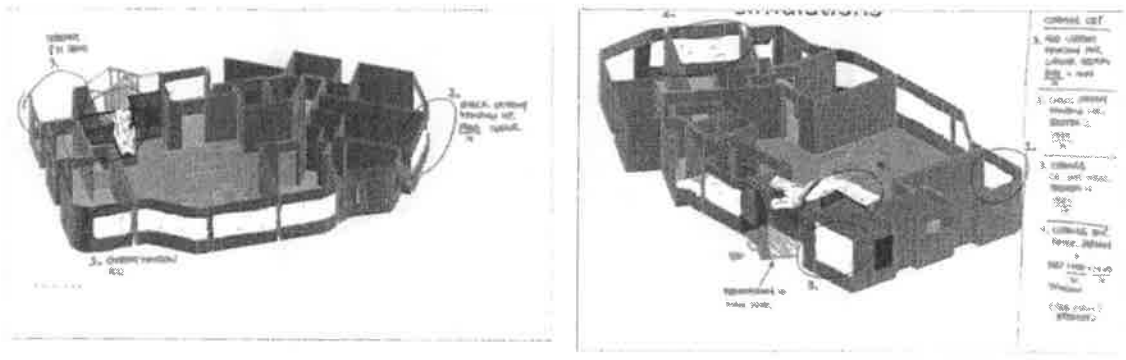
Revised design after CFD simulation

CFD simulation for building ventilation design – Example A



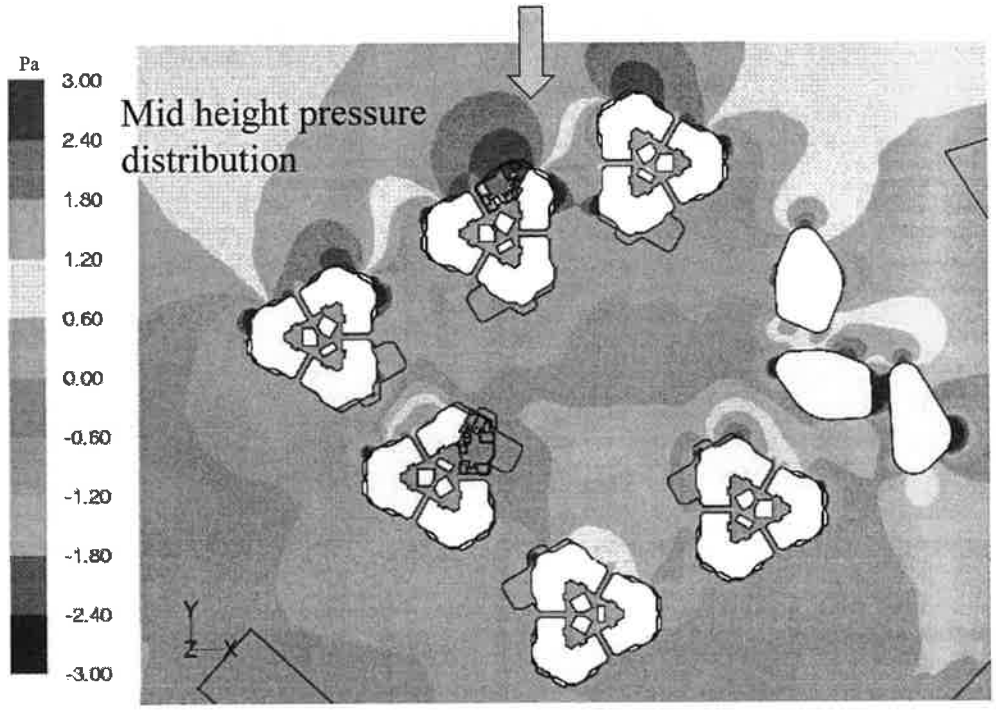
Revised design after CFD simulation

CFD simulation for building ventilation design – Example A



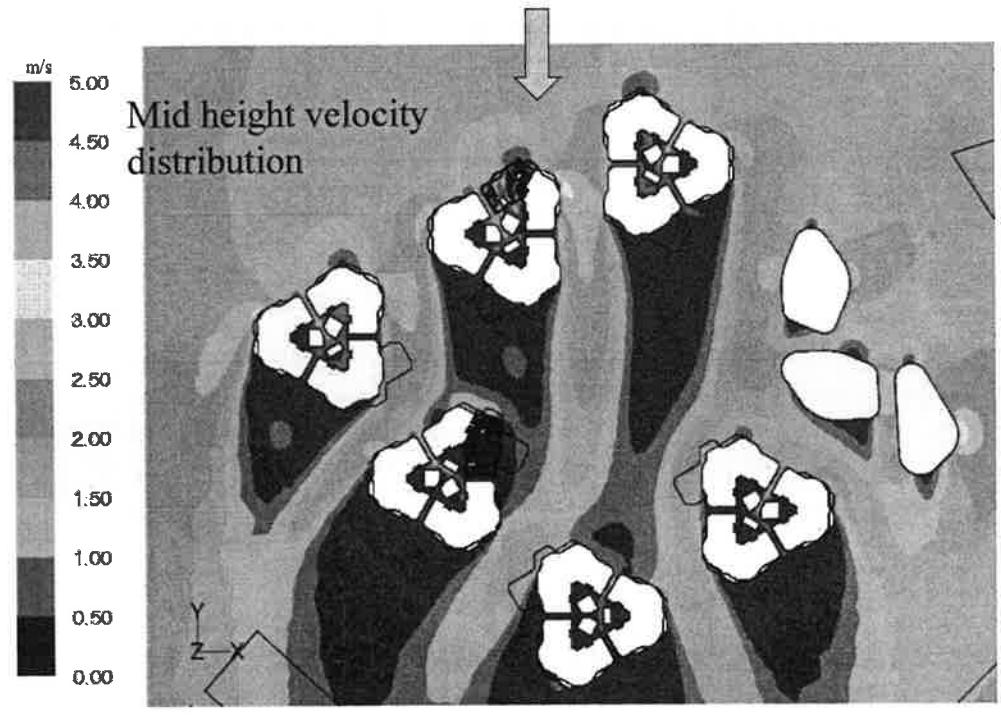
Revised design after CFD simulation

North wind



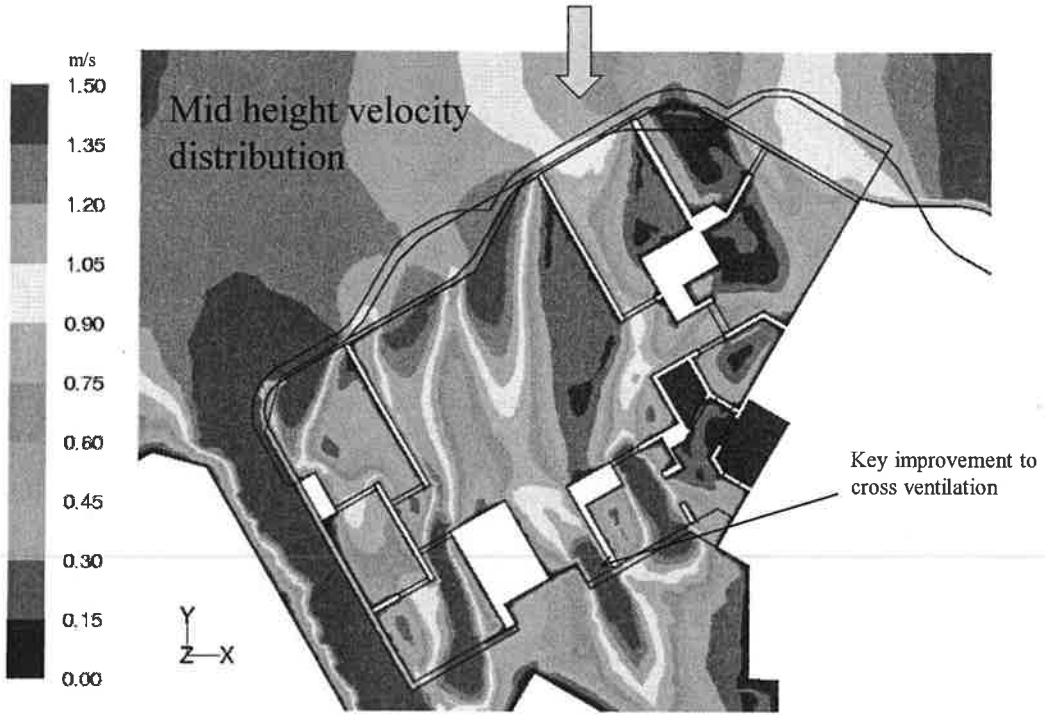
111

North wind

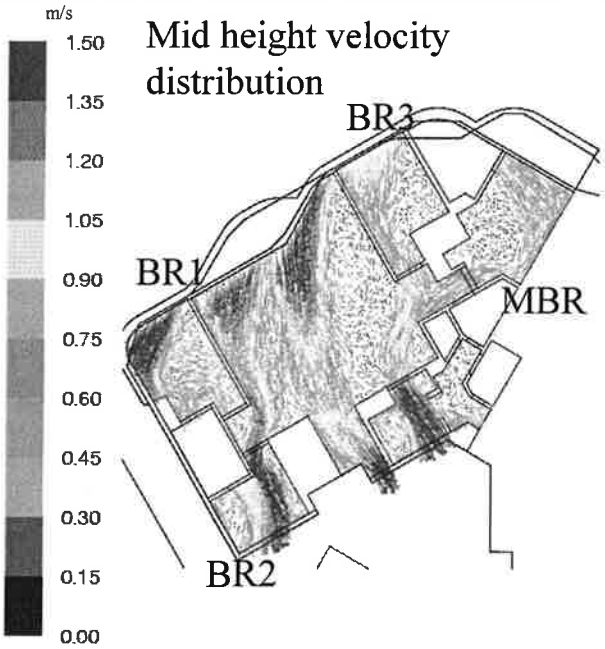


112

North wind

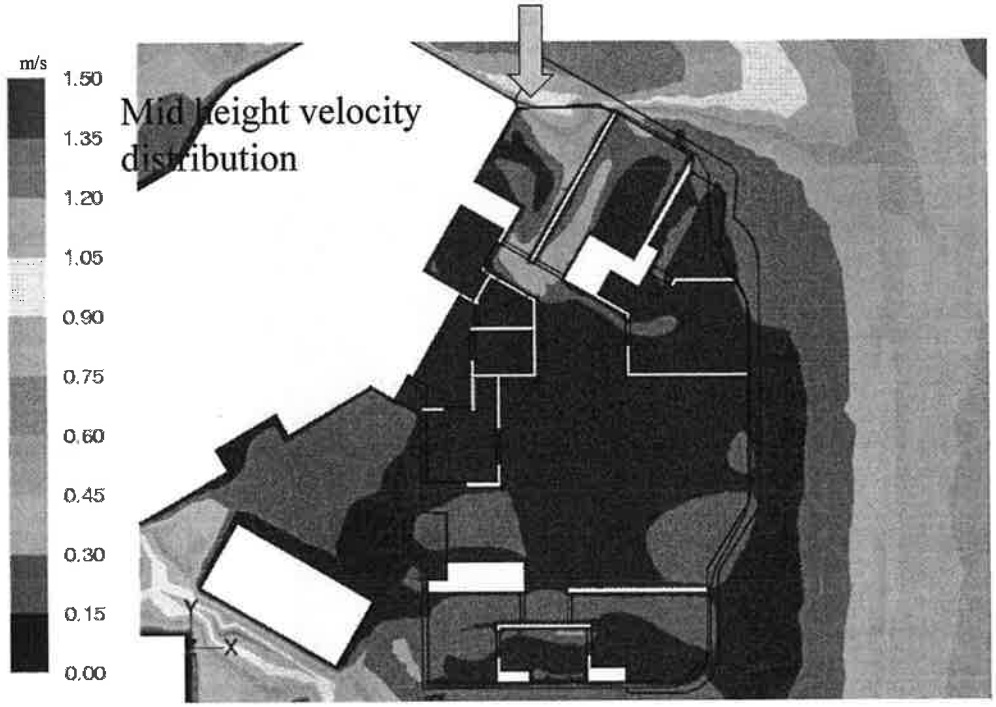


North wind

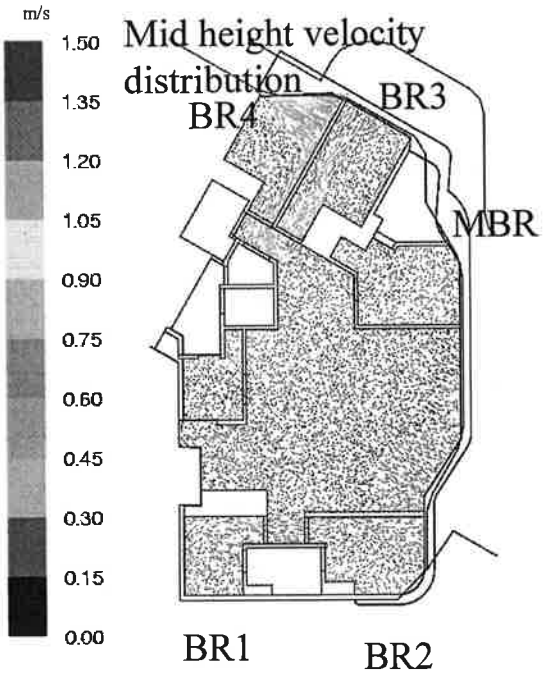


	Average velocity (m/s)
Kitchen	0.72
Living/Dining	0.76
Bedroom 1	0.75
Bedroom 2	0.98
Bedroom 3	0.55
Bedroom MBR	0.41
Average	0.70

North wind

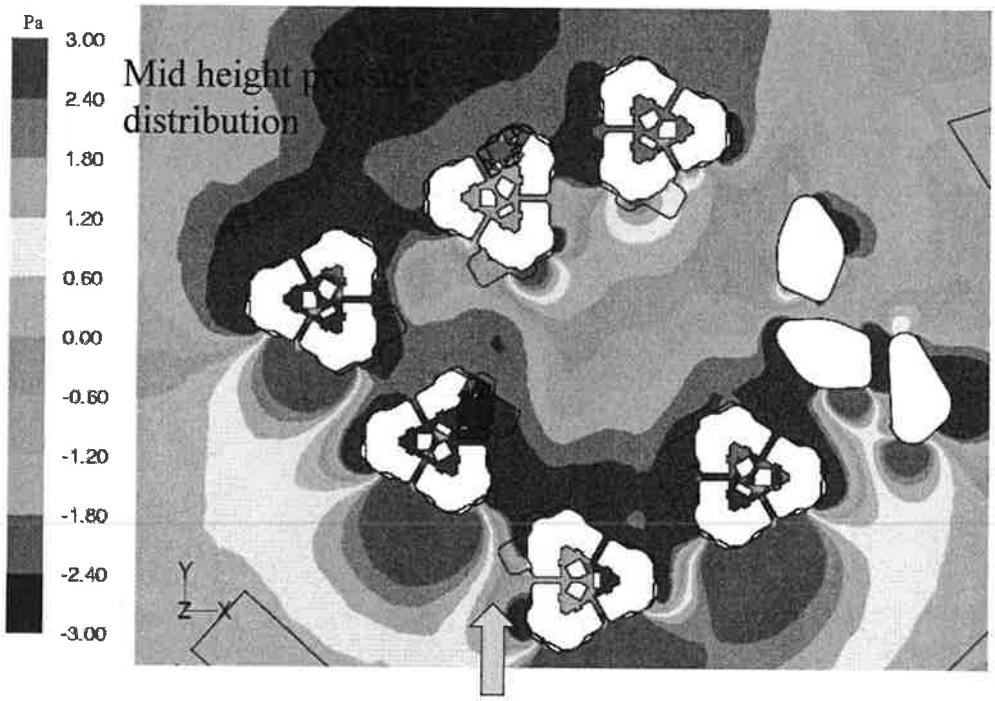


North wind



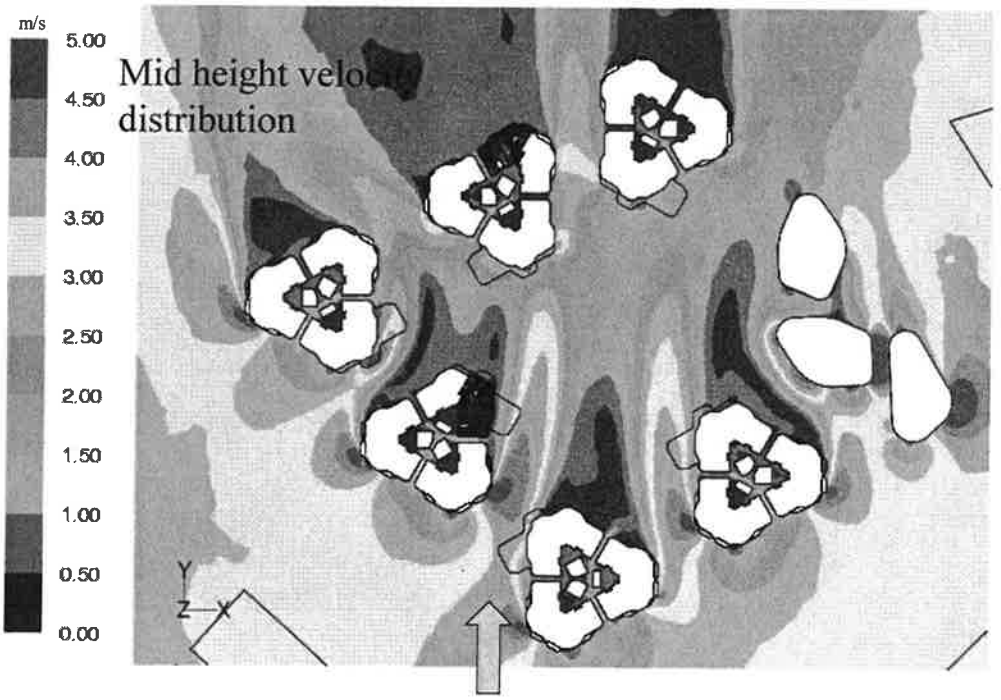
	Average velocity (m/s)
Kitchen	0.08
Living/Dining	0.12
Bedroom 1	0.17
Bedroom 2	0.12
Bedroom 3	0.21
Bedroom 4	0.32
Bedroom MBR	0.08
Average	0.14

South wind



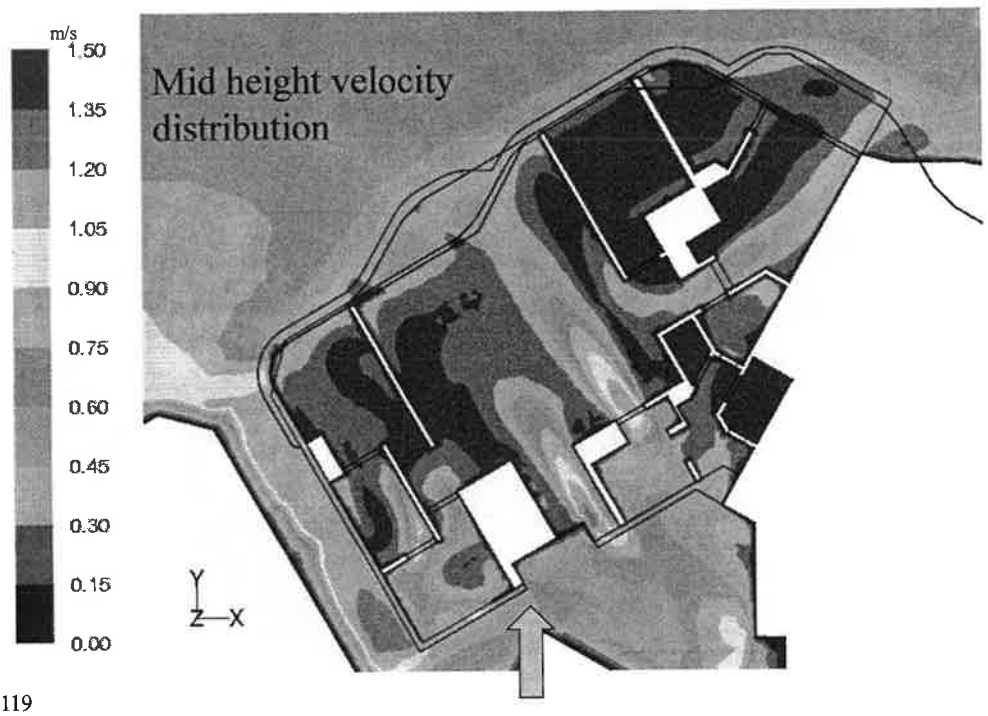
117

South wind

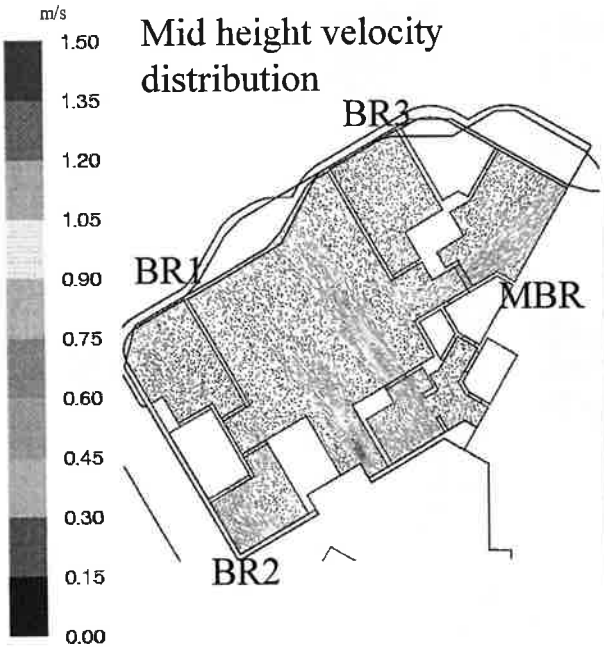


118

South wind

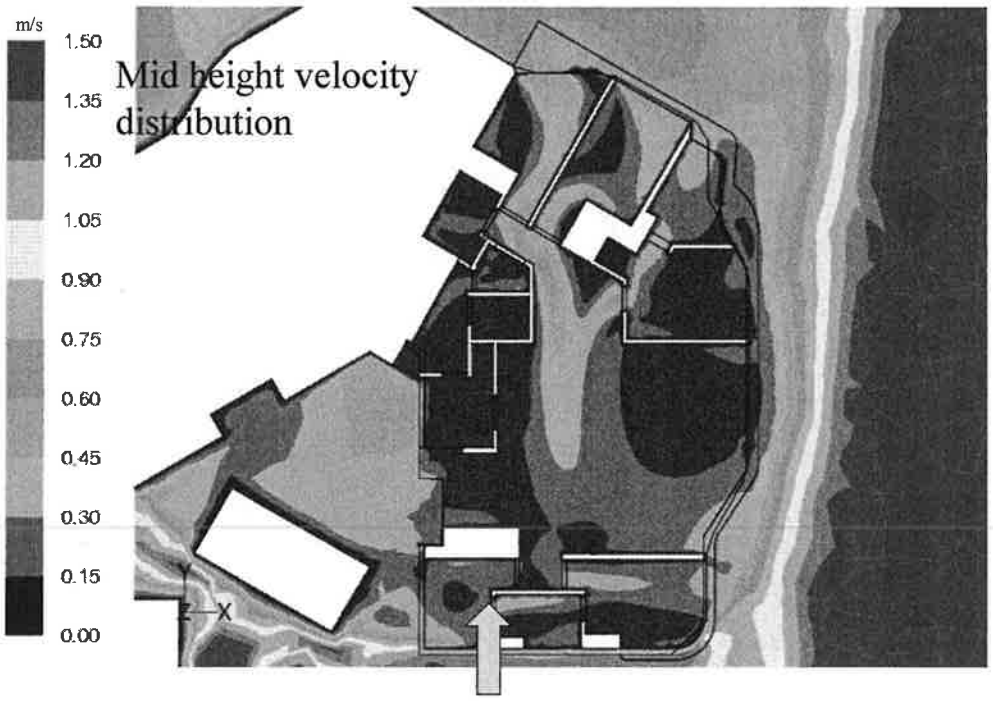


South wind

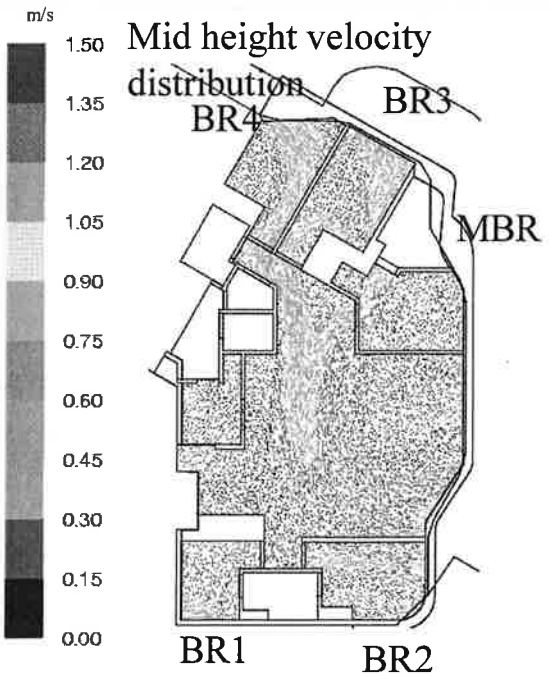


	Average velocity (m/s)
Kitchen	0.43
Living/Dining	0.32
Bedroom 1	0.19
Bedroom 2	0.52
Bedroom 3	0.09
Bedroom MBR	0.27
Average	0.30

South wind

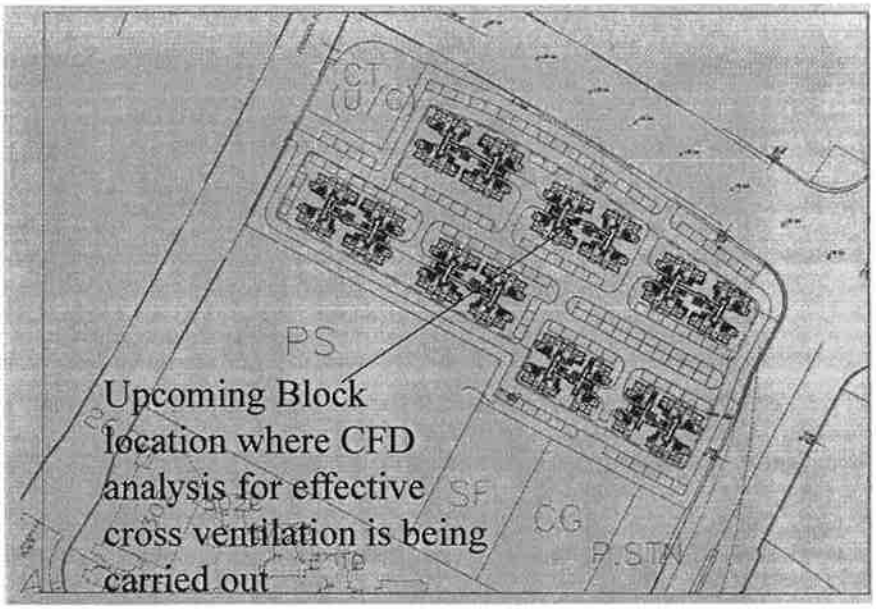


South wind

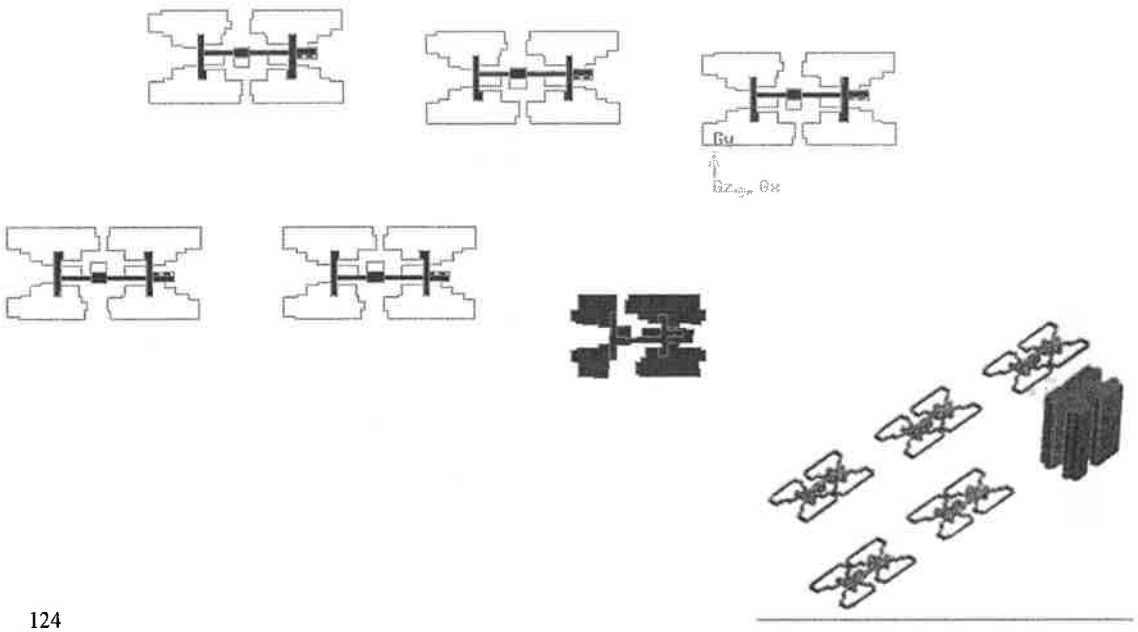


	Average velocity (m/s)
Kitchen	0.08
Living/Dining	0.19
Bedroom 1	0.25
Bedroom 2	0.19
Bedroom 3	0.28
Bedroom 4	0.29
Bedroom MBR	0.13
Average	0.20

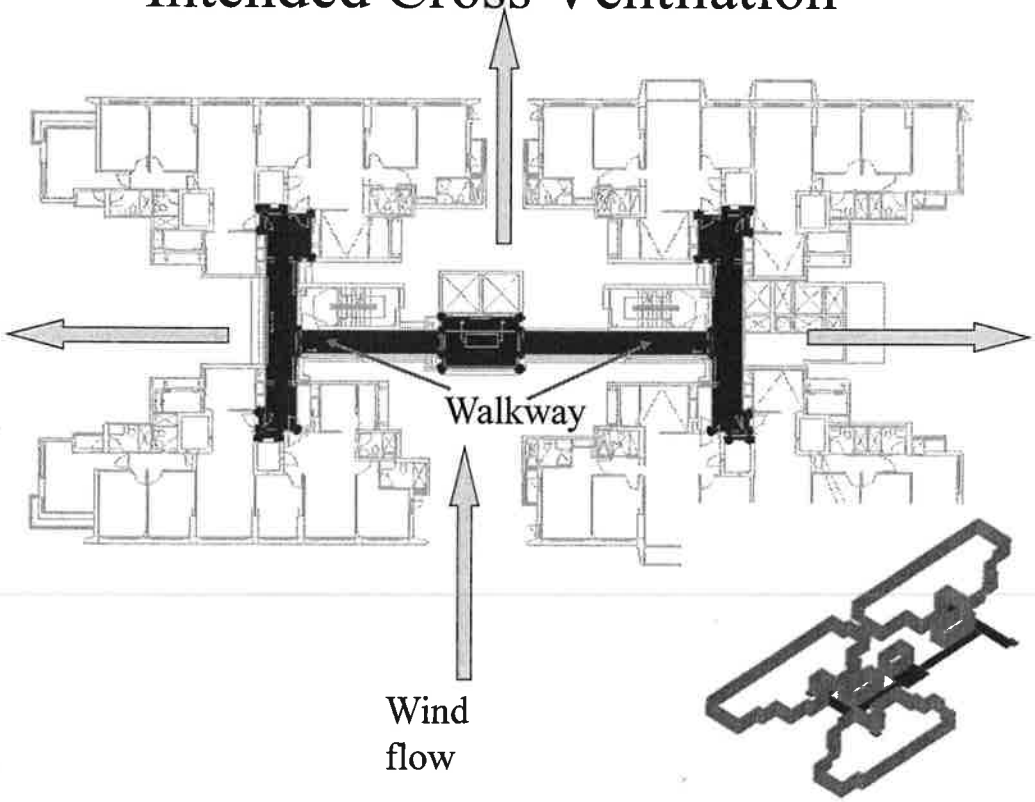
CFD simulation for building ventilation design – Example C

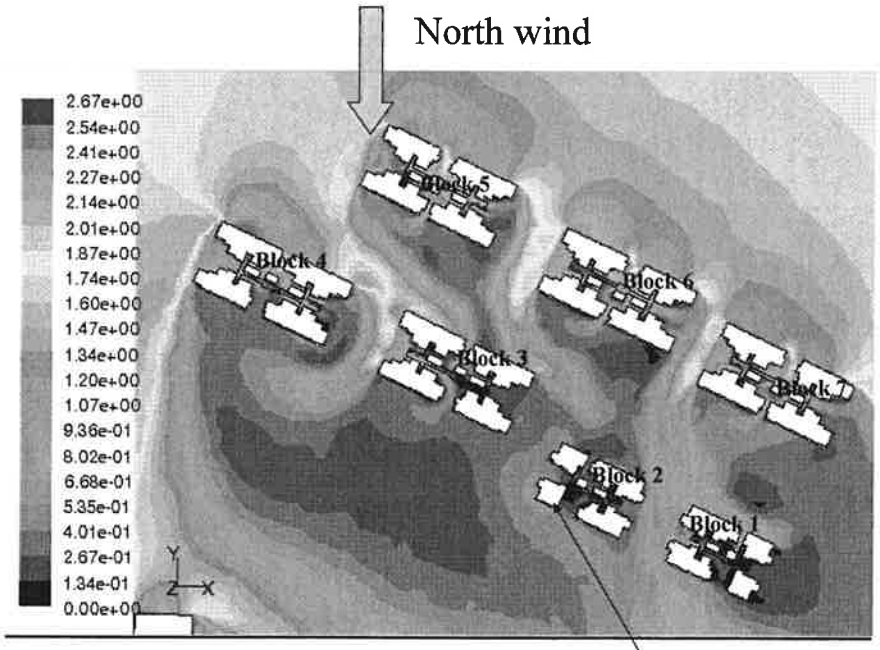


CFD simulation for building ventilation design – Example C



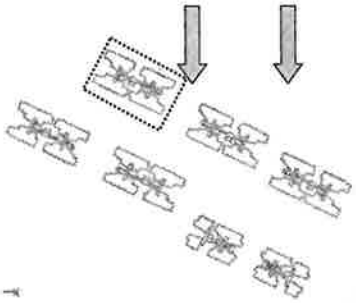
Intended Cross Ventilation



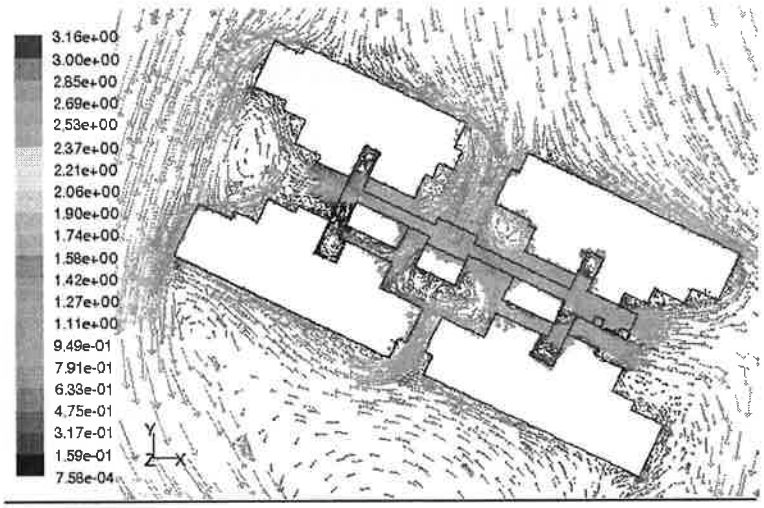


Mid height velocity distribution

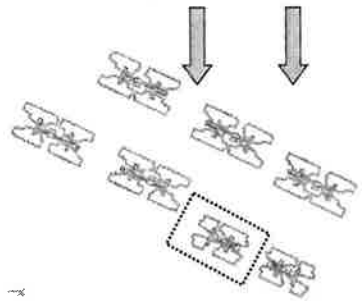
Block 2, minimum ambient wind



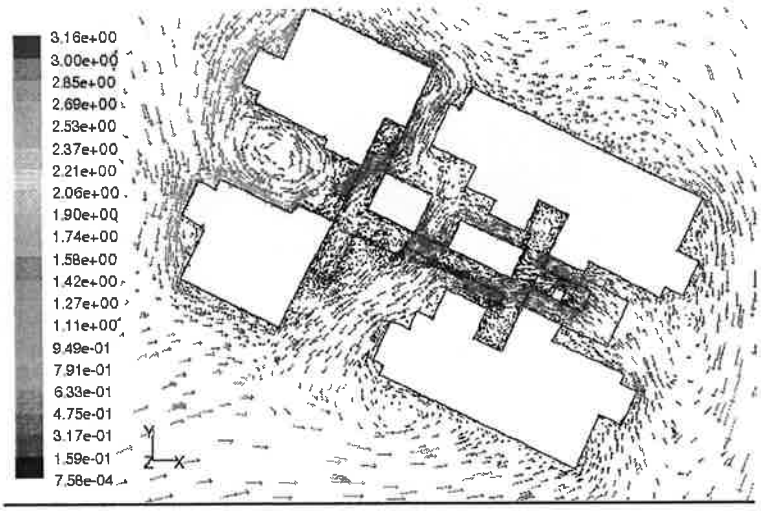
Block 5



127



Block 2



128

Day 2 (2:15pm – 3:45pm)

- Demo to CFD software

Demo to CFD Software

- Getting Started to CFD software
- Launching of CFD User Interface
- Reading, Writing and Managing Files

What is CFD?

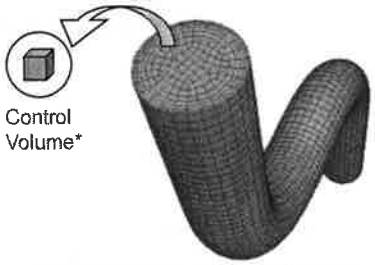
- **Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations**
  - **Conservation of mass**
  - **Conservation of momentum**
  - **Conservation of energy**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

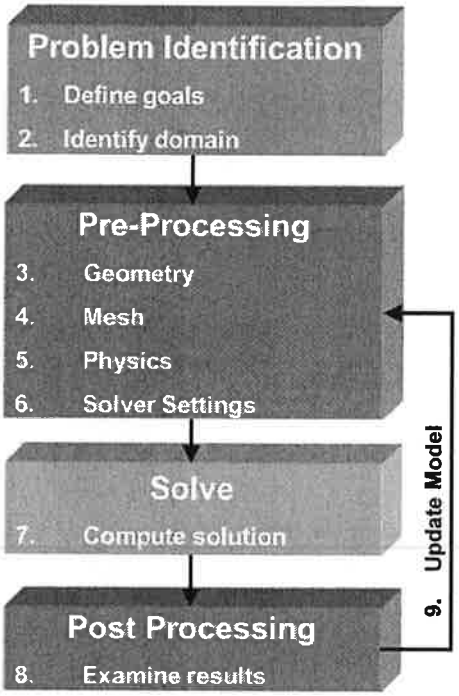
$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi\, dV}_{\text{Unsteady}} + \underbrace{\oint_A \rho\phi\mathbf{V}\cdot d\mathbf{A}}_{\text{Convection}} = \underbrace{\oint_A \Gamma_\phi \nabla\phi \cdot d\mathbf{A}}_{\text{Diffusion}} + \underbrace{\int_V S_\phi\, dV}_{\text{Generation}}$$

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

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

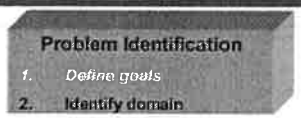
* FLUENT control volumes are cell-centered (i.e. they correspond directly with the mesh) while CFX control volumes are node-centered

CFD Modeling Overview



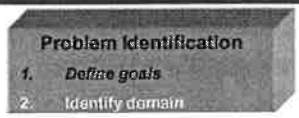
- **Problem Identification**
  1. Define your modeling goals
  2. Identify the domain you will model
- **PreProcessing and Solver Execution**
  3. Create a solid model to represent the domain
  4. Design and create the mesh (grid)
  5. Set up the physics (physical models, material properties, domain 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.

1. Define Your Modeling Goals

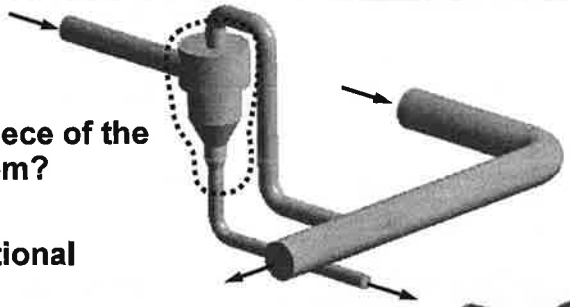


- **What results are you looking for (i.e. pressure drop, mass flow rate), and how will they be used?**
  - **What are your modeling options?**
    - **What physical models will need to be included in your analysis (i.e. turbulence, compressibility, radiation)?**
    - **What simplifying assumptions 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?**

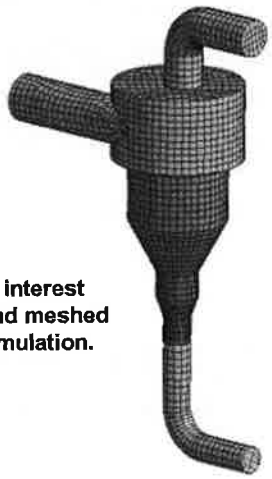
2. Identify the Domain You Will Model



- **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?**

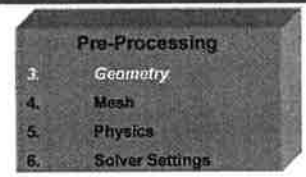


Domain of Interest as Part of a Larger System (not modeled)

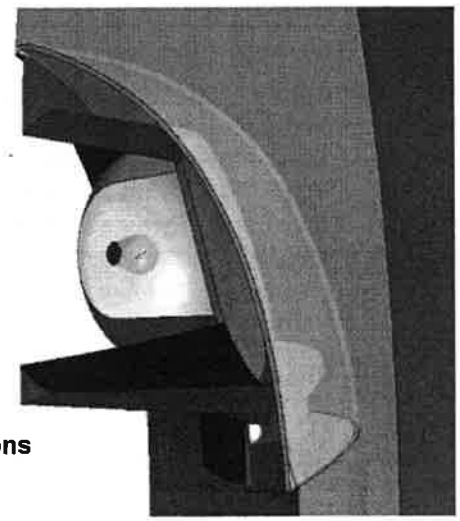


Domain of interest isolated and meshed for CFD simulation.

3. Create a Solid Model of the Domain

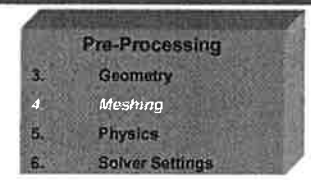


- **How will you obtain a solid model of the *fluid* region?**
  - **Make use of existing CAD models?**
    - **Extract the fluid region from a solid part?**
  - **Create from scratch?**
- **Can you simplify the geometry?**
  - **Remove unnecessary features that would complicate meshing (fillets, bolts…)?**
  - **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

4. Design and Create the Mesh

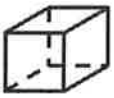


A mesh divides a geometry into many elements. These are used by the CFD solver to construct control volumes


Triangle


Quadrilateral


Tetrahedron


Hexahedron

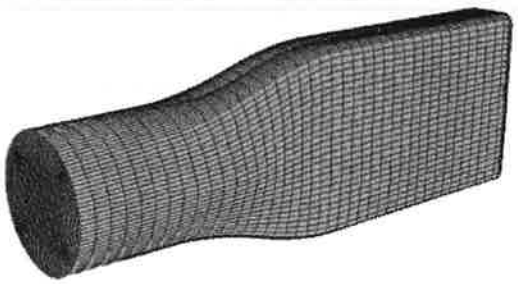

Pyramid


Prism/Wedge

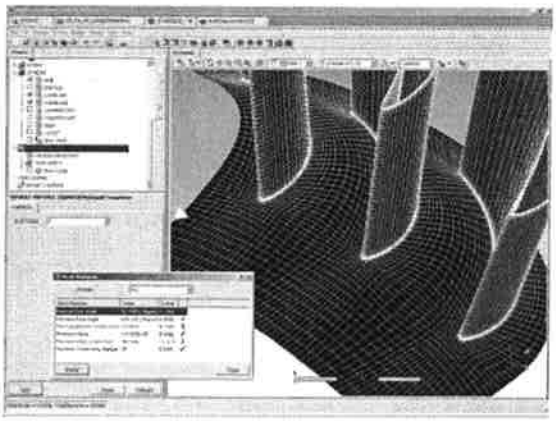
- **What degree of mesh resolution is required in each region of the domain?**
  - **The mesh must resolve geometric features of interest and capture gradients of concern, e.g. velocity, pressure, temperature gradients**
  - **Can you predict regions of high gradients?**
  - **Will you use adaption to 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.**

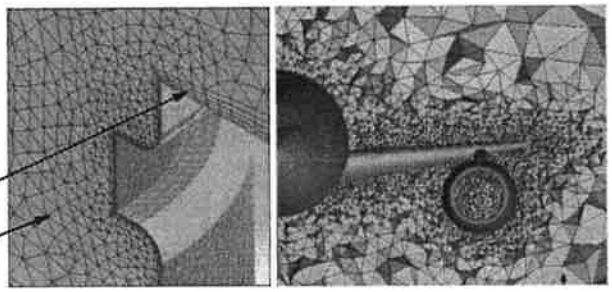


Tri/Tet vs. Quad/Hex Meshes

- **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.**



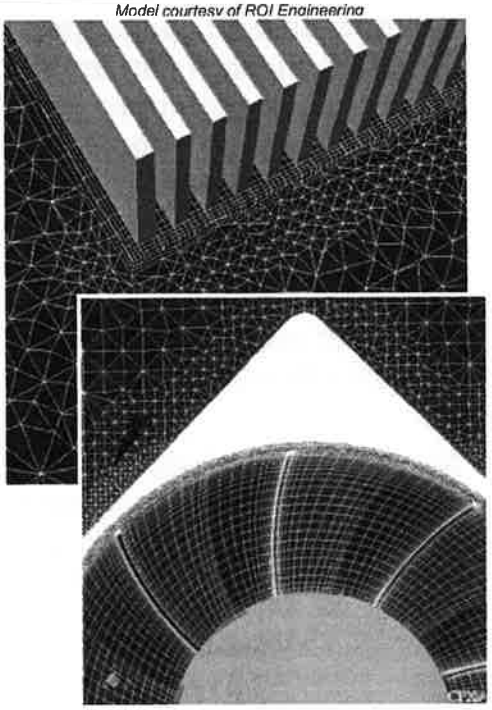
Multizone (or Hybrid) Meshes

- **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.**

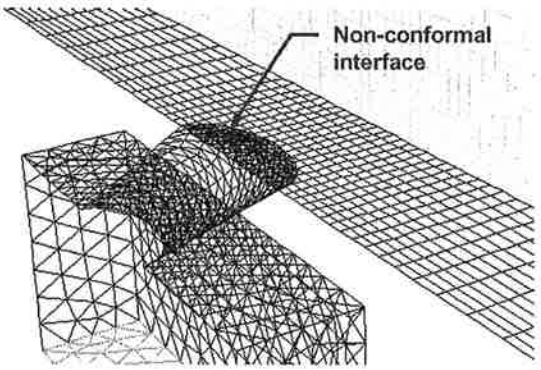
- **When the nodes do not match across the regions, a non-conformal interface can be used.**

Model courtesy of ROI Engineering

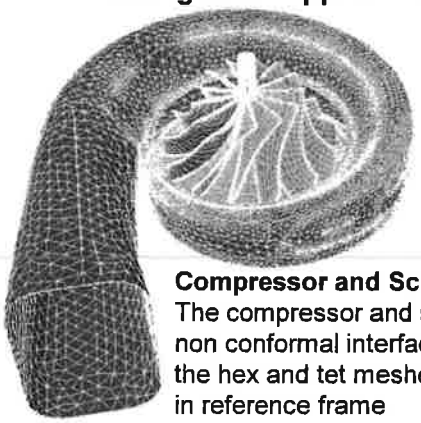


Non-Conformal Meshes

- **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**

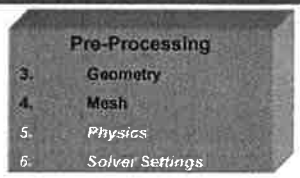


3D Film Cooling
Coolant is injected into a duct from a plenum. The plenum is meshed with tetrahedral cells while the duct is meshed with hexahedral cells



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

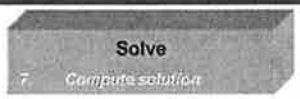
Set Up the Physics and Solver Settings

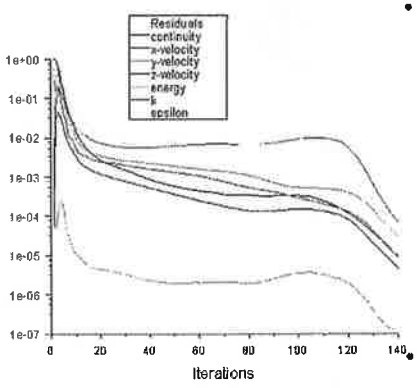


For complex problems solving a simplified or 2D problem will provide valuable experience with the models and solver settings for your problem in a short amount of time.

- **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**

Compute the Solution



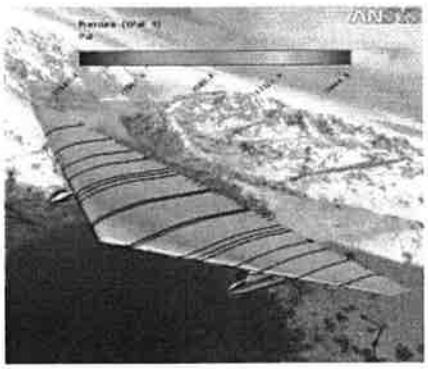


A converged and mesh-independent solution on a well-posed problem will provide useful engineering results!

- **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

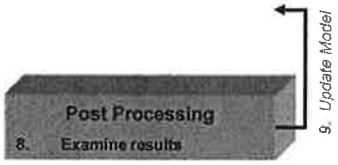




Examine results to ensure property conservation and correct physical behavior. High residuals may be caused by just a few poor quality cells.

- **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**

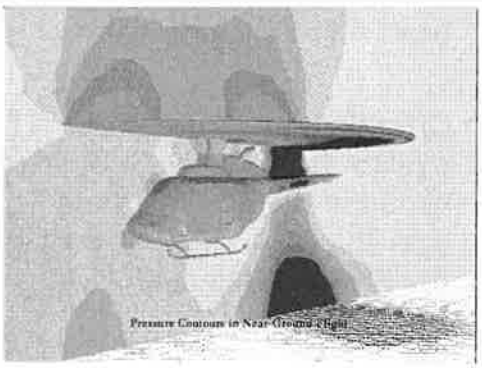
Consider Revisions to the Model



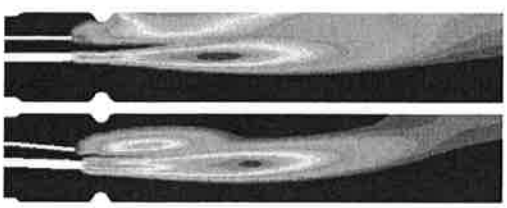
- Are the physical models appropriate?
  - Is the flow turbulent?
  - Is the flow unsteady?
  - Are there compressibility effects?
  - Are there 3D effects?
- Are the boundary conditions correct?
  - Is the computational domain large enough?
  - Are boundary conditions appropriate?
  - Are boundary values 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?

Models Available in FLUENT 12

- Fluid flow and heat transfer
  - Momentum, continuity, energy equations
  - Radiation
- Turbulence
  - RANS-based models (Spalart-Allmaras, k–ε, k–ω, Reynolds stress)
  - Large-eddy simulation (LES) and detached eddy simulation (DES)
- Species transport
- Volumetric reactions
  - Arrhenius finite-rate chemistry
  - Turbulent fast chemistry
    - Eddy Dissipation, non-Premixed, premixed, partially premixed
  - Turbulent finite-rate chemistry
    - EDC, laminar flamelet, composition PDF transport
  - Surface Reactions



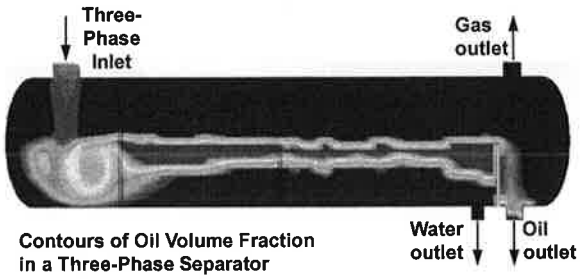
Pressure Contours in Near-Ground Flight



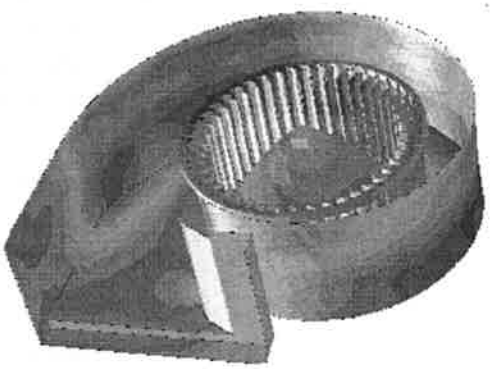
Temperature Contours for Kiln Burner Retrofit

Models Available in FLUENT 12

- **Multiphase flows**
  - **Discrete Phase Model (DPM)**
  - **Volume of Fluid (VOF) model for immiscible fluids**
  - **Mixtures**
  - **Eulerian-Eulerian and Eulerian-granular**
  - **Liquid/Solid and cavitation phase change**
- **Moving and deforming mesh**
  - **Moving zones**
    - **Single and multiple reference frames (MRF)**
    - **Mixing plane model**
    - **Sliding mesh model**
  - **Moving and deforming (dynamic) mesh (MDM)**
- **User-defined scalar transport equations**



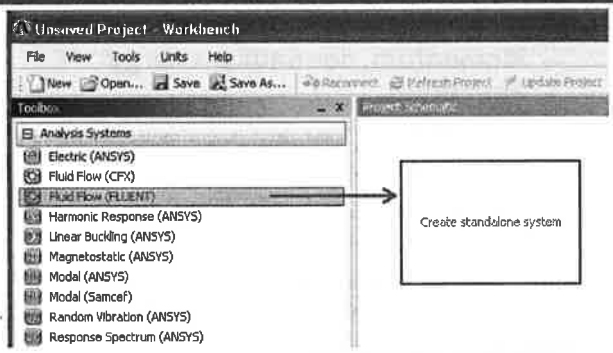
Contours of Oil Volume Fraction in a Three-Phase Separator

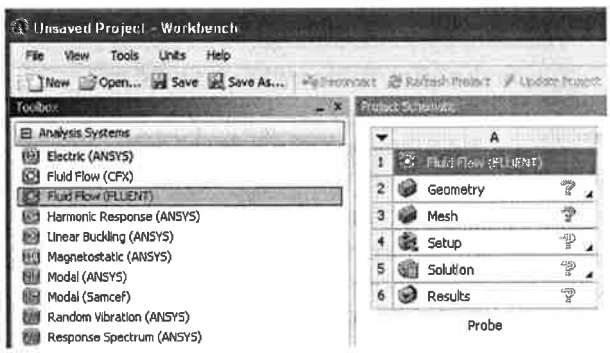


Pressure Contours in a Squirrel Cage Blower (Courtesy Ford Motor Co.)

FLUENT CFD Workflow under Workbench 2

- **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.**





Probe

Import the Geometry

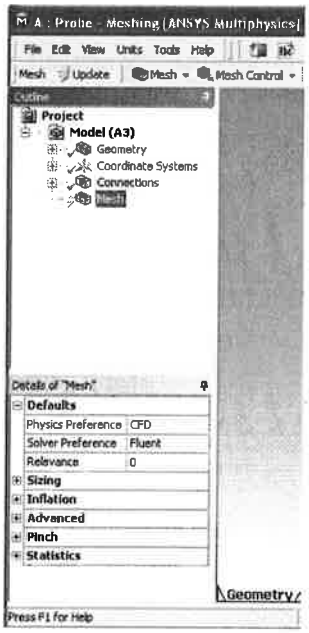
- **Right-click on Geometry cell A2 and select Import Geometry**
- **Import the geometry file (CAD model or DesignModeler .agdb file)**
- **You can also link the FLUENT simulation to an existing DesignModeler session.**



Probe

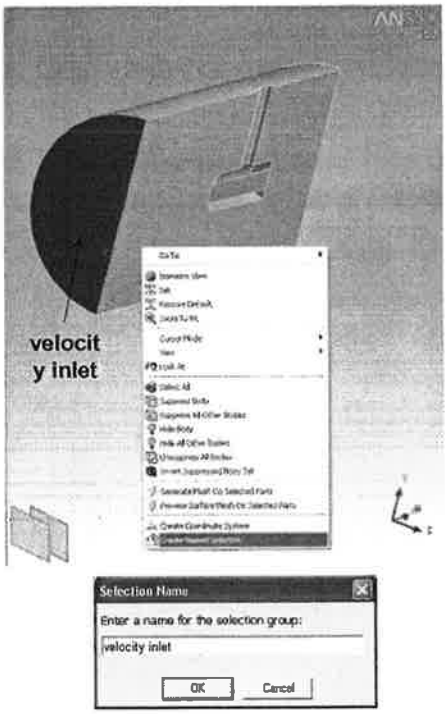
Generate a Mesh

- **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.

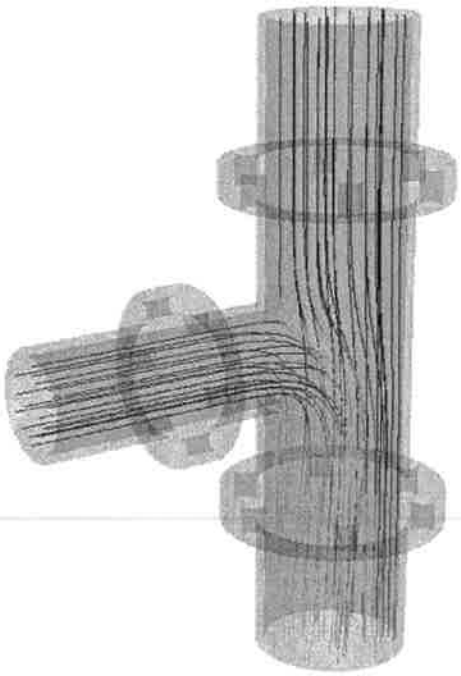


Set Up and Run FLUENT

- 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

Demonstration of FLUENT Software

- Start FLUENT (assume the mesh has already been generated).
  - Set up a simple problem.
  - Solve the flow field.
  - Postprocess the results.
- Online help and documentation is available on each panel by pressing the help button
  - Requires that you have the documentation installed and properly connected to your web browser.



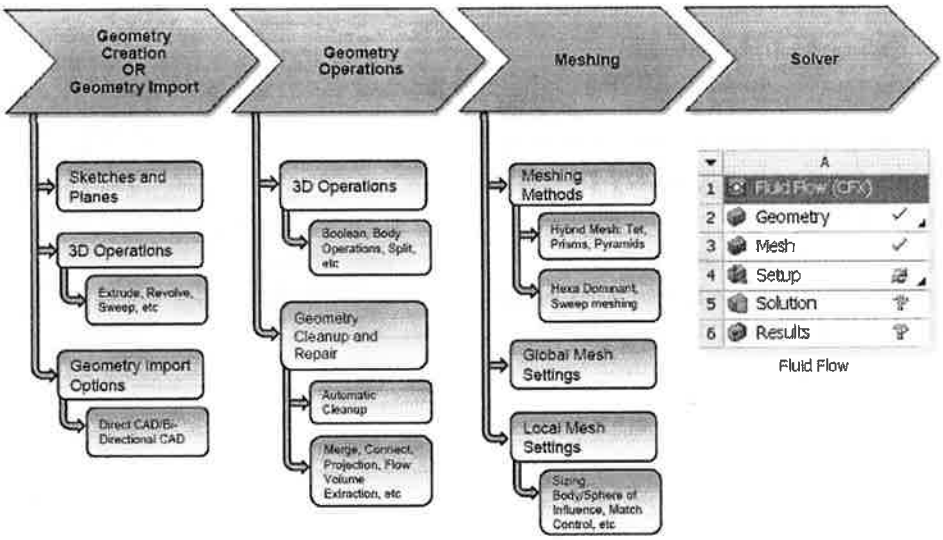
Navigating the PC at Fluent

- Log in to your workstation
  - Login name: `fluent`
  - Password: `fluent`
- Directories
  - Tutorial mesh/case/data files can be found in `c:\Student Files\fluent\tut\`
  - We recommend that you save your work into a central working folder: `c:\users`
  - Working folder shown on the desktop is a shortcut to c:\users
- To start FLUENT and/or Workbench, use the desktop icons.
- Your support engineer will save your work at the end of the week.
- It is recommended that you restart FLUENT and/or Workbench for each tutorial to avoid mixing solver settings from different workshops.

Creating 3D geometry model

ANSYS DesignModeler

PreProcessing Workflow



Creating 3D geometry model

ANSYS DesignModeler

Modelling Method

Feature/History based Modeling

- CAD like approach to create new models with sketch-to-3D operations
  OR
- Import models from CAD and perform various geometric operations to prepare it for meshing
- Fully parametric
  - Captures engineering intent
  - Supports native CAD parameters
  - Drives changes by changing a few parameters in an automated way
- Feature tree
  - Maintains history of all features
- Analysis focused
  - Custom tools to repair, prepare and optimize models for analysis
- ANSYS Design Modeler

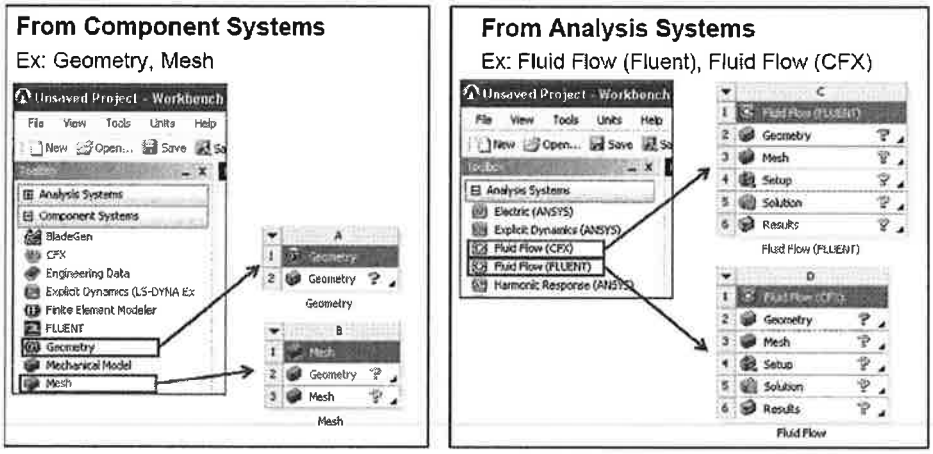
Direct Modeling

- Import models from CAD and perform various geometric operations to prepare it for meshing
- Locally parametric
- More flexibility to make unplanned and local changes
  - No features + no constraints = no regeneration failures
  - Easy to edit "dumb" neutral files
  - Easy to repair, prepare and optimize models for analysis
- Analysis focused
  - Custom tools to repair, prepare and optimize models for analysis
- GAMBIT (legacy ANSYS product), ANSYS SpaceClaim

Creating 3D geometry model

ANSYS DesignModeler

Launching DesignModeler



DesignModeler cannot be launched in a standalone mode, it can be only launched within Workbench as shown in the above example

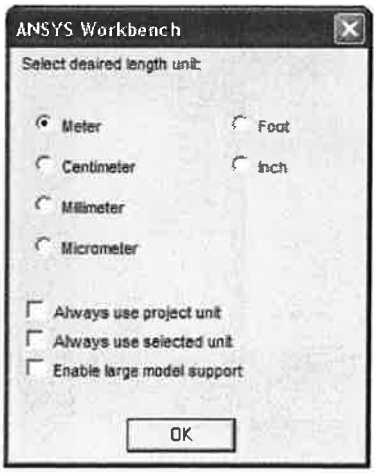
Creating 3D geometry model

ANSYS DesignModeler

Setting Up Working Unit

- Unit selection menu pops up immediately after DM launches

- **Always use project unit** : Project units set in Workbench will be used
- **Always use selected unit** : Unit selected from the panel will be used
- **Enable large model support** : enable this to create large models within a bounding box of 1000 cubic km.



DM will no longer show the Units pop up panel in subsequent DM sessions if 'Always use project unit 'or 'Always use selected unit' is chosen.

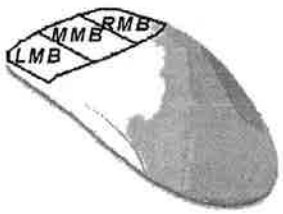
Units cannot be changed mid-session

Creating 3D geometry model

ANSYS DesignModeler

Basic Mouse Control

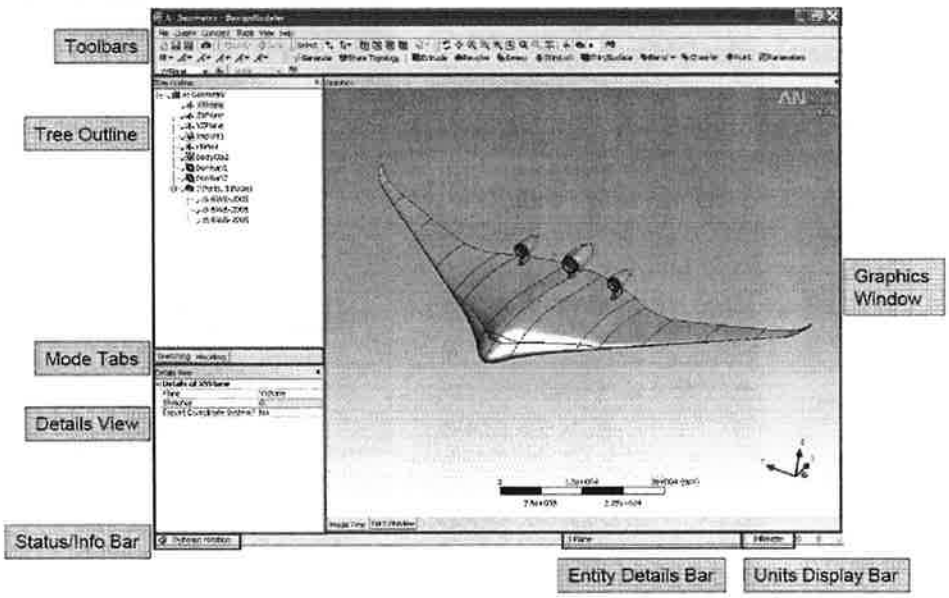
- LMB
  - Geometry selection
  - <CTRL> + LMB adds/removes selected entities
  - Hold LMB and sweep cursor = continuous selection
- MMB
  - **Rotate**
  - **Shift + MMB** : Zoom
  - **Ctrl + MMB** : Pan
- RMB
  - Box Zoom
  - Open context menus



Creating 3D geometry model

ANSYS DesignModeler

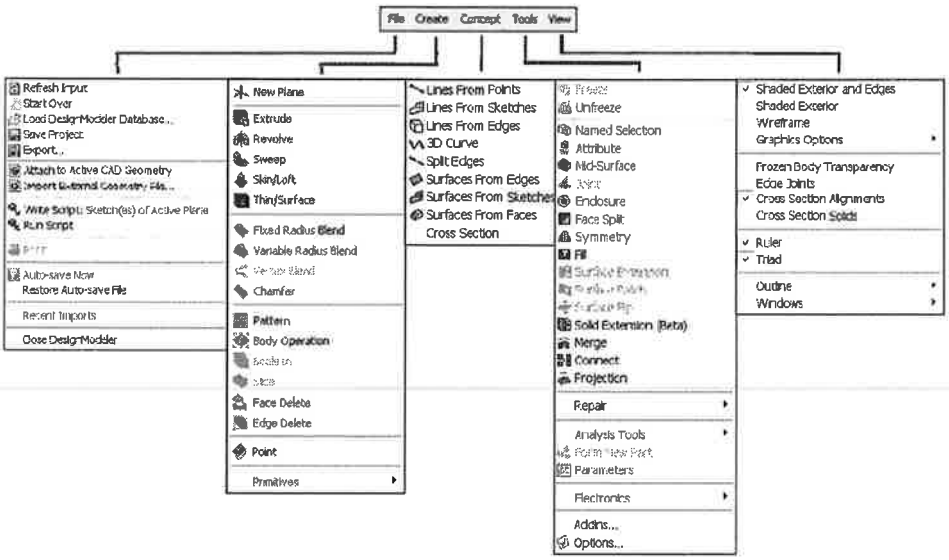
DesignModeler Interface



Creating 3D geometry model

ANSYS DesignModeler

Menu Toolbar

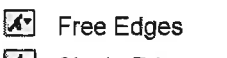


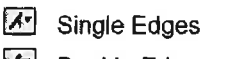
Creating 3D geometry model

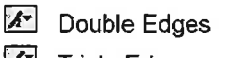
ANSYS DesignModeler

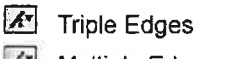
Graphics Option Toolbar



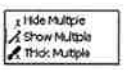




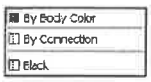












Creating 3D geometry model

ANSYS DesignModeler

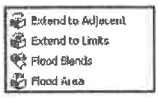
Selection Toolbar



- Select Mode –
  - Select single entity
  - Select multiple entities
- Select Points
- Select Edges
- Select Faces
- Select Bodies

(Selection Filters)

- Extend Selection for multiple entities –

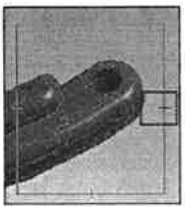


Tricks on 'Selection'

- To select multiple entities, drag the mouse to create a box enclosing the entities



Drag from left to right to select the entities, completely enclosed in the box



Drag from right to left to select the entities, completely and partially enclosed in the box

Hash lines on the edges of box help to determine the type of selection

Creating 3D geometry model

ANSYS DesignModeler

View Toolbar



1. **Rotate** : For dynamic rotation of the object
2. **Pan**: To scroll the object around
3. **Zoom**: Zoom in or Zoom out the view by dragging LMB up or down respectively
4. **Box Zoom**: Zooms to the view enclosed by the box created by dragging the LMB
5. **Fit Model to Screen**: Zooms in to fit the full model in the screen
6. **Magnifier Window**: Zoomed in model will be displayed in a new window
7. **Previous View**: Works as Undo for the Displays
8. **Next View**: Works as Redo for the displays
9. **Set Iso View**: Set the model in Isometric view
10. **Display Plane**: Toggle to display the plane on or off
11. **Display Model**: Toggle to display the model on or off
12. **Display Points**: Toggle to display the points on or off
13. **Look at**: Orients the view normal to the selected Face or active Plane, or active Sketch

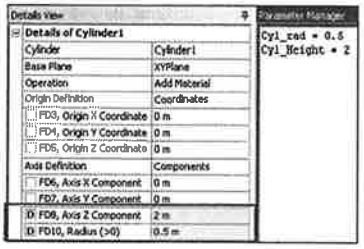
Creating 3D geometry model

ANSYS DesignModeler

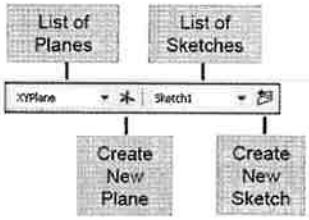
Feature Operation Toolbar



- Add/modify solid bodies
- Option to parameterize



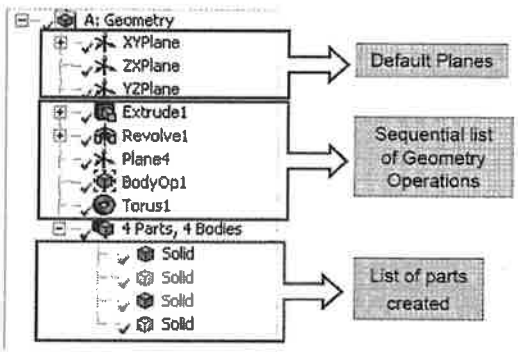
- Plane Toolbar
  - Create new plane and new sketch
  - List of Planes and sketches



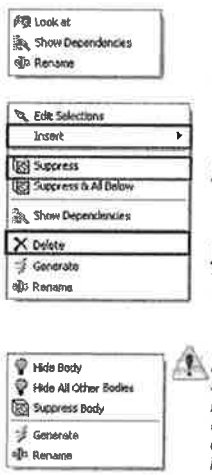
Creating 3D geometry model

ANSYS DesignModeler

Tree Outline



Contextual menu upon RMB click



Insert can be used to add operations anywhere in the tree

Suppress can be used to deactivate a selected operation

Delete can be used to delete a selected operation

It is recommended to suppress an operation instead of deleting it it helps the user with the option of unsuppressing it if needed in the future

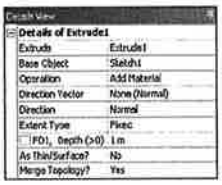
Creating 3D geometry model

ANSYS DesignModeler

Details View

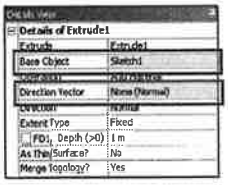
- Details of Feature operations selected in Tree outline are displayed
- Input the details for new Feature operations to be performed
- Edit details of existing Feature operations if needed

Details View of new Feature to be performed



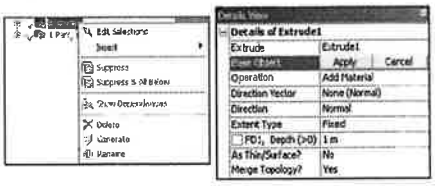
Specify details for feature creation

Details View of existing Feature



Gray colored details of previously created feature cannot be modified

Details View of existing Feature through Edit Selections option

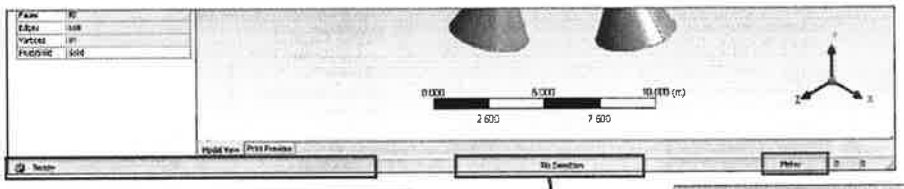


All details of previously created feature can be modified

Creating 3D geometry model

ANSYS DesignModeler

Status/Info Bar



Status/Info Bar
- Displays Status of geometry model
- Displays dynamic instructions
- Ex: Ready
  Extrude Creation – Click the Generate button to complete the extrude

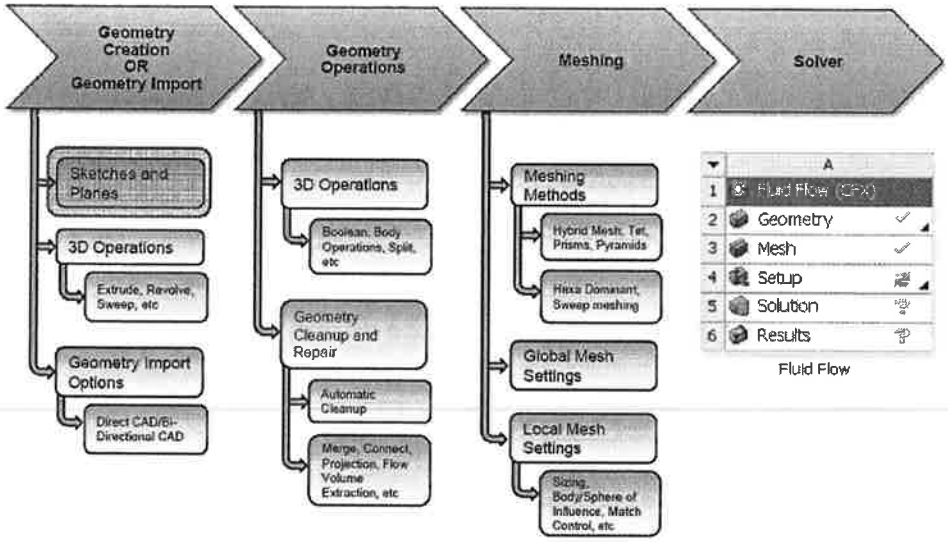
Units Display Bar
- Displays Unit used in current project

Entity Details Bar
- Displays details of entity selected in Graphics Window or Tree Outline.
- Ex: 1 Face: Area = 100 m²

Creating 3D geometry model

ANSYS DesignModeler

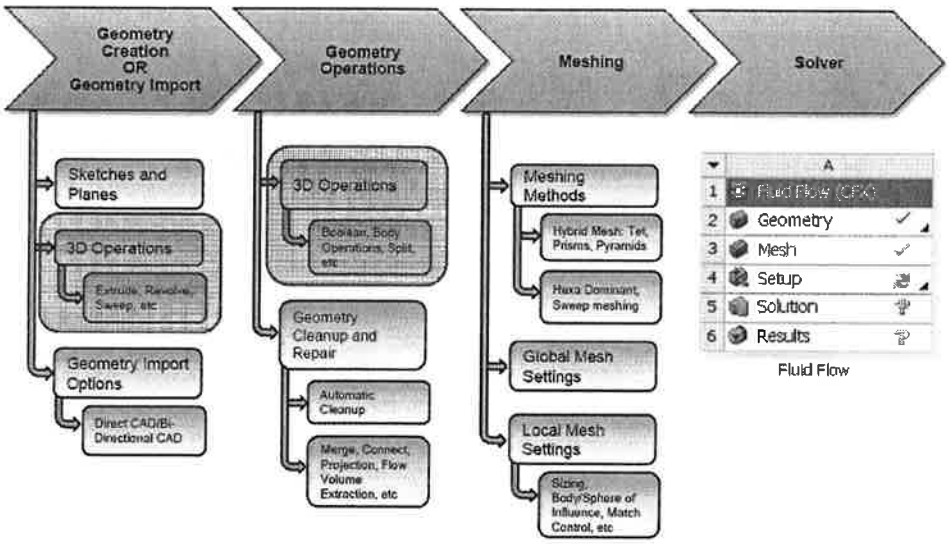
Planes and Sketches



Creating 3D geometry model

ANSYS DesignModeler

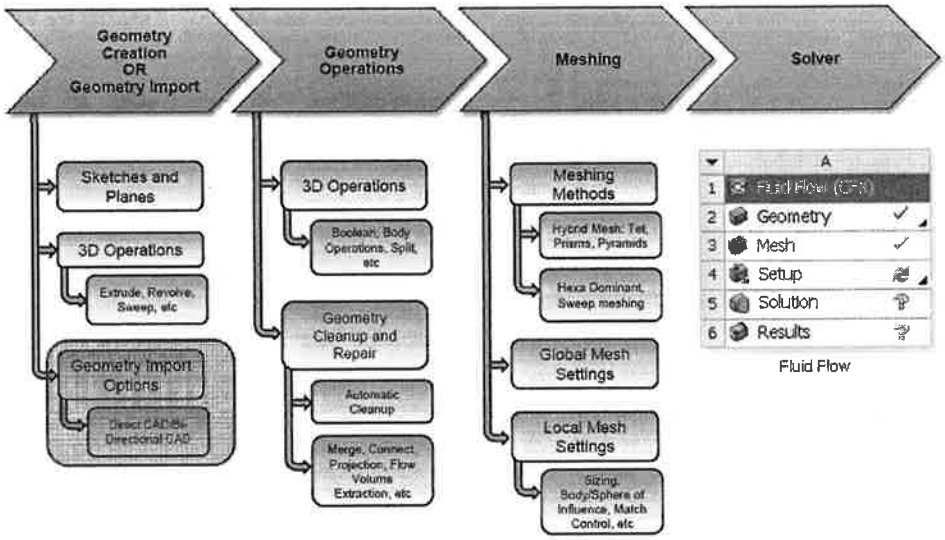
3D Modeling & Operations



Creating 3D geometry model

ANSYS DesignModeler

Geometry Import – CAD Connections & Parameterization

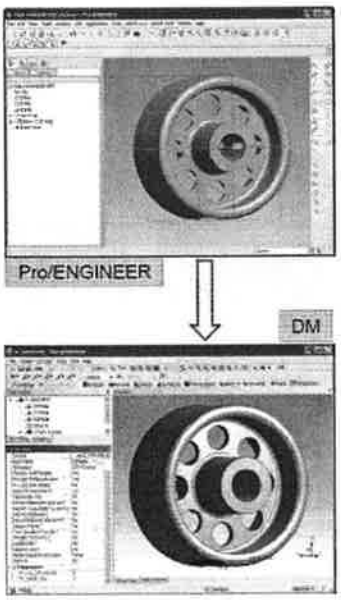


Creating 3D geometry model

ANSYS DesignModeler

CAD Connections

- –CAD Connections
  - Transfer CAD geometry to DM/AMP
- –Two approaches
  - Approach I (Import)
    - –Import neutral files in STEP, IGES etc. formats as well as native CAD formats
    - –Uni-directional connectivity
  - Approach II (Direct Connection)
    - –ANSYS Direct CAD Interface
    - –Bi-directional CAD connectivity

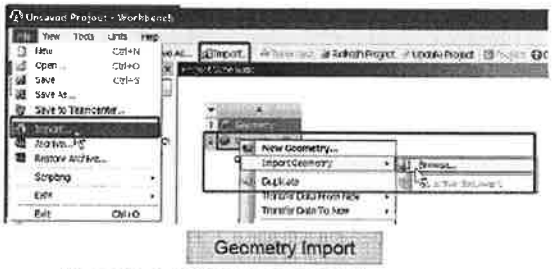


Creating 3D geometry model

ANSYS DesignModeler

CAD Connections : Approach 1

- Three ways to import file in WB
  - Import option on project page
  - File → Import
  - Geometry cell on project page
- Import Options
  - Neutral file formats like STEP, IGES etc.
  - ANSYS Legacy formats like Bladegen (*.bgd), DM (*.agdb) etc.
  - CAD formats like UG, ProE, CATIA etc.



Geometry Import



Geometry Import Options

Needs license for CAD & ANSYS CAD interface

- Please see documentation for more information

Creating 3D geometry model

ANSYS DesignModeler

CAD Connections : Approach 1

- CAD import in existing DM session
  - Appends the geometry file to existing geometry in DM
  - Two ways
    - Attach to Active CAD Geometry
      - Imports active CAD geometry in DM
      - Can attach only saved CAD sessions
      - Can import Parameters and Named Selections
    - Import External Geometry File
      - ACIS (extension .sat)
      - BladeGen (extension .bgd)
      - Monte Carlo N-Particle (extension .mcnp)
      - CATIA V5 (extension .CATPart and .CATProduct)
      - IGES (extension .igs or .iges)
      - Parasolid (extension .x_t and .xmt_txt for text files, extension .x_b and .xmt_bin for neutral binary files)
      - STEP (extension .step and .stp)



Attach and Import option in DM



Attached and imported models in model tree

Creating 3D geometry model

ANSYS DesignModeler

CAD Connections : Approach 2

- Launch WB from CAD system
  - Bi-Directional CAD Connectivity
  - Can export parameters from CAD to WB
  - Parameter change in CAD reflects in WB and vice versa
- Parameter Key
  - Prefix key for parameters to import
  - By default the filter is set to DS
  - If the filter is set to an empty string, all applicable entities will be imported as parameters



Direct CAD Connection in UG NX

8	Solid Bodies	☑
9	Surface Bodies	☑
10	Line Bodies	☐
11	Parameters	☑
12	Parameter Key	DS
13	Attributes	☐
14	Named Selections	☐
15	Material Properties	☐

Parameter Key

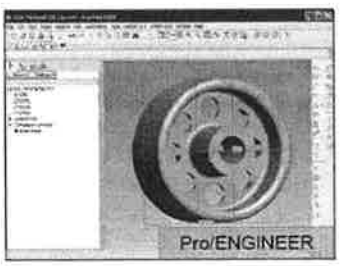
Creating 3D geometry model

ANSYS DesignModeler

CAD Connections : Bidirectional CAD

- Available for
  - UG NX
  - Pro/ENGINEER
  - SolidWorks
  - CoCreate Modeling
  - Inventor
  - Mechanical Desktop
  - Solid Edge
  - CATIA



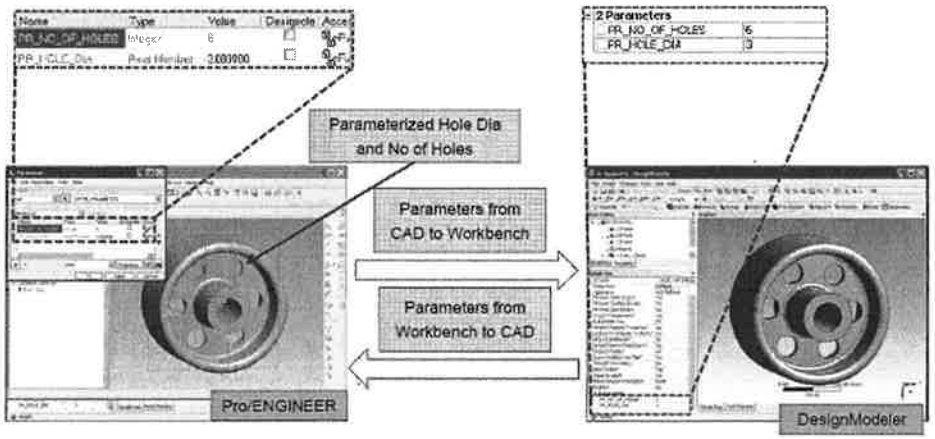


Creating 3D geometry model

ANSYS DesignModeler

CAD Connections : Bidirectional CAD

Pro/ENGINEER - Bi-Directional CAD Connection

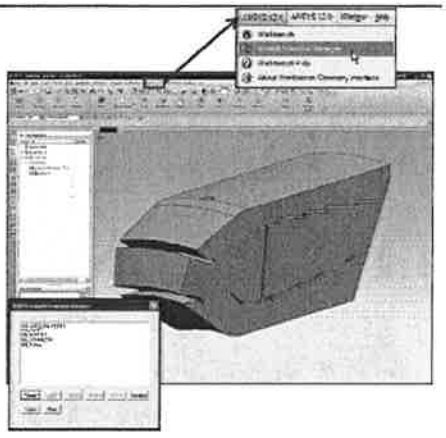


Creating 3D geometry model

ANSYS DesignModeler

CAD Connections : Named Selection Manager

- Named selection
  - Tag a boundary/face with a name
- Define in CAD & transfer to WB
- Define directly under WB
- Named Selection *filter* under WB
  - By default the filter is set to NS
  - Available if the "Import Named Selections" option is set to Yes
  - If the filter is set to an empty string, all entities with name selection/boundary definition will be transferred



Named Selection Manager in UG NX

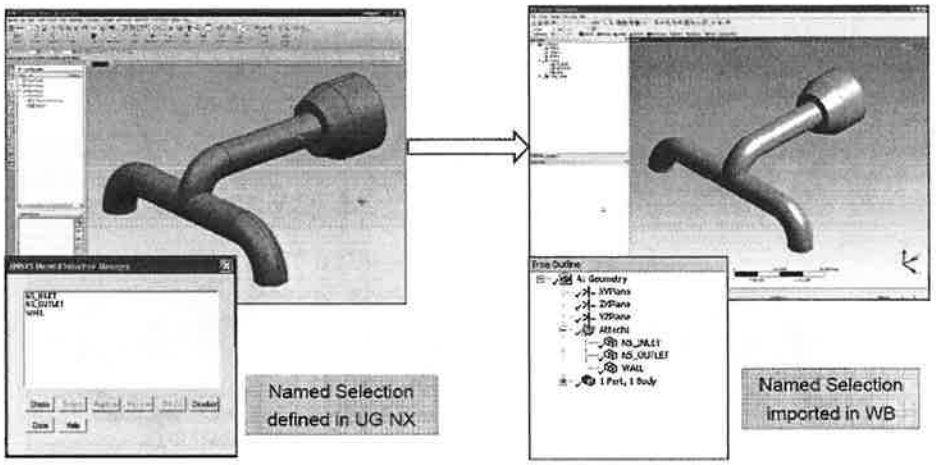


Creating 3D geometry model

ANSYS DesignModeler

CAD Connections : Named Selection Manager

UG-NX Named Selection Manager



Creating 3D geometry model

ANSYS DesignModeler

Parameterization

What is Parameterization?

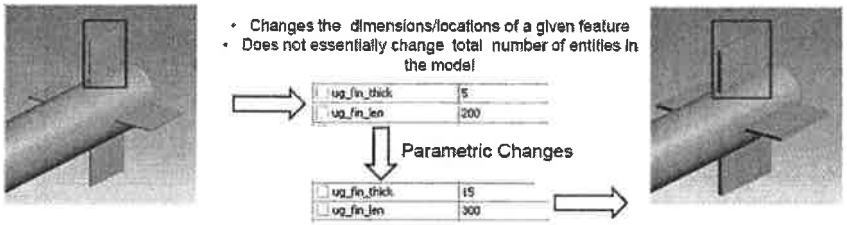
- Parameterization helps to execute a set of operations repeatedly with different set of values
  - Important features, such as diameter of cooling holes, camber etc can be made parametric
  - Very useful for analyzing various design points by modifying geometry/topology parameters
- DM supports sketch-to-3D approach to create parametric models in addition to accessing direct CAD parameters
  - Plane creation inputs, Sketch dimensions, 3D operation limiting values , etc can be treated as parameters
  - The parameters can be linked with each other through expressions

Creating 3D geometry model

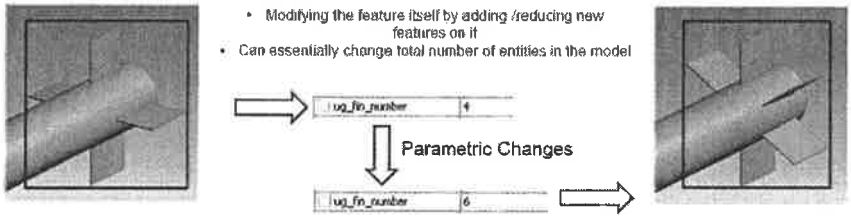
ANSYS DesignModeler

Type of Parameters in DM

- Geometric Parameters:



- Topological Parameters:

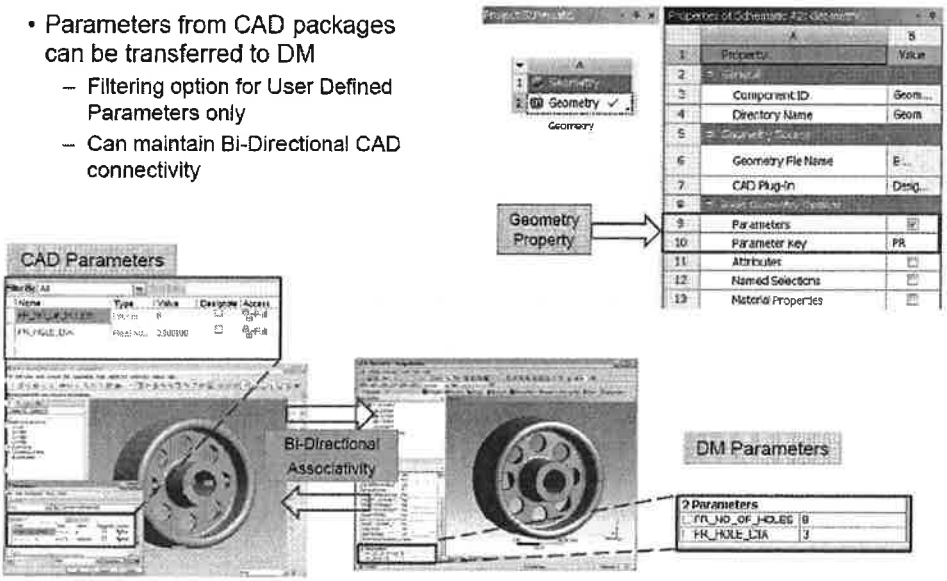


Creating 3D geometry model

ANSYS DesignModeler

Parameters from CAD

- Parameters from CAD packages can be transferred to DM
  - Filtering option for User Defined Parameters only
  - Can maintain Bi-Directional CAD connectivity

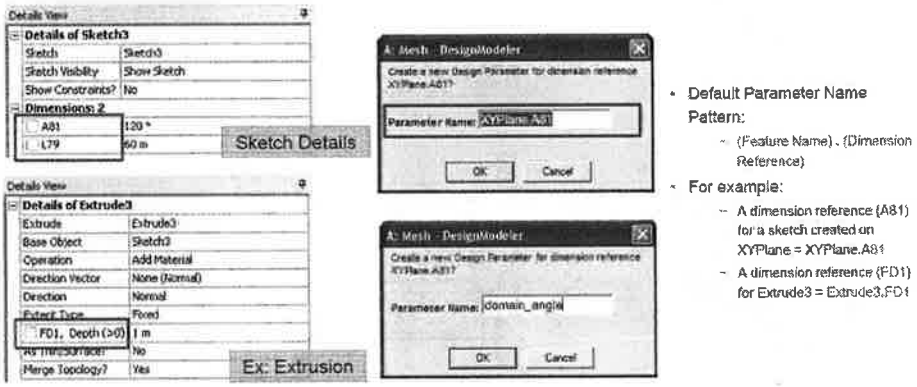


Creating 3D geometry model

ANSYS DesignModeler

Parameters under DM

- Most of the DM operations can be parameterized
  - Any input under 'details view' with "□" sign as prefix can be parameterized
  - Toggle "□" sign to "D" to parameterize that particular input
- The parameters can be renamed



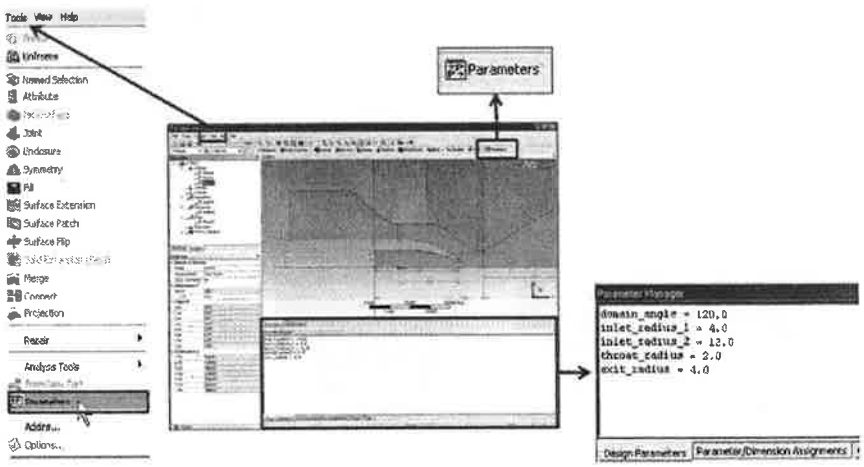
- Default Parameter Name Pattern:
  - (Feature Name) - (Dimension Reference)
- For example:
  - A dimension reference (A81) for a sketch created on XYPlane = XYPlane.A81
  - A dimension reference (FD1) for Extrude3 = Extrude3.FD1

Creating 3D geometry model

ANSYS DesignModeler

Parameter Manager

- When a parameter is defined, it is added under Parameter Manager
  - Can edit parameters

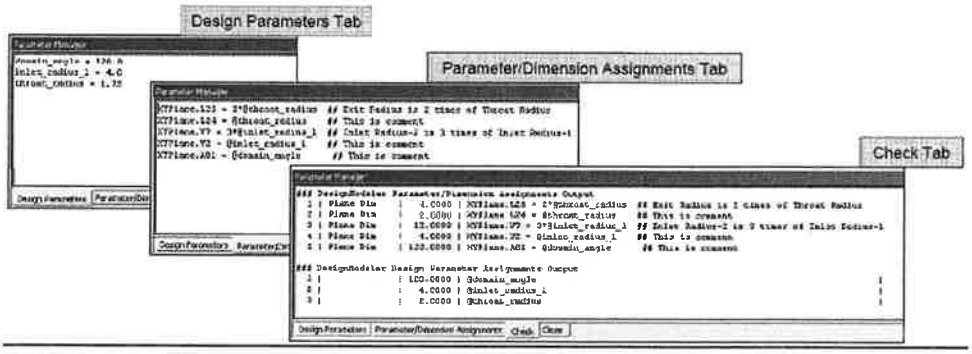


Creating 3D geometry model

ANSYS DesignModeler

Parameter Manager

- Parameter Manager contains tabs to create/edit parameters and its relations
  - Independent variables are defined under "Design Parameters" tab
    - These parameters appear in Project Schematic of WB under "Parameter Set"
  - Dependent variables are defined under "Parameter/Dimension Assignments" tab
    - Expressions can be used
  - Expressions can be verified under "check" tab



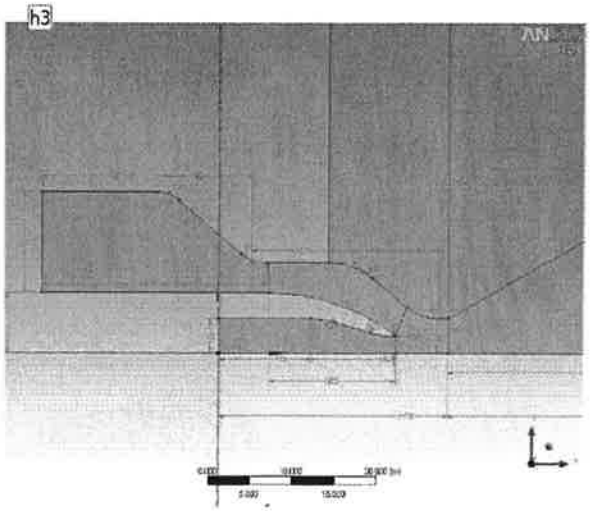
Creating 3D geometry model

ANSYS DesignModeler

Example

- Defining parameters for eductor case using sketches

Parameterization



Day 2 (2:15pm – 3:45pm)

- Hands on with CFD software
- CFD simulation for building ventilation design – Part 1
  - Tutorial for Practice 1 - Simulation of air flow at estate level (step by step guideline)

Day 3 (9:00am – 10:30am)

- Introduction to CFD
  - Mathematical Background
  - Turbulence Model
  - Wind Effect on Building Structure
  - Verification and Validation in CFD

Mathematical Background

From the general description

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

to a two-dimensional time dependent transport equation

$$\frac{\partial c}{\partial t} + u\frac{\partial c}{\partial x} + v\frac{\partial c}{\partial y} = D\left(\frac{\partial^2 c}{\partial x^2} + \frac{\partial^2 c}{\partial y^2}\right)$$

Mathematical Background

- General advection-diffusion equation

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

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	$c_p T$	k	Q
Species	c/ρ	d	Q_m

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

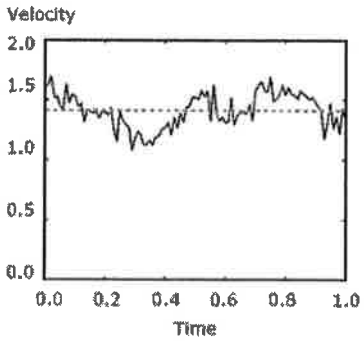
$$\varphi_j=k\left(\frac{\partial T}{\partial x_j}\right) \qquad -\overline{u_j'T'}=k_t\left(\frac{\partial \overline{T}}{\partial x_j}\right)$$

- 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

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

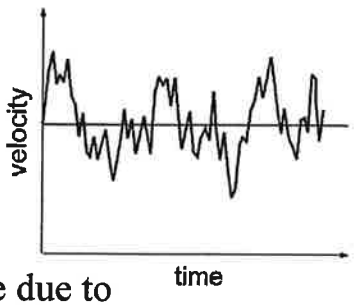
Turbulence Models

- 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 $\overline{\rho u_i' u_j'} = -\mu_t \frac{\partial \bar{u}}{\partial y}$
   - 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

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

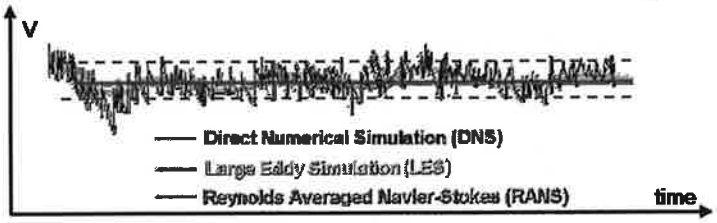
Turbulence Models

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) → For fire/smoke modelling (with buoyancy)
   - 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



LES

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

Spatial Filtering

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

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

Time-Averaging of the Instantaneous Flow Equations / …

The following mathematical rules (called ***Reynolds' Rules***) are used to time-average the instantaneous flow equations (2-D representation):

$$\overline{u+v}=\bar{u}+\bar{v} \qquad \overline{\frac{\partial u}{\partial x_i}}=\frac{\partial\bar{u}}{\partial x_i} \qquad \overline{uu'}=0$$

$$\overline{u\cdot v}=\bar{u}\cdot\bar{v} \qquad \overline{u\cdot v}=\bar{u}\cdot\bar{v}+\overline{u'v'}$$

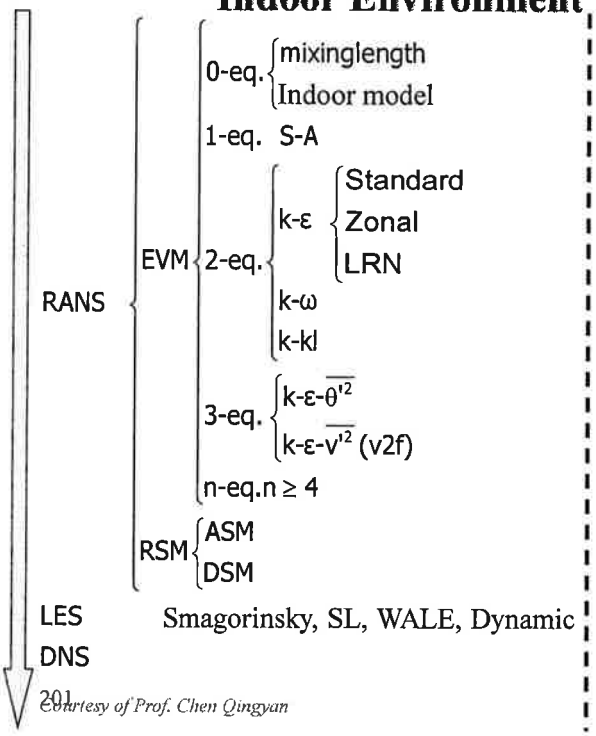
Turbulent shear Stress

For non-isothermal flows, the Boussinesq hypothesis (i.e. $\rho \alpha 1/T$) is applied. This means that the effect of density variations in all equations except in the *y*-***momentum*** equation are neglected, and the density variations in the buoyancy term induce the vertical motion of the fluid.

For two-dimensional Newtonian flow the resulting equations are given in the next slide (after dividing by ρ).

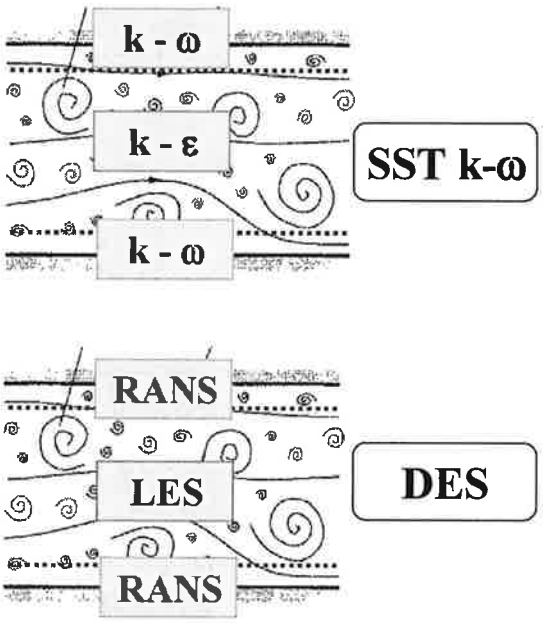
Different Turbulence Model

Indoor Environment Turbulence Models



Courtesy of Prof. Chen Qingyan

Coupling between models



Different Turbulence Model

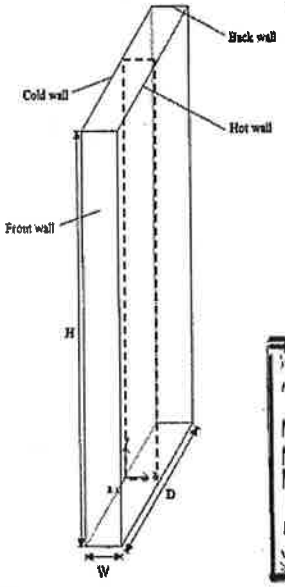
Indoor Environment Modeling: Models Tested

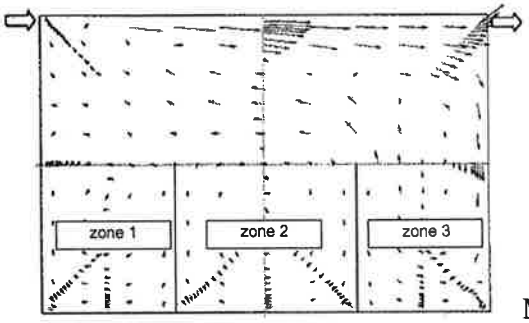
Group		Tested for indoor airflows	Selected for comparison
EVM	Zero eqn	Indoor zero-eq. (Chen and Xu, 1999)	Indoor zero eq.
	Two eqns	Standard k-ε (Launder and Spalding, 1974) RNG k- ε (Yakhot and Orszag, 1986) realizable k- ε (Shih et al., 1995)	RNG k- ε
		Launder-Sharma (1974) LRN k- ε Lam-Bremhorst (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

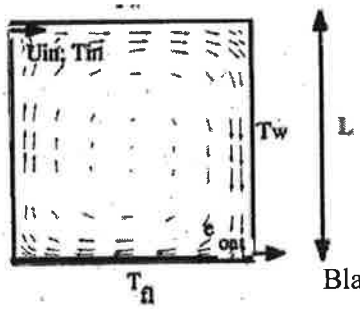
Betts and Bokhari (2000)

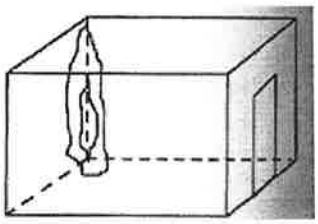




Ito et al (2000)

Murakami et al. (1995)





Blay et al (1992)

Different Turbulence Model

- k-ε models
  - Standard k-ε
  - Renormalization-group (RNG) k-ε
  - Realizable k-ε
- The three models above are different as follows
  1. Method of calculaitng turbulent viscosity
  2. Turbulent Schmidt numbers governing turbulent diffusion of k and ε
  3. Generation and destruction terms in ε equations
- Reynolds Stress Model (RSM)
- Large Eddy Simulation (LES)
- *Model listed above are by no means exhaustive

Different Turbulence Model

Turbulence Modelling – Turbulence Viscosity Models (RANS)

The turbulent shear stresses in the previous equations:

$$-\overline{u'_i u'_j} \qquad -\overline{u'_i T'} \qquad -\overline{u'_i C'}$$

are unknown, and they must be represented using a ***Turbulence Model***.

The ***Turbulence Viscosity*** (μ_t) concept leads to: $\rho\overline{u'v'} = -\mu_t \frac{\partial \bar{u}}{\partial y}$

Using, e.g. the ***k-ε*** turbulence model, μ_t is given by: $\mu_t = c_\mu \rho \frac{k^2}{\varepsilon}$

This leads to solving two additional equations for ***k*** and ***ε***.

***k*-equation** & **ε-equation**

Different Turbulence Model

Turbulence Models (RANS)

- ***k-ε* Turbulence Model:**
  Simple and widely used but not accurate in some situations, e.g. flow separation, buoyancy, etc.
- **Shear Stress Transport (SST) Model:**
  Eddy viscosity model which includes *a combination of the **k-ω** (where **ω=ε/k**) model (in the inner boundary layer) and the **k-ε** model (in the outer region of and outside of the boundary layer).*
- **Renormalized Group (RNG) Model:**
  Represents the effects of the small-scale turbulent motion by a large-scale turbulence and a modified viscosity

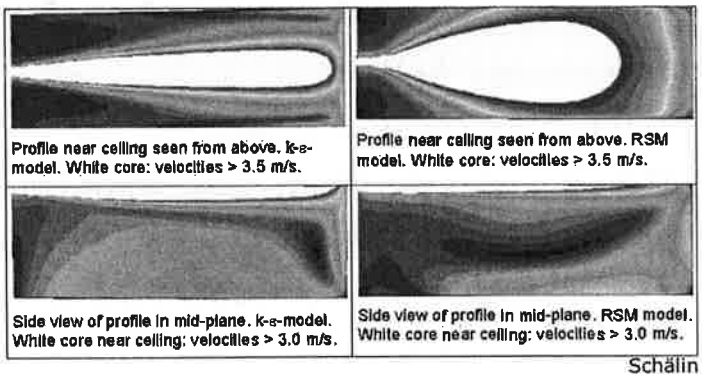
Different Turbulence Model

Turbulence Models/...

- **Reynolds Transport Stress (RTS) Model:**
  Non-Isotropic model that includes six unknown stresses, namely:
  $\overline{u'u'}$ $\overline{v'v'}$ $\overline{w'w'}$ $\overline{u'v'}$ $\overline{u'w'}$ *and* $\overline{v'w'}$
- **Large Eddy Simulation (LES) Model:**
  Only the turbulent fluctuations which are smaller than the control volume are modelled with a turbulence model (subgrid-scale) fluctuations and the turbulence scales that are larger than the control volume - the grid-scales (GS) - are obtained from the unsteady Navier-Stokes equation.

Different Turbulence Model

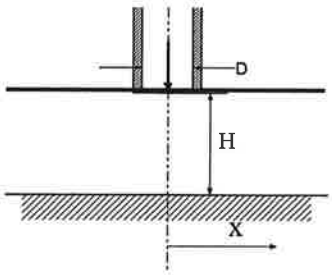
- 3D ceiling jet in a room simulated by a k-epsilon Model and a Reynolds Stress Model
- With k-ε, the ceiling jet becomes too thick. However, with RSM the RSM remains thin due to strong damping of turbulent fluctuations perpendicular to ceiling
- RSM provide better prediction on spreading rate of ceiling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

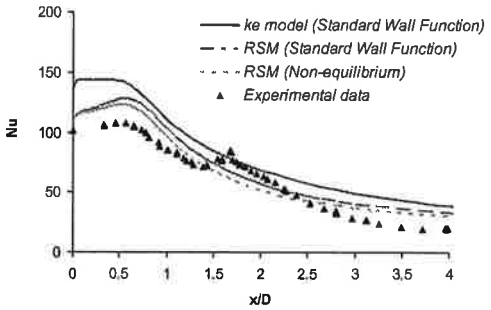


Schälin and Nielsen

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

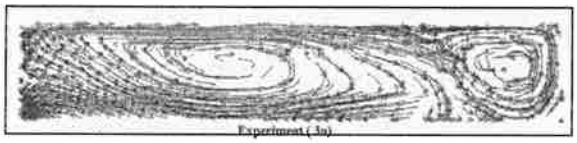




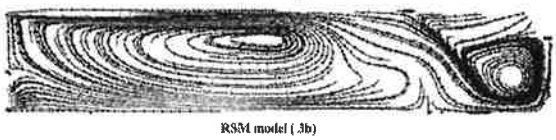
209

Different Turbulence Model

Turbulence Models

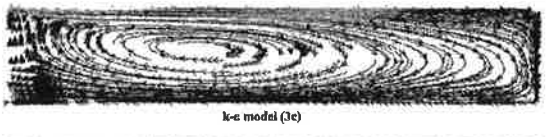


LDV Measurements



RSM model (.3b)

RSM Fine grid: 624,000 cells



k-ε model (3c)

Standard κ - ε



RNG k-ε model (3d)

RNG κ - ε



RSM model with a coarse grid (.3e)

RSM Coarse grid: 240 000 cells

Different Turbulence Model

Indoor Environment Modeling: Model Performance

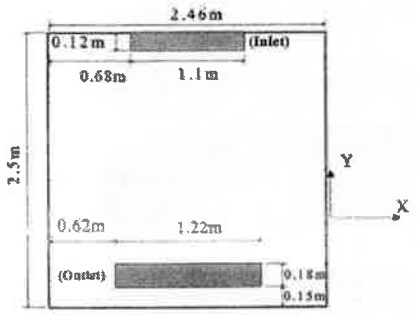
Cases	Compared items	Turbulence models							
		0-eq.	RNG k-e	SST k-w	LRN -LS	V2f-dav	RSM-IP	DES	LES
Natural convec-tion	Mean Temp.	B	A	A	C	A	A	C	A
	Mean Velo.	D	B	A	B	A	B	D	B
	Turbulence	n/a	C	C	C	A	C	C	A
Forced convec-tion	Mean Velo.	C	A	C	A	A	B	C	A
	Turbulence	n/a	B	C	B	B	B	C	B
Mixed convec-tion	Mean 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	

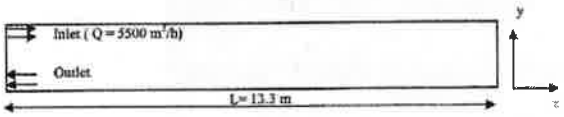
21

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

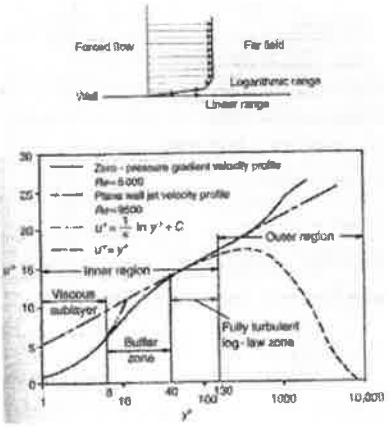
213

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

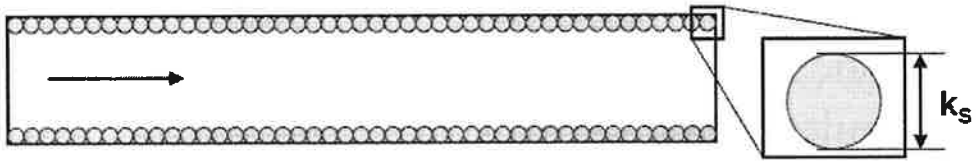
Wall Boundary

- 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



Surface Roughness in Urban Canopy Flow

- 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)

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

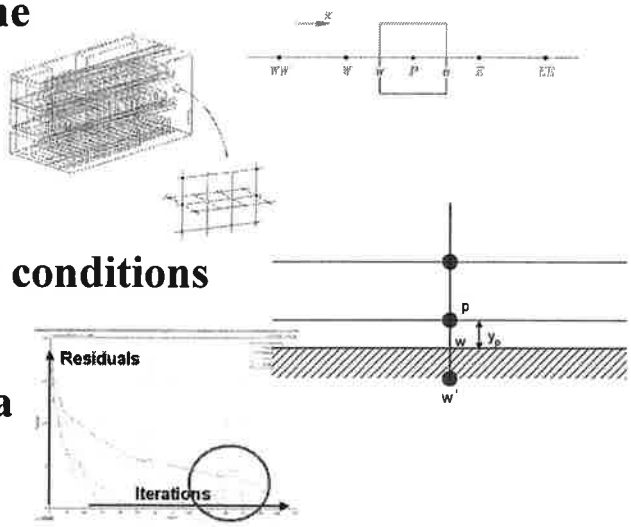
- Atmospheric flows: large-scale roughness (expressed by aerodynamic roughness length y_0)
- Many commercial CFD codes (Fluent, Ansys-CFX, …) provide only the first type of roughness modification.

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

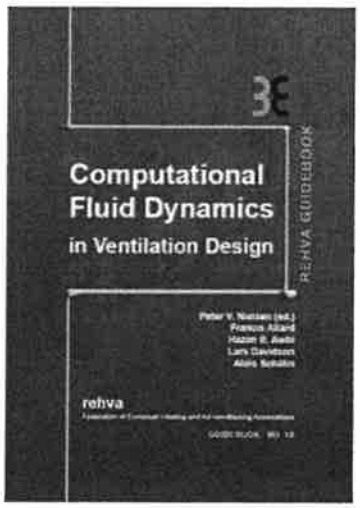
The accuracy of a CFD solution depends on a number of factors and in particular the following:

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

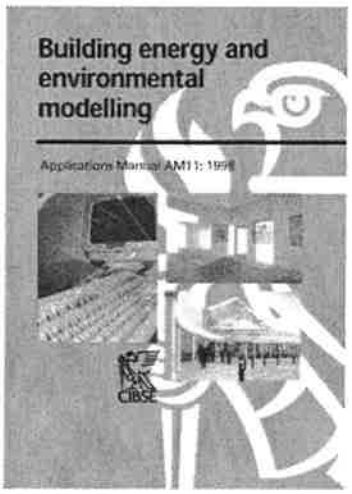


Modelling Guide Books

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 13 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

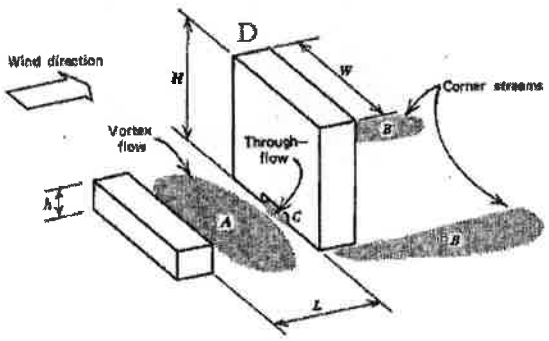
Accurate CFD simulation of coupled outdoor wind flow and indoor air flow is essential for the design and evaluation of natural cross-ventilation strategies for buildings. It is widely recognized that CFD simulations can be very sensitive to the large number of computational parameters that have to be 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.

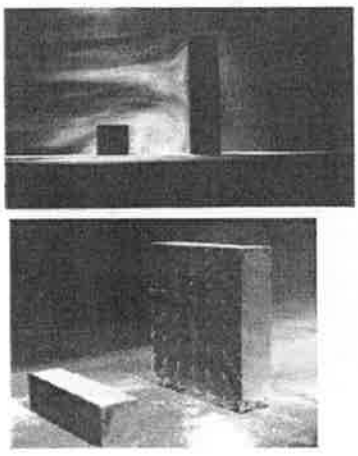
© 2012 Elsevier Ltd. All rights reserved.

21 *Ramponi & Blocken, Building and Environment, Vol. 53, pg. 34 – 48, 2012*

Wind Effects on Building Structure

- 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





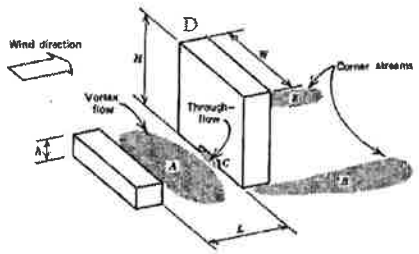
220 * E. Simiu and R. Scanlan, Wind Effects on Structures

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

Verification and Validation in CFD

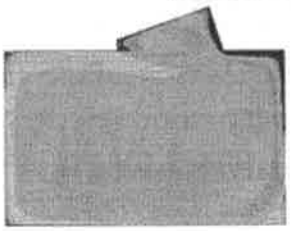
- Definition given in AIAA guide (1998) & Roache (1998)
  - Verification
    - Procedure to ensure the program solves the equations correctly
  - Validation
    - Procedure to test the extent to which the model accurately represent reality
  - Calibration
    - Procedure to assess CFD code to predict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

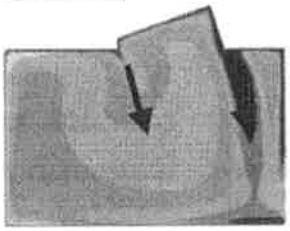
Verification and Validation in CFD



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







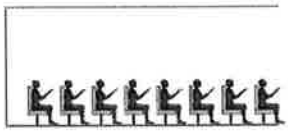
2D steady state 3D transient

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







Verification and Validation in FDS

- Definition given in ASTM E 1355, "Standard Guide for Evaluating the Predictive Capability of Deterministic Fire Models (2004)
  - Verification
    - The process of determining that the implementation of a calculation method accurately represents the developer's conceptual description of the calculation method and the solution to the calculation method - check of the math
  - Validation
    - The process of determining the degree to which a calculation method is an accurate representation of the real world from the perspective of the intended uses of the calculation method - check of the physics

Verification in FDS

- Analytical tests
  - 1D heat conduction, Radiation solver with simple flat plate objects with heat convection turned off
- Numerical tests - Sensitivity analysis
  - Grid size – most important numerical parameters in the model; FDS is second order accurate in spatial and temporal, halving grid size reduce discretization error by factor of 4, but increase computational time of factor of 16
  - LES, Radiation parameters, Thermo-physical properties of solid fuels
- Code checking on different hardware and OS
- Basics Hydrodynamic Solver, Thermal radiation, Heat Conduction, Pyrolysis, Droplets

Validation in FDS

- Broadly two classes
  1. Fire is specified as input to the model – design application
  2. Fire is predicted by the model – forensic reconstruction
- Selected Validation cases
  - Fire Plume, Pool Fires, Growing Fires, Flame Spread, Compartment Fires & Airflows
  - Displacement ventilation, wind engineering
  - Tunnel fires
  - Sprinklers Water Suppression, Smoke Detection
  - Reconstruction of actual fires
- It is the responsibility of model user to decide if the model is appropriate for the given application

Numerical Methods

- One-dimensional case
  - For room air distribution, it is IMPOSSIBLE to make direct analytical solution from the differential equation systems
  - Need to reformulate the differential equations into difference equations for which solution can be obtained through numerical method
  - One dimensional convection-diffusion equation

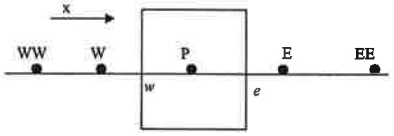
$$\rho u \frac{dc}{dx} = \Gamma_c \frac{d^2c}{dx^2} + S_c$$

Numerical Methods

$$\rho u\frac{dc}{dx}=\Gamma_c\frac{d^2c}{dx^2}+S_c$$

- Finite Volume

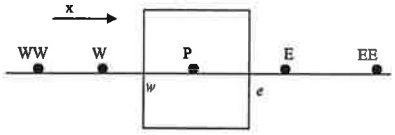
$$\rho(u_e c_e-u_w c_w)=\Gamma_c\left[\left(\frac{dc}{dx}\right)_e-\left(\frac{dc}{dx}\right)_w\right]+S_c\Delta x$$



  - Replace the values at the cell surfaces with values at the grid points (assume *u* constant and source term is zero)
  - Assume piecewise linear profile between grid points to obtain values on cell surfaces and the gradient – central difference scheme - 2nd order accurate

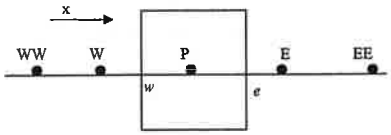
$$\left(\frac{dc}{dx}\right)_w=\frac{c_W-c_P}{\Delta x} \qquad c_w=(c_W+c_P)/2$$

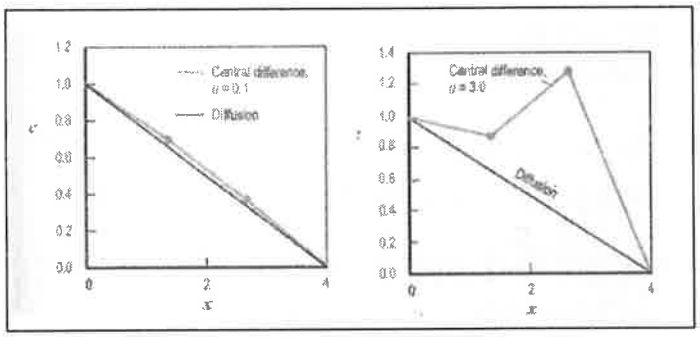
Numerical Methods



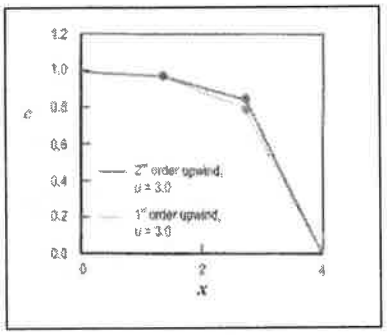
- Solution of transport equation
  - With central difference scheme in convection term, Peclet number ($Pe=\frac{\rho\Delta xu}{\Gamma_c}$)must be smaller than 2 to ensure convergence and stable solutions
  - Convection is asymmetric phenomenon (upstream conditions have greater influence than downstream condition) – 1st order accurate Upwind scheme – unconditional stable $c_w=c_W;u\geq 0$ $c_w=c_W;u<0$
  - 1st order Upwind scheme has error for a flow which has an angle to grid line; maximum error at 45^0

Numerical Methods



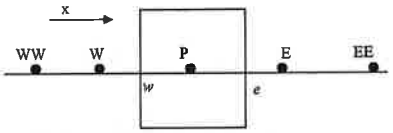


- Effect of Peclet number on central difference scheme



- Upwind Scheme

Numerical Methods



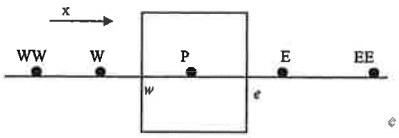
- False diffusion
  - 2nd order upwind scheme (values on cell surface are based on values in two upstream notes)

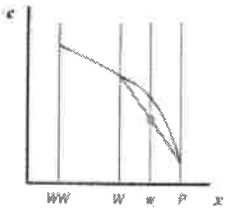
$$c_w = \frac{3}{2}c_W - \frac{1}{2}c_{WW}; u \geq 0 \qquad c_w = \frac{3}{2}c_P - \frac{1}{2}c_E; u < 0$$

  - QUICK scheme – central difference scheme with stabilizing upstream weighted curvature correction arising from 2nd order polynomial fit

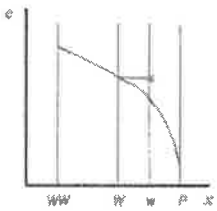
$$c_w = \frac{3}{4}c_W + \frac{3}{8}c_P - \frac{1}{8}c_{WW}; u \geq 0 \qquad c_w = \frac{3}{4}c_P + \frac{3}{8}c_W - \frac{1}{8}c_E; u < 0$$

Numerical Methods

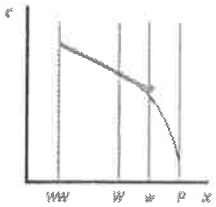




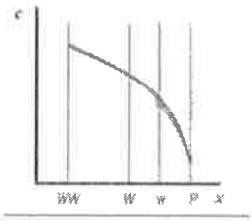
Central difference



1st order Upwind

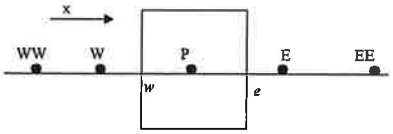


2nd order Upwind

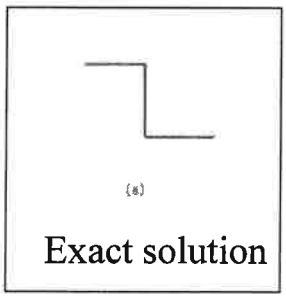


QUICK

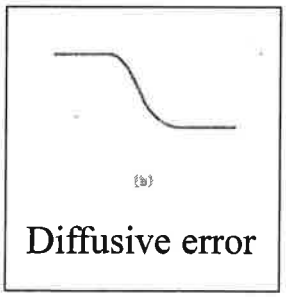
Numerical Methods



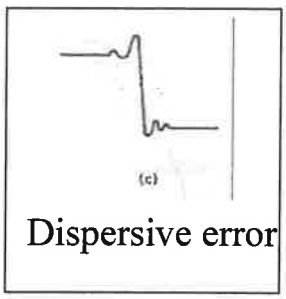
- 1st order Upwind – diffusive error
- 2nd order Upwind – dispersive error



Exact solution



Diffusive error



Dispersive error

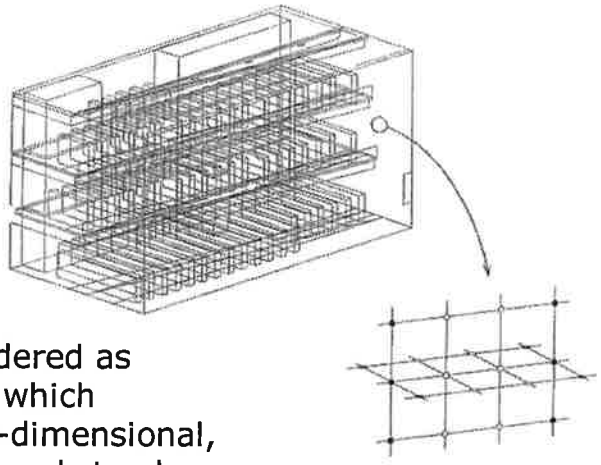
Numerical Methods

- Grid
  - More cells near wall region to reduce turbulence modeling error – e.g. 20 cells inside boundary layer for low Re κ-ε turbulence flow
  - Fine grids used in areas with large gradients to minimize false diffusion and dispersive error
  - Multi-block approach is used – combination of structured and unstructured grid
  - Adaptive Grid
    - Boundary ,Velocity, Gradient, Region, Volume
  - Geometry modeling and grid generation is the most labor intensive job in CFD simulation – 80% job scopes for complex geometry

Numerical Methods

- Iteration Procedure
  - Gauss-Seidel
    - Visiting each grid points in certain order
  - Tri-Diagonal-Matrix Algorithm (TDMA)
    - Direct solution of equation system
  - Pressure Velocity Coupling – SIMPLE
    - Continuity equation is rewritten for pressure correction
    - Staggered grid to avoid non-physical oscillations
  - Multi-grid Acceleration Method
    - Numerical wiggling are damped for periodicity same length as grid distance
    - V-cycle, W-cycle
    - Performed without user interaction

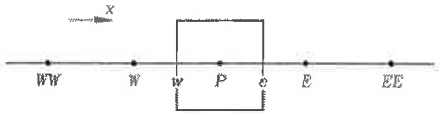
One-Dimensional Case



The case can be considered as a small part of a flow, which in certain areas is one-dimensional, parallel with grid lines and steady.

$$\rho u \frac{dc}{dx} = \Gamma_c \frac{d^2 c}{dx^2} + S_c$$

One-Dimensional Discretization Equation



Control volume formulation:

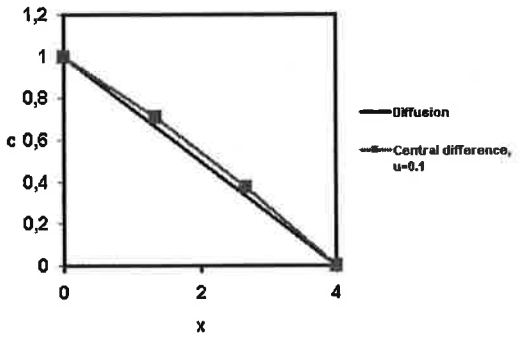
$$\rho(u_e c_e - u_w c_w) = \Gamma_c \left[\left(\frac{dc}{dx} \right)_e - \left(\frac{dc}{dx} \right)_w \right] + S_p \Delta x$$

It is necessary to replace values at the cell surfaces *e* and *w* with values from the grid points *WW*, *W*, *P*, *E* and *EE* to have a final version of the discretization equation.

Different Discretization Equations, 1

The flow is studied in a case where the length x is equal to 4. The boundary values c_o and c_3 are equal to 1.0 and 0.0.

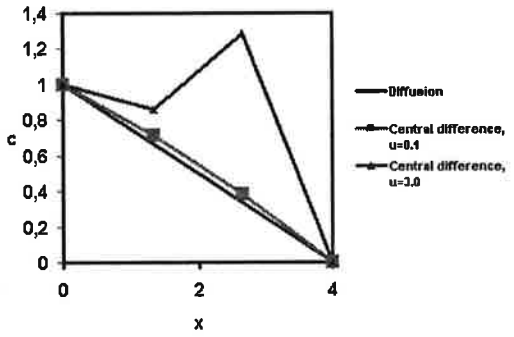
Central differences is e.g. $c_e = (c_P + c_E)/2$ and $u = 0.1$



Different Discretization Equations, 2

Central differences e.g. $c_e = (c_P + c_E)/2$ and $u = 3.0$

Wiggly for $Pe = \frac{\rho \Delta x u}{\Gamma_c}$ larger than 2



History and Numerical Schemes, 1

The sixties
The central difference scheme becomes unstable (wiggly) when the Peclet number is large. The cure is to decrease the grid size.

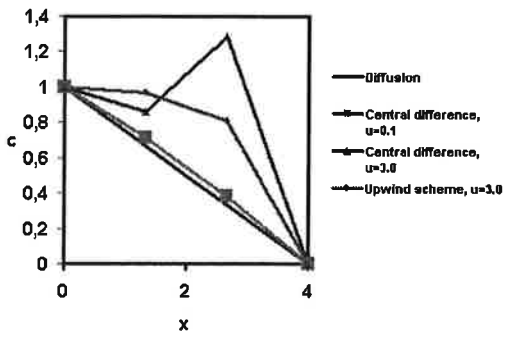
The seventies
Upwind difference opened the way for infinitely high Reynolds numbers, but false diffusion could in many cases be larger than diffusion of physical kind.

The eighties and the nineties
Second order schemes decreased the effect of false diffusion.

Different Discretization Equations, 3

The flow is studied in a case where the length x is equal to 4. The boundary values c_o and c_3 are equal to 1.0 and 0.0.

Upwind scheme e.g. $c_e = c_P$ and $u = 3.0$

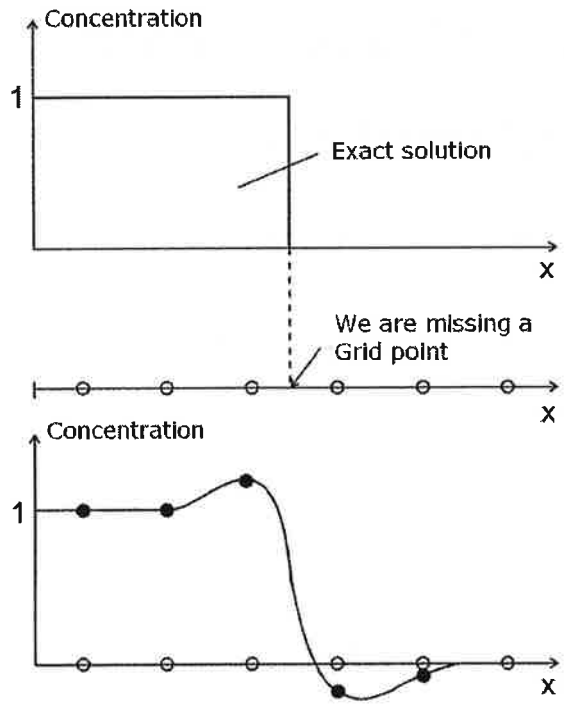


History and Numerical Schemes, 2

The sixties
The central difference scheme becomes unstable (wiggly) when the Peclet number is large. The cure is to decrease the grid size.

The seventies
Upwind difference opened the way for infinitely high Reynolds numbers, but false diffusion could in many cases be larger than diffusion of physical kind.

The eighties and the nineties
Second order schemes decreased the effect of false diffusion.

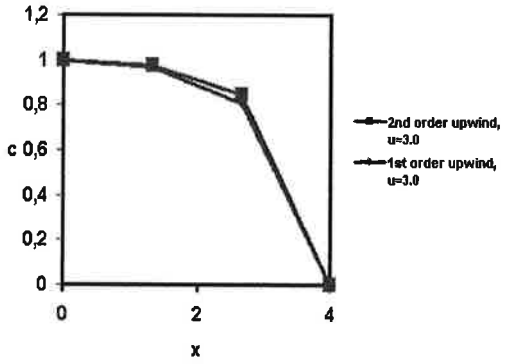


False Diffusion

Higher Order Schemes

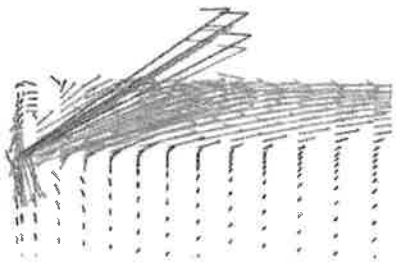
Second order upwind scheme is an example of a new scheme developed in the middle of the seventies.

The one-dimensional case with the velocity u = 3.0 and 1st and 2nd order upwind scheme

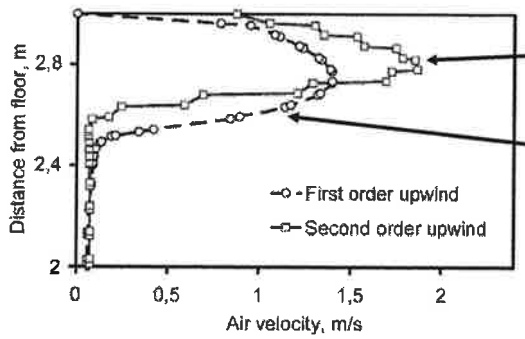


False Diffusion and Order of the Schemes

Flow from an opening, which is inclined at ~30 deg. to the mesh. Three-dimensional flow.



Profile at the upper surface at a distance of 1 m from the opening.



Dispersive error

Diffusive error

Svidt

History and Numerical Schemes, 3

The sixties
The central difference scheme becomes unstable (wiggly) when the Peclet number is large. The cure is to decrease the grid size.

The seventies
Upwind difference opened the way for infinitely high Reynolds numbers, but false diffusion could in many cases be larger than diffusion of physical kind.

The eighties and the nineties
Second order schemes decreased the effect of false diffusion.

Day 3 (10:45am – 12:15pm)

- Creating 3D geometry model
  - Simplifying architecture data
  - Creating water tight model
  - Creating model from primitive data
  - Preparing the model for meshing
  - Using BIM for CFD simulation
- Meshing computational model
  - Avoiding highly skewed geometry
  - Ensuring good quality mesh
  - Mesh densities
  - Different meshing topology

CFD Model Preparation

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

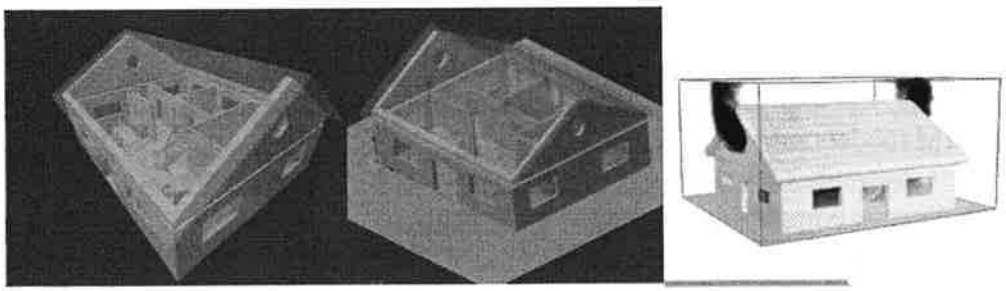
CFD Model Preparation

- 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

CFD Model Preparation - Case studies - Fire modeling in a house

- Operational procedure



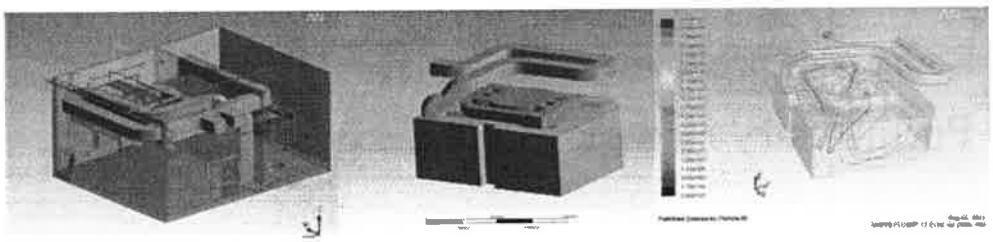


* Open-source codes, toll free

CFD Model Preparation - Case studies - Operating Theatre

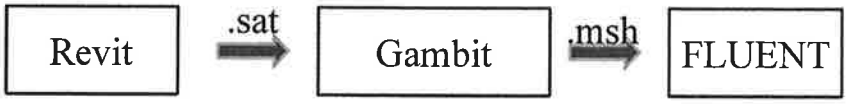
- Operational procedure

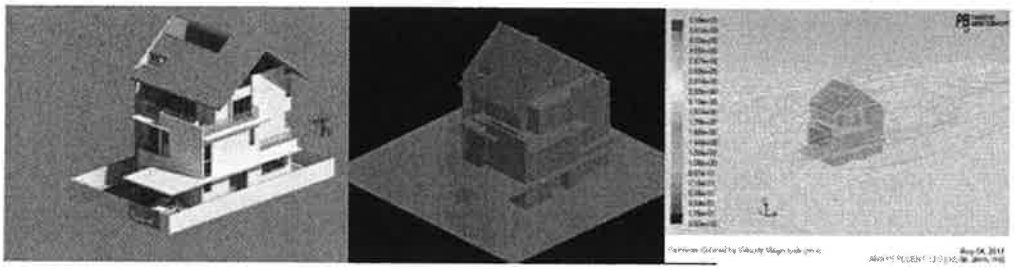




CFD Model Preparation - Case studies - Airflow around a private house

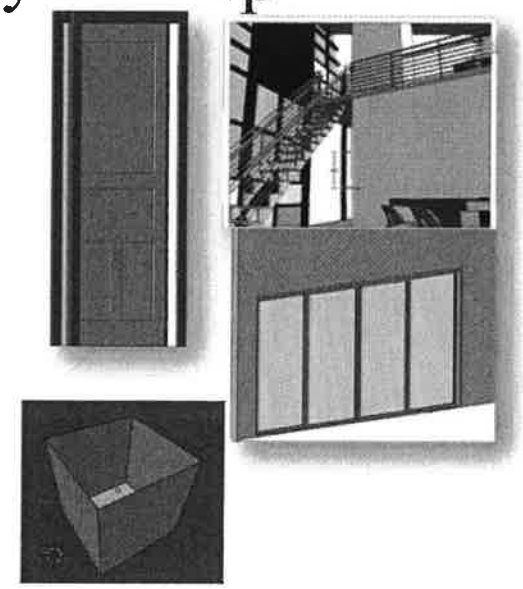
- Operational procedure





CFD Model Preparation 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



CFD Model Preparation - Sources of problems

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

CFD Model Preparation - Remark

- 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

Creating 3D geometry model

- Importing third party geometry data (Building Information Model, BIM)
- Simplifying architecture data
- Creating water tight model
- Creating model from primitive data
- Preparing model for meshing

Creating 3D geometry model

- The most time consuming part for CFD work – typically 70% of time spent on this part, especially geometry clean up
- Simplify the architecture model with reasonable justification and physical assumption

Creating 3D geometry model

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

Creating 3D geometry model

Building Good Models

- Good practice to build a model which does not have any unnecessary small gaps between objects
  - Retains desired connectivity
  - Avoids possible meshing problems
- Set tolerance for gaps using minimum object separation in the Edit Preferences Panel

Creating 3D geometry model

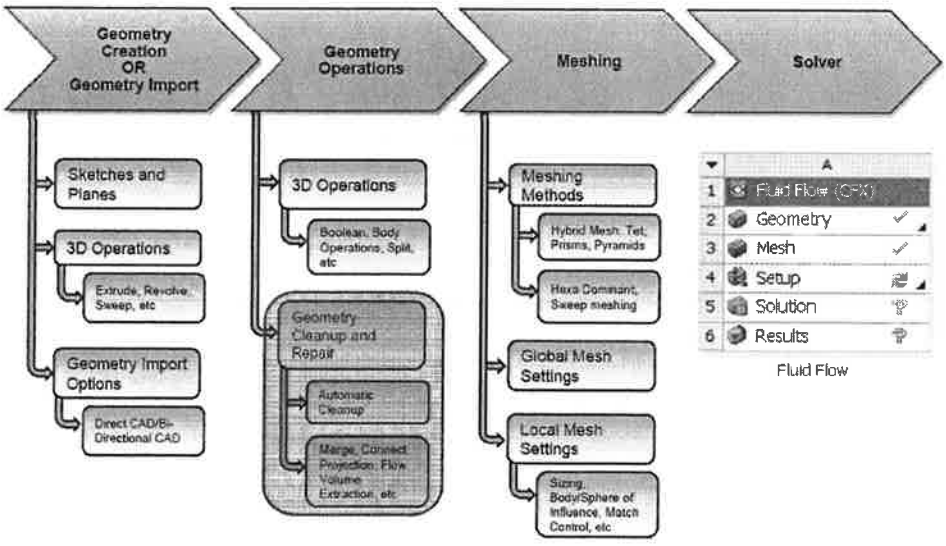
Model Building Considerations

- 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

Creating 3D geometry model

ANSYS DesignModeler

Geometry Clean Up & Repairs



Creating 3D geometry model

ANSYS DesignModeler

Geometry Clean Up & Repairs - Intro

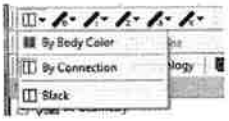
- Several translation methods available to enable data exchange with CAD/CAE systems
  - Direct integration
  - Import of generic CAD formats (IGES, ACIS etc)
- Translation can:
  - Return incomplete, corrupt, or disconnected geometry
  - Return geometry details unnecessary for CAE analysis
- Geometry cleanup refers to processes required to prepare geometry for meshing.
  - Fix incomplete or corrupt geometry and connect disconnected geometry
  - Remove unnecessary details (defeaturing)
  - Decompose geometry into meshable sections
- This lecture covers some of the tools available within DM to prepare the geometry, and examples of where they might be used

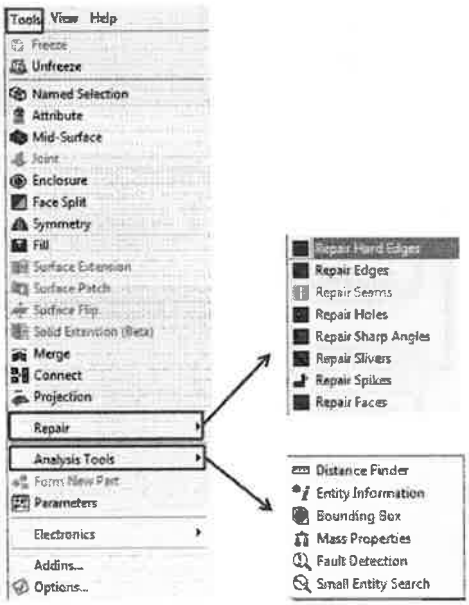
Creating 3D geometry model

ANSYS DesignModeler

Geometry Clean Up & Repairs - Intro

- DM provides geometry tools to:
  - Analyse
  - Repair
  - Modify/Simplify
- All these commands are available under the Tools Menu
- May need to view the model in wireframe mode to visualise the defects
- DM also give the ability to colour edges depending on connectivity



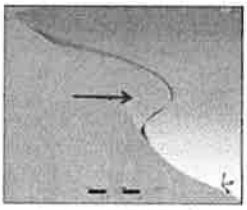


Creating 3D geometry model

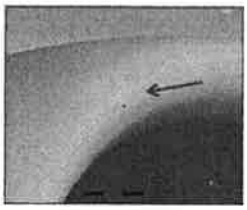
ANSYS DesignModeler

Typical Geometry Issue

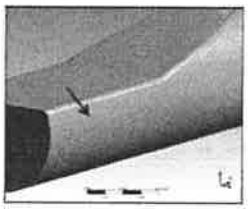
- Missing faces
- Sliver faces
- Hard edges
- Small edges
- Sharp angles
- Others …



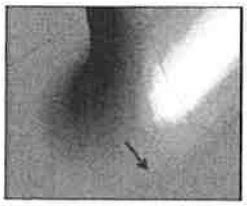
Missing faces



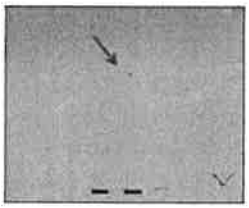
Sliver faces



Hard Edges



Small edges



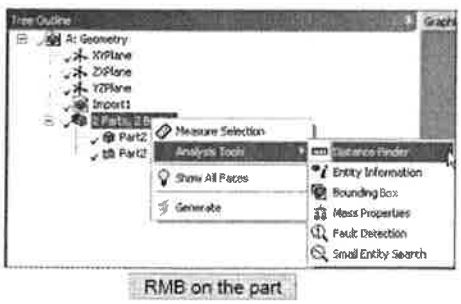
Sharp angles

Creating 3D geometry model

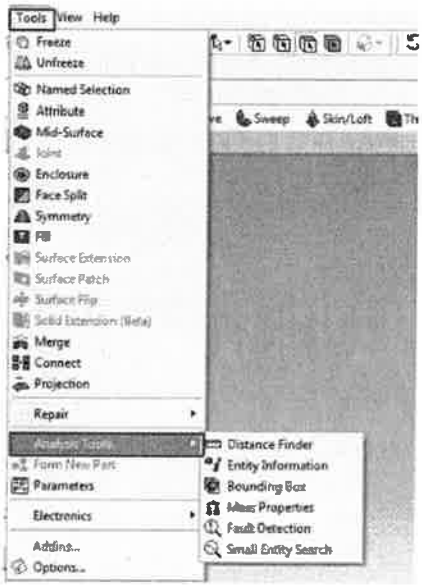
ANSYS DesignModeler

Analysis Tools

- Set of functions that gather information, detect faults in geometry



RMB on the part

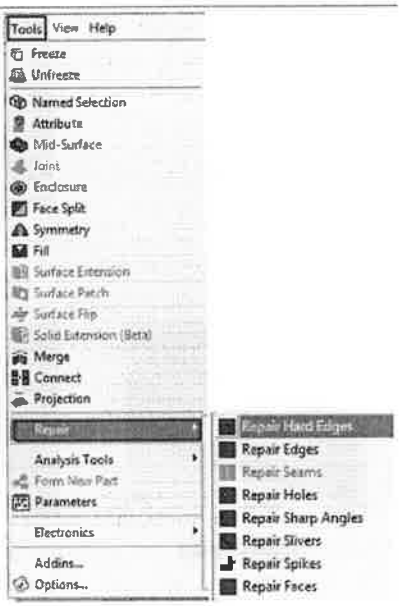


Creating 3D geometry model

ANSYS DesignModeler

Repair Tools

- Easy search and fix unwanted geometrical features or geometric errors
- Can be executed on both frozen and active bodies
- Repair Types
  - Repair Hard Edges
  - Repair Edges
  - Repair Seams
  - Repair Holes
  - Repair Sharp Angles
  - Repair Slivers
  - Repair Spikes
  - Repair Faces

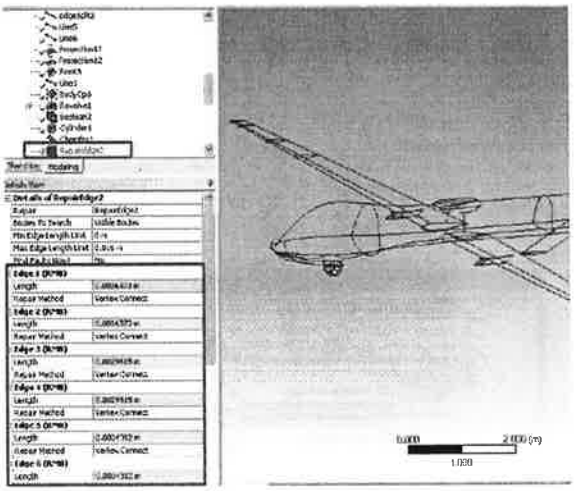


Creating 3D geometry model

ANSYS DesignModeler

Repair Tools - Automation

- Automatically detects and lists out geometrical issues
- Highlights the location of fault in the geometry
  - Zooms into the location, when clicked on the items in the list
- Proposes fixes to the issues
  - choose a different method to fix, if needed
- Analyze the list
- Fix issues all at once
  - Upon clicking "Generate"
  - Can't be done one after the other

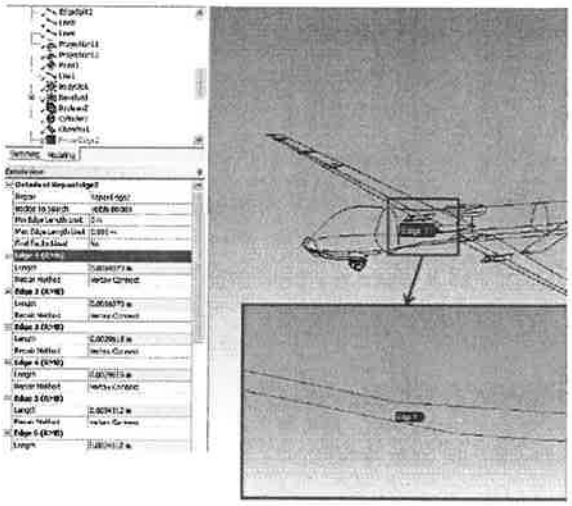


Creating 3D geometry model

ANSYS DesignModeler

Repair Tools - Automation

- Automatically detects and lists out geometrical issues
- Highlights the location of fault in the geometry
  - Zooms into the location, when clicked on the items in the list
- Proposes fixes to the issues
  - choose a different method to fix, if needed
- Analyze the list
- Fix issues all at once
  - Upon clicking "Generate"
  - Can't be done one after the other

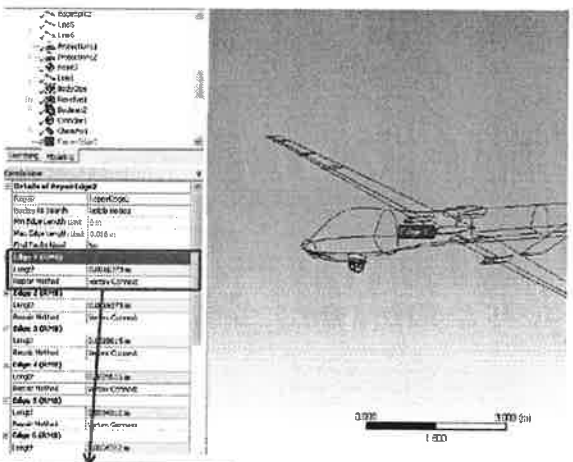


Creating 3D geometry model

ANSYS DesignModeler

Repair Tools - Automation

- Automatically detects and lists out geometrical issues
- Highlights the location of fault in the geometry
  - Zooms into the location, when clicked on the items in the list
- Proposes fixes to the issues
  - choose a different method to fix, if needed
- Analyze the list
- Fix issues all at once
  - Upon clicking Generate
  - Can't be done one after the other



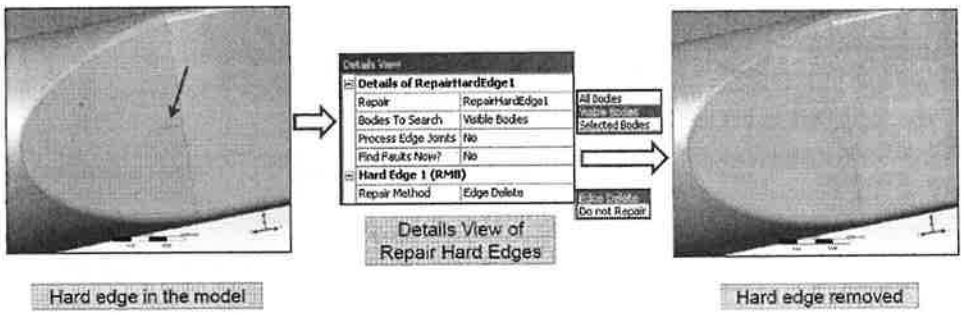
Edge 1 (RMB)	
Length	0.0016373 m
Repair Method	Vertex Connect
Edge 2 (RMB)	Vertex Connect
Length	Edge Delete
Repair Method	Face Merge
Edge 3 (RMB)	Do not Repair

Creating 3D geometry model

ANSYS DesignModeler

Repair Tools - Example

- Repair Hard Edges

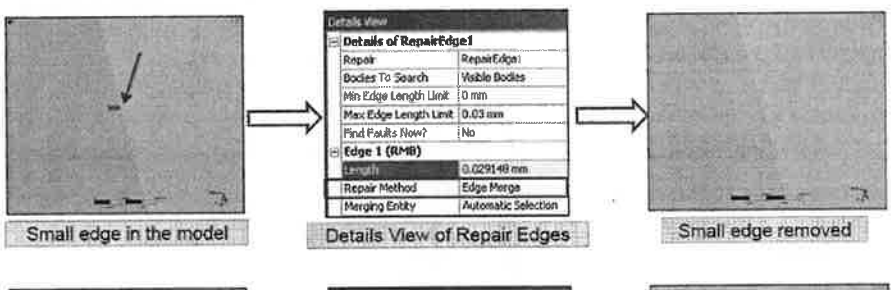


Creating 3D geometry model

ANSYS DesignModeler

Repair Tools - Example

- Repair Edges

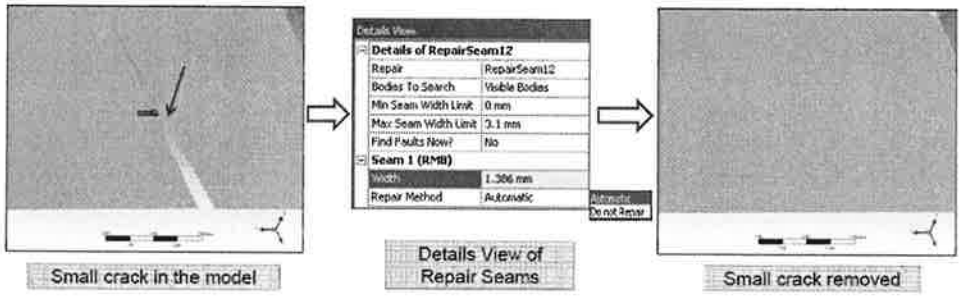


Creating 3D geometry model

ANSYS DesignModeler

Repair Tools - Example

- Repair Seams



Creating 3D geometry model

ANSYS DesignModeler

Repair Tools - Example

Repair Holes

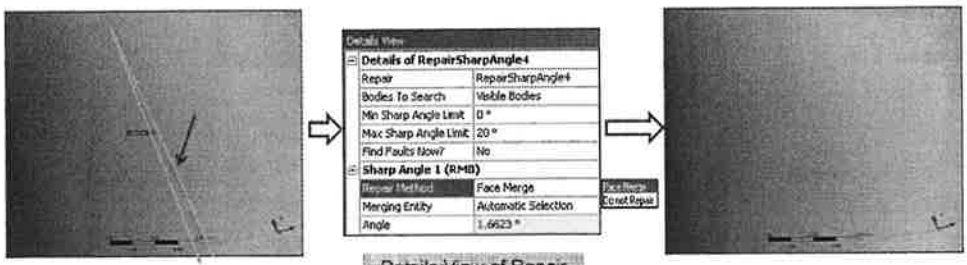


Creating 3D geometry model

ANSYS DesignModeler

Repair Tools - Example

- Repair Sharp Angles



Sharp angle in the model

Details View of Repair Sharp Angles

Sharp Angle removed

Creating 3D geometry model

ANSYS DesignModeler

Repair Tools - Example

- Repair Slivers



Sliver face in the model

Details View of Repair Slivers

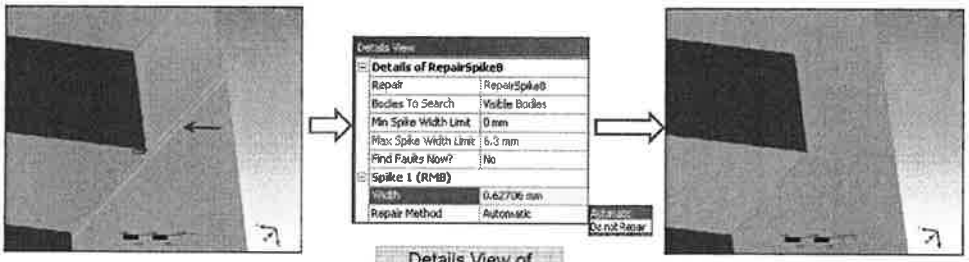
Sliver face removed

Creating 3D geometry model

ANSYS DesignModeler

Repair Tools - Example

- Repair Spikes



Spike in the model

Details View of Repair Spikes

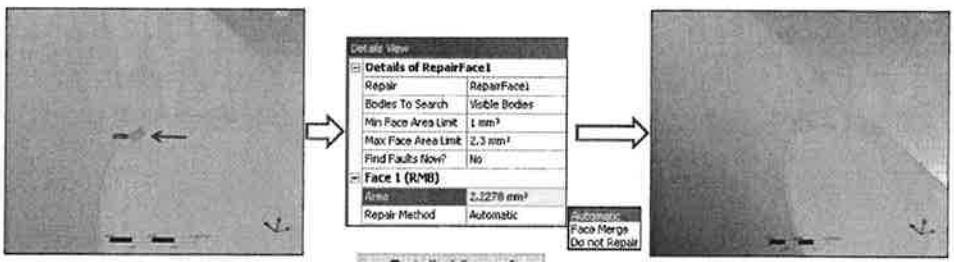
Spike removed

Creating 3D geometry model

ANSYS DesignModeler

Repair Tools - Example

- Repair Faces



Small face in the model

Details View of Repair Faces

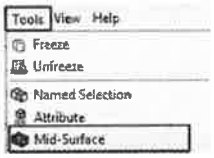
Small face removed

Creating 3D geometry model

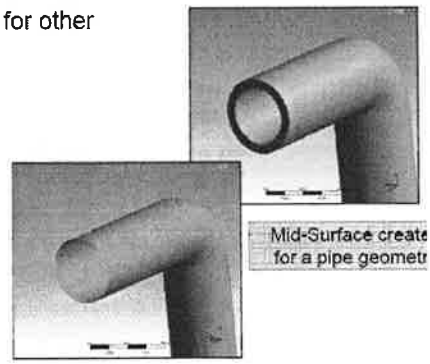
ANSYS DesignModeler

Modify/Simplify: Mid-Surface Extraction Tool

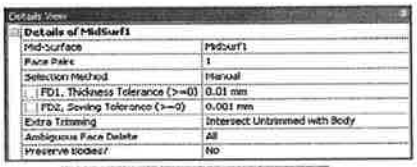
- Extract surface body that is midway between existing solid body faces



- Two Options:
  - Manual: Operates only on user specified face pairs
  - Automatic: Provides option to search for other matching face pairs



Mid-Surface create for a pipe geometr



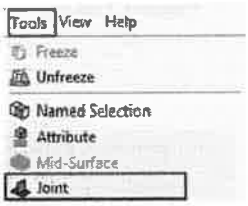
Details View of Mid-Surface

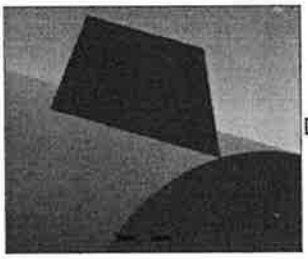
Creating 3D geometry model

ANSYS DesignModeler

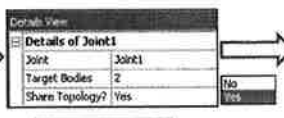
Joint Tool

- Joins surface bodies together such that their contact regions share common edge
  - Prerequisite for conformal meshes
- Takes two or more surface bodies as input
- Imprints edges on all bodies where they make contact

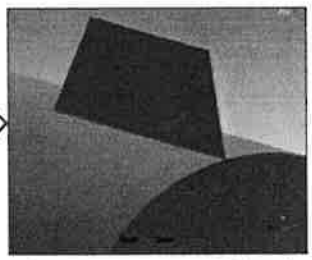




Surface bodies to be Joined



Details View of Joint



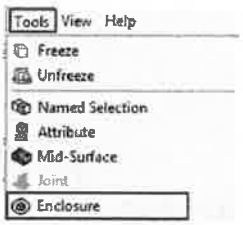
Surface bodies after Joint

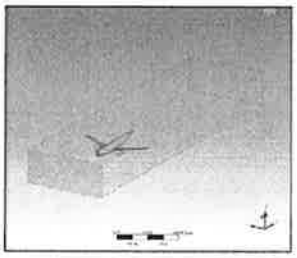
Creating 3D geometry model

ANSYS DesignModeler

Enclosure Tool

- Creates enclosure around the bodies
  - Useful for external aerodynamic analysis
  - Can be used for extracting internal flow domain
- Input
  - All or selected bodies
  - Shape of enclosure
  - Extent of enclosure
- Output
  - A frozen body around the selected parts with specified extents
  - Internal flow volume as a result of automatic Boolean subtract
- Supports symmetry models
- Full or partial models can be included in enclosure



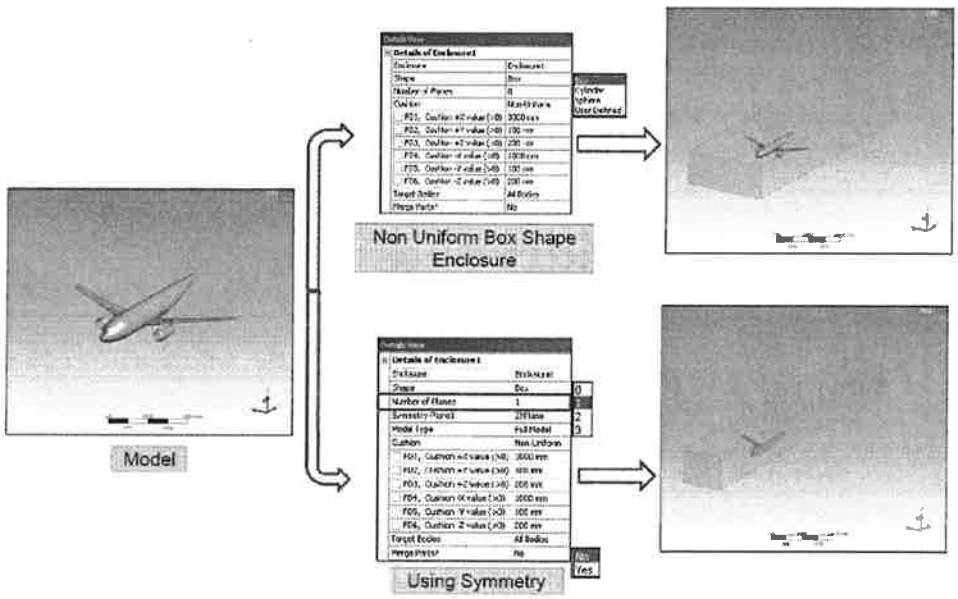


Enclosure created for an airplane

Creating 3D geometry model

ANSYS DesignModeler

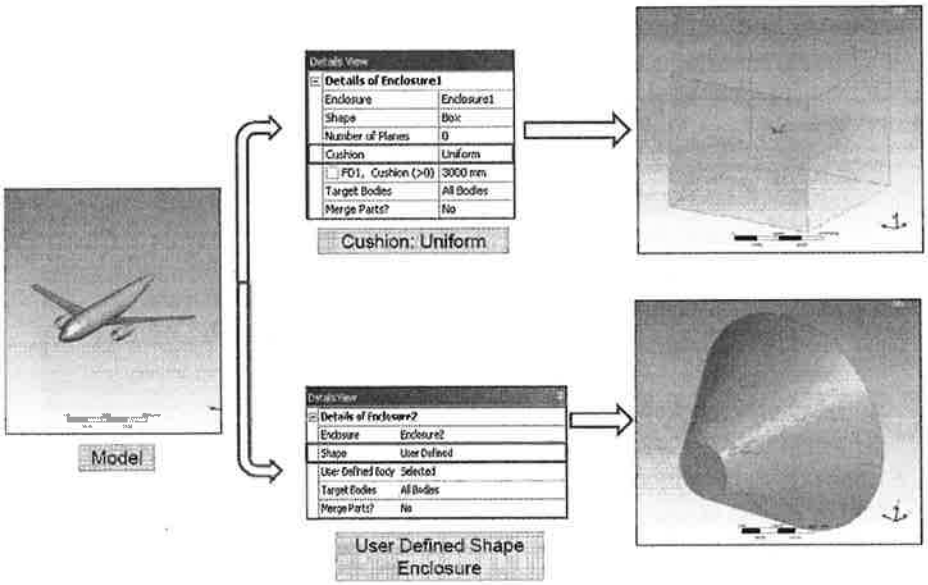
Enclosure Tool : Internal Flow Example



Creating 3D geometry model

ANSYS DesignModeler

Enclosure Tool : Internal Flow Example



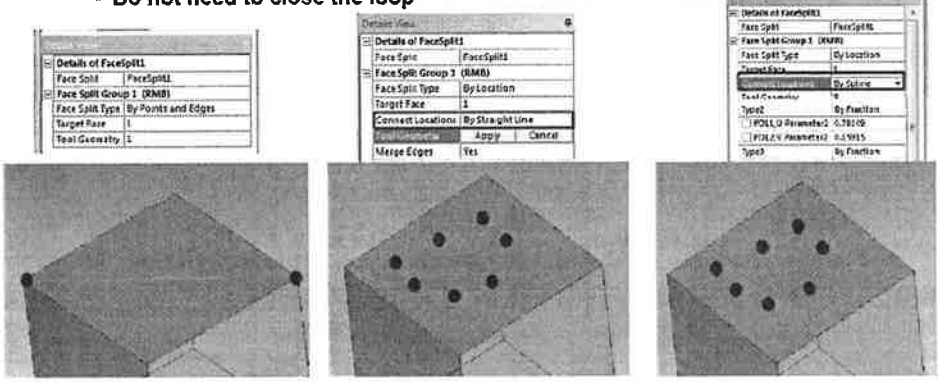
Creating 3D geometry model

ANSYS DesignModeler

Face Split

- **Allows faces to be broken down by:**
  - **Points and Edges**
    - **Combinations of sketch and 3D geometry**
  - **Locations**
    - **"Click" on the surface**
    - **Do not need to close the loop**





Creating 3D geometry model

ANSYS DesignModeler

Symmetry Tool

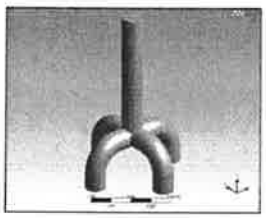
- Defines symmetry for the model
- Input
  - All or selected bodies
  - Up to three symmetry planes
- Operation
  - Symmetry planes slice the model
- Output
  - Sliced model (part lying on the positive side of the axis is retained)

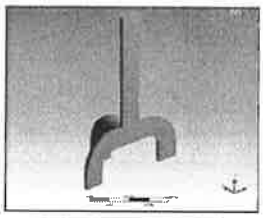


Details View

Details of Symmetry1	
Symmetry	Symmetry1
Number of Planes	2
Symmetry Plane1	XYPlane
Symmetry Plane2	ZXPlane
Model Type	Full Model
Target Bodies	All Bodies

Details View of Symmetry





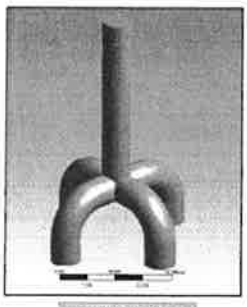
Using Symmetry

Tip: Faces that are coincident with the symmetry plane are tagged with a "Symmetry" named selection under AMP

Creating 3D geometry model

ANSYS DesignModeler

Symmetry Tool : Example

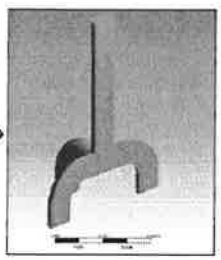


Full Model

Details View

Details of Symmetry1	
Symmetry	Symmetry1
Number of Planes	1
Symmetry Plane1	XYPlane
Model Type	Full Model
Target Bodies	All Bodies

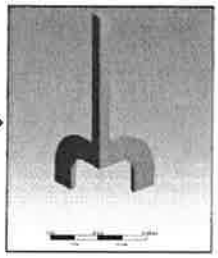
Symmetry using XY plane



Details View

Details of Symmetry1	
Symmetry	Symmetry1
Number of Planes	2
Symmetry Plane1	XYPlane
Symmetry Plane2	ZXPlane
Model Type	Full Model
Target Bodies	All Bodies

Symmetry using XY & ZX plane



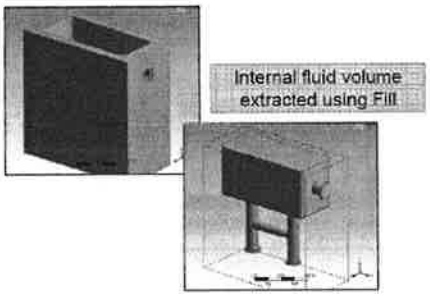
Creating 3D geometry model

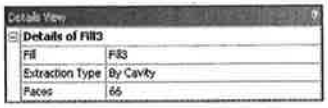
ANSYS DesignModeler

Fill Tool

- Creates the fluid volume using wetted solid faces
- Two options
  - By Cavity
  - By Caps







Details View of Fill

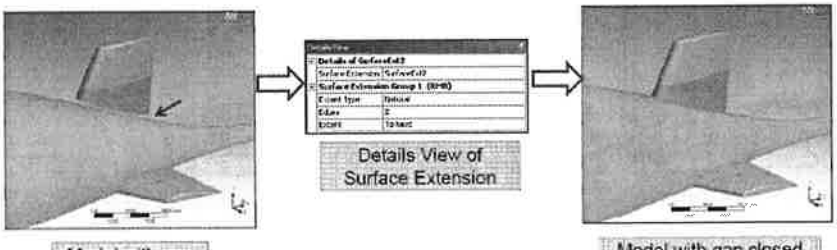
Creating 3D geometry model

ANSYS DesignModeler

Surface Extension Tool

- Allows extension of surface bodies to the desired extent
- Extension type
  - Natural: Based on gradients of selected edges
  - User Defined: User specifies the direction





Model with gap

Details View of Surface Extension

Model with gap closed

Creating 3D geometry model

ANSYS DesignModeler

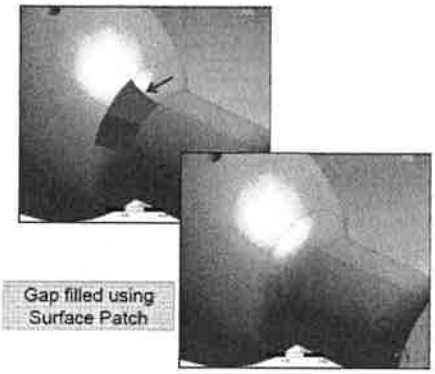
Surface Patch Tool

- Fills the gaps in surface bodies
- Requires closed loop of edges
- Edges can be from different surface bodies
- Allows selection of multiple patch operations in a single go





Details View of Surface Patch



Gap filled using Surface Patch

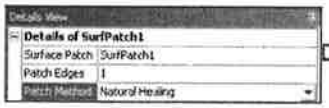
Creating 3D geometry model

ANSYS DesignModeler

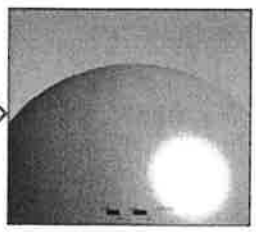
Surface Patch Tool : Example



Model with a hole

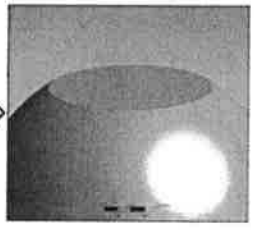


Natural Healing (maintains the curvature)





Patch Healing



Creating 3D geometry model

ANSYS DesignModeler

Merge Tool

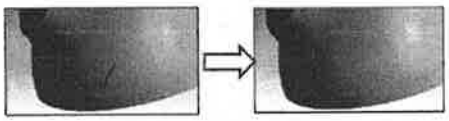
- Merges a set of edges or faces
- Useful to reduce model complexity
- Type
  - Edge merging
  - Face merging
- Method
  - Manual: user selects the entities to merge
  - Automatic: Clusters of entities to merge are automatically selected and highlighted.



Details of Merge2

Merge	Merge2
Merge Type	Edges
Selection Method	Manual
Edges	4
Minimum Angle [0, 180]	135 °
Edges Not Merged	
Edges	4

Details View of Merge



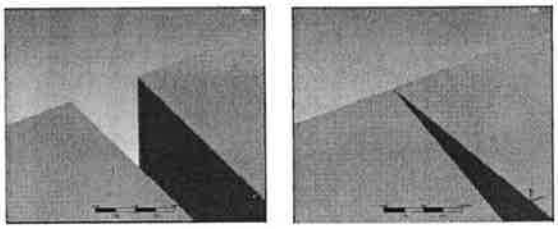
Edge Merging

Creating 3D geometry model

ANSYS DesignModeler

Connect Tool

- Aligns and possibly joins a set of vertices, edges or faces
- Properties
  - Location property
  - T-junction property



Vertex Connect



Details of Connect2

Connect	Connect2
Connection Type	Vertices
Vertices	2
Tolerance	0.1 in
Location	Interpolated
Merge Bodies	Yes

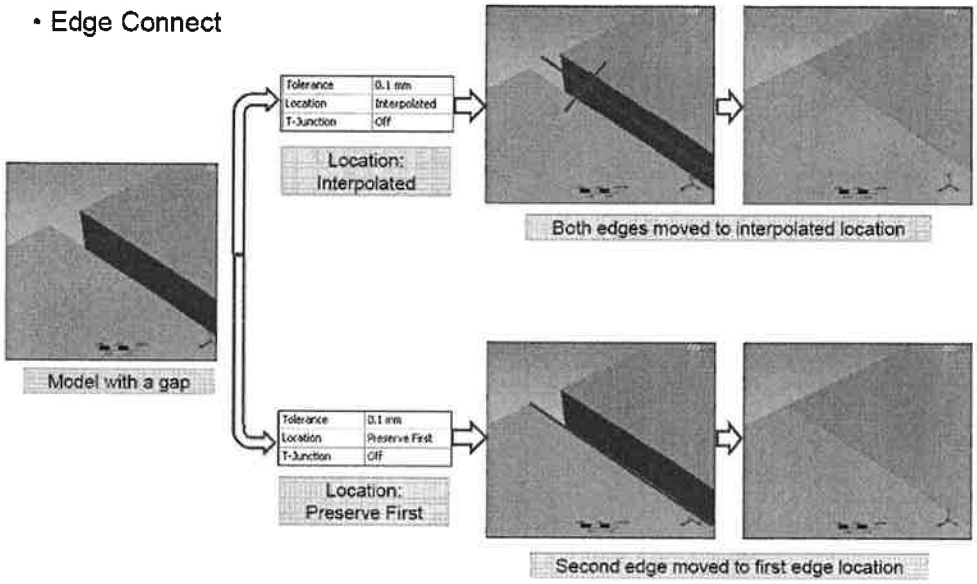
Details View of Connect

Creating 3D geometry model

ANSYS DesignModeler

Connect Tool : Example

• Edge Connect

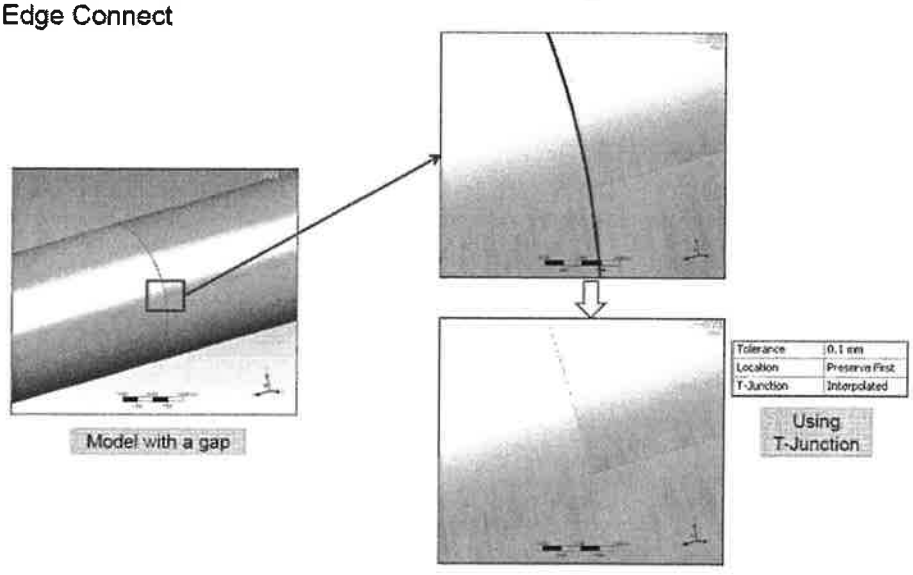


Creating 3D geometry model

ANSYS DesignModeler

Connect Tool : Example

• Edge Connect

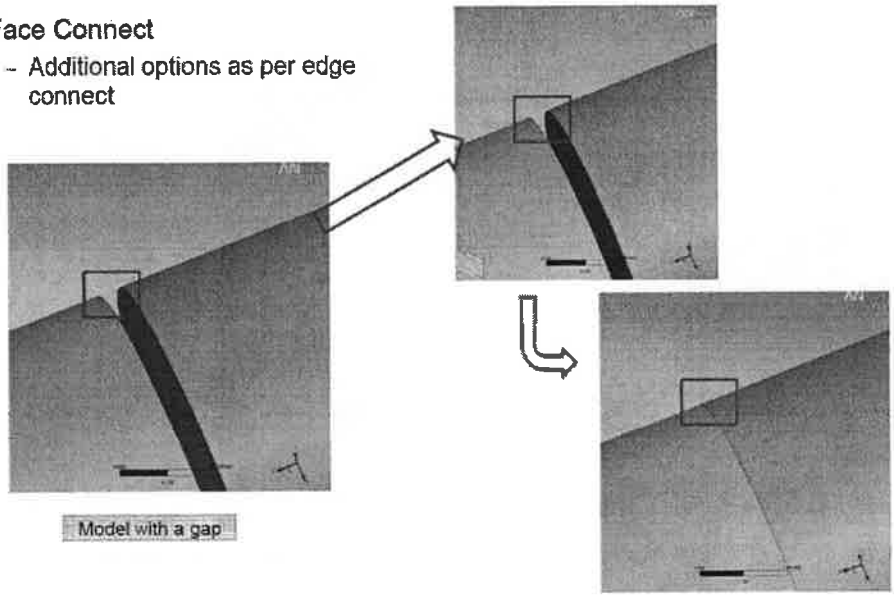


Creating 3D geometry model

ANSYS DesignModeler

Connect Tool : Example

- Face Connect
  - Additional options as per edge connect



Model with a gap

Creating 3D geometry model

ANSYS DesignModeler

Projection Tool

- Allows projection of points on edges/faces and edges on faces/bodies
- Options
  - Points on edge or face
  - Edge on face or body
- Works on both frozen as well as active bodies



Details of Projection1	
Projection	Projection1
Type	Edges On Face
Edges	3
Target	1
Direction Vector	None (Closest Direction)
Imprint	Yes
Extend Edges	No

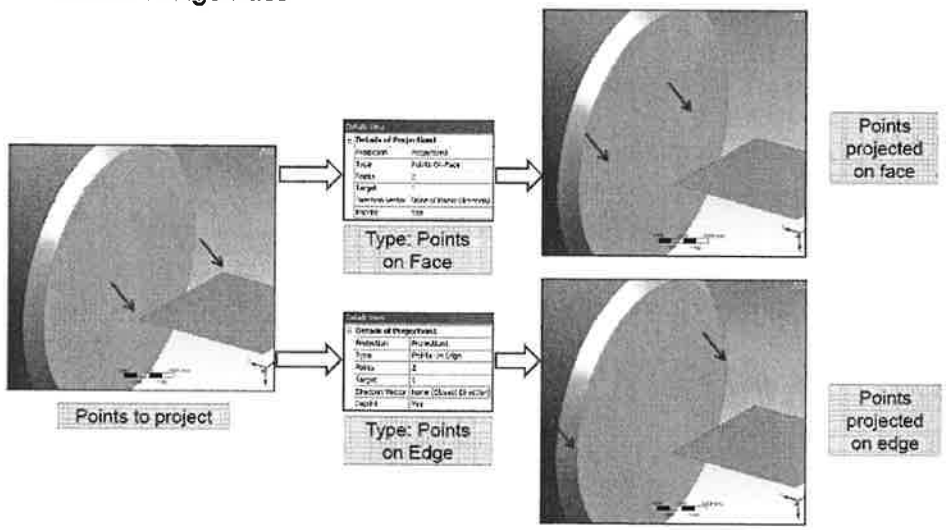
Details View of Projection

Creating 3D geometry model

ANSYS DesignModeler

Projection Tool - Example

- Points on Edge/Face

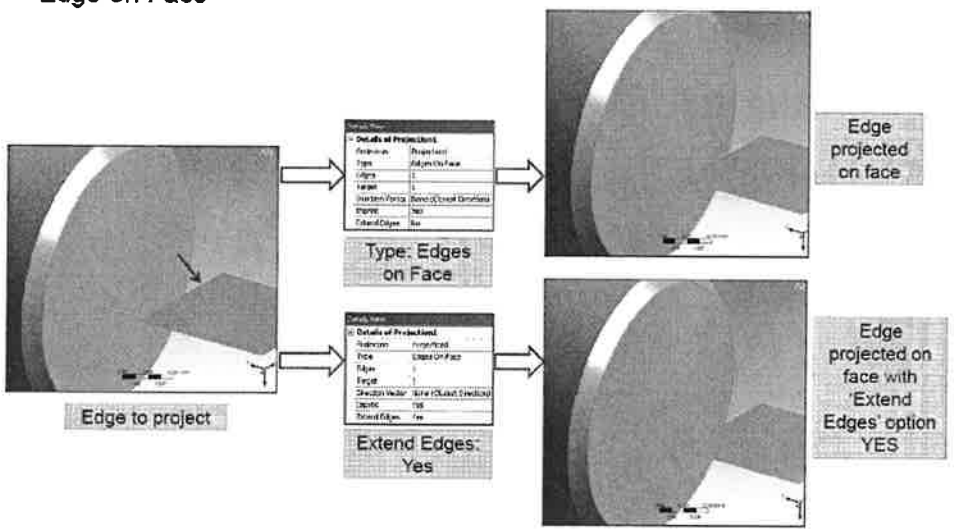


Creating 3D geometry model

ANSYS DesignModeler

Projection Tool - Example

- Edge on Face



Creating 3D geometry model

ANSYS DesignModeler

Geometry Clean Up & Repairs - Summary

- **CAD import/connection can produce**
  - **Corrupt and/or incomplete geometry**
  - **Excessive detail for CAE purposes**

- **Lecture has covered**
  - **Geometry analysis tools**
  - **Clean-up tools and shown examples on where to apply them**

- **You will now do Workshop ?title?**

Meshing computational model

- Avoiding highly skewed geometry
- Ensuring good quality mesh
- Mesh densities
- Different meshing topology

Meshing computational model

Meshing

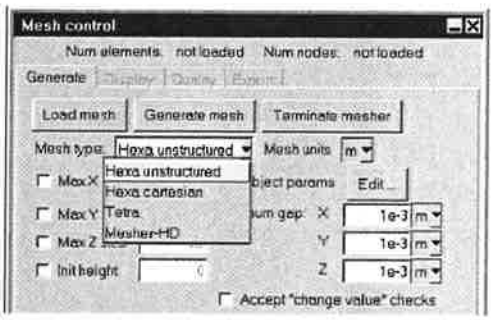
- Meshing is *the* most critical aspect of CFD modeling:
  - A good mesh is absolutely essential for a good solution
- Meshing in Airpak is fully automated and can generate meshes at varying levels of complexity
- Choice of four methods available for meshing
- Meshing parameters can be defined
  - On a global meshing basis
  - On an object by object basis
- After meshing, the mesh is loaded and can be examined for quality using a number of quality measures



Meshing computational model

Airpak Meshing Methods

- Choice of three meshers available:
  - Two Hexahedral meshers
    - Unstructured - It can be used for most applications satisfactorily
    - Cartesian
  - Tetrahedral mesher
    - Intended for some complex geometry
    - Must be used for ellipsoids, elliptical cylinders, or polygonal ducting
  - Mesher-Hex Dominant: Mixed Meshing
  - Hexa, tetra and cartesian mesh can all be mixed

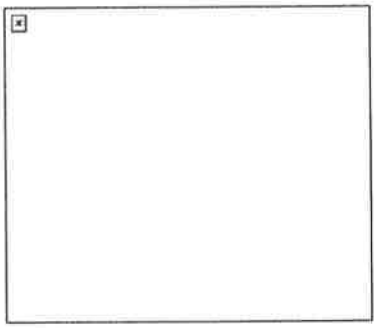


Meshing computational model

Hexa-unstructured Meshing Methodology

Airpak operates on a **"cocooning"** methodology

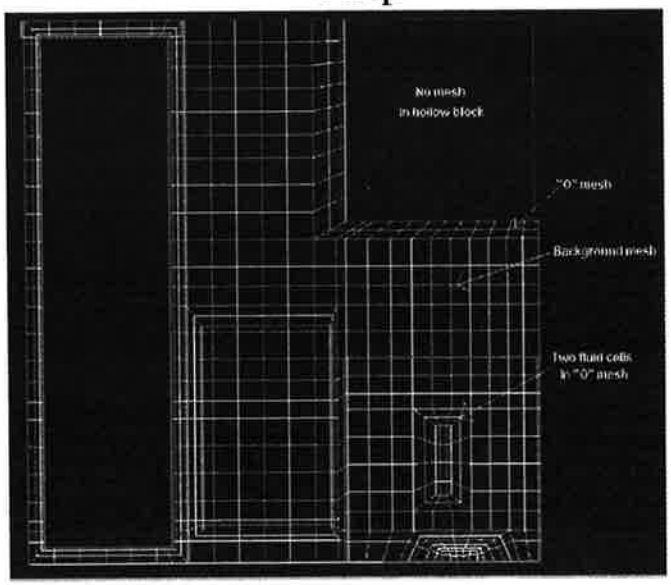
- Each object is meshed individually as close to the specifications defined as possible
- First combines adjacent objects together
  - Creates single group of multiple objects to "O" grid together
- Airpak then creates the background Cartesian mesh
  - Max Cell size is controlled by Max x, Max y and Max z values
- Next, Airpak cuts out a section of the mesh around the grouped objects and replaces the mesh with an unstructured mesh - the "O"-Mesh region
  - This allows for a finer mesh to be generated close to objects
  - In open spaces between objects the mesh is tessellated with larger elements - Minimizes computational costs



Airpak

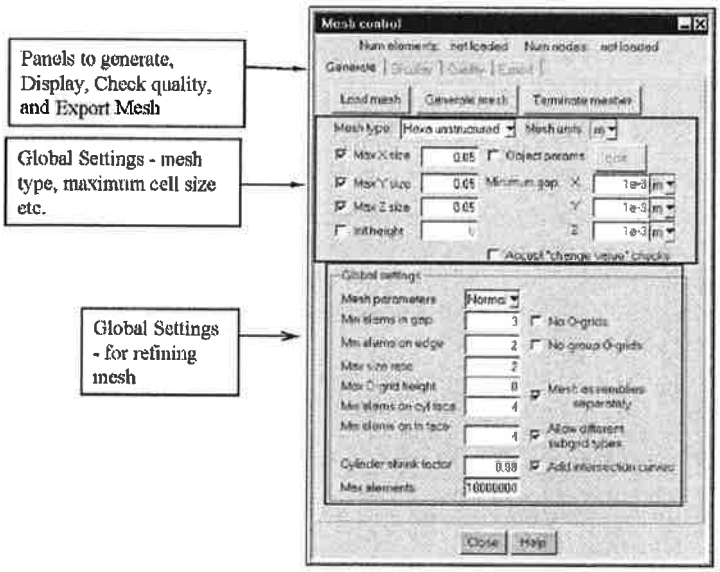
Meshing computational model

Elements of an Airpak mesh



Meshing computational model

Hexahedral Meshing: Generation Panel



Meshing computational model

Global Settings

- Mesh Type
  - Hexa-Unstructured
  - Hexa-Cartesian
  - Tetrahedral
- Mesh Units
  - Apply to all mesh control parameters with length unit
  - O-grid height is an exception (always in meter)
- Object Parameters
  - Mesh control parameters on individual object
- Minimum gap (Separation)
  - Tolerance used to remove small gaps/misalignments during meshing
  - Independent specification of minimum gap sizes in each direction (X,Y,Z)
  - **Accept "change value" check:** Toggle on in order to avoid any minimum gap related warnings in advance

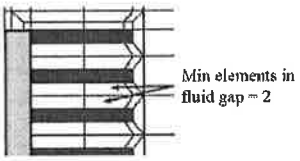


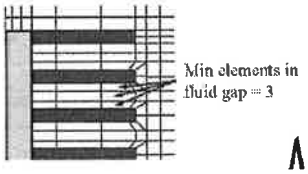
Meshing computational model

Global Settings

- Mesh default settings
  - Normal mesh - Provides an acceptable mesh for most situations provided background grid is reasonable
  - Coarse mesh - Provides a "first cut" coarse mesh that can be refined locally

- Mesh control parameters
  - Max X, Max Y, Max Z - Control the largest cell size in X, Y and Z direction
  - Init height - Maximum height of first cell away from surface of all objects within the model
  - Min elements in fluid gap - Minimum number of cells in between adjacent faces of closely spaced objects

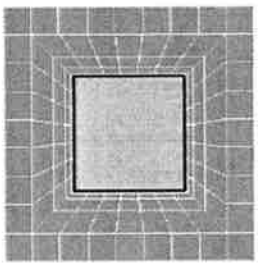




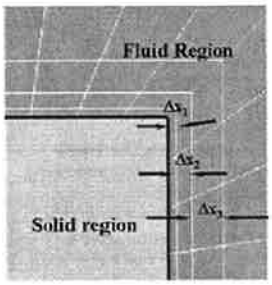
Meshing computational model

Global Settings

- Min elements on solid edge - Minimum number of cells to be placed on the edge of an object
- Max O-grid height - Maximum height of the O-grid region from the surface of an object
- Maximum Size Ratio - Controls the growth rate of cell size



O-grid height

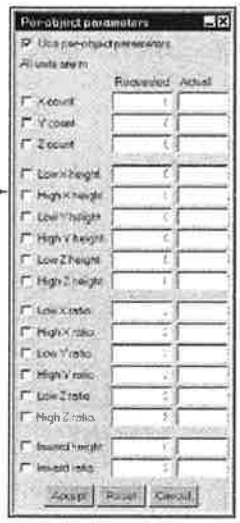


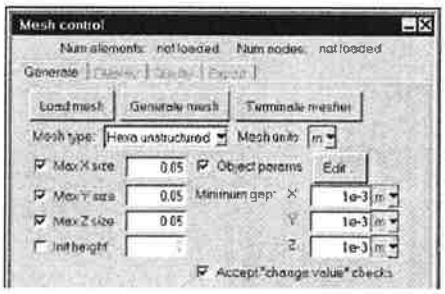
Size ratio $\Delta x_{i+1}/\Delta x_i$

Meshing computational model

Using Per-Object Parameters

- User can reach per-object parameters for an individual object using either of the two methods.
  - Use Global Mesh Control window and Toggle on Object Parameters, then select the object in per-object meshing parameters window
  - Select the object in model tree and click on right mouse and use Edit Mesh Parameters option to open up Per-Object Parameters window



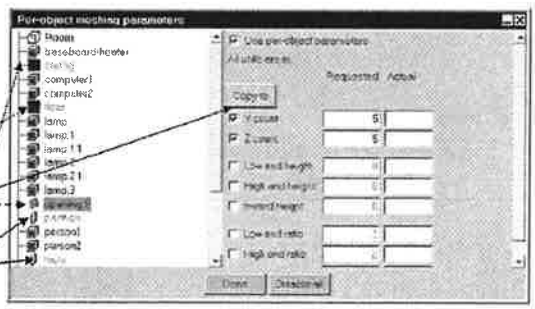


Airr

Meshing computational model

Copying Per-Object Parameters

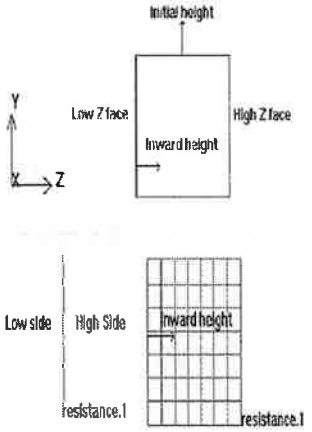
- Settings maybe copied to like objects (e.g. Prism to prism) using **Copy to:**
  - For example: settings for high/low height on an object can be copied to other objects
  - Method:
    - Set the parameters on one object
    - Click on the **Copy to** button
    - Airpak highlights the objects you can copy these parameters to
    - Click on each object you want to inherit the mesh parameters
    - Once finished click on **Copy to** for a second time to end the operation.



Meshing computational model

Per-Object Parameters - Rectangular

- Rectangular Prism objects
  - X,Y,Z count
    - Number of cells across in each co-ordinate direction
  - Low/High X,Y,Z height
    - Height of first cell away from and object
  - Low/High X,Y,Z ratio
    - Growth of aspect ratio away from an object
  - Inward height/ratio
    - Height/growth of first cell into an object
- Rectangular planar objects
  - X&Y, Y&Z, X&Z count
    - Cell count in each co-ordinate direction
  - Low/High end height
    - Height of first cell either side of planar object
  - Low/High end ratio
    - growth rate either side of object
  - Inward height/ratio
    - Height/growth of cells into the object

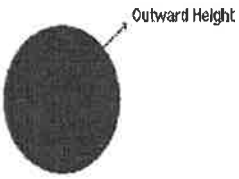


Ai

Meshing computational model

Per-Object Parameters - Cylindrical/Circular

- Cylindrical objects
  - Additional settings are:
    - Side count
      - ▲ Number of elements along the curved surface of the cylinder
    - Side height/ratio
      - ▲ Height/expansion ratio of first cell into the solution domain.



- Circular planar objects
  - Additional settings are:
    - Diameter count
      - ▲ Number of cells across the diameter
    - Outward height/ratio
      - ▲ Height and aspect ratio control away from the object

Airpak

Meshing computational model

Per-object Parameters: Other Shapes

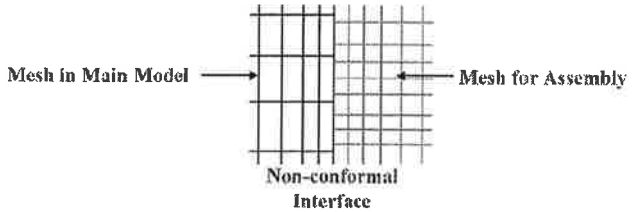
- Polygonal shapes
  - Analogous to prism blocks/plates
  - 3 controls:
    - side counts – number of cells in that direction. side "n" = from vertex "n" to vertex "n+1"
    - Side heights – first cell height outside the specific side
    - Side ratio – growth ratio on the specific side

- Object specific mesh control parameters
- Controls are intuitive:
  - Counts – number of elements on specified component of the object
  - Heights – first cell height outside the concerned component of the object
  - Ratio – growth ratio on the component of the object

Meshing computational model

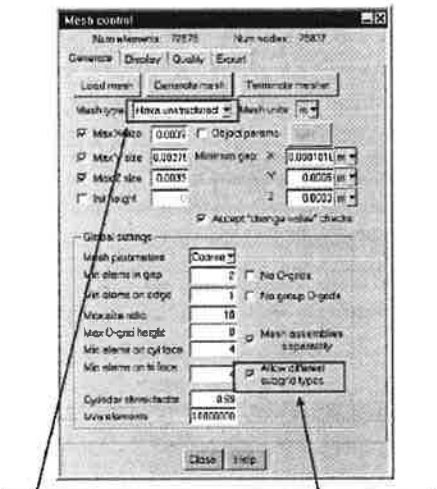
Non-Conformal Mesh

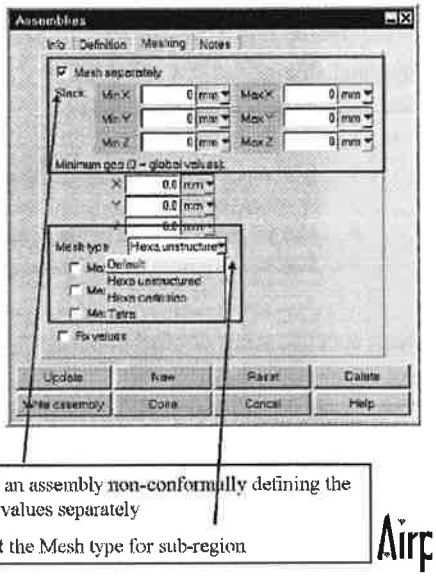
- A **non-conformal mesh** is one where there is one or more regions which have a different mesh refinement
- At the interface with the main model, there is a many-one transition between the mesh inside the region and the mesh outside it
  - In effect, we are embedding a fine mesh within the sub-region
- A non-conformal mesh is created by meshing assemblies of objects, so that there is a fine mesh where it is needed and a coarse mesh in the rest of the model
- Both the Hexa-Unstructured and Hexa-Cartesian methods can be used to mesh assemblies separately to create non-conformal meshes



Meshing computational model

Creating Non-Conformal Mesh





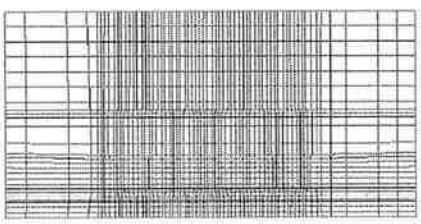
Airf

Meshing computational model

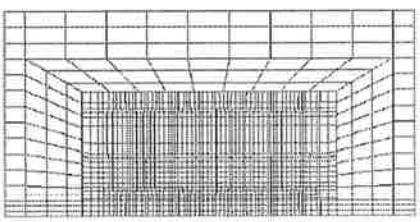
Non-Conformal Mesh

- If assemblies are not meshed separately, then the mesh is a **Conformal Mesh**
  - Throughout the model there is a one-one cell transition.
- Conformal meshes are usually considerably larger than meshes that make sensible use of non-conformal meshing

Conformal Mesh - 111,329 cells



Non-Conformal Mesh - 77,149 cells

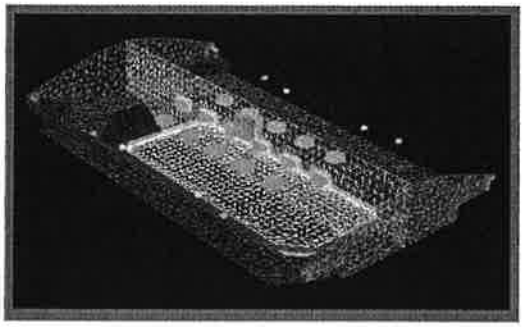


Airf

Meshing computational model

Tetrahedral Mesher

- Tetrahedral mesher should be used for meshing complicated geometries when the hexa mesher fails or does not produce an acceptable mesh
- When many objects are involved that have small dimensions (such as thin partitions), the tetrahedral mesher may lead to very large meshes



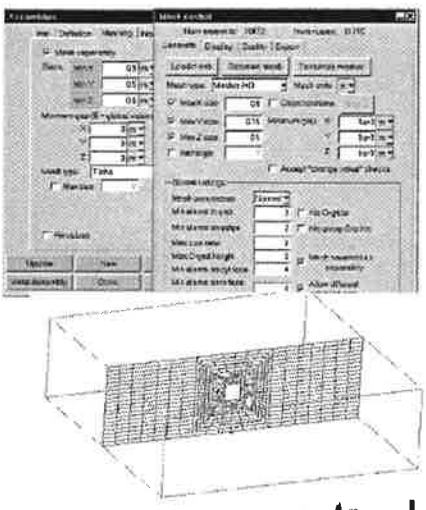
- **Global mesher controls:**
  - Max tetra size:
    - identifies the maximum size of the tetrahedral element to be created
    - maximum length of the sides of the element
  - Max Expansion ratio:
    - Controls rate at which cell size increases
  - Smooth mesh:
    - activates a smoothing algorithm to improve mesh quality
  - Coarsen mesh:
    - elements are combined to form a larger element where possible
  - Generally have the smooth and coarsen options active.
  - Mesh Assemblies Separately:
    - Create non-conformal mesh

Airpal

Meshing computational model

Hex-Dominant Meshing

- The hex-dominant meshing algorithm (Mesher-HD) allows the usage of multiple mesh types within the same model.
  - Hex-unstructured and Cartesian
  - Tetrahedral and hex combinations
- Hex-dominant meshing is used on assemblies to create different meshes in the assemblies and outside.
  - Eg: a Tetrahedral mesh inside the assembly and hex mesh outside or vice versa.
- Both conformal and non-conformal meshes can be generated.
  - In the main meshing panel, the "Mesh Assemblies separately" and "Allow different subgrid types" options have to be turned on to create a non-conformal mesh.
- The "Mesher-HD" algorithm will generate a hex-unstructured mesh where other mesh types have not been set.



Airpak

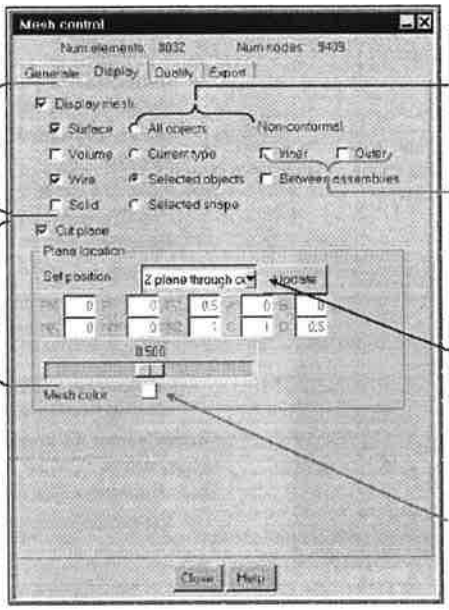
Meshing computational model

Displaying the Mesh

Display format:

surface/ volume/ solid/ cut plane options

Cut plane location/ mesh color



Limit Display of the mesh to current type, or particular object

View the inside and outside surface mesh on Non-Conformal Interface

Screen select options for cut plane

Change the colour of the mesh on the plane cut

Meshing computational model

Using Quality Diagnostics

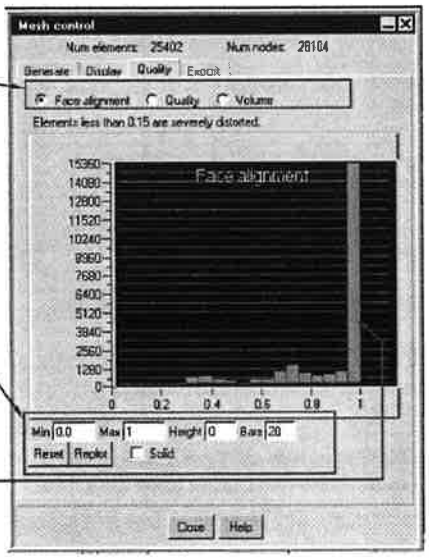
Choose the quality measure - Airpak calculates the quality of each cell, and reports min-max values in the messages window. Histogram is created for each quality measure

Change the Max value to limit the range Then click replot- e.g.: set Max to 0.15 to visualise cells with poor face alignment

Can also change the number of bars in the histogram

Displaying the cells as solid can aid in viewing

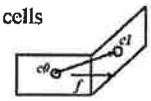
To see cell locations - click on the green histogram bar. Bar colour changes to pink and Airpak displays the cells for that range



Meshing computational model

Mesh Quality Is Important

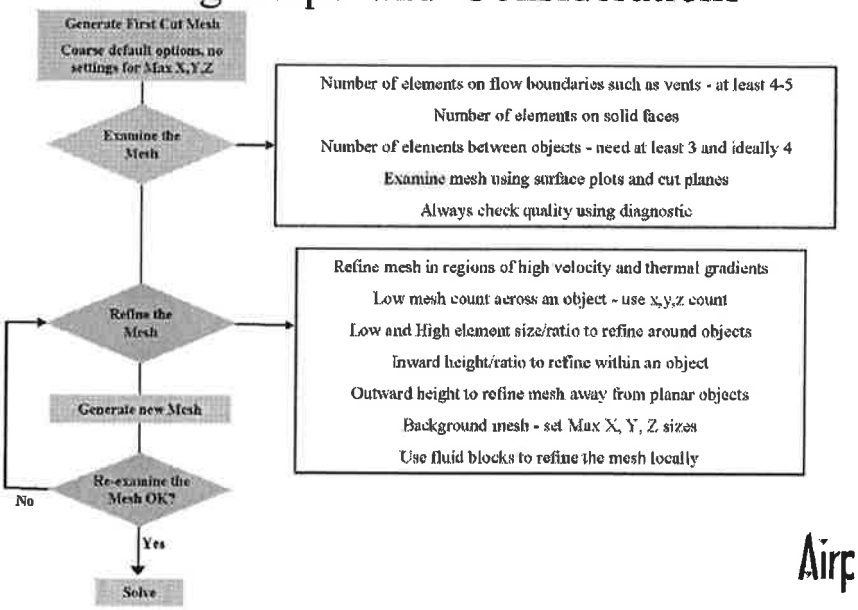
- **Poor quality meshes usually lead to trouble converging the solution and may lead to inaccuracies**
  - Mesh quality metrics
    - Aspect ratio
      - Ratio of shortest edge width to the longest edge length (0=bad, 1=good)
    - Face Alignment
      - Measures the deviation from 90 degree angled perfect cells
      - Range from 0 (bad) to 1 (good)
    - Quality metrics should be > 0.15
    - Volume - must be greater than 1e-12



  - Use mesh diagnostic tool
    - quantify quality of mesh
    - identify poor quality elements by clicking on green histogram bar

Meshing computational model

Meshing Steps and Considerations



Day 3 (1:15pm – 3:15pm)

- Applying appropriate boundary condition and physical assumption
  - Valid boundary condition
  - Reasonable physical assumption
- CFD computer number crunching
  - Quality control
  - Setting criteria for convergence
- Analyzing CFD result for design and improvement
  - Checking solution
  - Post-processing
  - Analysis and Interpretation
  - Documentation

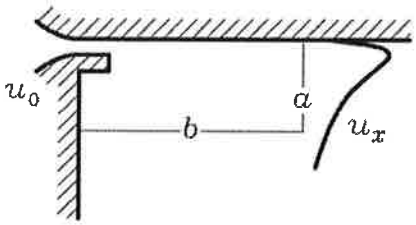
Applying appropriate boundary condition & physical assumption

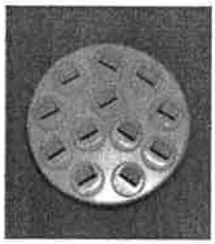
- Valid boundary condition
- Reasonable physical assumption
- Remember – any modeling results can only be as good as assumptions

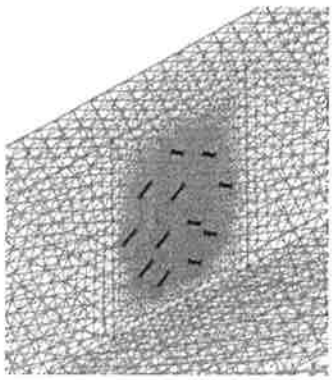
Boundary Conditions

- Wall boundary
- Free boundary
- Plane of symmetry
- Air supply opening
- Air exit opening
- Obstacle boundary

Air supply opening

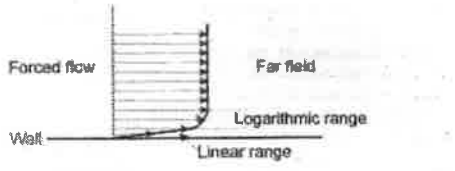


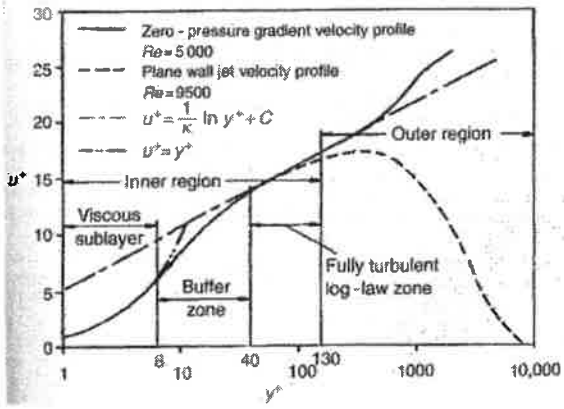




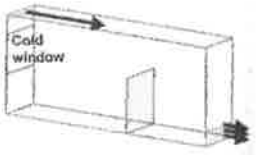
Boundary Conditions

- Wall boundary
  - Laminar sub-layer: $y^+ < 8$
  - Buffer zone: $8 < y^+ < 40$
  - Log-law fully turbulent zone (inner): $40 < y^+ < 130$
  - Outer turbulent zone: $y^+ > 130$



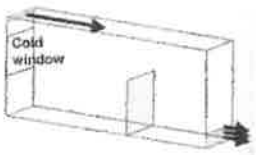


Boundary Conditions



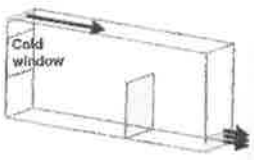
- Wall boundary
  - Momentum
    - No slip condition, $u_w = 0$
    - Velocity at first node point from wall, u_p, is calculated using "wall function"
  - Turbulence, κ-ε
    - $\kappa_w = 0$, κ_p is calculated from transport equation
    - ε_w is maximum, $\left(\partial\varepsilon/\partial y\right)_w = 0$ ε_p is calculated empirically
  - Low Re number turbulence model with extremely fine grid close to surface can adequately describe damping effect of the wall

Boundary Conditions



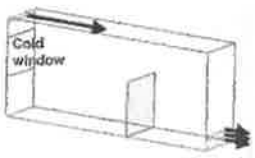
- Wall boundary
  - Heat transfer
    - Mainly by convection in the turbulent bl, can ignore diffusion
    - Prescribe temperature/heat flux
    - For radiation, T_w is determined by heat exchange between convection and radiation
    - T_w is used to calculate T_p, typical value of $y_p \sim 5$mm
  - Contaminant emission
    - Model pollutant sources, e.g. VOCs $S(t) = S_o e^{-Kt}$
    - To solve concentration within solid phase and in the air simultaneously
    - Adsorption and desorption of room surfaces

Boundary Conditions



- Free boundary
  - Boundary pressure are known, e.g. free stream or stagnant surrounding fluid
  - Applications include buoyancy driven flow, external flow around buildings, air jet diffusion to stagnant surroundings
  - Specified at long distance (~ 5 times characteristic length) from the flow regime of interest
- Plane of symmetry
  - Save time and computing power
  - No fluid and scalar quantities cross boundary

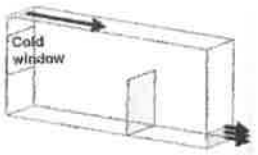
Boundary Conditions



- Air supply openings
  - Prescribe fluid velocity, temperature, concentration level and turbulence quantities
  - Turbulence intensity, I, is used to describe turbulence kinetic energy level $\kappa_o = 1.5 I^2 u_o^2$ 5~10% for NV model
  - Hydraulic length of opening, l_o, is used to describe turbulence dissipation rate $\varepsilon_o = C_\mu \kappa_o^{1.5} / l_o$
  - Simplification for flow from diffuser
    - Box Method – ignore flow details in the immediate vicinity of supply opening
    - Prescribed Velocity Method – decouple mass and momentum flow of diffuser from CFD and use the initial values as boundary condition

— for Coanda effect.

Boundary Conditions



- Air exit openings
  - Transverse velocity components set to zero
  - Longitudinal velocity is calculated from mass balance
  - Exit temperature and concentration are calculated from balance law for the whole flow field
  - Exit pressure is not required unless mechanical extract fan is used (with known static pressure)
  - Exit values for turbulence quantities are not required due to high Re number
- Obstacle boundaries
  - Special attention is required for wall function to use for obstacle boundaries

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**

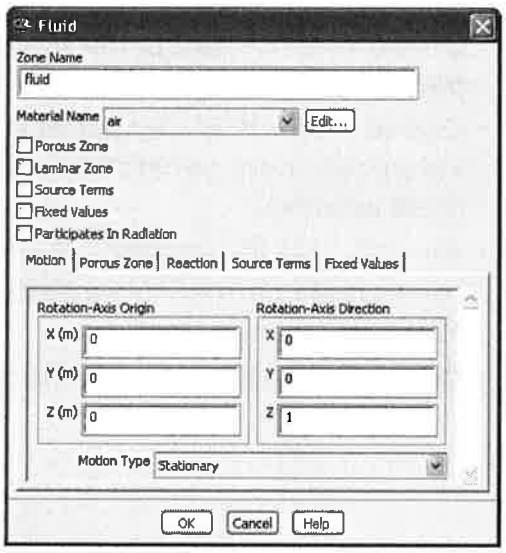
Boundary Conditions

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.**

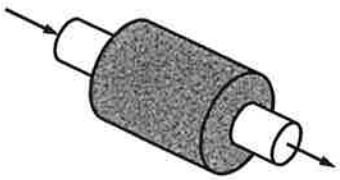
- **Optional inputs**
  - **Porous region**
  - **Source terms**
  - **Laminar region**
  - **Fixed Values**
  - **Radiation**



Boundary Conditions

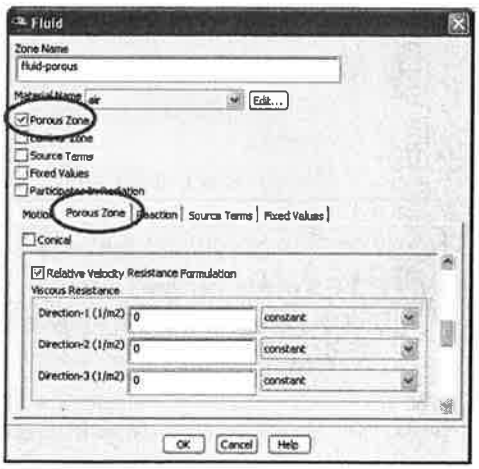
Porous Media

- **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**

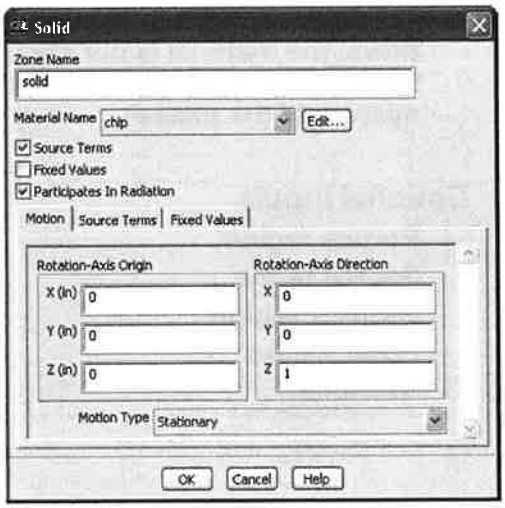
- **Inputs are directional viscous and inertial resistance coefficients.**



Boundary Conditions

Cell Zones – Solid

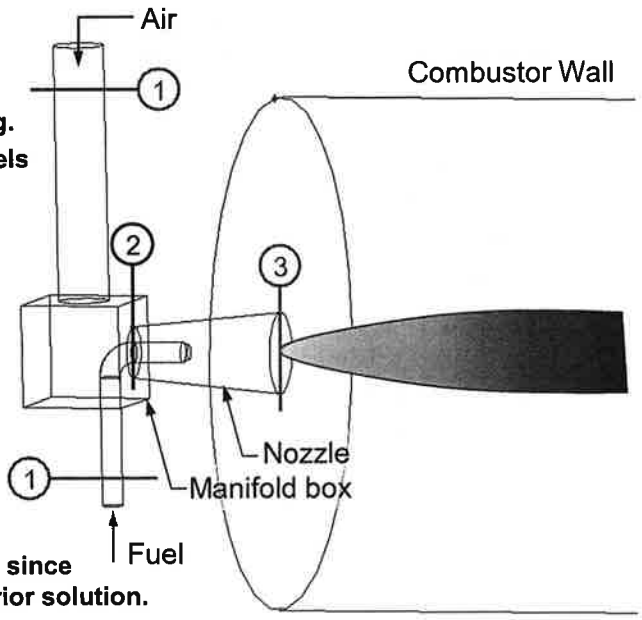
- A solid zone is a group of cells for which only heat conduction problem solved. Flow equations are not solved.
- Only required input is the material name (defined in the Materials panel).
- Optional inputs allow you to set volumetric heat generation rate (heat source).
- Need to specify rotation axis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



Boundary Conditions

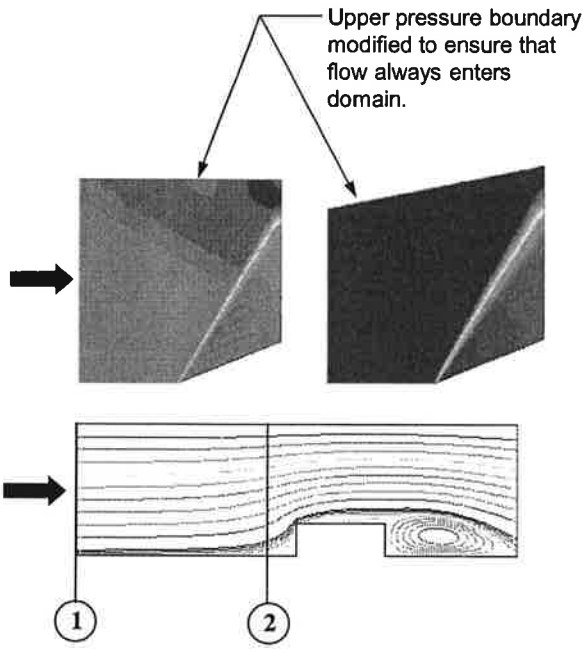
Locating Boundaries – An Example

- 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.



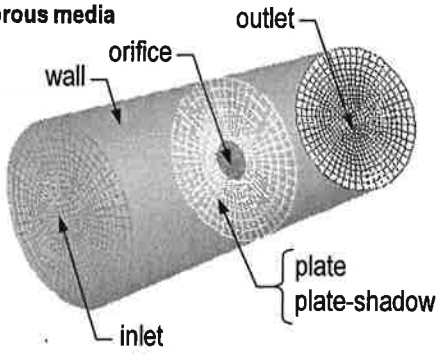
General Guidelines

- 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.



Available Boundary Condition Types

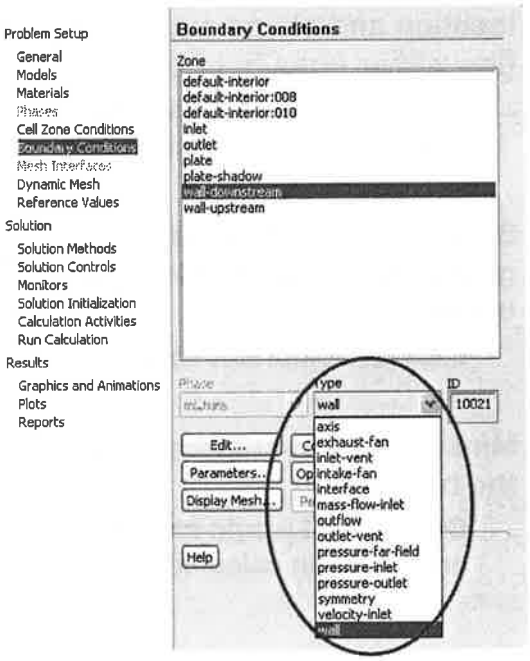
- External Boundaries
  - General
    - Pressure Inlet
    - Pressure Outlet
  - Incompressible
    - Velocity Inlet
    - Outflow (not recommended)
  - Compressible
    - Mass Flow Inlet
    - Pressure Far Field
  - Other
    - Wall
    - Symmetry
    - Axis
    - Periodic
  - Special
    - Inlet /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



Boundary Conditions

Changing Boundary Condition Types

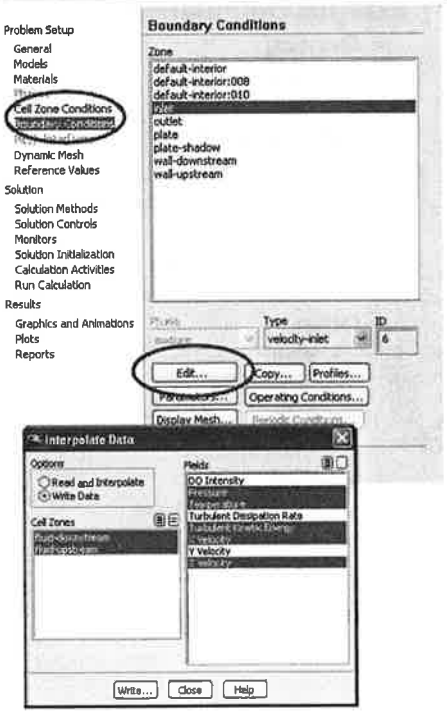
- **Zones and zone types are initially defined in the preprocessing phase.**
- **To change the boundary condition type for a zone:**
  - **Choose the zone name in the Zone list.**
  - **Select the type you wish to change it to in the Type pull-down list.**



Boundary Conditions

Setting Boundary Condition Data

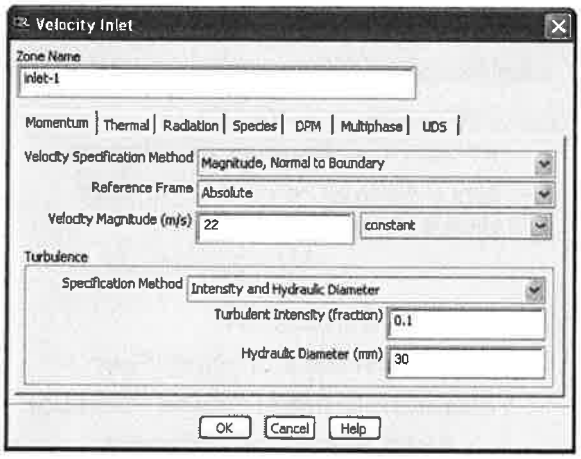
- **Explicitly assign data in BC panels.**
  - **To set boundary conditions for particular zone:**
    - **Select Boundary Conditions in the project tree.**
    - **Choose the boundary name in the Zone list.**
    - **Click the Edit... button.**
  - **Boundary condition data can be copied from one zone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



Boundary Conditions

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.**

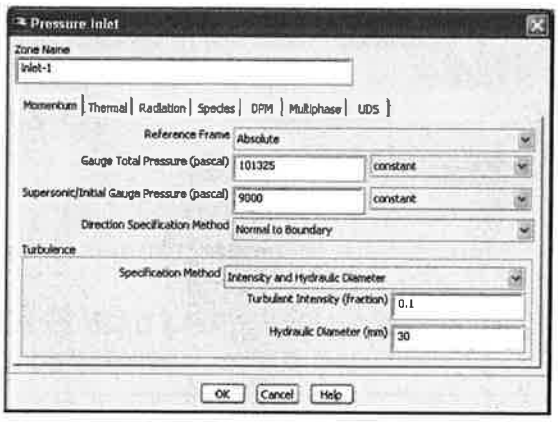
- **Velocity inlets are intended for use in incompressible flows and are not recommended for compressible flows.**

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

Boundary Conditions

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

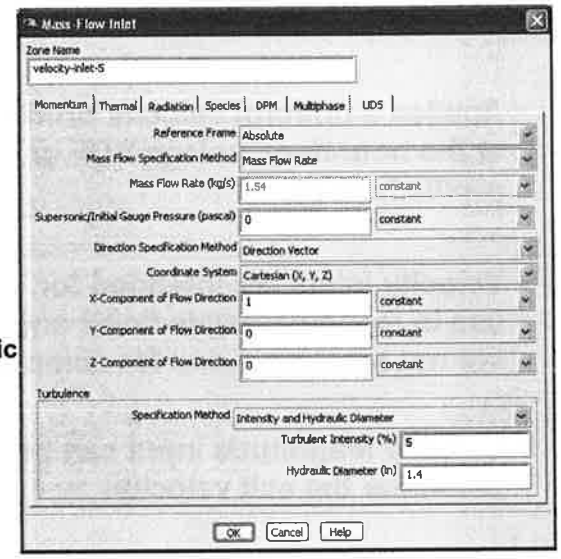
Compressible: $p_{total,abs} = p_{static,abs}\left(1 + \frac{k-1}{2}M^2\right)^{\frac{k}{k-1}}$

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

Boundary Conditions

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.

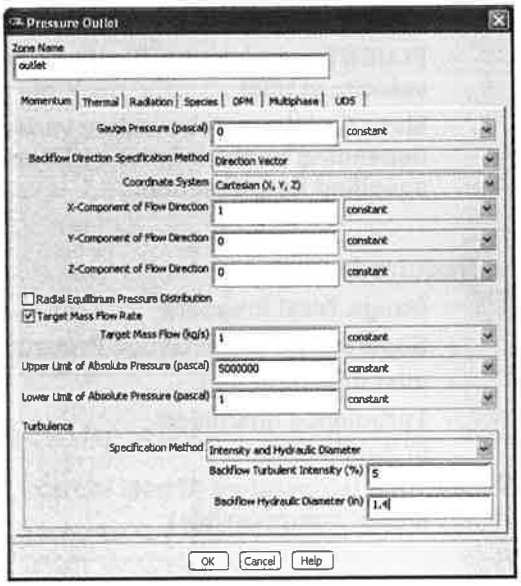


- Required information
  - Mass Flow Rate or Mass Flux
  - Supersonic/Initial Gauge Pressure
    - Static pressure where flow is locally supersonic; ignored if subsonic
    - Will be used if flow field is initialized from this boundary.
  - Total Temperature (on Thermal tab)
    - Used as static temperature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

Boundary Conditions

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.



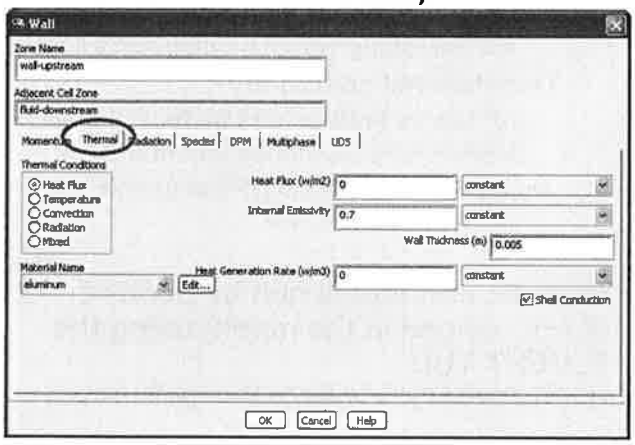
- Required information
  - Gauge Pressure (static) – static pressure of the environment into which the flow exits.
  - Backflow quantities – Used as inlet conditions if/when backflow occurs (outlet acts like an inlet).

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

Boundary Conditions

Wall Boundaries

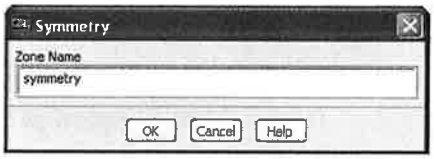
- In viscous flows, no-slip conditions are applied at walls.
  - Shear stress can be applied.
- Thermal boundary conditions
  - Several types of thermal BCs are available.
  - Wall material and thickness can be defined for 1D or shell conduction heat transfer calculations (details will be discussed in 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.

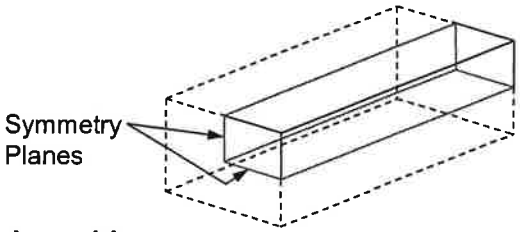


Boundary Conditions

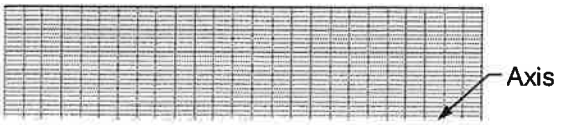
Symmetry and Axis Boundaries

- 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.





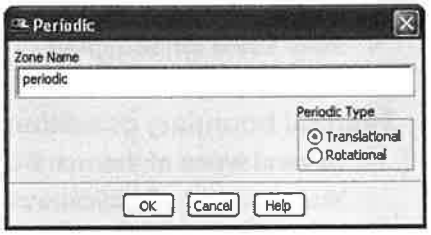
- Axis Boundary
  - Used at the center line for axisymmetric problems.
  - No user inputs required.
  - Must coincide with the positive x direction!

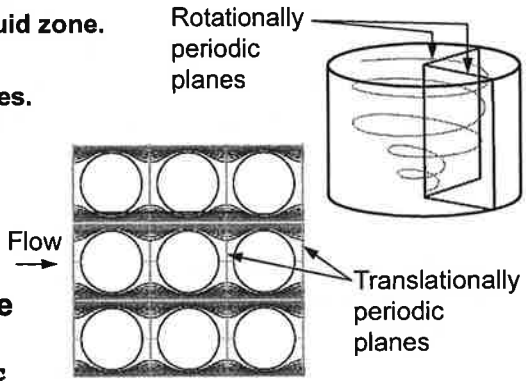


Boundary Conditions

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.**
- **Periodic conditions can be defined (if not defined in the mesh) using the FLUENT TUI:**
  `/mesh/modify-zones/make-periodic`





2D Tube Heat Exchanger

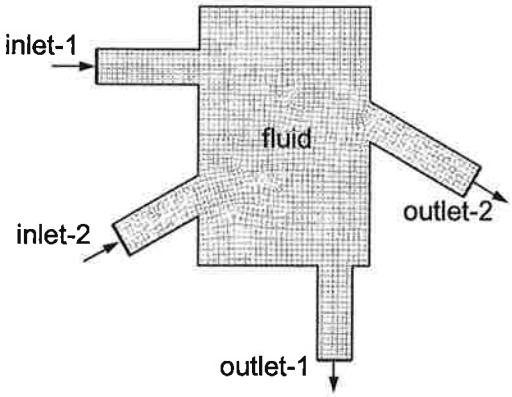
Boundary Conditions

Internal Face Boundaries

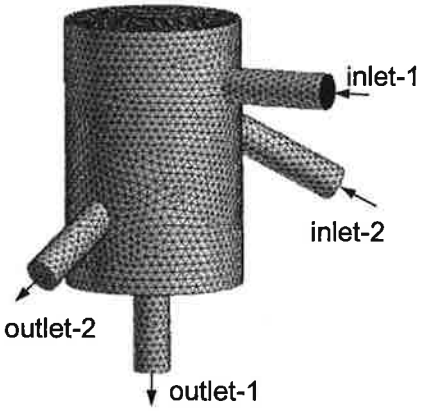
- **Defined on the cell faces only:**
  - **Thickness of these internal faces is zero**
  - **These internal faces provide means of introducing step changes in flow properties.**
- **Used to implement various physical models including:**
  - **Fans**
  - **Radiators**
  - **Porous-jump models**
    -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

Case Setup Replication

- **To replicate a case setup:**
  - **Use the read/write boundary conditions feature via TUI command:**
    `/file/write-bc` **Creates a BC file**
    `/file/read-bc` **Reads an existing BC file**
  - **You can transfer settings from a 2D case to a 3D case!**



2D Flow Domain (approximation)



Actual 3D Flow Domain

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 (see appendix for more information about these).**
  - **Pressure Far Field**
  - **Exhaust Fan / Outlet Vent**
  - **Inlet Vent / Intake Fan**
  - **Outflow**

Boundary Conditions

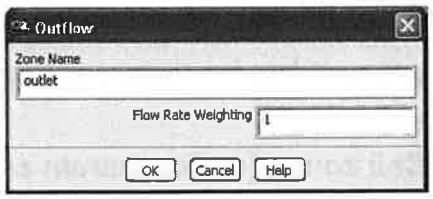
Other Inlet / Outlet Boundary Conditions

- Pressure Far Field
  - Used to model free-stream compressible flow at infinity, with prescribed static conditions and the free-stream Mach number.
  - Available only when density is calculated using the ideal gas law.
- Target Mass Flow Rate option for pressure outlets (not available for the multiphase models)
  - Provides the ability to fix the mass flow rate on a pressure outlet (either constant or via UDF hook)
  - Options to choose iteration method in TUI
- Exhaust Fan / Outlet Vent
  - Models an external exhaust fan or outlet vent with specified pressure rise / loss coefficient and ambient (discharge) pressure and temperature.
- Inlet Vent / Intake Fan
  - Models an inlet vent / external intake fan with specified loss coefficient / pressure rise, flow direction, and ambient (inlet) pressure and temperature.
- Inlet boundary conditions for large-eddy / detached-eddy simulations are covered in the Turbulence Modeling lecture.

Boundary Conditions

Outflow

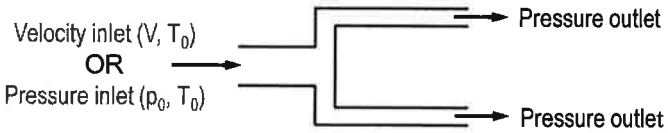
- No pressure or velocity information is required.
  - Data at exit plane is extrapolated from interior.
  - Mass balance correction is applied at boundary.



- Flow exiting outflow boundary exhibits zero normal diffusive flux for all flow variables.
  - Appropriate where the exit flow is fully developed.
- The outflow boundary is intended for use with incompressible flows.
  - Cannot be used with a pressure inlet boundary (must use velocity-inlet).
    - Combination does not uniquely set pressure gradient over whole domain.
  - Cannot be used for unsteady flows with variable density.
- Poor rate of convergence when backflow occurs during iterations.
  - Cannot be used if backflow is expected in the final solution.

Modeling Multiple Exits

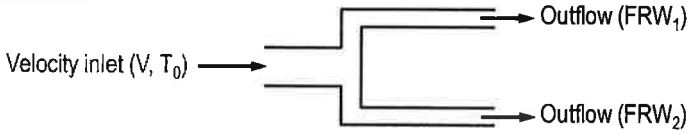
- **Flows with multiple exits can be modeled using pressure outlet or outflow boundaries, depending on the information you know.**
  - **Pressure outlets – requires knowledge of downstream pressures; FLUENT calculates the fraction of total flow through each branch.**



  - **Outflow:**
    - **Mass flow rate fraction determined from Flow Rate Weighting (FRW) by**

$$\frac{\dot{m}_i}{\dot{m}_{total}} = \frac{FRW_i}{\sum_i FRW_i}$$

    - **Static pressure varies among exits to accommodate the prescribed flow distribution.**



CFD computer number crunching

- Quality control
- Setting criteria for convergence

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

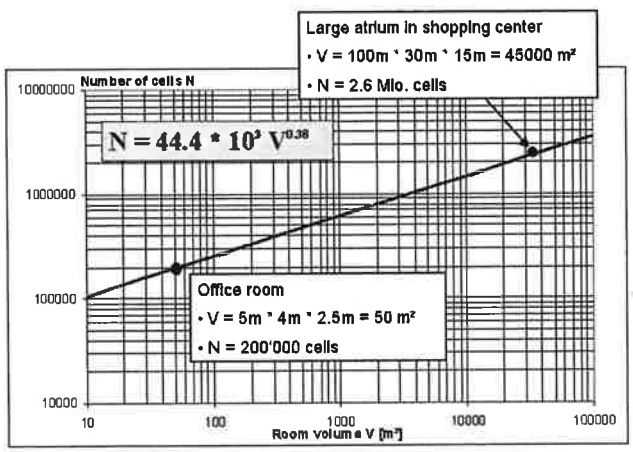
- Ensure quality

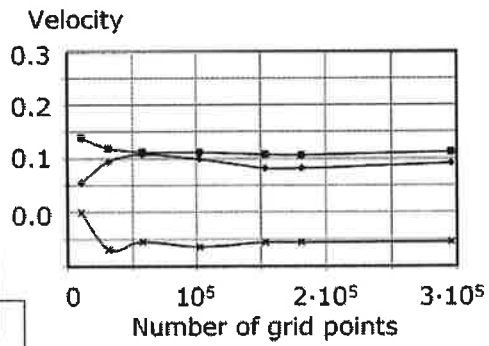
2. Computational Grid
   - Number of cells
     - Grid resolution usually cannot be too high
     - Typical cell numbers for two equation turbulence modeling with standard wall functions (German guideline VDI6019)

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

   - 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





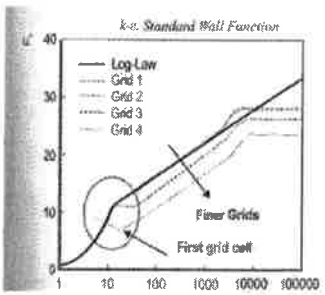
Monitoring velocities versus number of cells in the prediction

Quality Controls

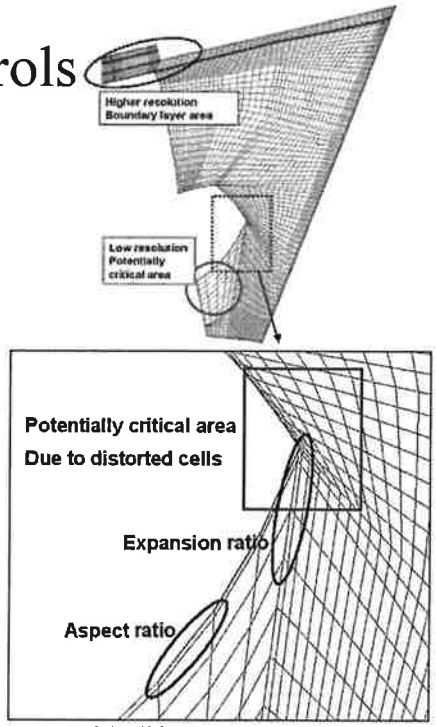
- Ensure quality
  2. Computational Grid
     - Cell quality
       - Distorted, squeezed or degenerated cells introduce numerical error
       - Criteria: angles, determinants, aspect ratio, expansion ratio etc
         - Angles between 20° and 160°
         - Expansion ratio < 2
         - Aspect ratio < 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

Solver Settings

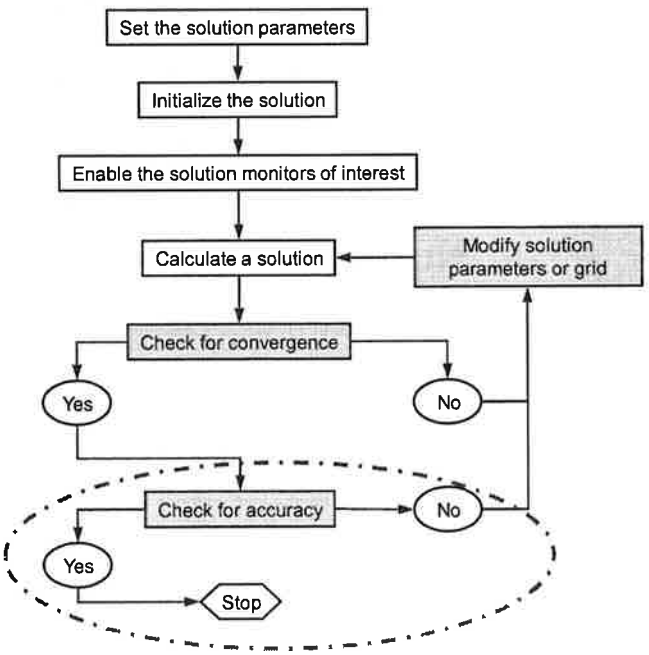
Outline

- **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**

Solver Settings

Solution Procedure Overview

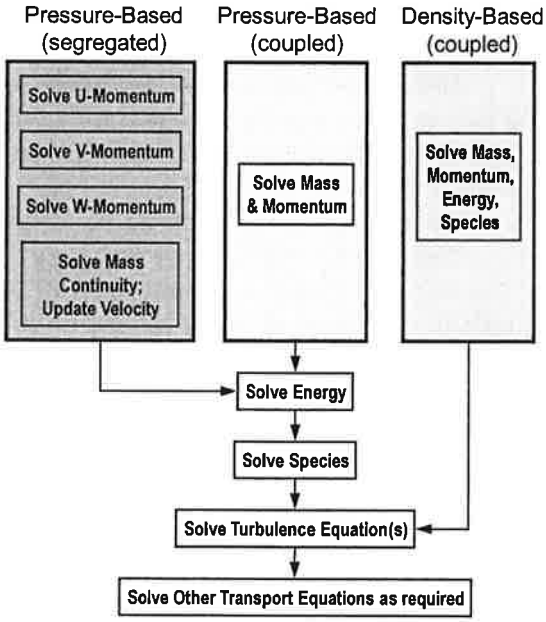
- **Solution parameters**
  - **Choosing the solver**
  - **Discretization schemes**
- **Initialization**
- **Convergence**
  - **Monitoring convergence**
  - **Stability**
    - **Setting Under-relaxation**
    - **Setting Courant number**
  - **Accelerating convergence**
- **Accuracy**
  - **Grid Independence**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



Solver Settings

Available Solvers

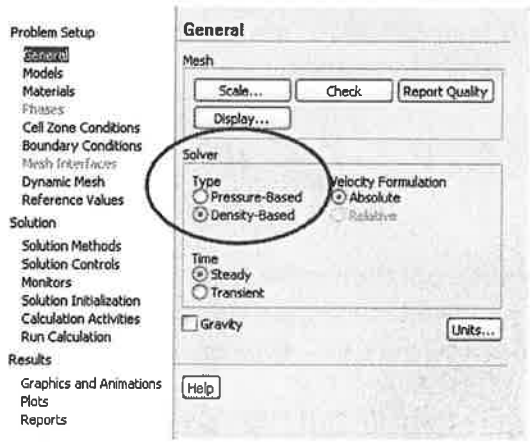
- There are two kinds of solvers available in FLUENT – Pressure based and Density based.
- The pressure-based solvers take momentum and pressure (or pressure correction) as the primary variables.
  - Pressure-velocity coupling algorithms are derived by reformatting the continuity equation
- Two algorithms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

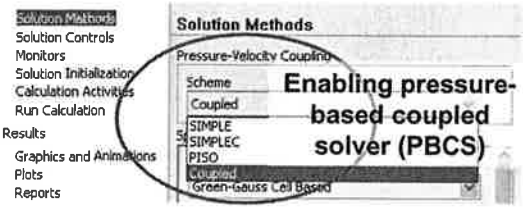


Solver Settings

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





Solver Settings

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).

Solver Settings

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

Solver Settings

Pressure-Velocity Coupling

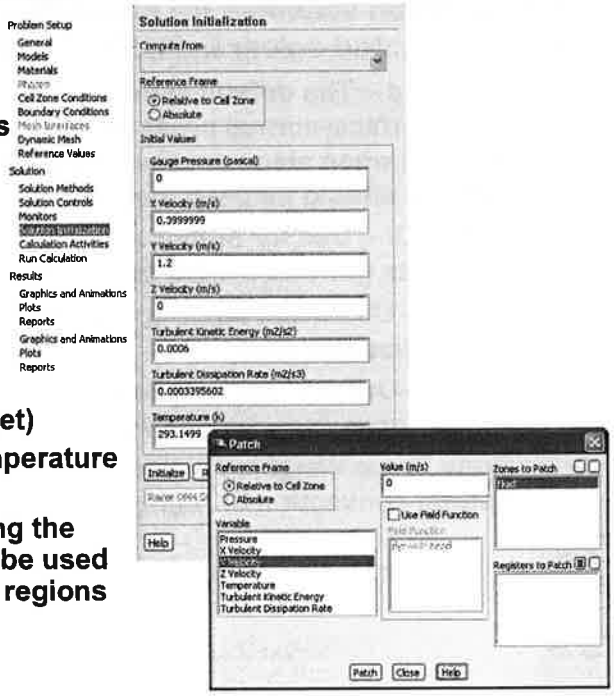
- **Pressure-velocity coupling refers to the numerical algorithm which uses a combination of continuity and momentum equations to derive an equation for pressure (or pressure correction) when using the pressure-based solver.**

- **Four algorithms are available in FLUENT.**
  - **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.**

Solver Settings

Initialization

- **FLUENT requires that all solution variables 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.**

Solver Settings

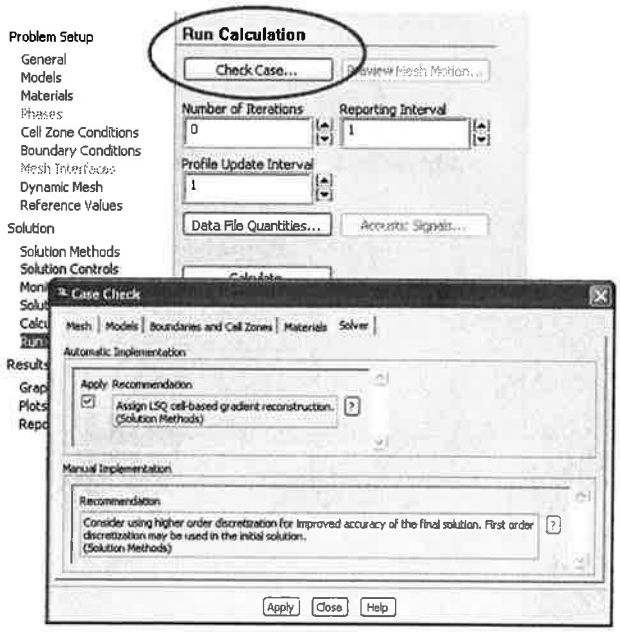
FMG Initialization

- Full MultiGrid (FMG) Initialization can be used to create a better initialization of the flow field.
  - FMG Initialization is useful for complex flow problems involving large pressure and velocity gradients on large meshes.
  - Euler equations are solved with first-order accuracy on the coarse-level meshes.
  - It can be used with both pressure and density based solvers, but only in steady mode.
- To enable FMG initialization:
  - Pressure-based solver: `/solve/init/fmg-initialization`
  - Density-based solver: Enabled in the GUI if using density-based solver.
- FMG uses the Full Approximation Storage (FAS) MultiGrid method to solve the flow problem on a sequence of coarser meshes, before transferring the solution onto the actual mesh.
  - Settings can be accessed by the TUI command `/solve/init/set-fmg-initialization`

Solver Settings

Case Check

- Case Check is a utility in FLUENT which searches for common setup errors and inconsistencies.
  - Provides guidance in selecting case parameters and models.
- Tabbed sections contain recommendations which the user can optionally apply or ignore.



Solver Settings

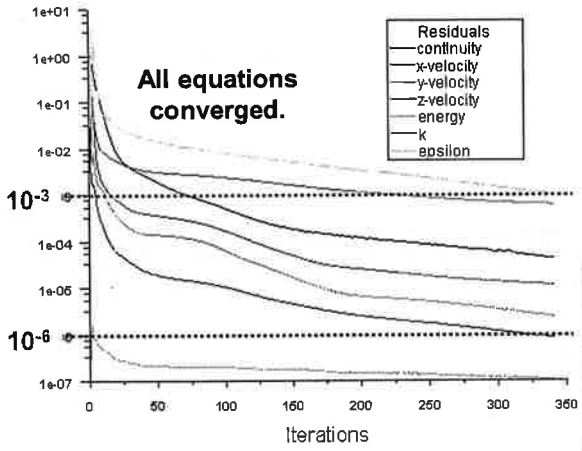
Convergence

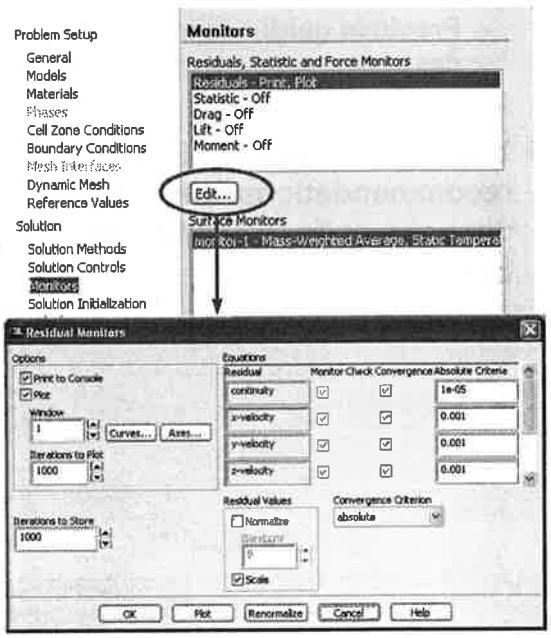
- At convergence, the following should be satisfied:
  - All discrete conservation equations (momentum, energy, etc.) are obeyed in all cells to a specified tolerance OR the solution no longer changes with subsequent iterations.
  - Overall mass, momentum, energy, and scalar balances are achieved.
- 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:
  - Monitor other relevant key variables/physical quantities for a confirmation.
  - Ensure that overall mass/heat/species conservation is satisfied.

Solver Settings

Convergence Monitors – Residuals

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



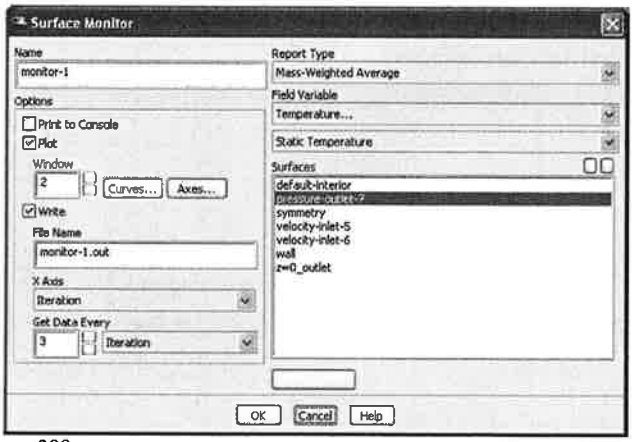


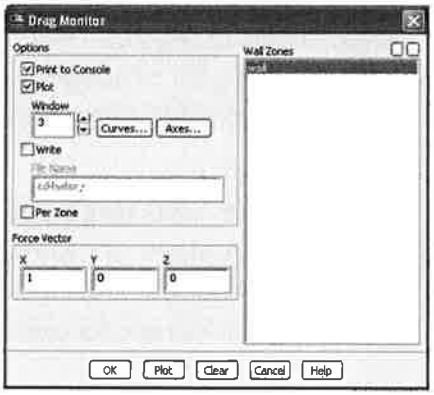
Solver Settings

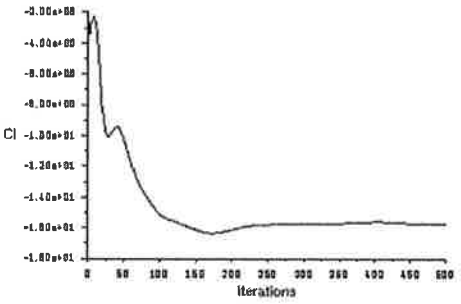
Convergence Monitors – Forces and Surfaces

- **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.**





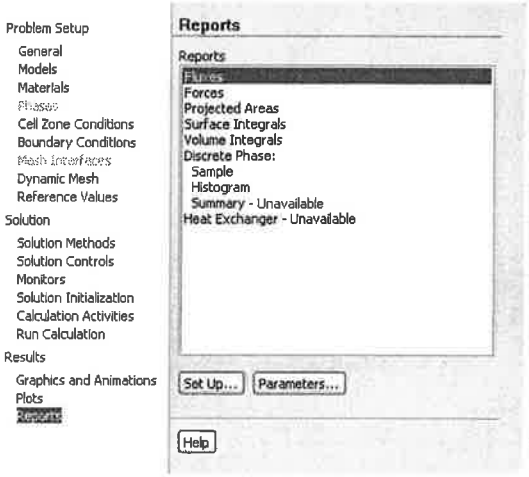


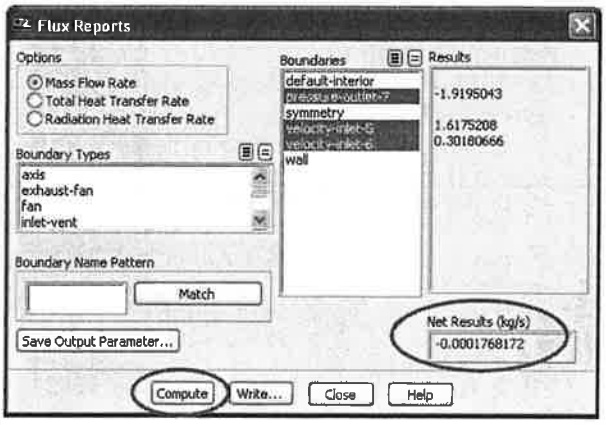
Solver Settings

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 (shown in the GUI as Net Results) should be less than 1% of the smallest flux through the domain boundary**





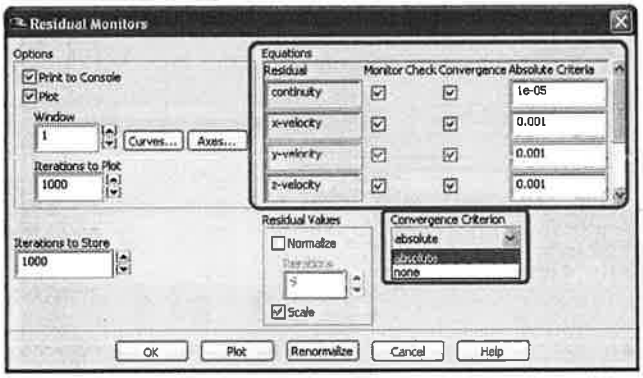
Solver Settings

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.**

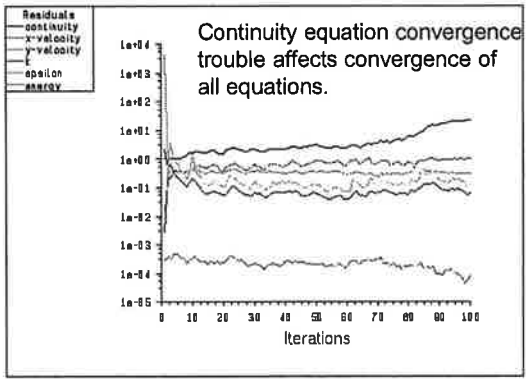


Solver Settings

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!**



Solver Settings

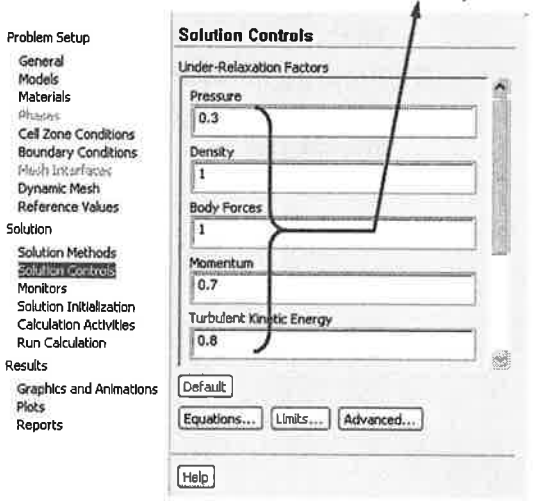
Modifying Under-Relaxation Factors

- 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.

- 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.

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

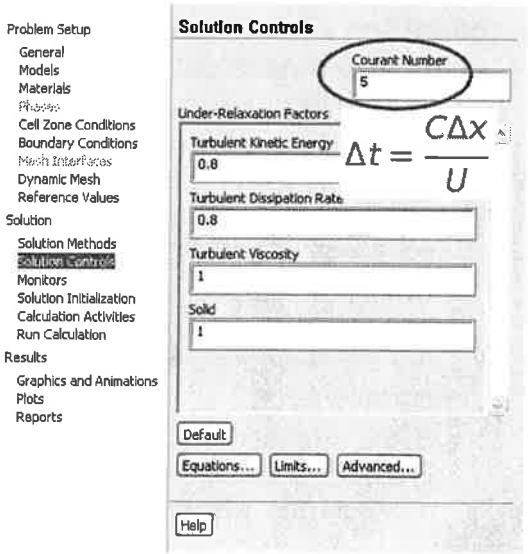


Solver Settings

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.

$$\Delta t = \frac{C \Delta x}{U}$$



Solver Settings

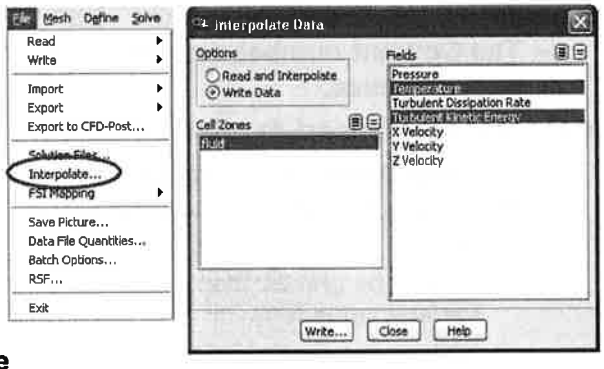
Accelerating Convergence

- **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.**

Solver Settings

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
      - FLUENT provides capability to 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

Solver Settings

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 in FLUENT.
       - 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. FLUENT offers 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.

- To use a different mesh on a single problem, use the TUI commands `file/write-bc` and `file/read-bc` to facilitate the setup of a new problem.
  - Better initialization can be obtained via interpolation from existing case/data by using solution data interpolation
- A web-based training module is available to train users in replication of case setup and solution data interpolation.

Solver Settings

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.

Solver Settings

Governing equation for compressible flow

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u}{\partial x} = 0, \tag{1a}$$

$$\frac{\partial \rho u}{\partial t} + \frac{\partial}{\partial x}(\rho u^2 + p) = 0, \tag{1b}$$

$$\frac{\partial \rho e_T}{\partial t} + \frac{\partial}{\partial x}(\rho e_T + p)u = \dot{Q} + \dot{W}q, \tag{1c}$$

$$\frac{\partial \rho Y}{\partial t} + \frac{\partial \rho Y u}{\partial x} = -\dot{W}. \tag{1d}$$

The equation of state and reaction rate are defined by

$$p = (\gamma - 1)\left(\rho e_T - \frac{1}{2}\rho u^2\right), \tag{2}$$

$$\dot{W} = B\rho Y \exp(-E/T), \tag{3}$$

Table 1. Nondimensional variable definitions (primes denote dimensional quantities).

Position	$x = x'/l'$
Time	$t = t'/t'_a$
Velocity	$u = u'/a'_o$
Pressure	$p = p'/\gamma p'_o$
Density	$\rho = \rho'/\rho'_o$
Temperature	$T = T'/T'_o$
Total Energy	$e_T = e'_T/a'^2_o$
Activation energy	$E = E'/R'T'_a$
Heat of reaction	$q = q'/a'^2_a$
External power deposition	$\dot{Q} = \dot{Q}'t'_a/\rho'_o a'^2_o$
Pre-exponential factor	$B = B't'_A$

401

Solver Settings

Non-Iterative Time Advancement (NITA)

- **What is NITA? What are its benefits?**

NITA stands for non-iterative time advancement. In many cases, the NITA scheme will

- significantly reduce the CPU time required for transient simulations, without sacrificing the overall time accuracy of the solution.
- Solution speedups of 4-5x are normally seen for cases where NITA is applicable.

- **How do I use NITA?**

There are two pressure-velocity coupling options available with NITA - **PISO** and the fractional step method (**FSM**).

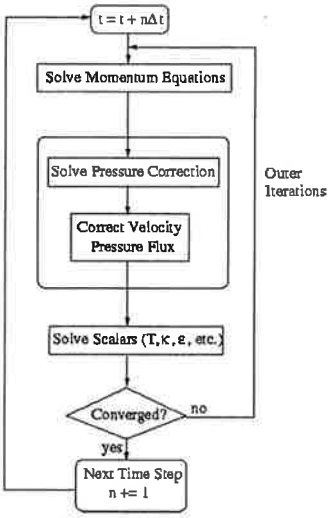
- **Are there any limitations when using NITA?**

Recommended for

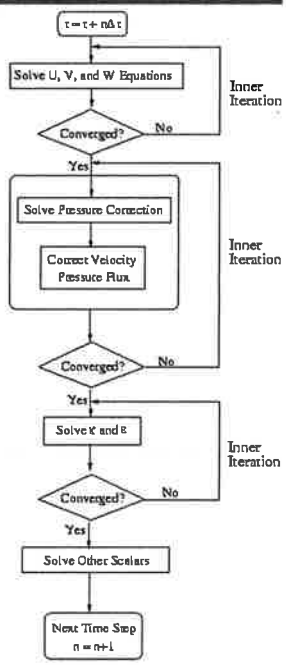
- incompressible and weakly compressible laminar or turbulent flows.

Not available with

- multiphase models (except VOF), radiation models, reacting species and all combustion models (including PDF), discrete phase model, IC engine spark and crevice models, phase change (solidification and melting), UDS transport, inviscid ideal gas, and porous media, porous jump, and fan models.
- In general, the NITA solver is not recommended for highly viscous fluid flow.



Overview of ITA Solution Method For the Segregate Solver



Overview of NITA Solution Method

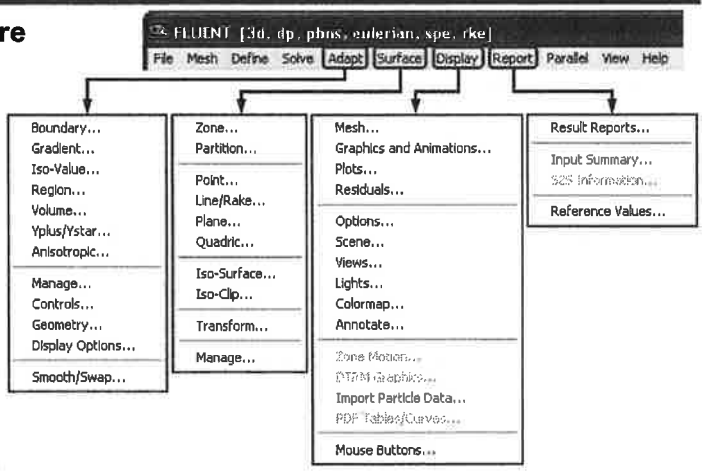
402

Overview

- There are two ways to postprocess CFD results from FLUENT
  - FLUENT postprocessing tools – legacy tools integrated in the FLUENT solver.
  - ANSYS CFD-Post application – a state-of-the-art postprocessor for ANSYS CFD products. CFD-Post can run as a standalone postprocessor, or within Workbench
- Both postprocessors include many tools for analyzing CFD results
  - Isosurfaces
  - Vector Plots
  - Contour plots (shaded and graded)
  - Streamlines and pathlines
  - XY plotting
  - Animation creation
- Contains the ability to postprocess any user-defined quantity either defined through custom field functions or user-defined memory (UDF).

Postprocessing in FLUENT

- Many post processing tools are available in FLUENT:
  - Surface creation
  - Display types
  - Rendering options
  - Plots of solution data
  - Flux reports and Integral calculations
- Post processing functions typically operate on surfaces, which can be automatically created by FLUENT from existing zones or by the user.



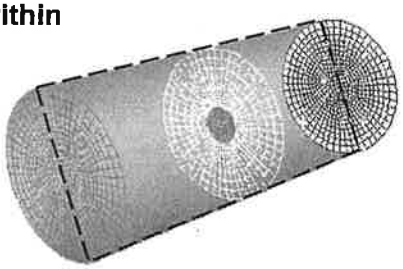
- For more information, please refer to the web-based lecture, "Post-Processing in FLUENT" on the FLUENT User Services Center `www.fluentusers.com`

Postprocessing

Surface Creation

- Fluent allows you to select portions of the CFD domain, called surfaces, to be used for visualizing and plotting the flow field.

- 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)



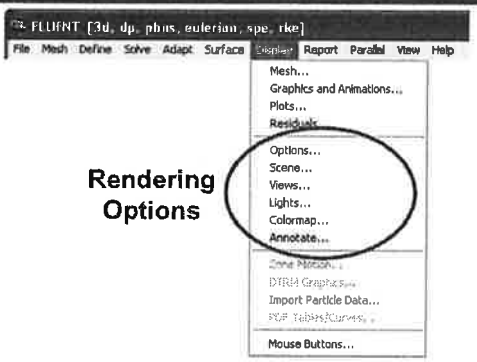
Iso-Surface of a constant grid coordinate can be created for viewing data within a plane.

- Surfaces can be renamed, deleted or moved and used to write profile files.

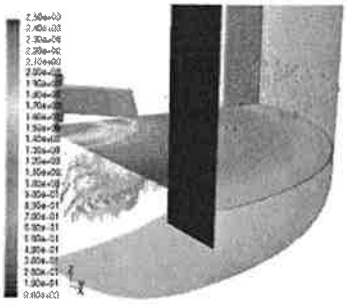
Postprocessing

Post-Processing Rendering Options

- The rendering options in FLUENT allow 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)



Rendering Options

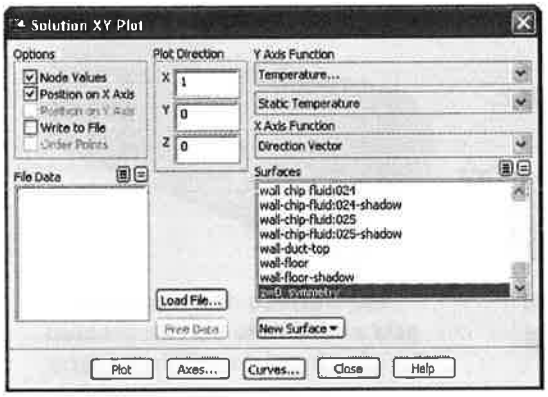


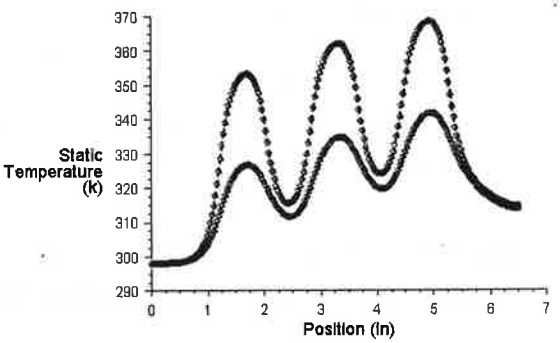
Example of scene composition: Overlay of contour and vector plot with transparent walls to show internal details.

Postprocessing

Plots

- FLUENT provides 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.

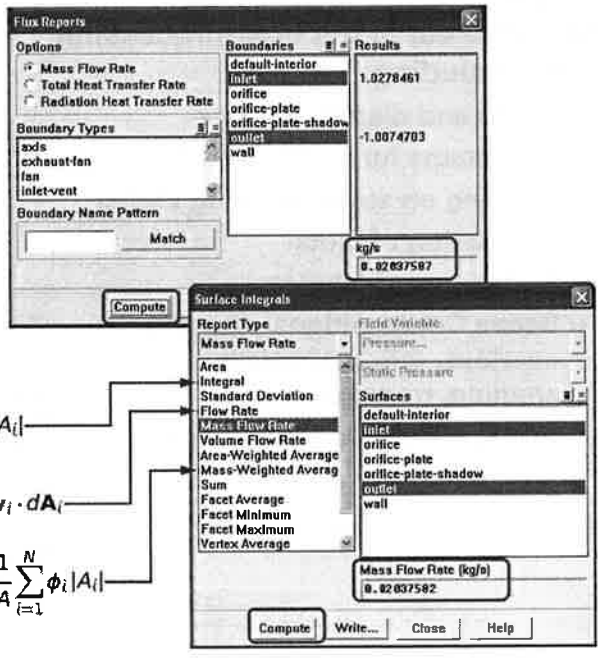




Postprocessing

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



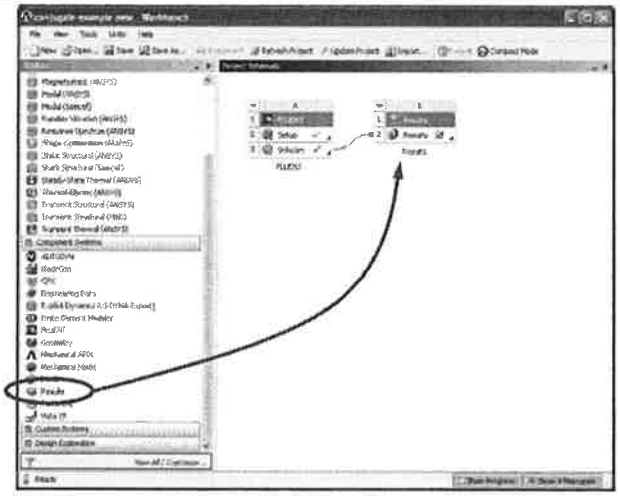
$$\int \phi \, dA = \sum_{i=1}^{N} \phi_i |A_i|$$

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

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

Postprocessing

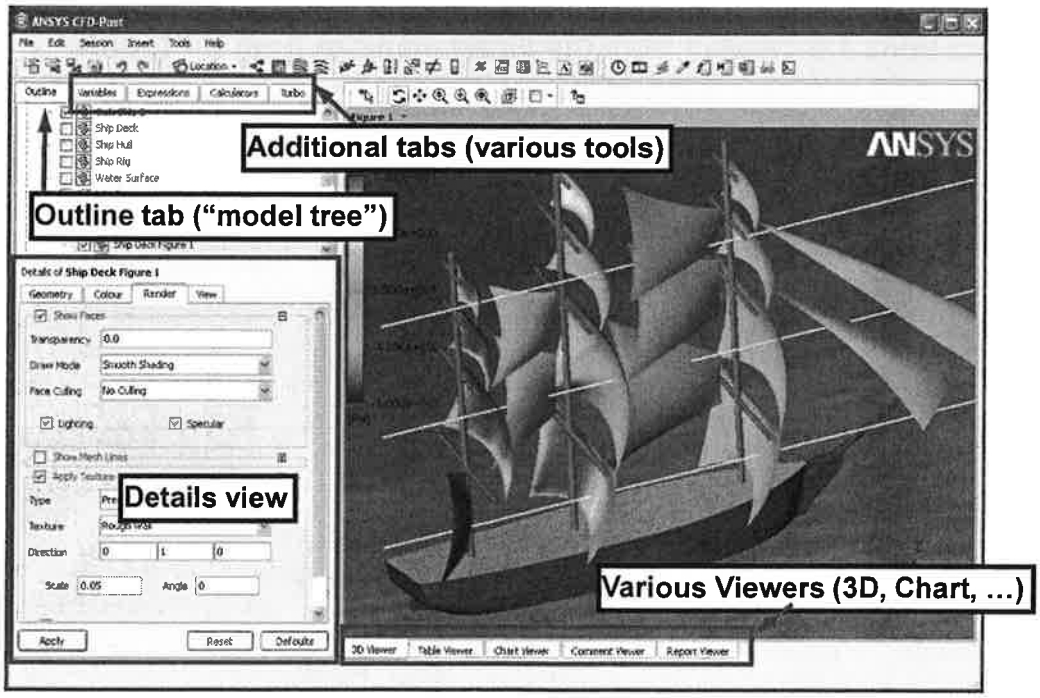
Starting CFD-Post



- **Within ANSYS Workbench**
  - **Drag the CFD-Post icon in the Component Systems list to the project tree.**
  - **OR, create a standalone CFD-Post session.**
- **From the Start Menu or Command Line**
  - **Start > Programs > ANSYS 12.0 > ANSYS CFD-Post**
- **CFD-Post can also be started from the CFX-Solver Manager or the CFX Launcher**

Postprocessing

GUI Layout

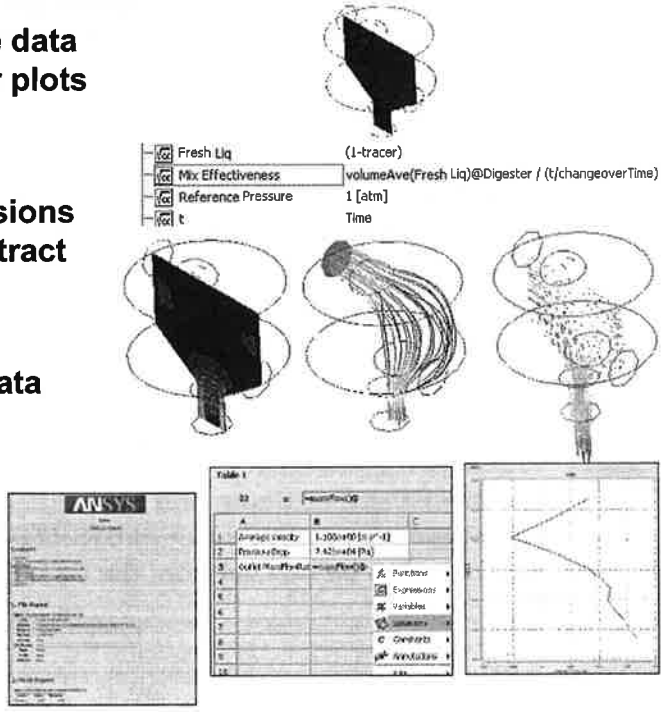


Postprocessing

CFD-Post General Workflow

1. **Prepare locations where data will be extracted from or plots generated**
2. **Create variables/expressions which will be used to extract data (if necessary)**
3. **i) Generate qualitative data at locations**

   ii) Generate quantitative data at locations
4. **Generate Reports**



Postprocessing

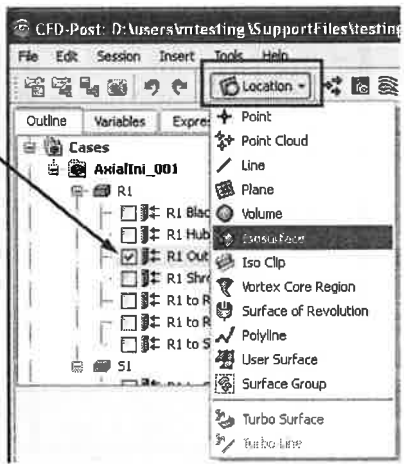
Creating Locations

- **Locations are created from the *Insert* menu or from the toolbar**
- **Once created, all Locations appear as entries in the *Outline* tree**

Use the check boxes next to each object in the *Outline* tree to quickly control visibility

Double-click objects in the *Outline* tree to edit

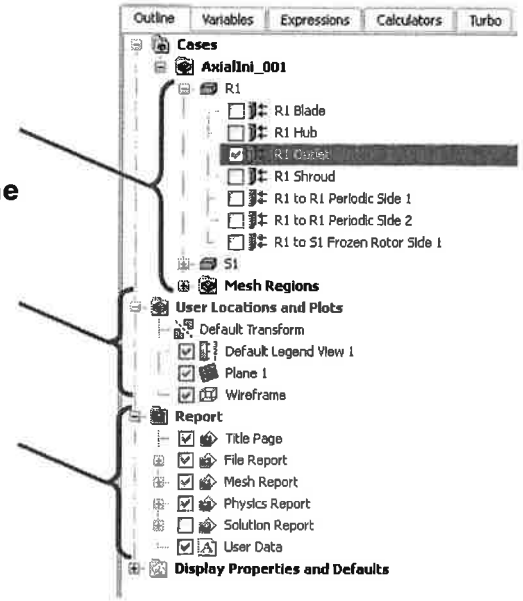
Right-click objects in the *Outline* tree to *Duplicate* or *Delete*



Postprocessing

Creating Locations

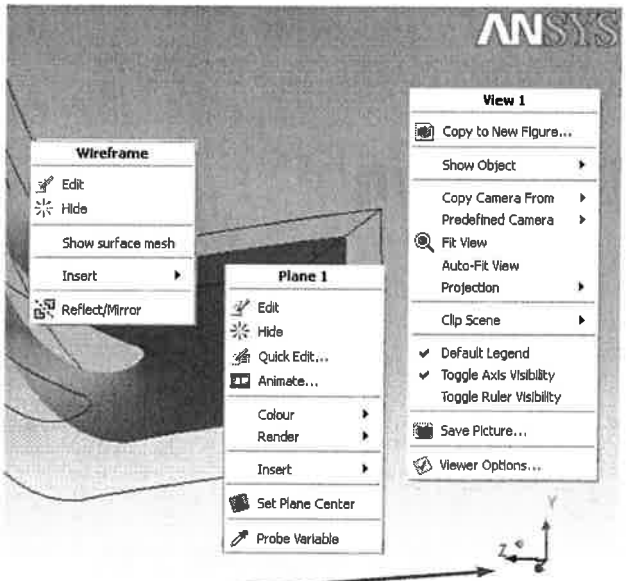
- **Domain, Subdomain, Boundary and Mesh Regions are always available**
  - **Boundary and Mesh Regions can be edited and colored by any variable**
  - **Mesh Regions provides all available interior/exterior 2D/3D regions from the mesh**
- **All Locations you create and listed under *User Locations and Plots***
- **All items contained in the Report are listed here**



Postprocessing

Viewer Right-click Menus

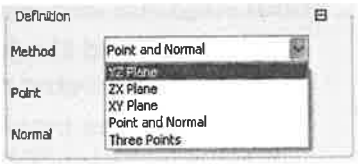
- **Right-clicking in the Viewer provides a context-sensitive menu.**
  - **Right-clicking on an object (e.g. Wireframe, Plane) shows options for that object**
  - **Can also insert new objects based on the current location (such as a vector plot on a plane, contour plot on a surface, etc.)**
  - **Right-clicking in empty space shows options for the current View**



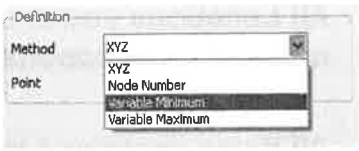
- **Click on the axes to orient the view**

Location Types

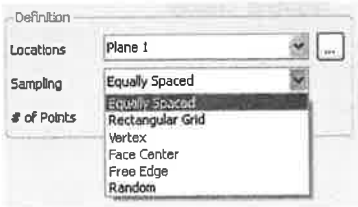
- **Planes**
  - ***XY Plane, Point and Normal*, etc.**
  - **Can define a circle or rectangle to bound the plane, otherwise it's bounded only by the solution domain(s)**



- **Point**
  - ***XYZ*: At coordinates. Can pick from Viewer**
  - ***Node Number*: Some solver error messages give a node number**
  - ***Variable Max / Min*: Useful to locate where max / min values occur**



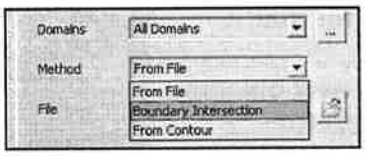
- **Point Cloud**
  - **Create multiple points**
  - **Usually used as seeds to streamlines, vectors**

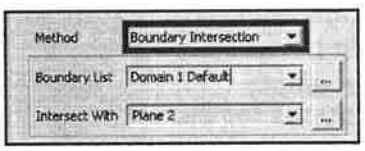


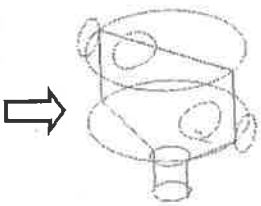
Location Types

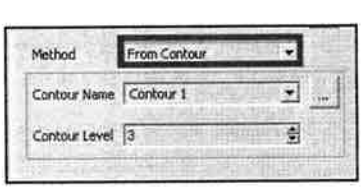
- **Lines**
  - **Straight line between two points**
  - **Usually used as the basis for an XY Chart**

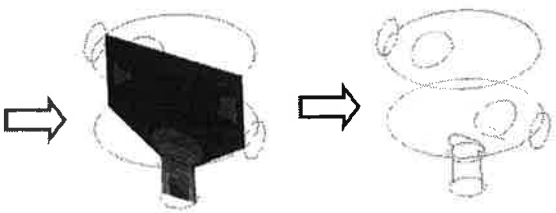
- **Polylines**
  - **Also used for Charts**
  - **Read points from a file**
  - **Use the line of intersection between a boundary and another location**
  - **Extract a line from a contour plot**







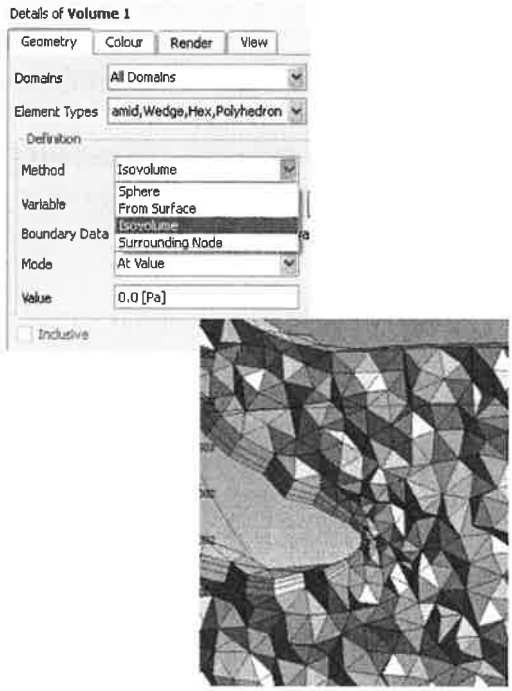




Postprocessing

Location Types

- **Volumes**
  - **Elements are either in or out**
    - **No cut volumes**
  - **From Surface**
    - **A volume is formed from all elements touching (or above / below) the selected location**
    - **Can be useful for mesh checking**
  - **Isovolume**
    - **Base on a variable at, above or below a given value, or between two values**

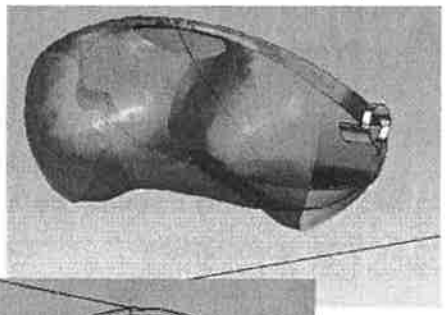


Postprocessing

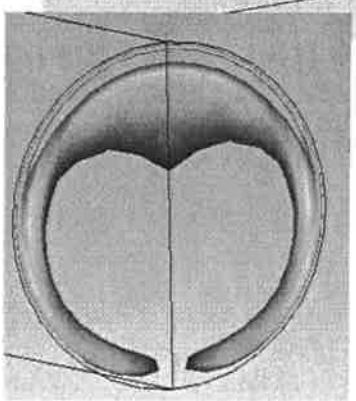
Location Types

- **Isosurfaces**
  - **Surface of a variable at a specified value**

Isosurface of pressure behind a flap valve

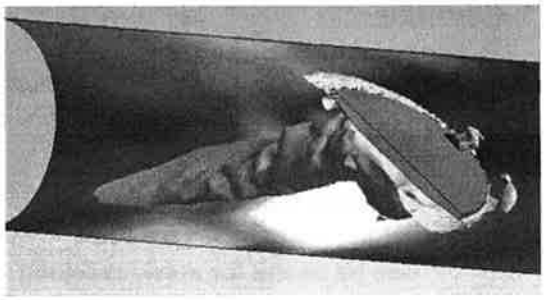


- **Iso Clip**
  - **An Iso Clip takes a copy of any existing location and then clips it using one or more criteria**
    - **E.g. a outlet boundary plot which is then clipped by Velocity >= 10 [m/s] and Velocity <= 20 [m/s]**
  - **Can clip using any variable, including geometric variables**

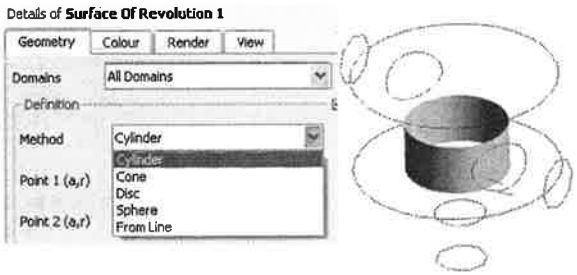


Location Types

- **Vortex Core Region**
  - **Used to automatically identify vortex regions**
  - **Best method is case dependent**
  - **See documentation for details on the different methods**

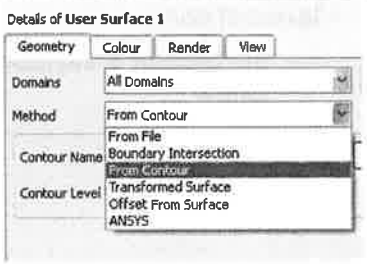


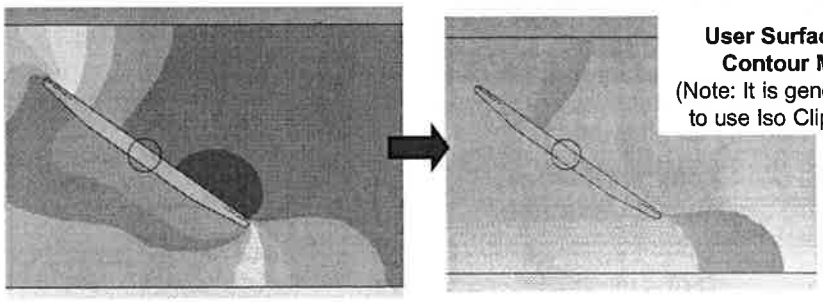
- **Surface of Revolution**
  - **Predefined options for *Cylinder*, *Cone*, *Disc* and *Sphere***
  - ***From Line* is much more general**
    - **Any line (existing Line, Polyline, Streamline, Particle Track) is rotated about an axis**



Location Types

- **User Surface**
  - **Provides a number of additional surface creation options including**
  - ***From File*: reads point data from a text file; usually export this file from a different case**
  - ***From Contour*: extract a contour level**
  - ***Transformed Surface*: rotate, translate or scale and existing surface**
  - ***Offset From Surface*: offset an existing surface in either the Normal direction or by Translating**



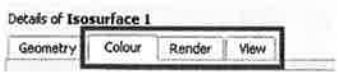


User Surface: From Contour Method
(Note: It is generally easier to use Iso Clips instead)

Postprocessing

Color, Render and View

- All Locations have similar *Color*, *Render* and *View* settings



- Color
  - Select the variable with which to color the location
  - Set the *Range* (*Global*, *Local*, *User Specified*)
  - Pick a *Color Map*

- Render
  - *Draw Faces*: shows solid surface
  - *Draw Lines*: shows mesh edges or intersecting lines between mesh edges and the plot
  - *Transparency*, *Lighting*, *Texture*...

- View
  - Apply Rotation, Translations, Reflection, Scaling
  - Pick a different *Instance Transform*

Postprocessing

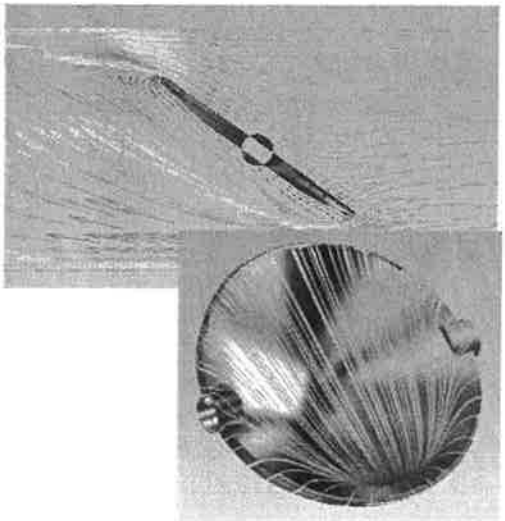
Other Graphics Objects

- Insert from the toolbar or the Viewer right-click menus



- Vectors, Contour and Streamlines use existing locations as a base

- Vector Plot
  - Can plot any vector variable; usually velocity
  - Can project vectors Normal or Tangential to the base object

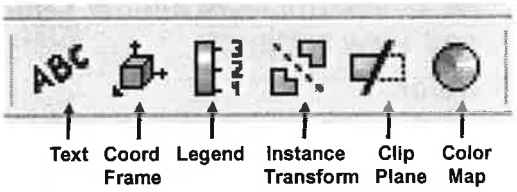


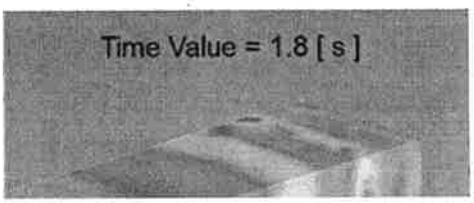
- Streamlines
  - Can proceed forwards and/or backwards from a seeding location
  - Use the *Surface Streamline* option to visualise velocity "on" walls

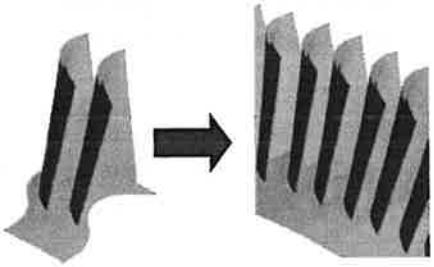
Postprocessing

Other Graphics Objects

- **Text: add your own labels to the Viewer**
  - **Auto-text allows you to show time step/values, expressions, filenames and dates that change automatically**
- **Coord Frame**
- **Legend**
  - **Create addition legend that are tied to a specific plot (the default legend changes automatically with the active plot)**
- **Instance Transform**
  - **Usually used to re-create full plots from symmetric/periodic solution data**



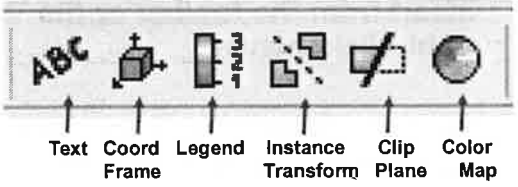




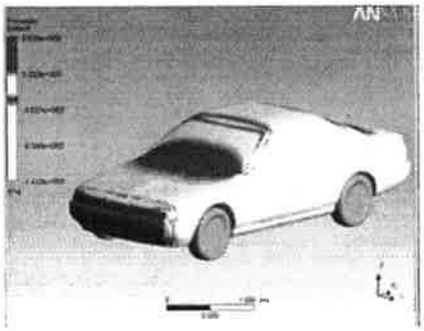
Postprocessing

Other Graphics Objects

- **Clip Plane**
  - **Define a plane; when active all viewer object in front / behind this plane are hidden**
- **Color Map**
  - **Create custom color maps**



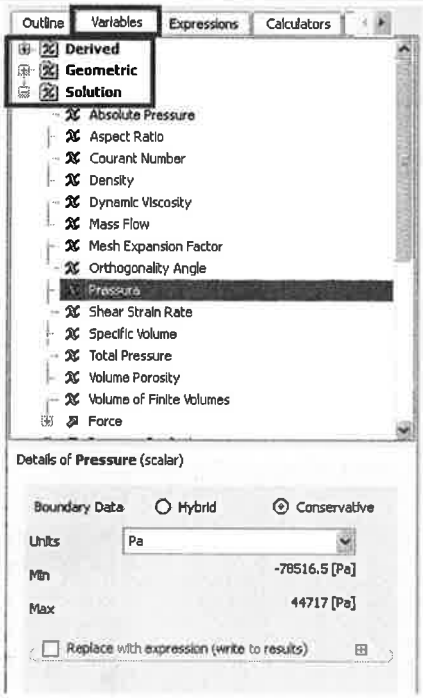




Postprocessing

Variables Tab

- The *Variables* Tab shows information about all available variables
- *Derived* variables
  - Calculated by CFD-Post – they are not contained in the results file
- *Geometric* variables
  - X, Y, Z, Normals , mesh quality data
- *Solution* variables
  - From the results file
- *User Defined* variables
  - Create new derived variables
- *Turbo* variables
  - Additional variables automatically created for turbomachinery cases

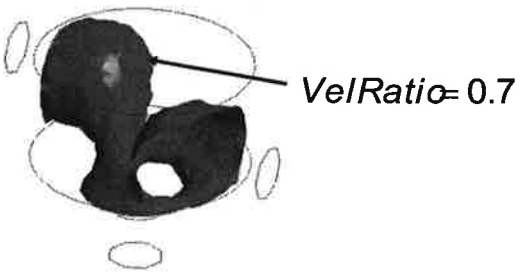


Postprocessing

User Defined Variables Example

- Goal: Plot an isosurface at VelRatio = 0.7

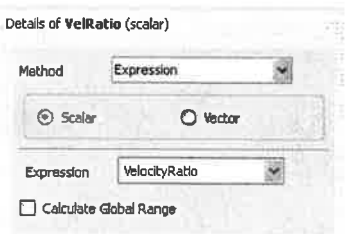
where $VelRatio = \frac{V_{local}}{V}$



1. On the *Expressions* tab create the expression for Velocity Ratio:



2. On the *Variables* tab create a new variable named *VelRatio* using *Method* = *Expression*

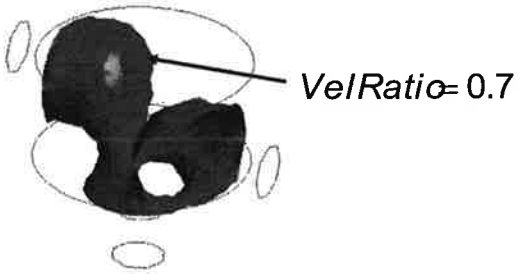


Postprocessing

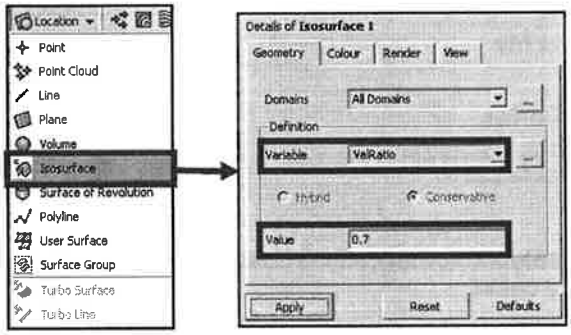
User Defined Variables Example

- **Goal: Plot an isosurface at VelRatio = 0.7**

where $VelRatio = \frac{V_{local}}{V}$



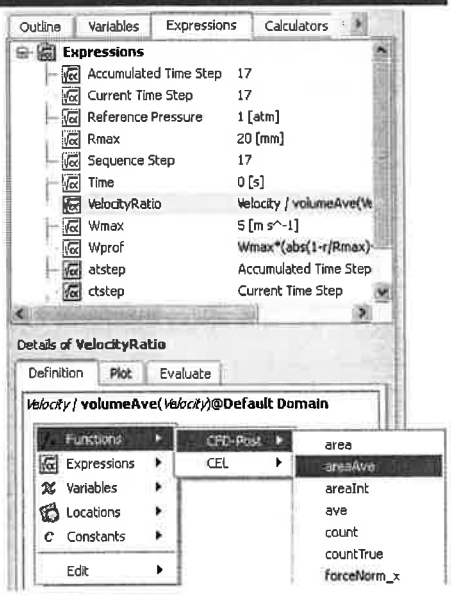
3. **Create an Isosurface using the variable *VelRatio* at a value of 0.7**

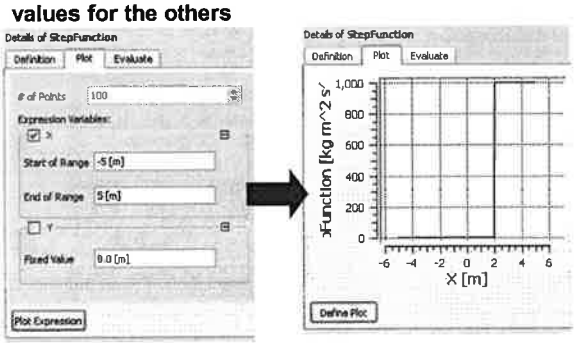


Postprocessing

Expressions Tab

- **The *Expression* tab shows all existing expressions and allows you to create new expressions**
  - **Right-click in the top area > *New***
- **Enter the new expressions on the *Definition* tab in the *Details* view**
  - **Right-click to view *Functions*, *Variables* etc. that can be used to build your expression**
- **Use the *Plot* tab to view an XY plot of the expression**
  - **Must enter a range for one of the variables and fixed values for the others**

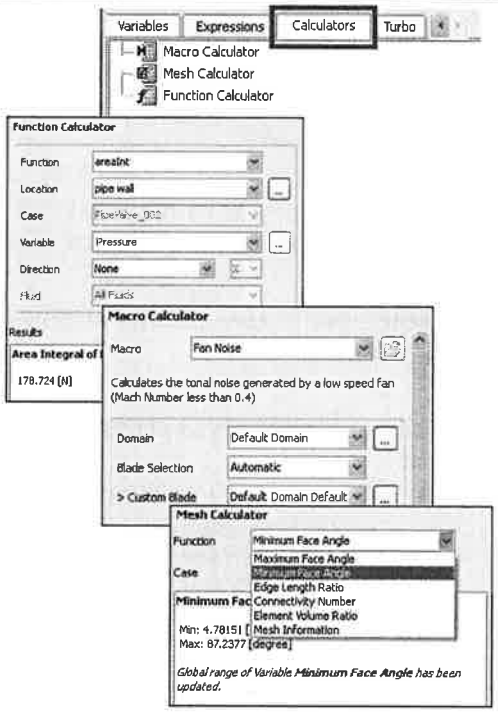




Postprocessing

Calculators Tab

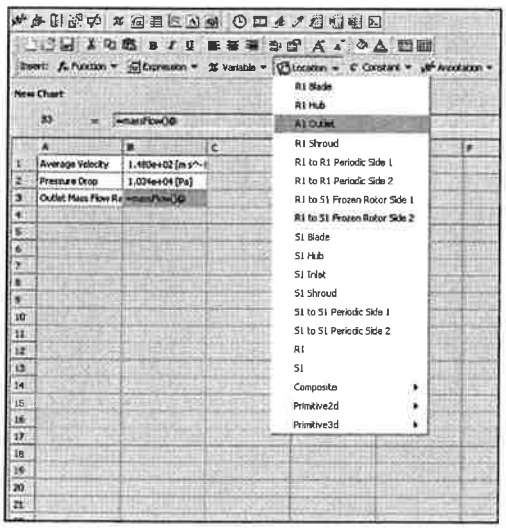
- **Function Calculator**
  - Extract engineering data from the results
  - Many functions, see doc to understand how they operate
  - Same function used as when creating expressions
- **Macro Calculator**
  - Run predefined Macros
  - Write your own Macros and have them appear here
  - More in Scripting lecture
- **Mesh Calculator**
  - Mesh quality metrics and stats
  - Field variables exist for all the metric and can be plotted

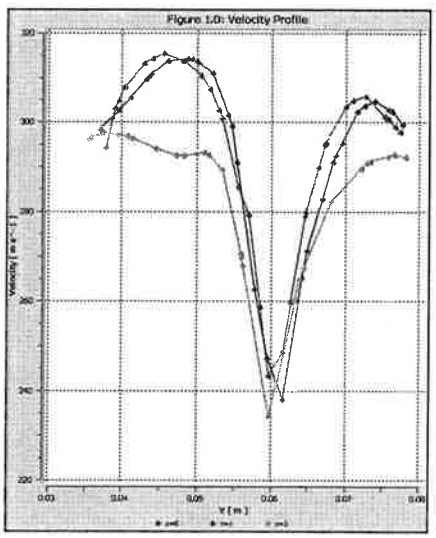


Postprocessing

Generating Tables and Charts

- **Tables and Charts can be created to format and present results**

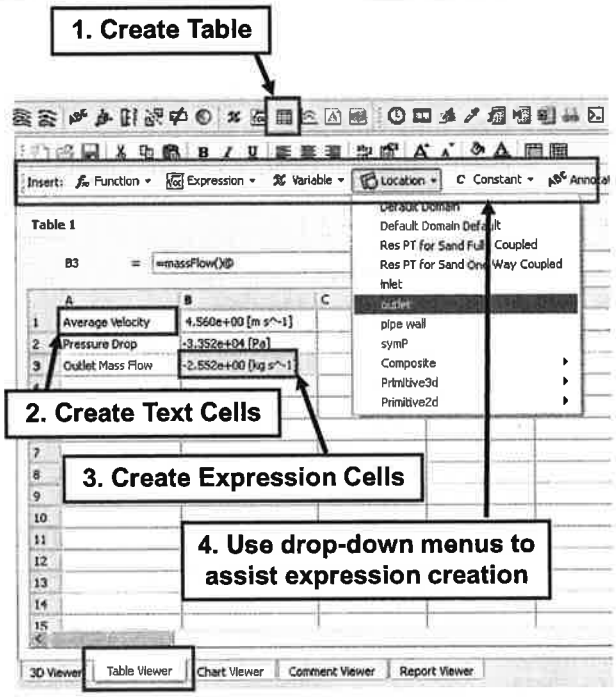




Postprocessing

Tables

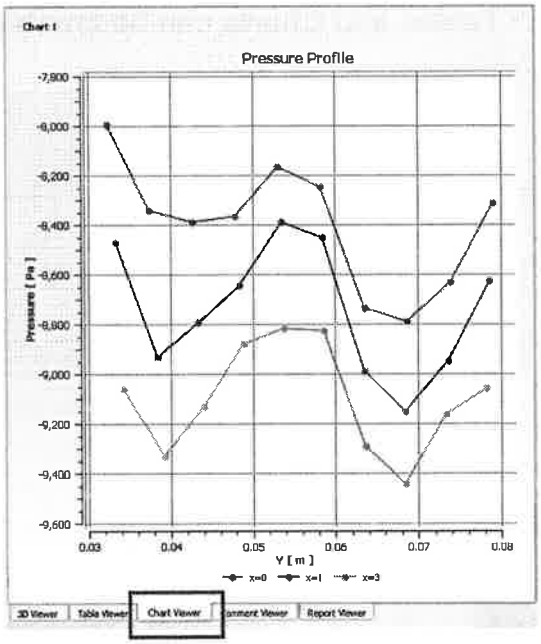
- Select *Insert* > *Table* or use the toolbar icon to create a new table
  - *3D Viewer* will switch over to the *Table Viewer*
- Tables allow you to display data and expressions in a tabular view
- Tables are automatically added to the Report
- Cells can contain expressions or text
  - Begin with "=" to distinguish
  - Expressions are evaluated and updated when variables and/or locations they depend on change
- This is not a spreadsheet
  - Cannot reference other cells



Postprocessing

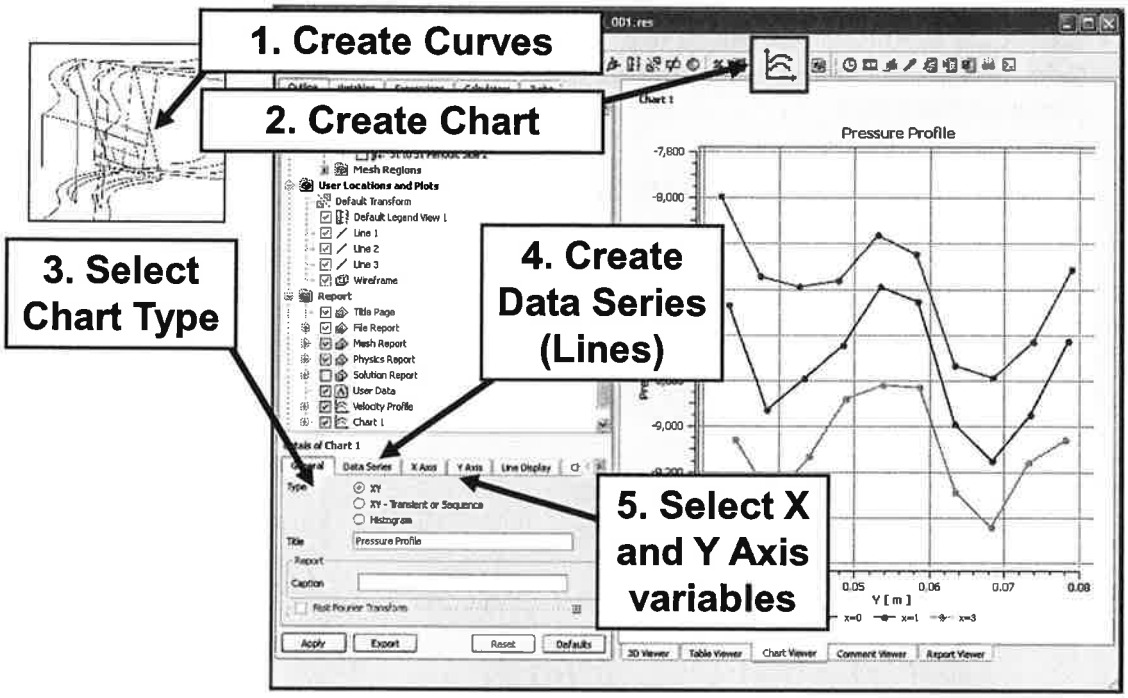
Charts

- Plot a relationship between two variables along a line/curve
  - Need to create the line first
  - Polyline, Boundary Intersection curve, Contour line, etc.
- Charts are automatically added to the Report
- Chart Points are not necessarily evenly spaced
  - Data points usually correspond to where the line/curve intersects a mesh face
- Multiple lines can be plotted on a single chart



Postprocessing

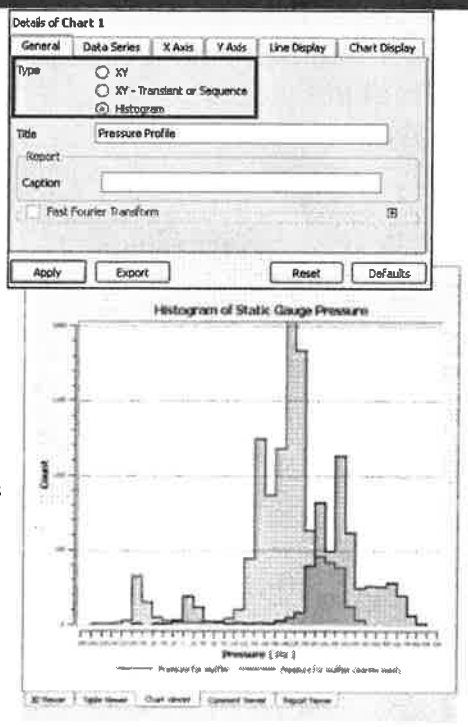
Charts



Postprocessing

Charts: Type

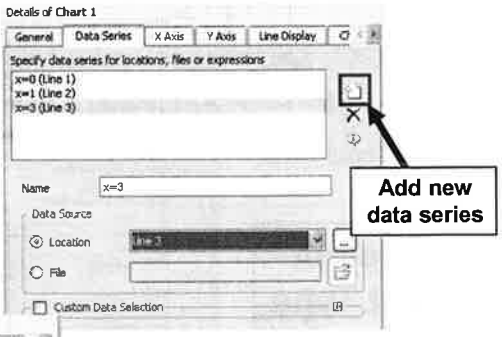
- **Charts can be one of three types:**
  - **XY**
    - **Standard XY plots based on line locators**
  - **XY – Transient or Sequence**
    - **Plots an expression (usually Time) versus a variable at a point locator**
    - **Typically used to show the transient variation of a variable at a point**
  - **Histogram**
    - **Can be based on any locator that contains multiple data locations – lines, surfaces, planes, domains (but not points)**
    - **Plots a variable divided into discrete bands on the X Axis versus the frequency of occurrence on the Y Axis**



Postprocessing

Charts: Data Series and Axes

- **Each data series corresponds to a location (line, point, etc.) which corresponds to a curve on the chart**
- **Use the *X* and *Y Axis* tabs to set the variables on the axes**
- **The remaining tab are for various display options**



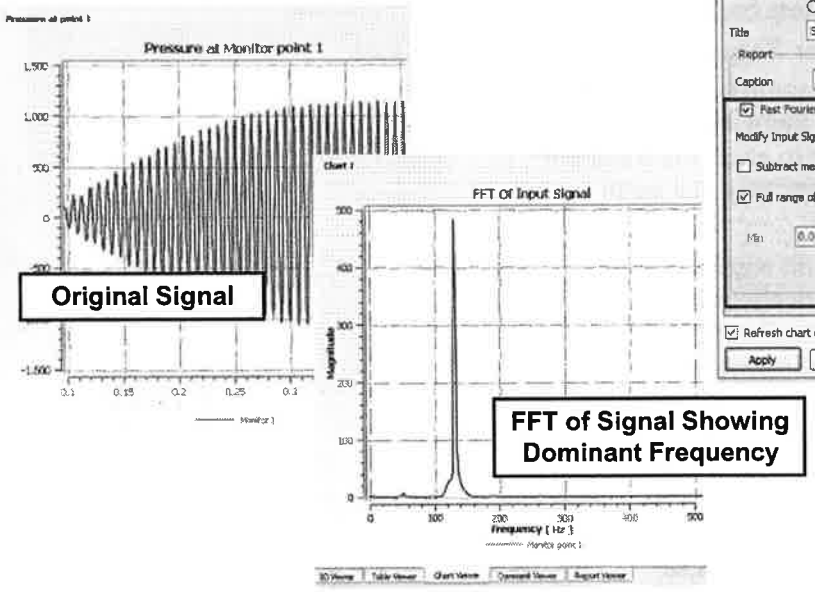


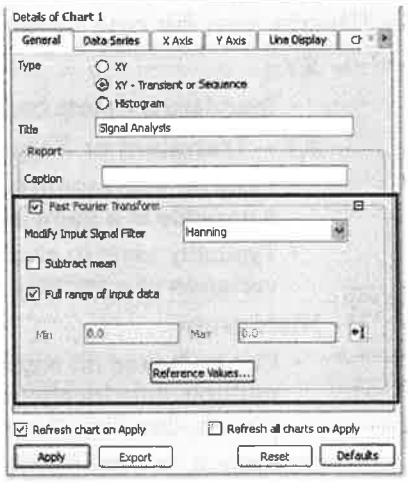


Postprocessing

Fast Fourier Transform

- **FFT can be applied to signals to extract frequency data**



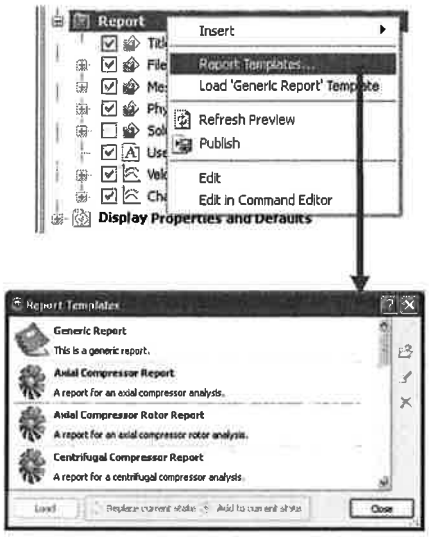


Postprocessing

Reports

- **CFD-Post has report generation tools which allow for rapid creation of customized reports**

- **Reports are template based**
  - **Depending on the information contained in a results file, a report template will be selected automatically**
  - **Right-click on *Report* to select a different template**
    - **You can create your own custom templates or modify existing templates**
    - **E.g. add you company logo, add Charts, Tables, Plots etc**



Postprocessing

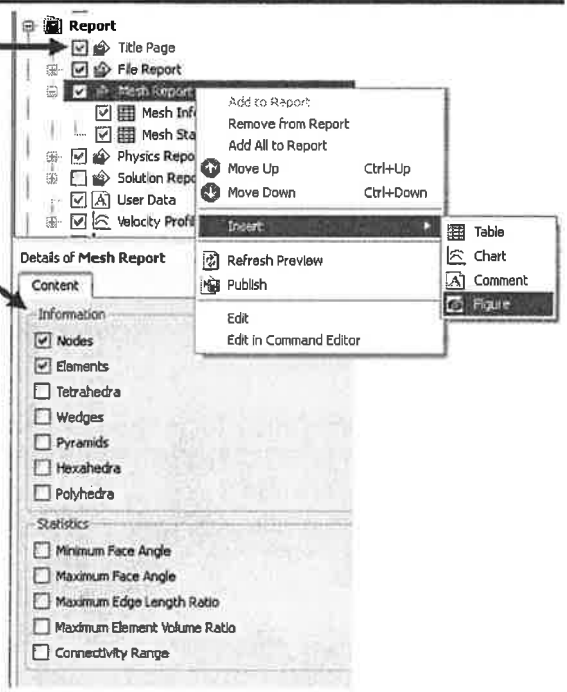
Reports

- **Use the check boxes to control what is included in the report**

- **Double-click items to edit**
  - **For example, editing the *Mesh Report* shows that additional items can be included**

- ***Tables* and *Charts* are automatically added to the report. Other items that can be added are *Comments* and *Figures*.**
  - ***Right-click* > *Insert* to add new items**

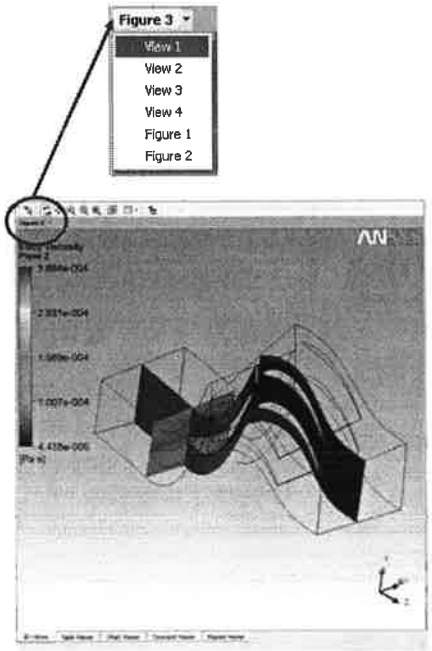
- **Can also right-click on each item to move it up or down in the report**



Postprocessing

Reports: Figures

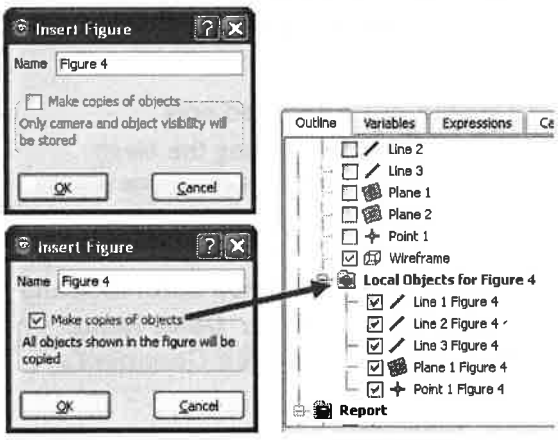
- When you add a new *Figure* it will be listed in the drop-down menu in the top corner of the Viewer
- Figures are not static, you can change them after they have been created
  - If you do not want to change a Figure, make sure one of *View 1 – View 4* is selected from the drop down menu
- To change the camera position for a figure (i.e. rotate / pan / zoom) select the figure from the Viewer drop down menu and move as necessary
  - All changes are automatically saved to the Figure



Postprocessing

Reports: Figures

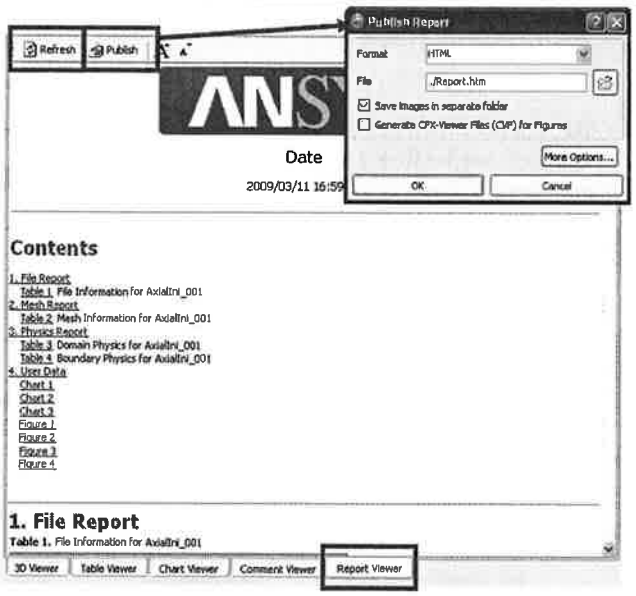
- When you create a Figure, you have the option to *Make copies of objects*
  - If you disable this only the camera and object visibility is stored with the figure
    - So changing global objects will *always* cause the Figure to change
    - Good if you want the Figure to update automatically
  - If you enable this a local copy of all the current objects is created and shown in the *Outline* tree
    - Changing global object will not change the Figure, you must edit the local objects
  - In both cases the camera position and object visibility can only be changed when the Figure is active



Postprocessing

Reports

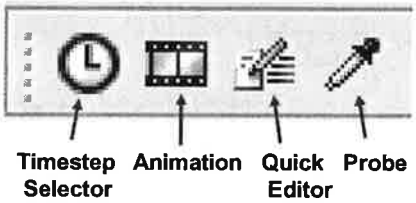
- To view the report the *Report Viewer* tab
- After making changes to objects contained in the report you will need to *Refresh*
- *Publish* writes out an HTML or Text copy of the report
  - You have the option can generate 3D Viewer files (see below) for all Figures



Postprocessing

Other Tools

- Timestep Selector
  - Transient results are post-processed by loading in the end results file, then selecting different timesteps from the *Timestep Selector*
- Animation
  - Animate objects, create MPEGs
  - More on next slide
- Quick Editor
  - Provides a very quick way to change the "primary" value associated with each object
- Probe
  - Pick a point from the Viewer and probe a variable value at that point

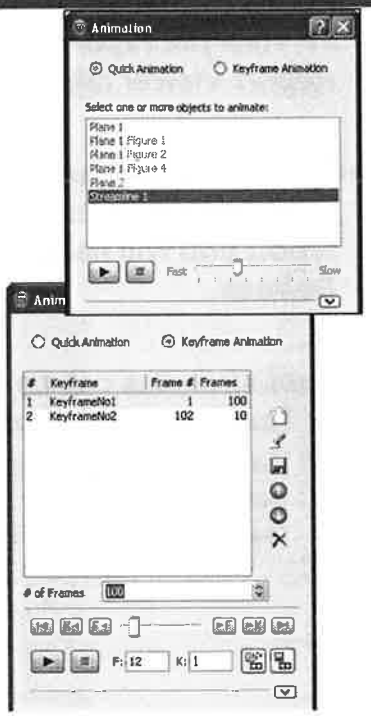




Postprocessing

Animations

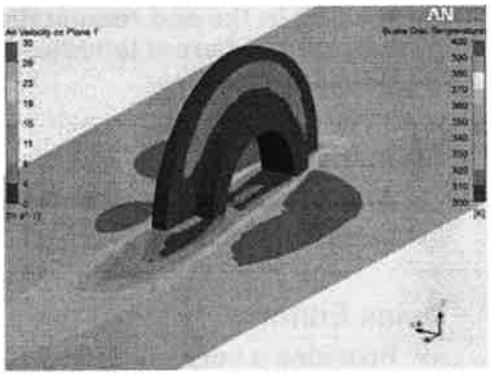
- **Animations have two modes, *Quick* and *Keyframe***
- **In *Quick Animation* mode you simply pick an object and click the *Play* button**
  - **The "primary" variable for that object is animated**
  - **Limited control**
- ***Keyframe* mode gives you much more control**
  - **When you create a Keyframe a snapshot of the current state is stored with that Keyframe**
  - **A series of Keyframes represent a series of different states**
    - **Camera position, object visibility, selected timestep, or anything else can be different between Keyframes**
  - **An animation needs at least two Keyframes (one for the start and one for the end)**
  - **Enter the *# of Frames* between each Keyframe**
    - **Everything that is different between the Keyframes gets interpolated over the # of Frame**



Postprocessing

Typical Keyframe Animation Procedure

1. **Using the *Timestep Selector* load the first timestep**
2. **Create necessary plots and position the view**
3. **Create the first *Keyframe***
4. **Load the last timestep**
5. **If necessary change the plots and the view**
6. **Create the second *Keyframe***
7. **Select the first *Keyframe* and set the *number of Frames***
   - **This is the number of Frames between the first and second Keyframes**
   - **If we have a total of 100 timesteps, then setting *# of Frames* to 98 will produce a total of 100 frames (98 plus first, plus last) and therefore 1 frame per timestep**
8. **Set the Movie options**
9. **Rewind to the first *Keyframe* and click *Play***

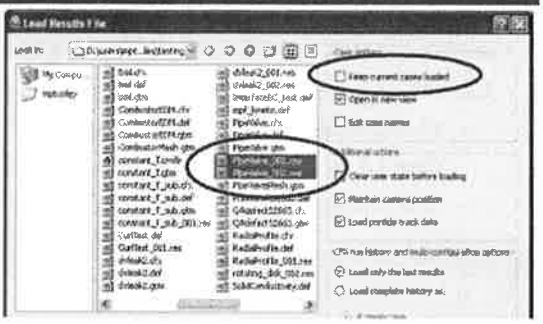


In this example the first and second Keyframes used a different view position and the Transparency of the Plane was changed from 0 to 1. The changes between Keyframes are interpolated gradually over each animation frame

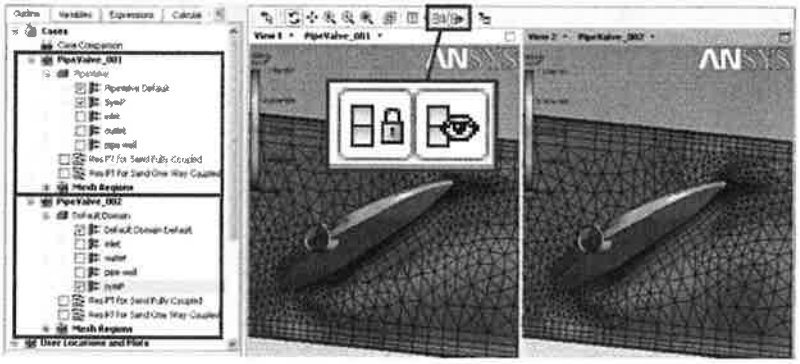
Postprocessing

Multi File Mode

- To post-process multiple files simultaneously you can:
  - Multi-select files when loading
  - Or enable the *Keep current cases loaded* toggle



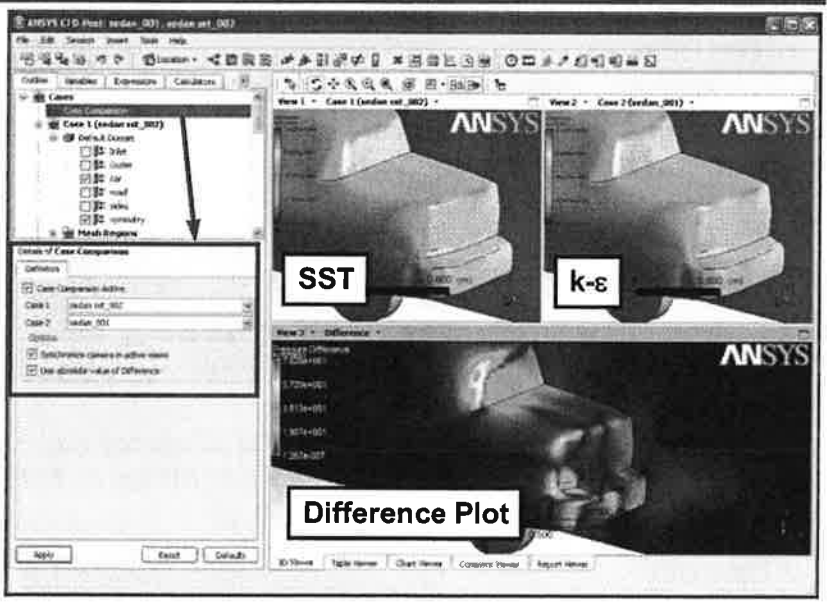
- Each file is shown separately in the *Outline* tree and the *Viewer*
-  Sync cameras
  - All Views move the same
-  Sync objects
  - The visibility of all *User Locations and Plots* is the same



Postprocessing

Case Comparison

- When multiple files are loaded you can select *Case Comparison* from the *Outline* tree
  - Automatically generates difference variables and plots

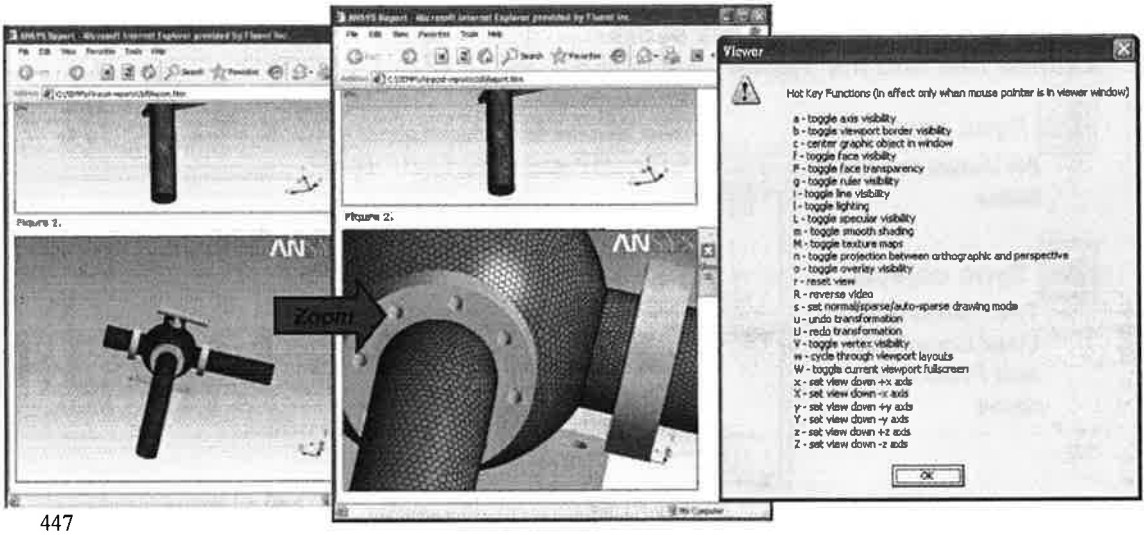


- Expression syntax:
  - `function()@CASE:#.Location`
  - E.g: `areaAve(Pressure)@CASE:1.Inlet`

Postprocessing

3D Viewer Files

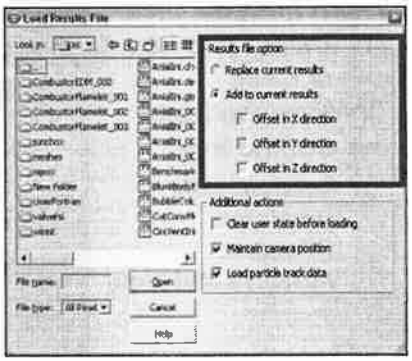
- ***Save Picture*** **in the** ***CFX Viewer State (3D)*** **file format (.cvf file)**
- **Can then use the stand-alone Viewer to view the file, rotate, pan, zoom, etc**
  - **Unlicensed and free to distribute to your customers**
  - **Can embed 3D Viewer files in PowerPoints and HTML files**
  - **Download from the Customer Portal under** ***Product Information*** **>** ***ANSYS CFX***



Postprocessing

Files

- **Results**
  - **ANSYS – CFD Post can read ANSYS results for temperature, velocity, acceleration, magnetic forces, stress, strain, and mesh deformation**



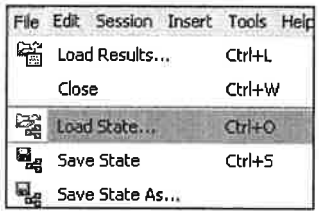
- **Import**
  - **Locations – .csv files which contain point data which defines a polyline or surface**
  - **ANSYS Surface Mesh (.cdb): To allow for export of data on a surface for use as a boundary condition in ANSYS**
- **Export**
  - **Profile Boundary Data: for use in CFX-Pre**
  - **General formatted results data**
  - **ANSYS Load Data: Written onto an imported ANSYS .cdb file**

Files

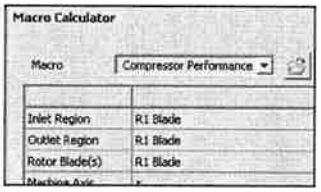
- **Session**
  - Session files can be used to quickly reproduce all the actions performed in a previous CFD-Post session
  - Session recording in CFX Command Language (CCL)



- **State**
  - Saves a snapshot of all objects
  - Excludes actions (e.g. file output)



- **Macros**
  - Useful for automating tasks.



Establishing result verification and validation

- Result verification – checking mesh independence and solver accuracy
- Result validation – comparing with experimental data

Day 3 (3:30pm – 5:30pm)

- Hands on with CFD software
- CFD simulation for building ventilation design – Part 2
  - Tutorial for Practice 1 - Simulation of air flow at unit level (step by step guideline)

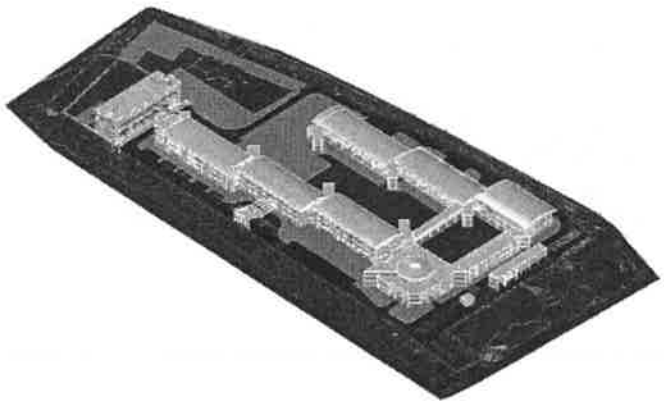
Day 4 (9:00am – 9:45pm)

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

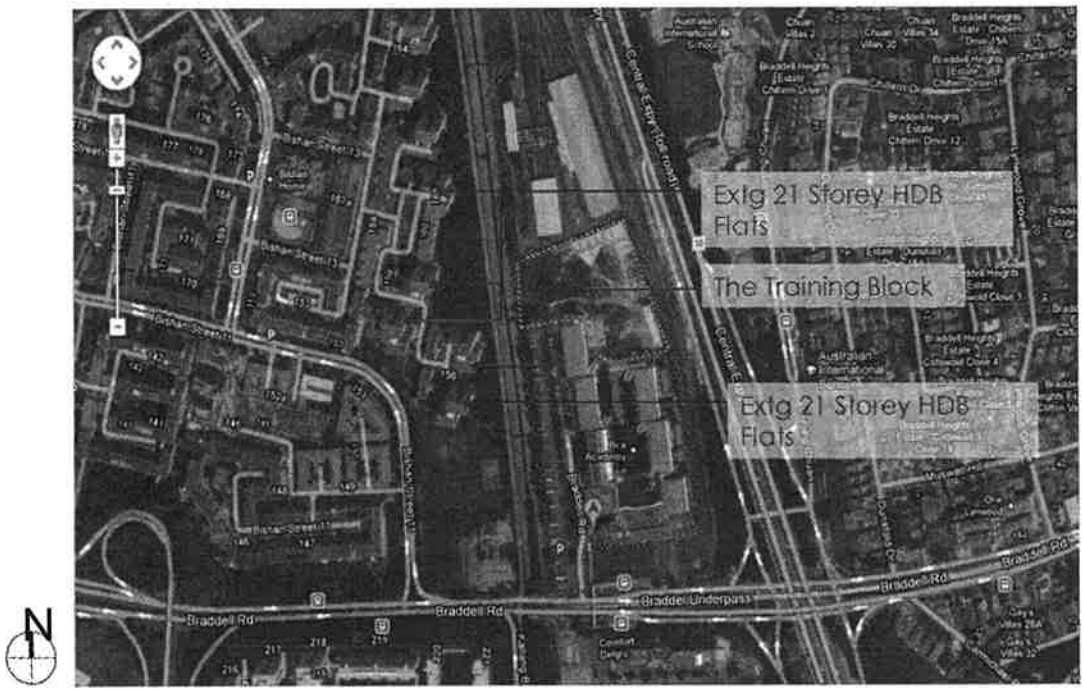
Natural Ventilation for BCAA Training Block

- 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

CFD Study of Air Flow around BCA Academy 2-Storey Training Block



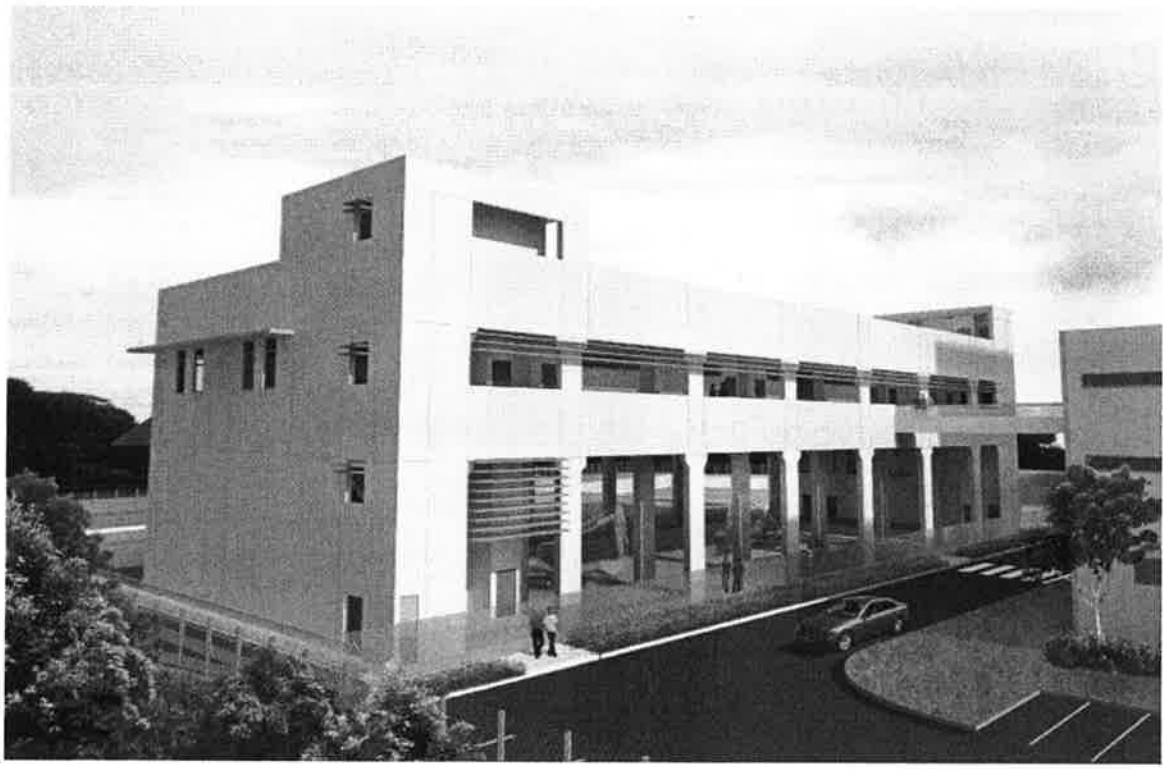
Site Information



455

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



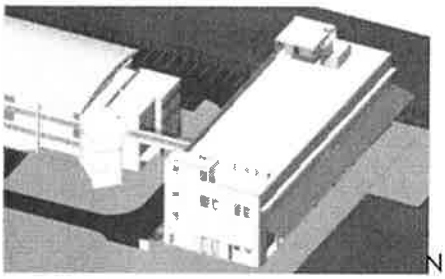


457

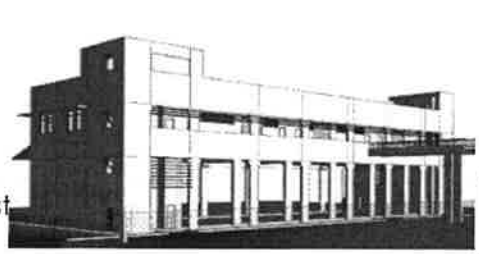
Project Information – 3D BIM IMAGES



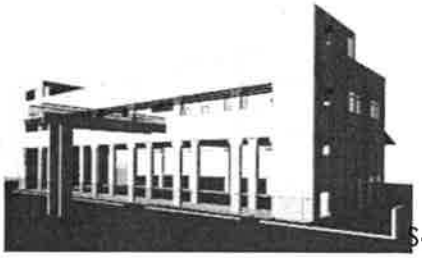
South-West



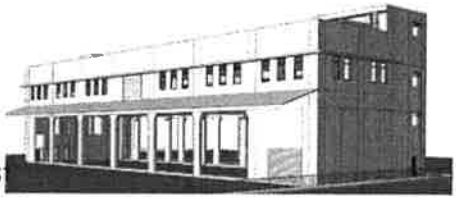
North-East



South-West



South-East



North-West

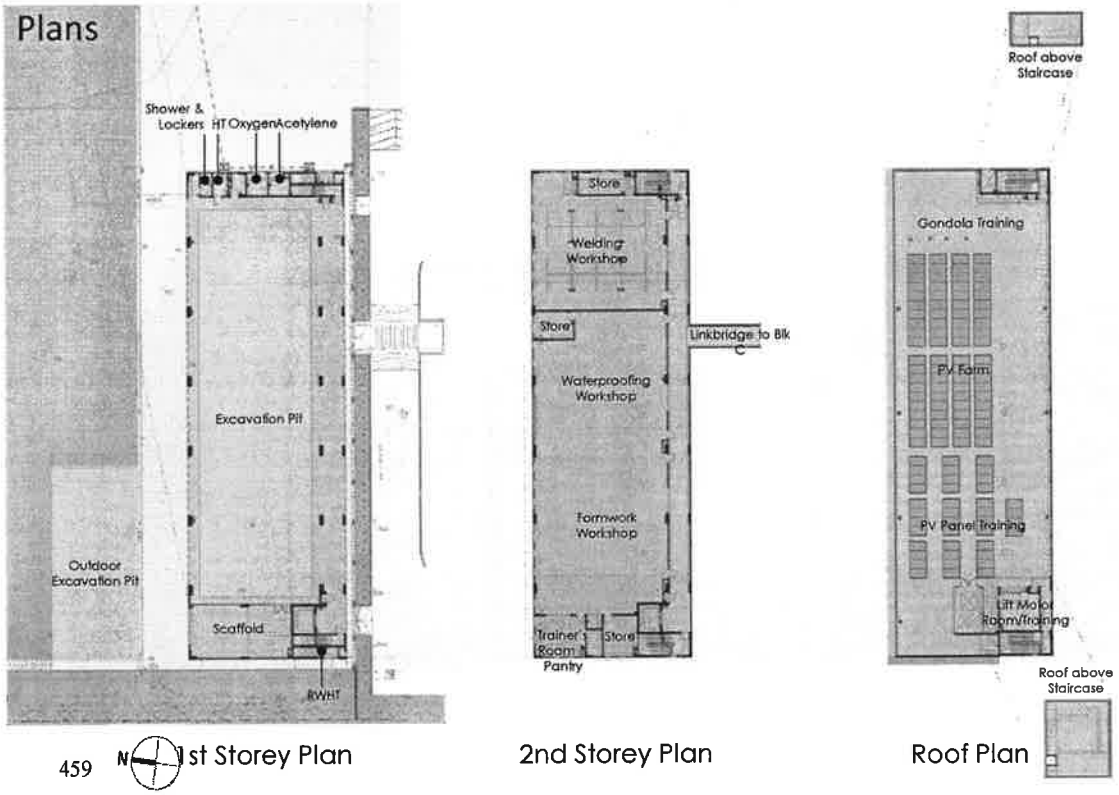


North-East

458

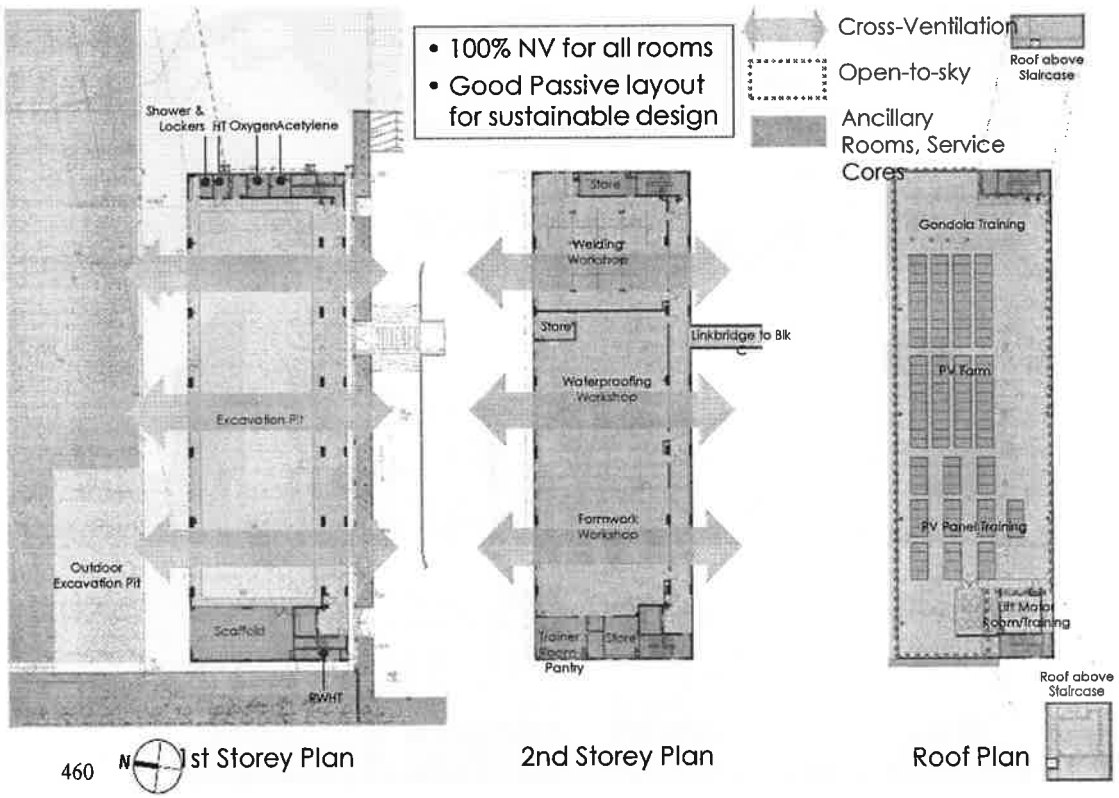
Project Information –

Plans



459 N 1st Storey Plan | 2nd Storey Plan | Roof Plan

01 Natural Ventilation



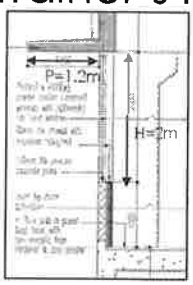
460 N 1st Storey Plan | 2nd Storey Plan | Roof Plan

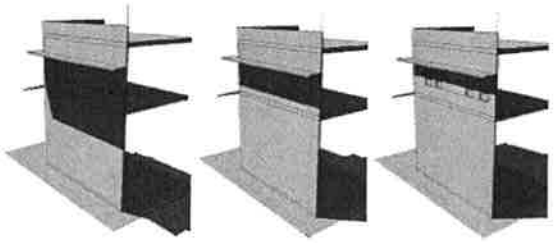
01 Natural Ventilation

100% naturally ventilated Building:

- RC ledge provides shade and glare-control throughout afternoon

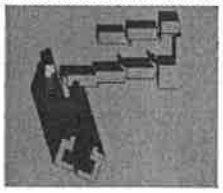
Trainer's Room





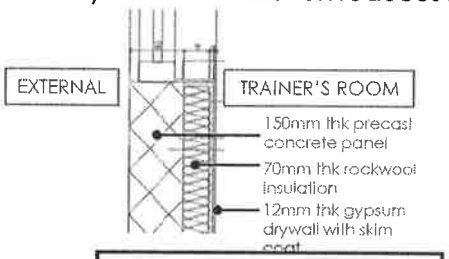
2pm 3pm 4pm



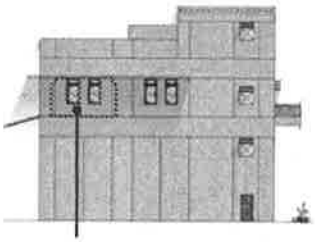


5pm (shaded by HDB Flats)

- Internal drywall with insulation reduces heat gain

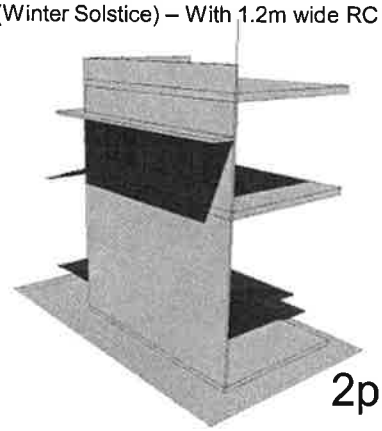


Wall U-Value:
0.435Wm2/K

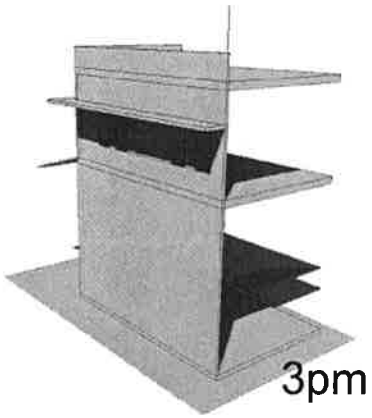


90mm thk drywall with insulation inside Trainer's Room

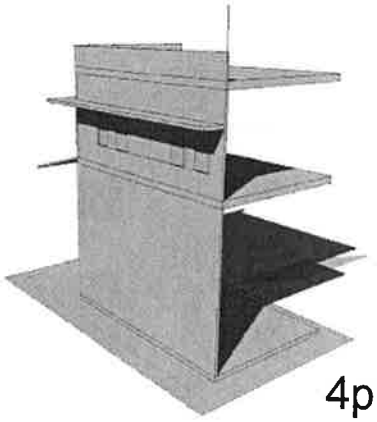
21 DECEMBER (Winter Solstice) – With 1.2m wide RC ledge



2pm

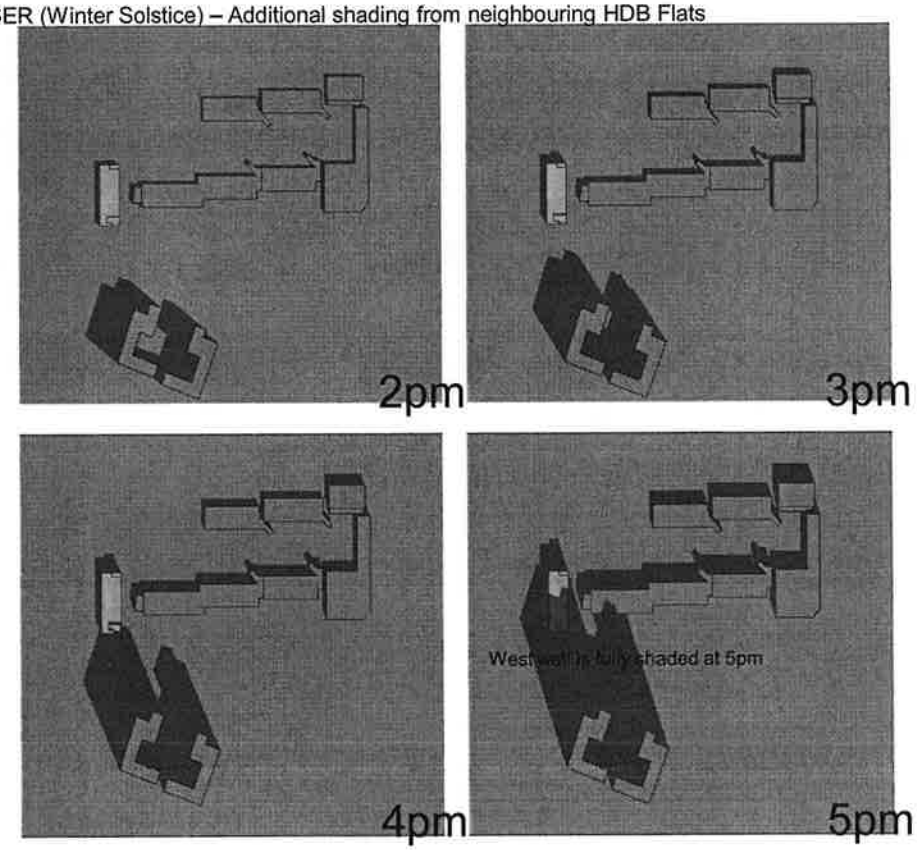


3pm



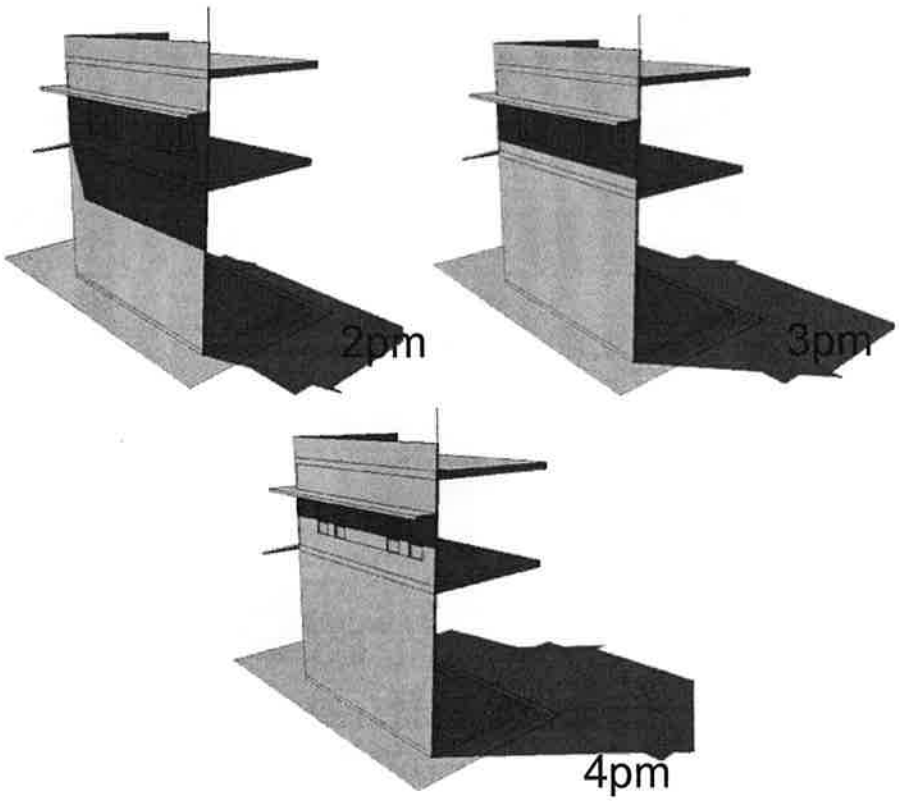
4pm

21 DECEMBER (Winter Solstice) – Additional shading from neighbouring HDB Flats



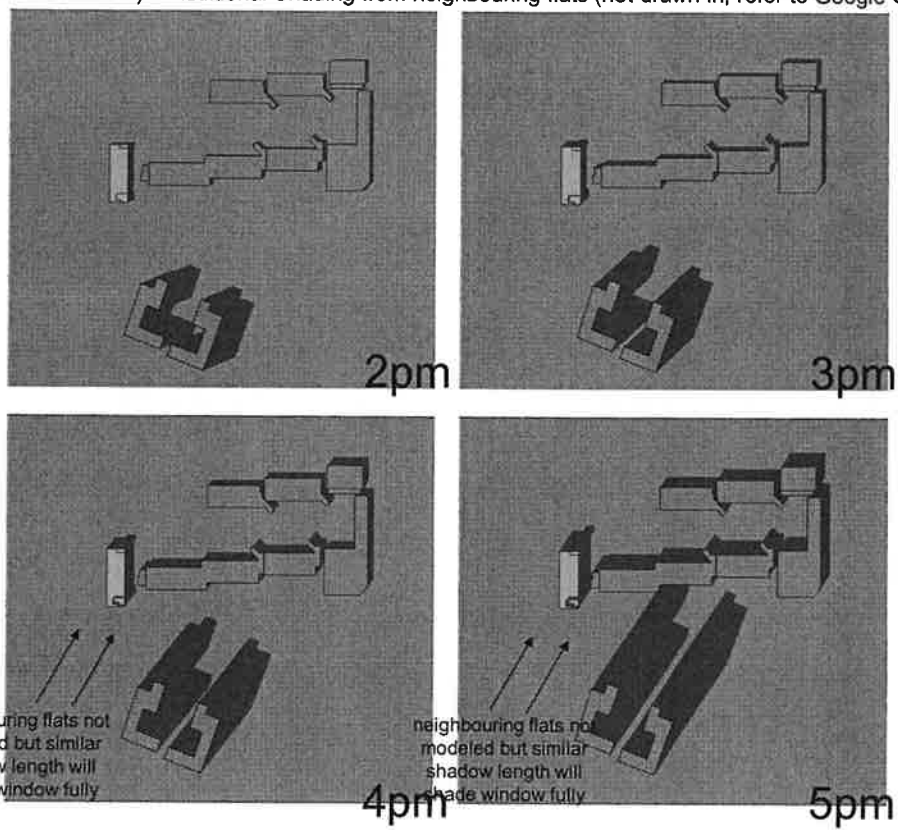
463

21 JUNE (Summer Solstice) – With 1.2m wide RC ledge



464

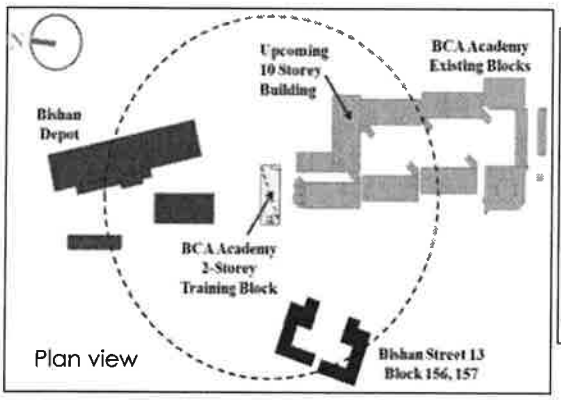
21 JUNE (Summer Solstice) – Additional Shading from neighbouring flats (not drawn in, refer to Google Site Map)

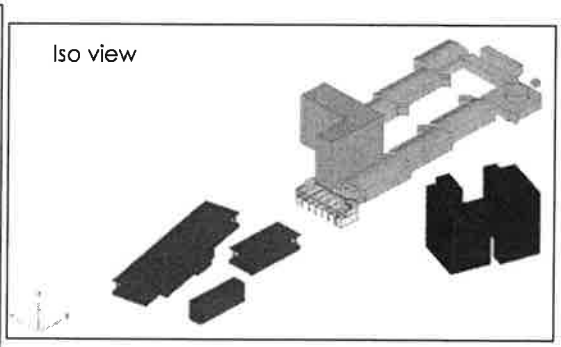


01 Energy Efficiency – Natural Ventilation

NRB	Criteria	Max Points	Points Achieved
1-4(a)(ii)	Use of ventilation simulation software	10	??

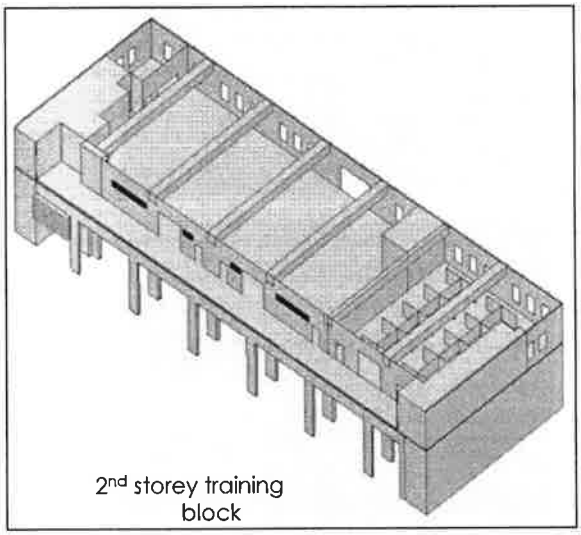
Computational Model

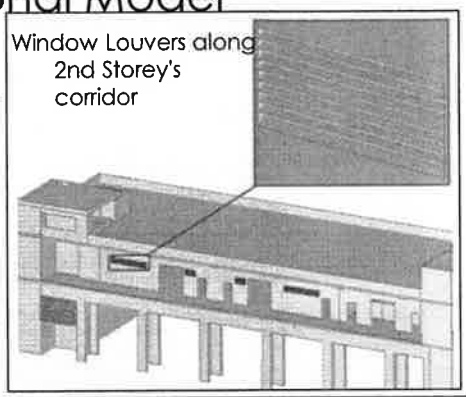


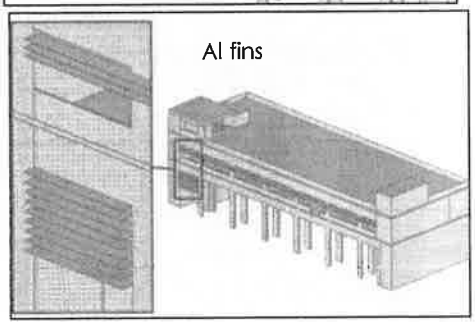


01 Energy Efficiency – Natural Ventilation

Computational Model





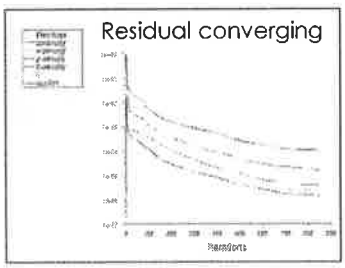


01 Energy Efficiency – Natural Ventilation

Simulation Methodology

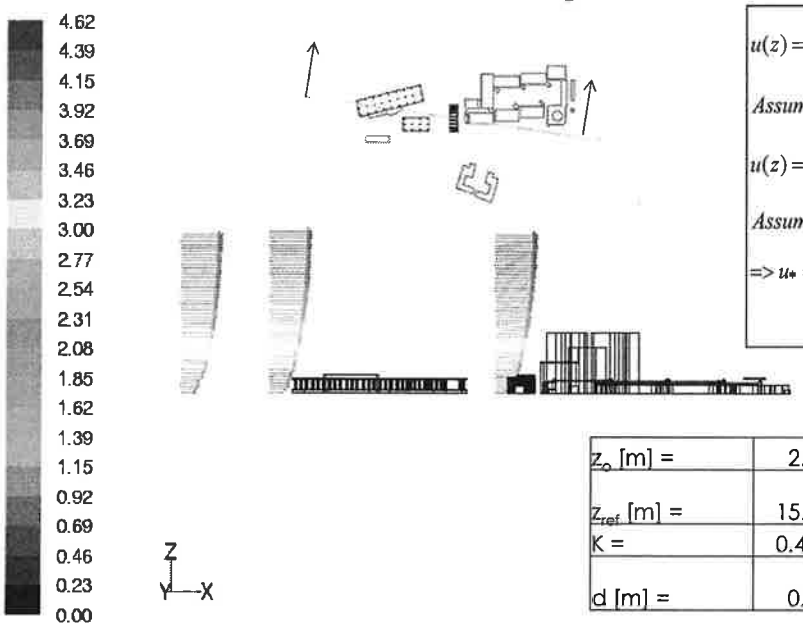
Software	FLUENT 6.3
Mesh Size	Approximately 23 million cells
Mesh Type	Unstructured tetrahedral / prisms
Turbulence Model	Standard *k-ε* turbulence model
Inlet Boundary	Logarithmic boundary layer profile
Reference Speed	North wind: 2.0m/s
Outlet Boundary	Pressure outlet
Domain Height	North wind direction: 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



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

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

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

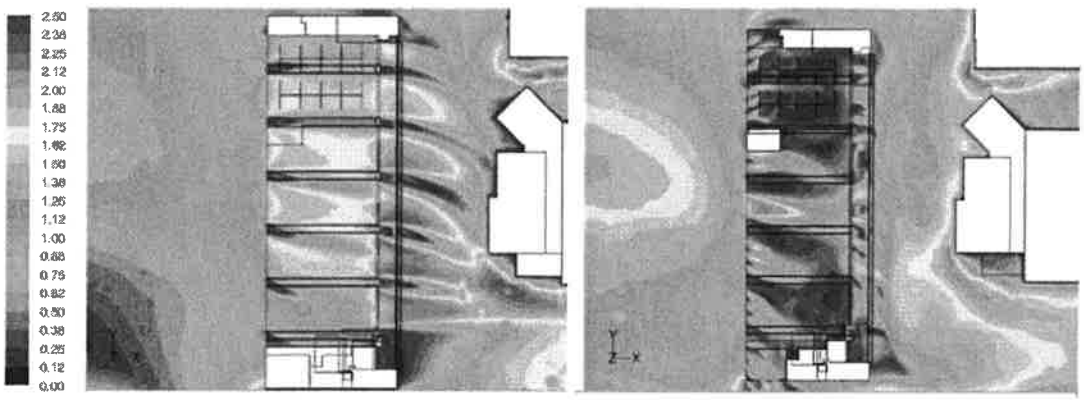
$$Assume:\ d = 0$$

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

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

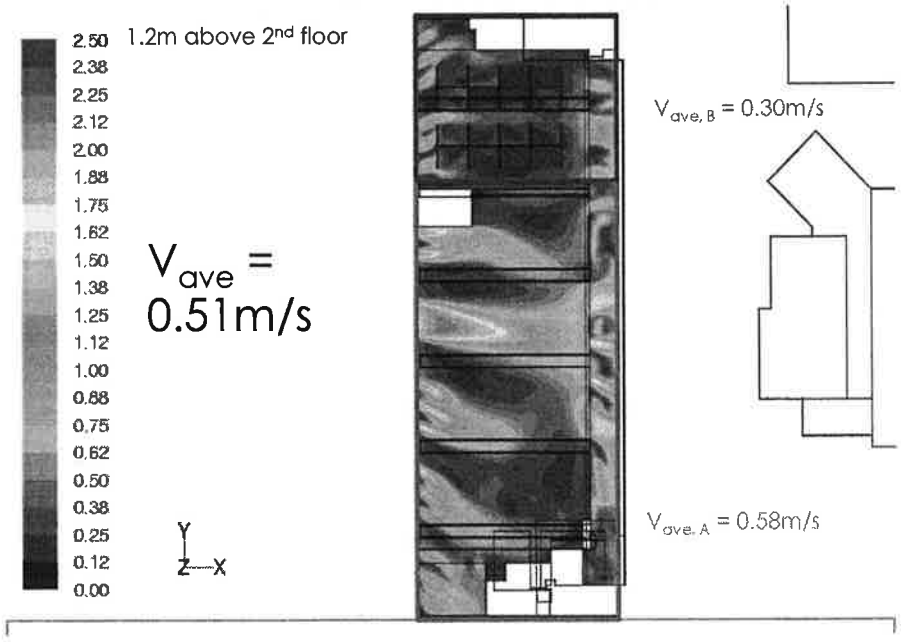


1.2m above ground floor

1.2m above 2nd floor

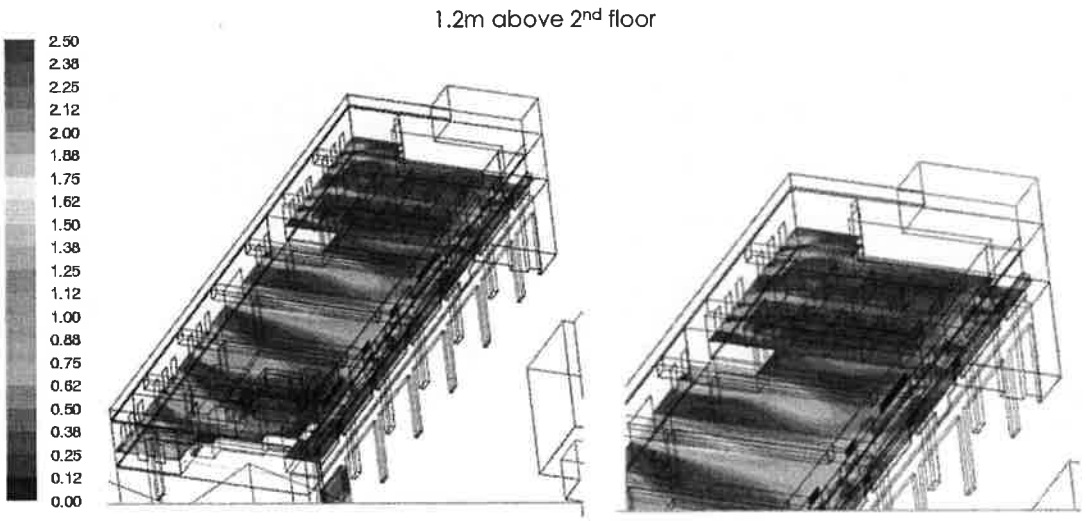
01 Energy Efficiency – Natural Ventilation

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

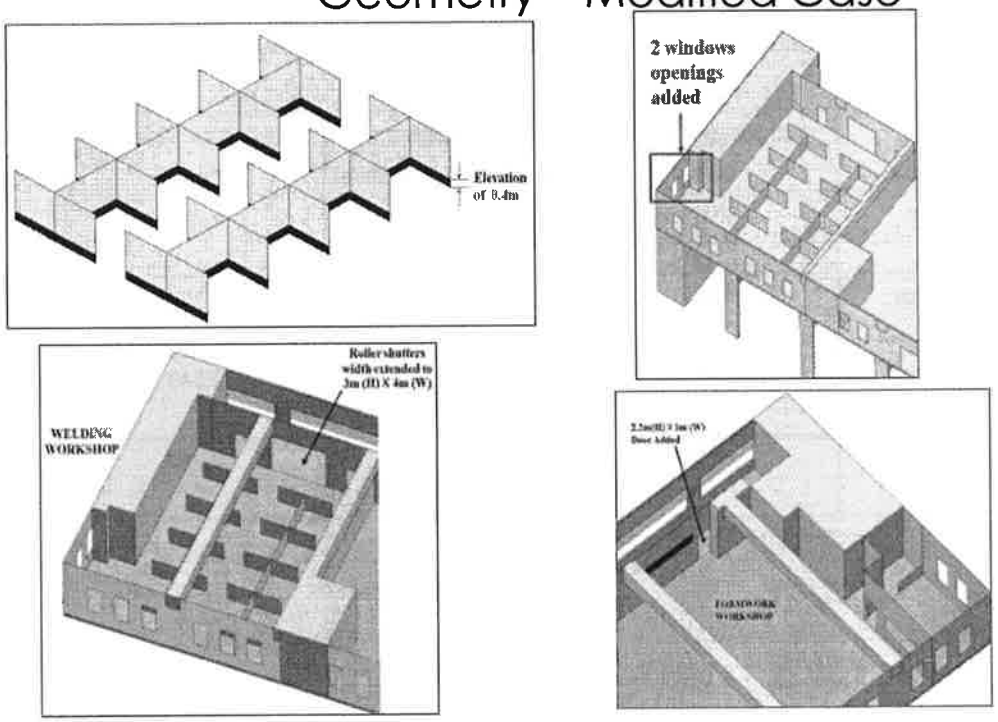
01 Energy Efficiency – Natural Ventilation

- 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

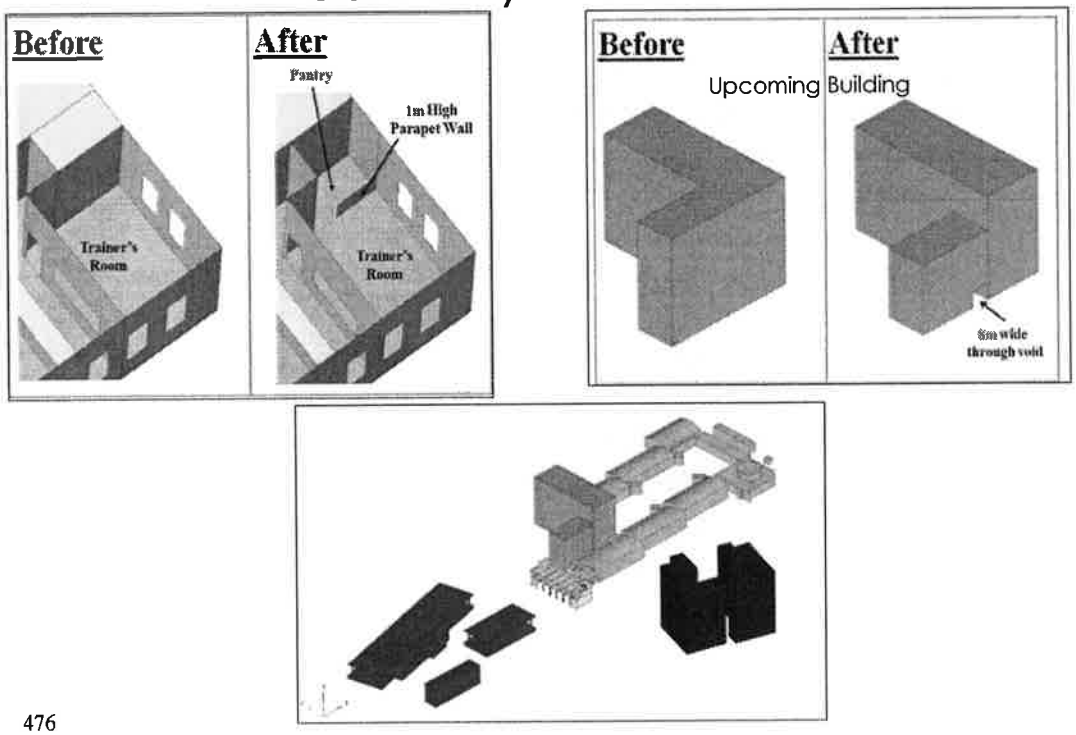
01 Energy Efficiency – Natural Ventilation

Geometry – Modified Case



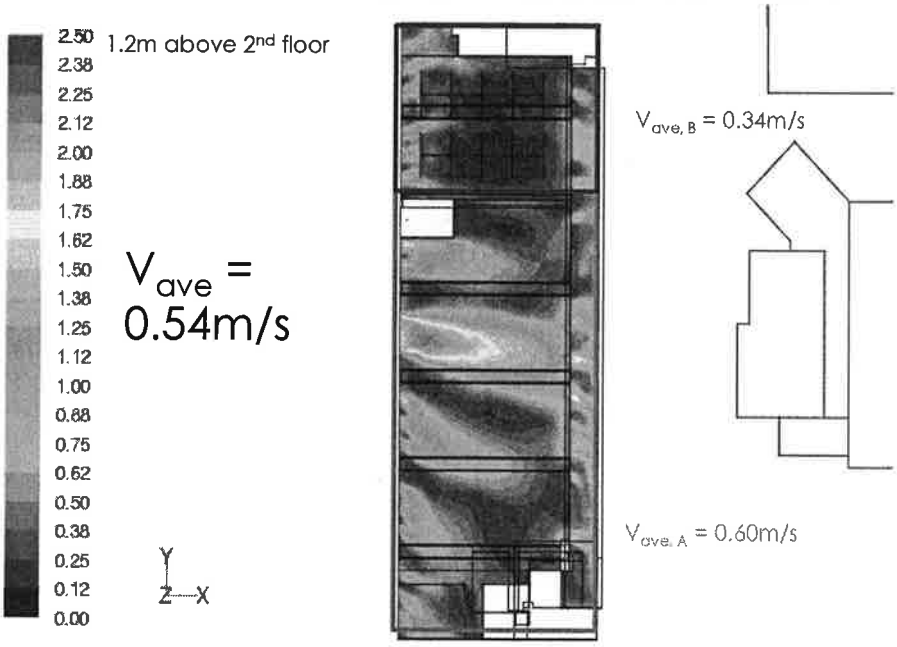
01 Energy Efficiency – Natural Ventilation

Geometry – Modified Case



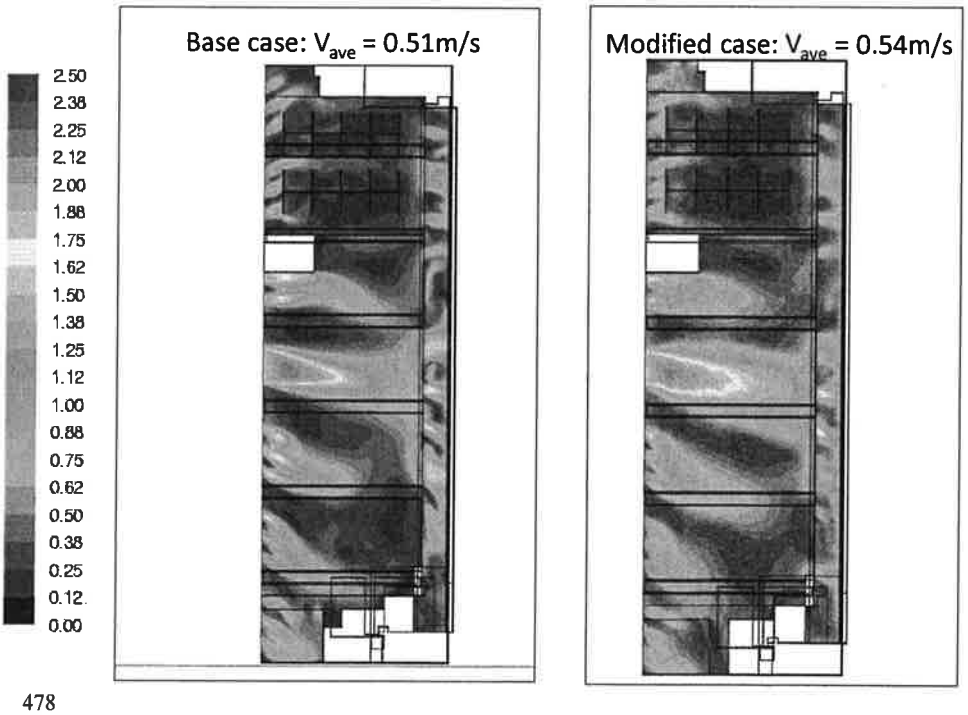
01 Energy Efficiency – Natural Ventilation

Simulation Result – Modified Case



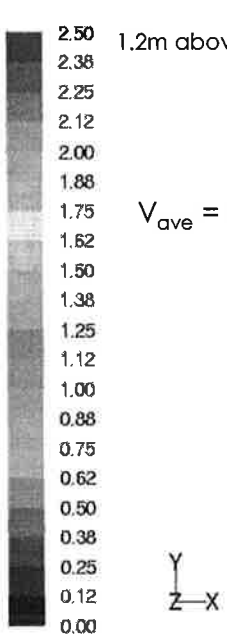
01 Energy Efficiency – Natural Ventilation

Simulation Result – Comparison



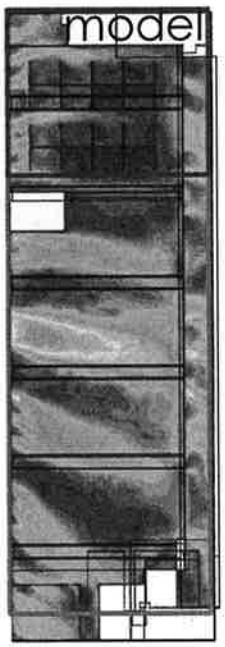
01 Energy Efficiency – Natural Ventilation

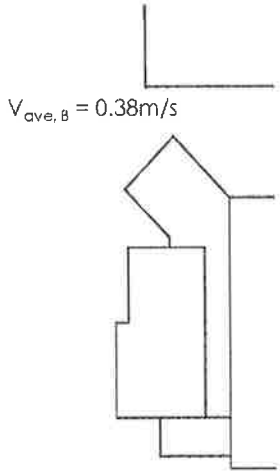
Simulation Result – Modified Case – with more advanced RNG K-Epsilon turbulence



1.2m above 2nd floor

V_{ave} = 0.55m/s





$V_{ave, B}$ = 0.38m/s

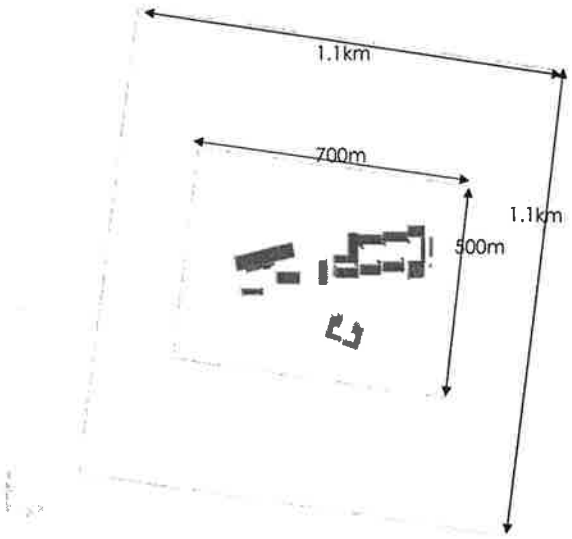
$V_{ave, A}$ = 0.62m/s

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

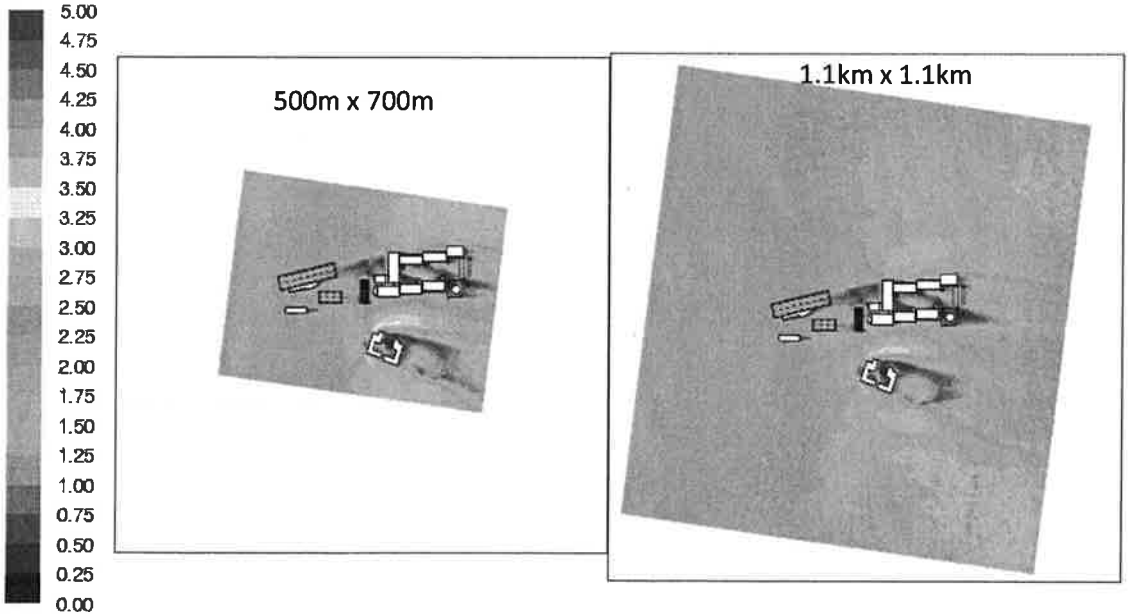


Computational height is also increased from 150m to 300m

23 million to 27 million cells

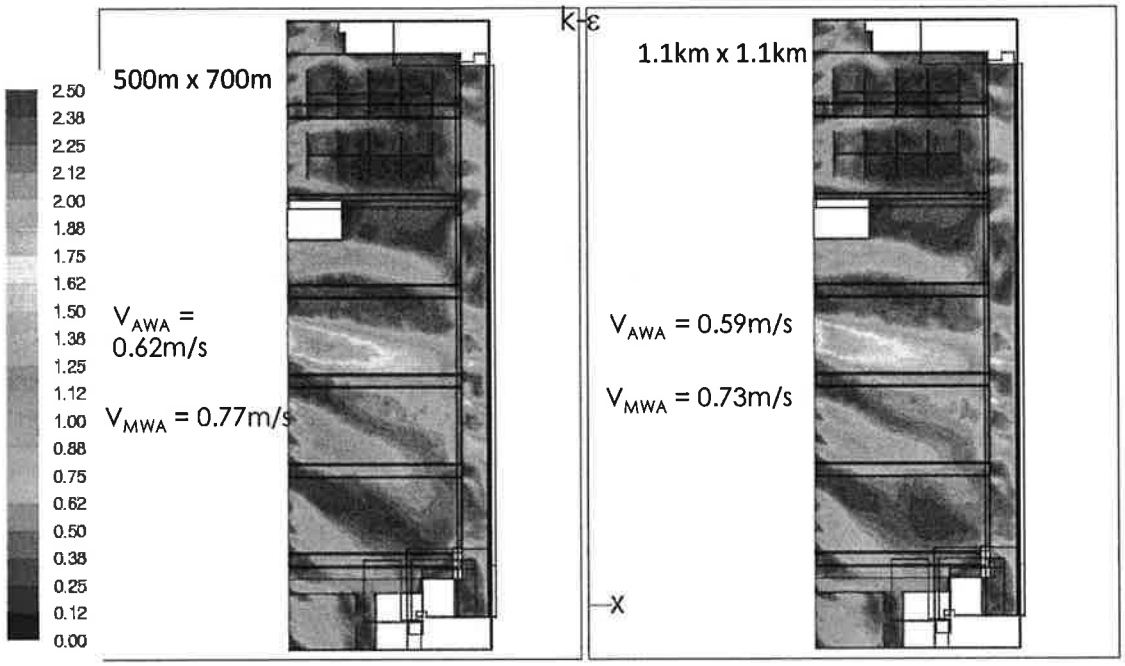
01 Energy Efficiency – Natural Ventilation – Platinum Requirement

Comparison for different domain size on North Wind Results with RNG k-ε



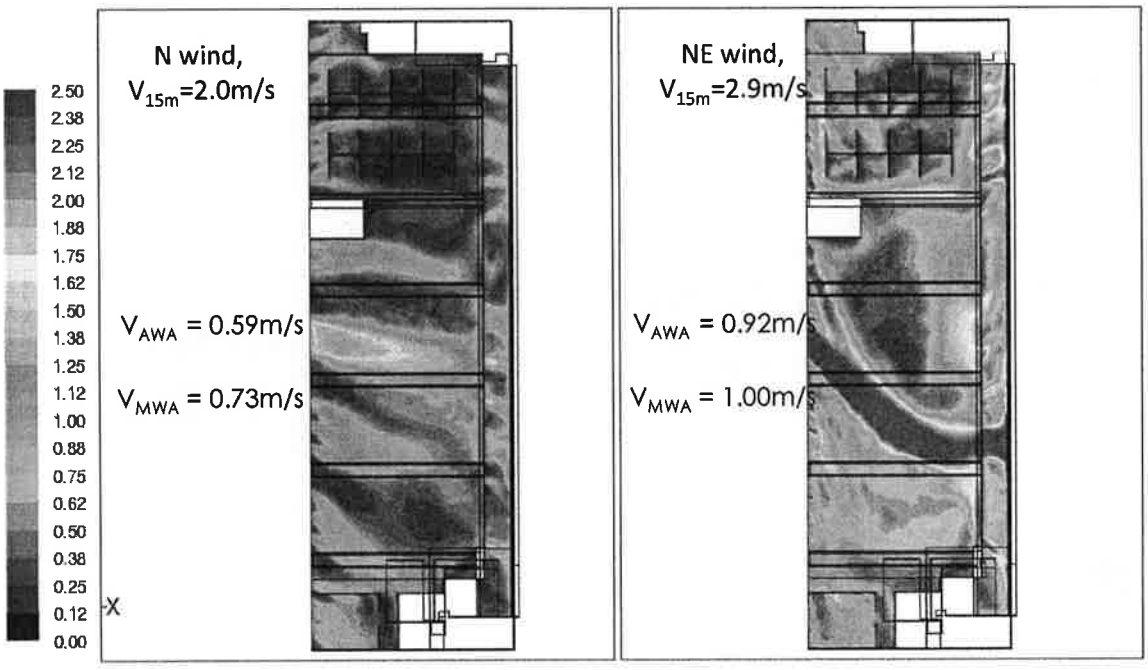
01 Energy Efficiency – Natural Ventilation – Platinum Requirement

Comparison for different domain size on North Wind Results with RNG



01 Energy Efficiency – Natural Ventilation – Platinum Requirement

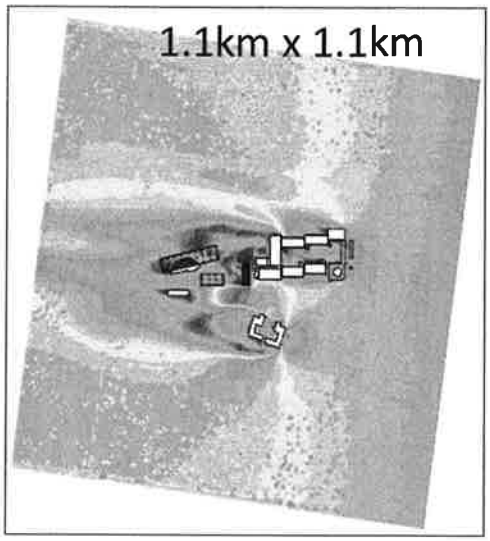
Simulation Result Comparison on North and North East Wind



01 Energy Efficiency – Natural Ventilation – Platinum Requirement

South Wind Results with RNG k-ε



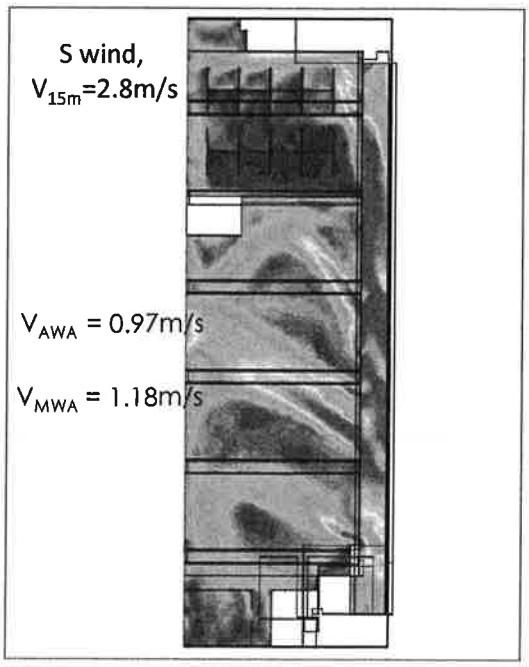


485

01 Energy Efficiency – Natural Ventilation – Platinum Requirement

Simulation Result on South Wind





486

01 Energy Efficiency – Natural Ventilation – Platinum

Requirement Quantification on Natural Ventilation

- The percentage of units achieving good natural ventilation

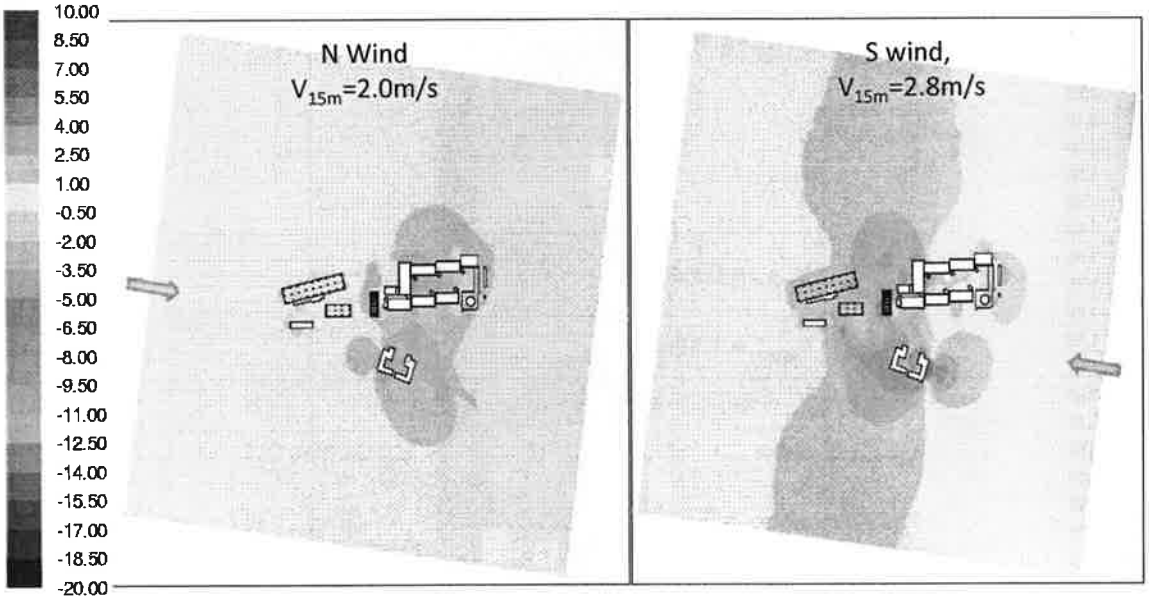
$$\frac{\sum(\text{No. selected units for Each Layout x Area Weighted Average Wind Velocity})}{\text{Total number of selected units x 0.6m/s}} \text{ x } 100\%$$

- 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

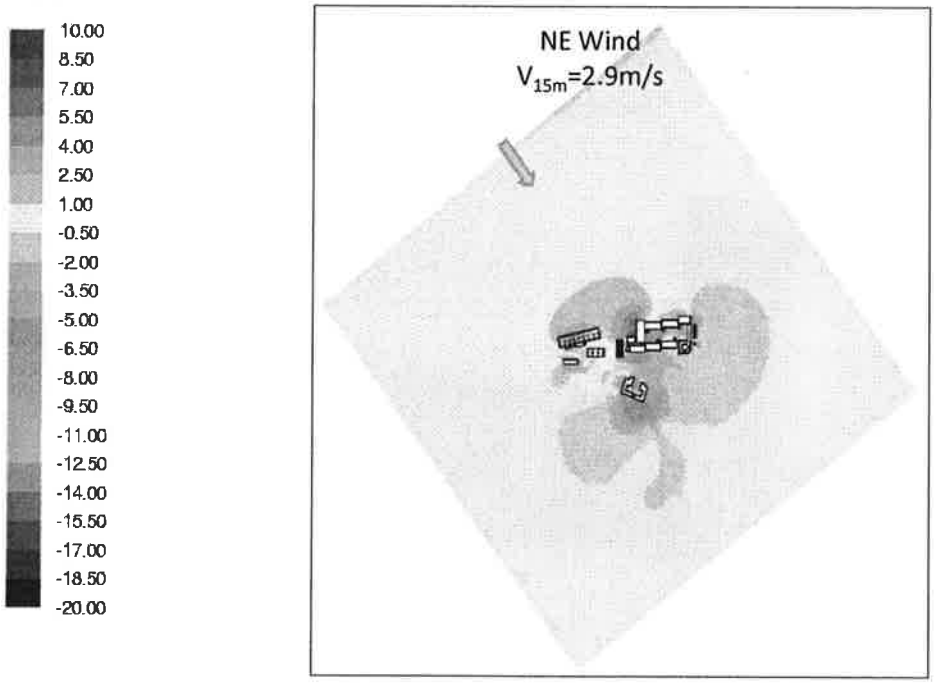
Requirement Pressure Distribution



01 Energy Efficiency – Natural Ventilation – Platinum

Requirement

Pressure Distribution

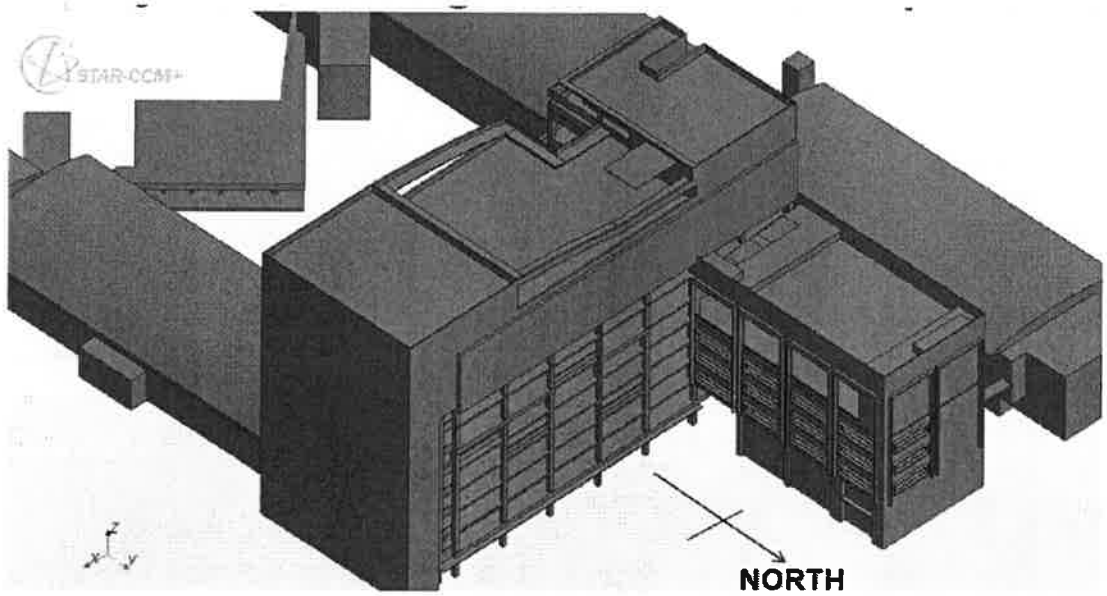


489

490

Case Study 1: BCA Academic Workshop

- Geometrical Description

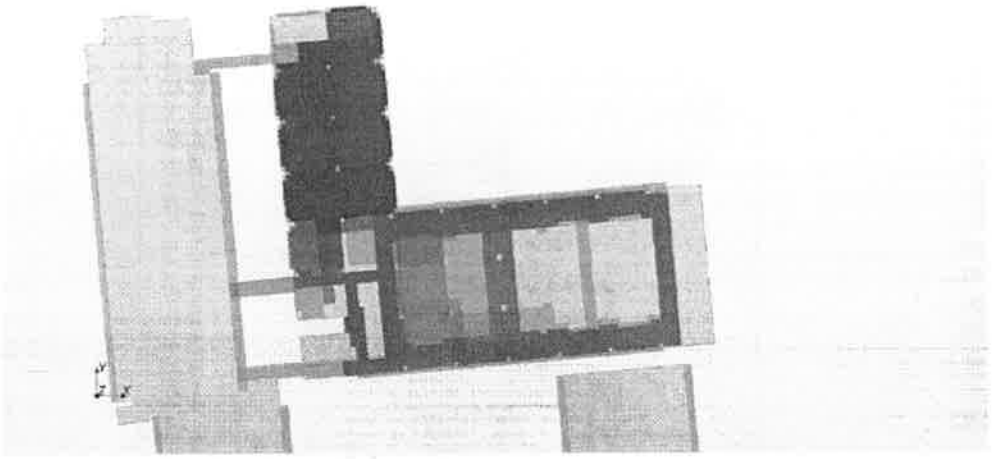


490

Case Study 1: BCA Academic Workshop

- Level 3 – Training Workshop

** Naturally-Ventilated Habitable Space is as shaded in Blue*



Case Study 1: BCA Academic Workshop

- CFD Result – Velocity during North Wind



Case Study 1: BCA Academic Workshop

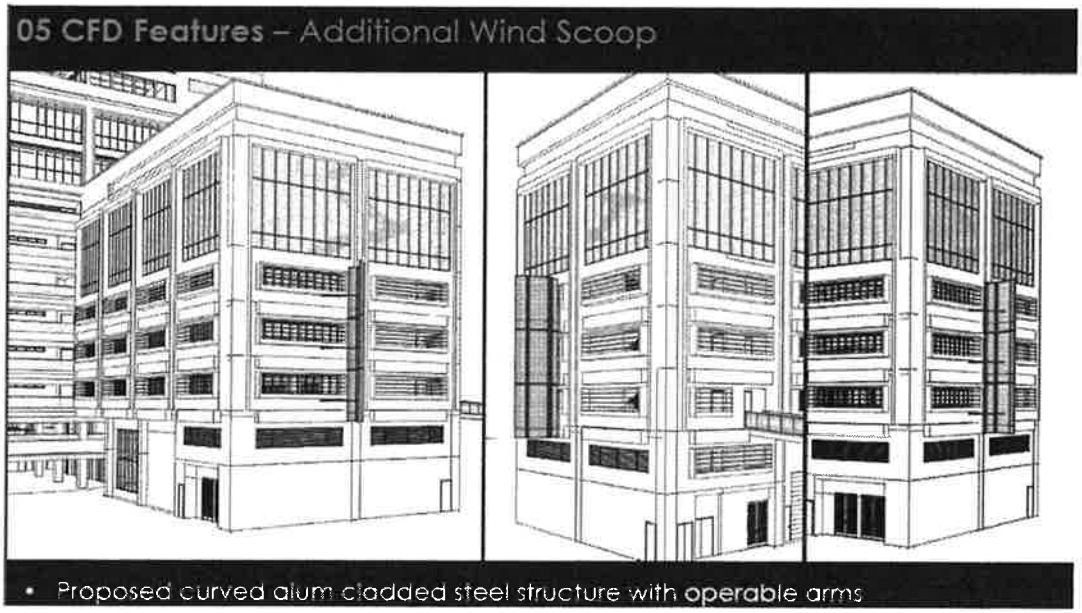
- Summary of CFD result for 4 prevailing wind condition

S.level	Type	Area, A	(NORTH), U_{NORTH}	(SOUTH), U_{SOUTH}	(NORTH-EAST), $U_{NORTHEAST}$	(SOUTH-EAST), $U_{SOUTHEAST}$	Average, $U_{AVERAGE}$	$U_{NORTH} * N$	$U_{NORTH} * A$	$U_{NORTHEAST} * A$	$U_{NORTHEAST} * N$
		[m²]	[m/s]	[m/s]	[m/s]	[m/s]	[m/s]	[m/s * m²]	[m/s * m²]	[m/s * m²]	[m/s]
1	Pavilion	397	0.290	0.576	0.453	0.463	0.446	115.2	228.4	179.9	183.8
3	Workshop	685	0.303	0.190	0.305	0.211	0.302	207.6	129.8	346.1	144.6
4	Workshop	773	0.380	0.202	0.570	0.208	0.335	293.7	148.4	418.1	152.7
5	Workshop	733	0.396	0.204	0.539	0.199	0.333	270.0	149.8	409.5	145.8
6	Basketball Court	936	0.764	0.492	0.665	0.513	0.609	495.8	312.6	422.3	325.9
4	Multi-Purpose/Discussion Area 1	343	1.869	2.173	1.353	2.068	1.759	672.9	760.1	490.0	707.1
6	Discussion Area 2	38	0.764	0.394	0.667	0.338	0.541	27.8	14.3	24.2	12.5
7	Discussion Area 3	40	0.786	0.554	0.906	0.468	0.653	31.1	21.9	31.3	18.4
8	Multi-Purpose/Discussion Area 4	179	0.141	0.559	0.184	1.473	0.589	25.2	100.3	33.0	264.0
	Sum:	3782									

$U_{NORTH\ AVE}$ [m/s]	$U_{SOUTH\ AVE}$ [m/s]	$U_{NORTHEAST\ AVE}$ [m/s]	$U_{SOUTHEAST\ AVE}$ [m/s]
0.529	0.439	0.607	0.517

Case Study 1: BCA Academic Workshop

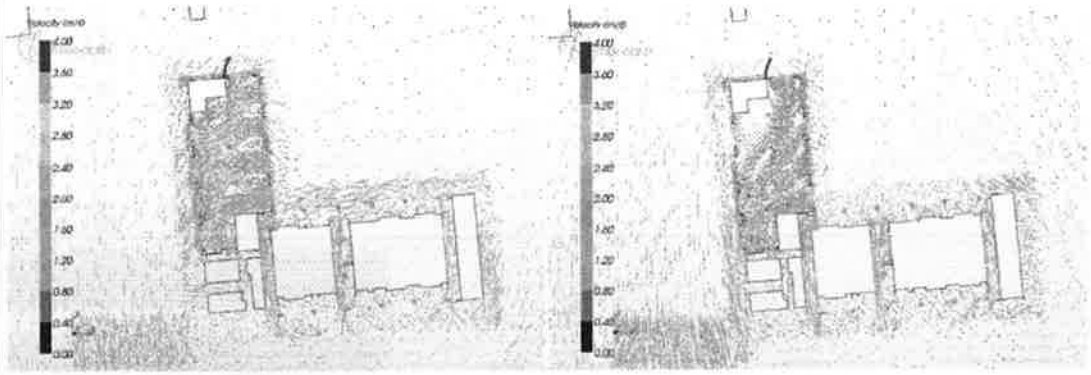
- Effectiveness of Incorporating Wind



Case Study 1: BCA Academic Workshop

Effectiveness of Incorporating Wind

05 CFD Features – Additional Wind Scoop



- A curved profile (base at 90 deg to the north facade and gradually curved to the northeast by 30 degree)
- Min 2 meter length
- Start from Level 3, and extend to the ceiling level

Case Study 1: BCA Academic Workshop

Effectiveness of Incorporating Wind

To further improve the 3 workshops (located on Levels 3, 4 & 5) of the BCAA AWB, the incorporation of a wind deflector is proposed. The proposed location of the wind deflector is illustrated in Figure 1 and Figure 2. The objective is to deflect the wind into the 3 workshops, under NORTH and NORTH-EAST wind directions. The geometric description of the wind deflector is illustrated in Figure 3.

CFD re-simulations, under NORTH and NORTH-EAST wind directions, are re-conducted, taking the incorporation of the wind deflector into account. The NV performance of the 3 workshops, under the analyzed NORTH and NORTH-EAST winds, with and without the incorporation of the wind deflector, is summarized in Table 1.

Table 1. NV Performance of the 3 Workshops, under NORTH and NORTH-EAST Wind Directions

			Without Wind Deflector		With Wind Deflector	
Level	Type	Area, A	(NORTH), U_{NORTH}	(NORTH-EAST), $U_{NORTH\text{-}EAST}$	(NORTH), U_{NORTH}	(NORTH-EAST), $U_{NORTH\text{-}EAST}$
		[m²]	[m/s]	[m/s]	[m/s]	[m/s]
3	Workshop	685	0.303	0.505	0.303	0.510
4	Workshop	733	0.360	0.570	0.361	0.577
5	Workshop	733	0.368	0.559	0.369	0.565

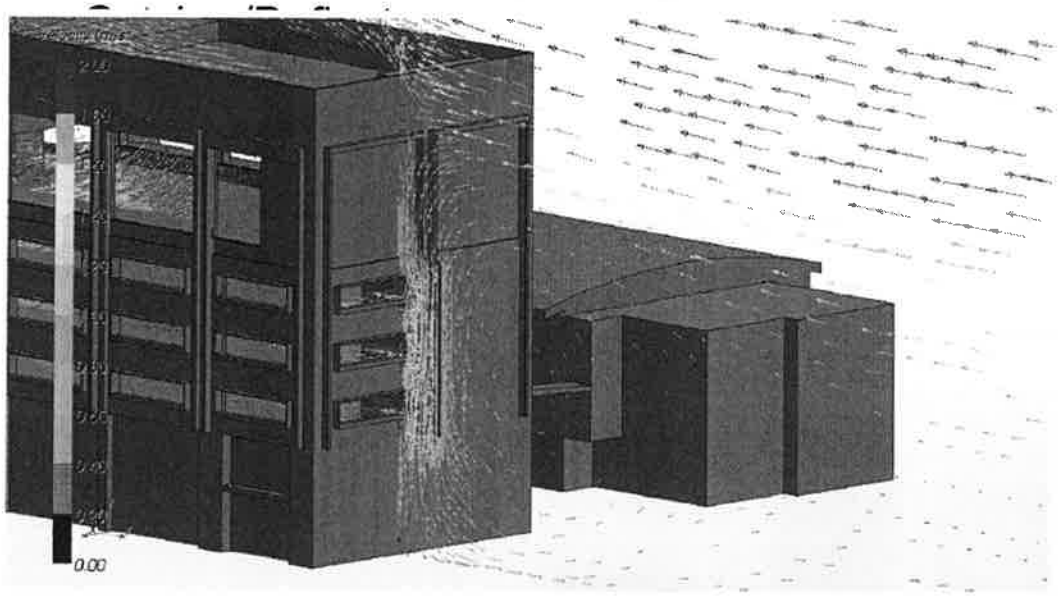
From Table 1, it is found that the improvement in NV performance of the 3 workshops with the incorporation of the wind deflector, is almost negligible.

The primary reason of its ineffectiveness is as follows:

- Under NORTH wind (Refer to Figures 4 through 6), the primary direction of wind flow movement around the wind deflector is vertically downward, thus rendering the wind deflector ineffective.
- Under NORTH-EAST wind (Refer to Figure 7), the wind deflector does deflect certain amount wind into the 3 workshops (through the north window opening), but its amount, when compared to the wind penetration through the west window openings, is so negligible, that the calculated improvement of NV performance is very small (Refer to Figure 8 and Figure 9).

Case Study 1: BCA Academic Workshop

- Effectiveness of Incorporating Wind



Case Study 1: BCA Academic Workshop

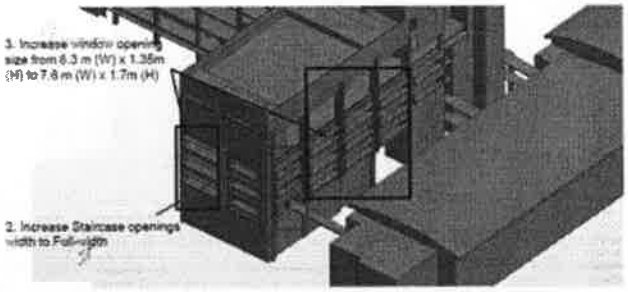
- Improvement of Natural Ventilation Performance of the Workshops, with the Proposed Design Changes

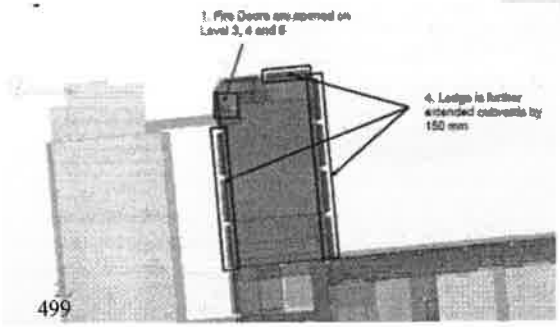
Design Change (Option 2)

1. Open Fire-Doors on Level 3, 4 and 5
2. Increase Staircase openings' width to Full-width
3. Increase window opening size from 6.3 m (W) x 1.35m (H) to 7.6 m (W) x 1.7m (H)
4. Ledge is further extended outwards by 150 mm

Case Study 1: BCA Academic Workshop

- Improvement of Natural Ventilation Performance of the Workshops, with the Proposed Design Changes

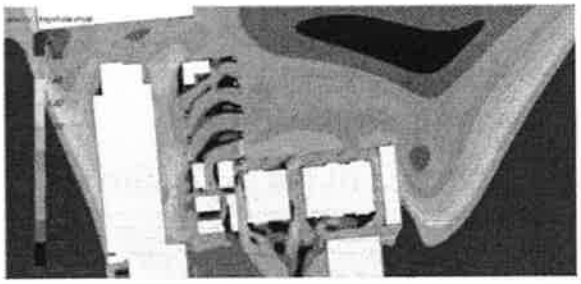




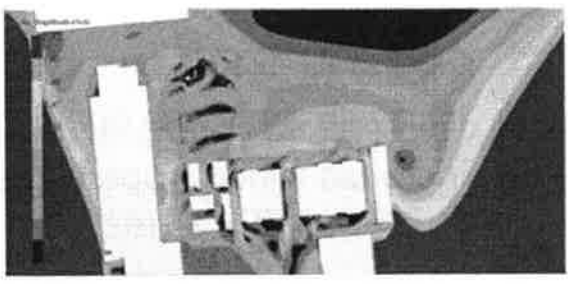
Case Study 1: BCA Academic Workshop

- Improvement of Natural Ventilation Performance of the Workshops, with the Proposed Design Changes

Original Design



Option 2



Case Study 1: BCA Academic Workshop

- Improvement of Natural Ventilation Performance of the Workshops, with the Proposed Design

Cl

Summary of Results

Level	Type	Area, A	Original Design (NORTH), U_{NORTH}	Original Design (NORTH-EAST), $U_{NORTH-EAST}$	Option 2 (NORTH), U_{NORTH}	Option 2 (NORTH-EAST), $U_{NORTH-EAST}$
		[m^2]	[m/s]	[m/s]	[m/s]	[m/s]
3	Workshop	685	0.303	0.505	0.328	0.567
4	Workshop	733	0.360	0.570	0.442	0.664
5	Workshop	733	0.368	0.559	0.428	0.654

Level	Type	Area, A	Original Design (SOUTH), U_{SOUTH}	Original Design (SOUTH-EAST), $U_{SOUTH-EAST}$	Option 2 (SOUTH), U_{SOUTH}	Option 2 (SOUTH-EAST), $U_{SOUTH-EAST}$
		[m^2]	[m/s]	[m/s]	[m/s]	[m/s]
3	Workshop	685	0.188	0.211	0.201	0.244
4	Workshop	733	0.202	0.208	0.226	0.252
5	Workshop	733	0.204	0.199	0.238	0.243

Level	Type	Area, A	Original Design Average U_{AVE}	Option 2 Average U_{AVE}
		[m^2]	[m/s]	[m/s]
3	Workshop	685	0.302	0.335 (Up 11.0%)
4	Workshop	733	0.335	0.396 (Up 18.3%)
5	Workshop	733	0.333	0.391 (Up 17.6%)

501

Day 4 (9:45am – 10:30am)

- CFD Case Studies for Urban and Built Environment – Researcher Perspective

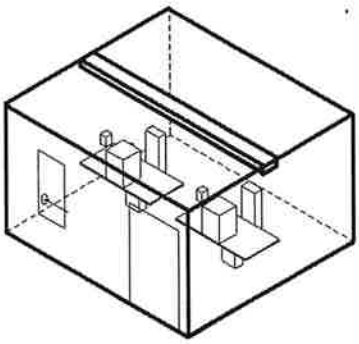
Case Studies on Different Air Distribution Systems

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.	

503

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).

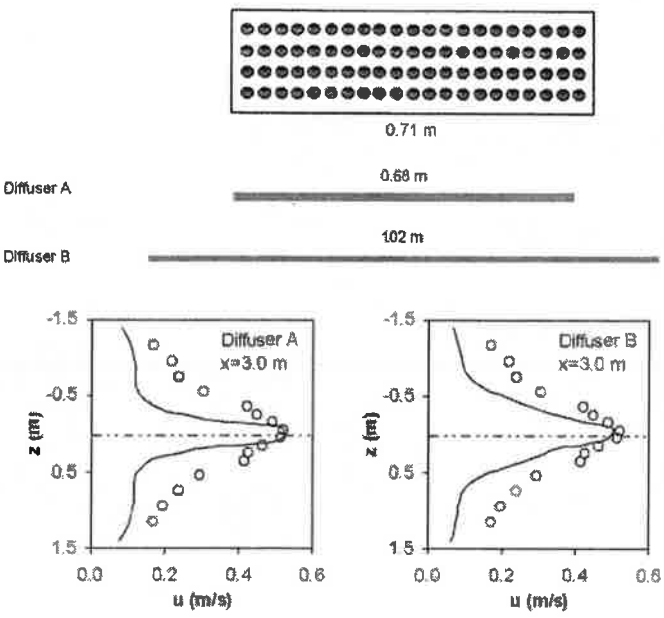
504

Case Study, Mixing Ventilation with Wall Mounted ATD



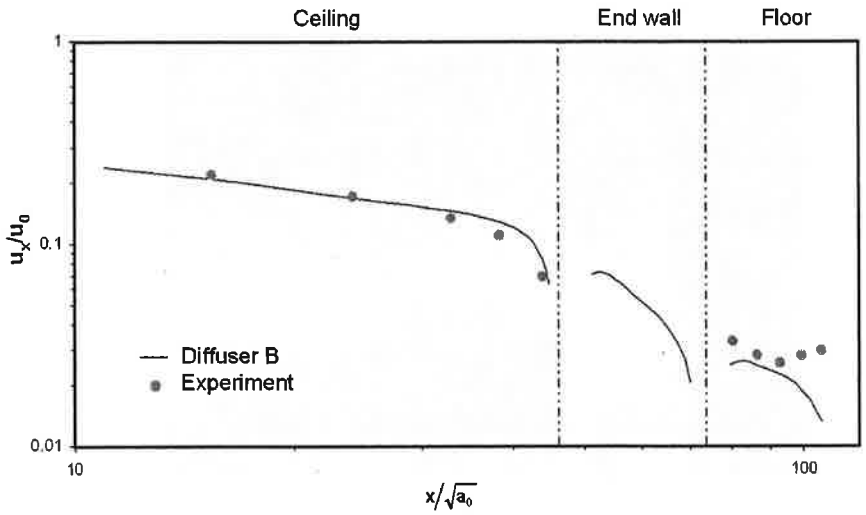
505

Simulation of the Diffuser



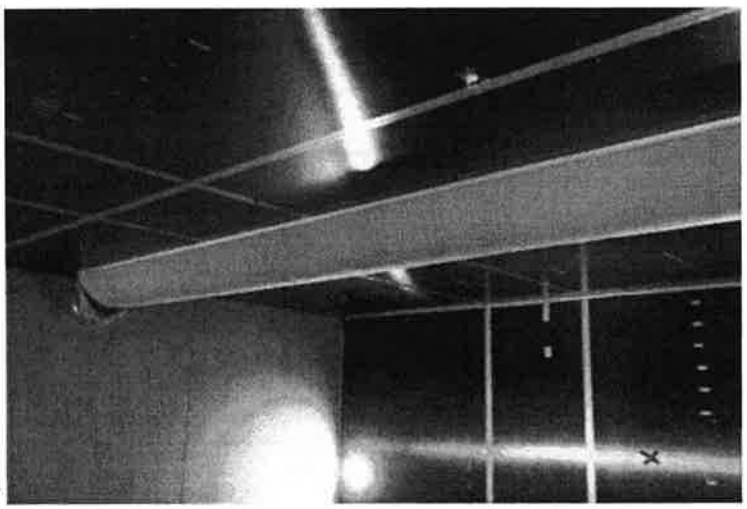
506

Mixing Ventilation with End Wall Mounted Diffuser



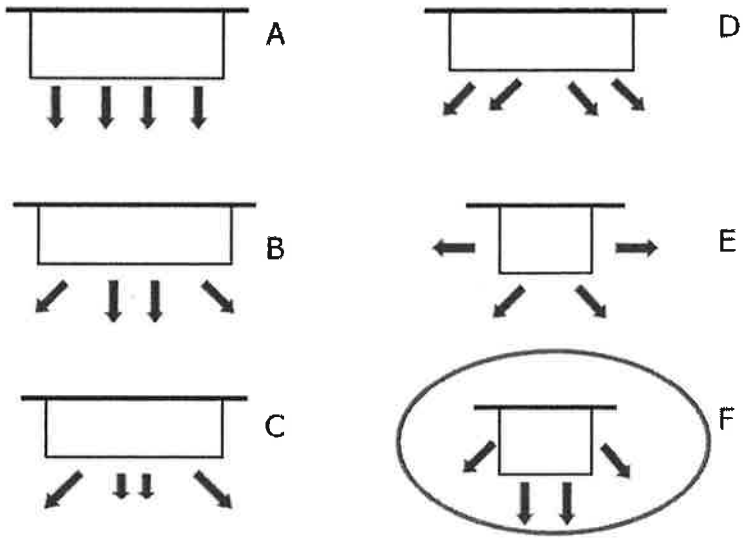
507

Case Study, Vertical Ventilation



508

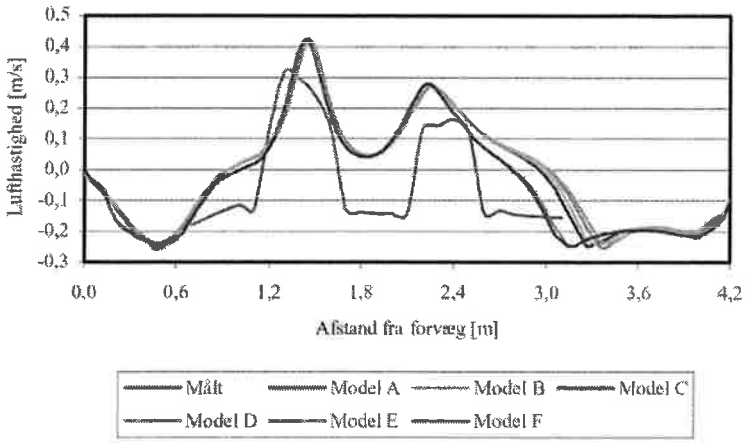
Vertical Ventilation, Diffuser



509

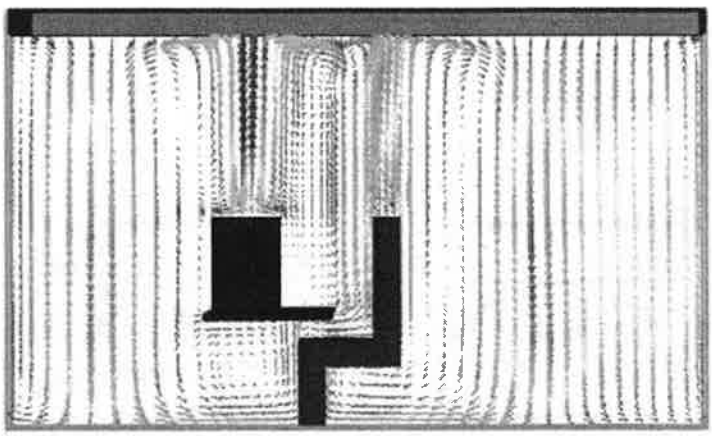
Vertical Ventilation, Diffuser

1st order steady state equations, *k-ε* turbulence model, 300,000 cells

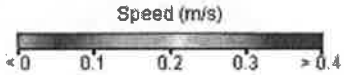


510

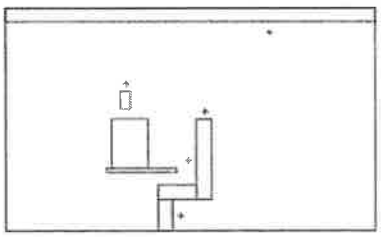
Vertical Ventilation



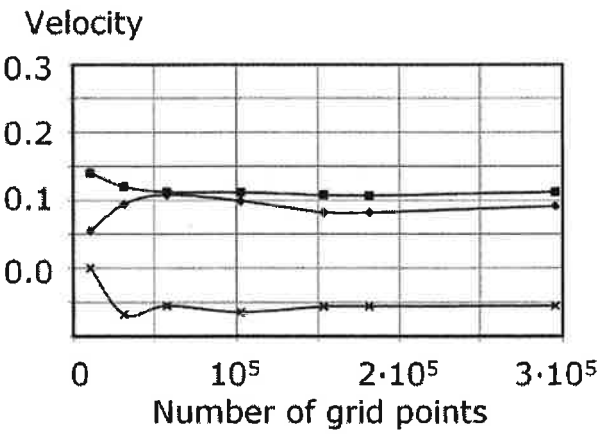
BC: Diffuser F



Vertical Ventilation, Quality Control

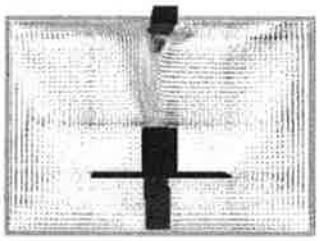


Monitoring points

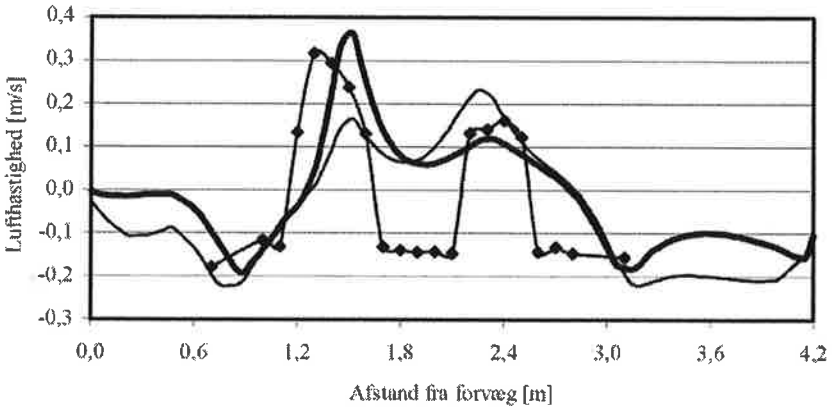


Velocity in y-direction

Vertical Ventilation

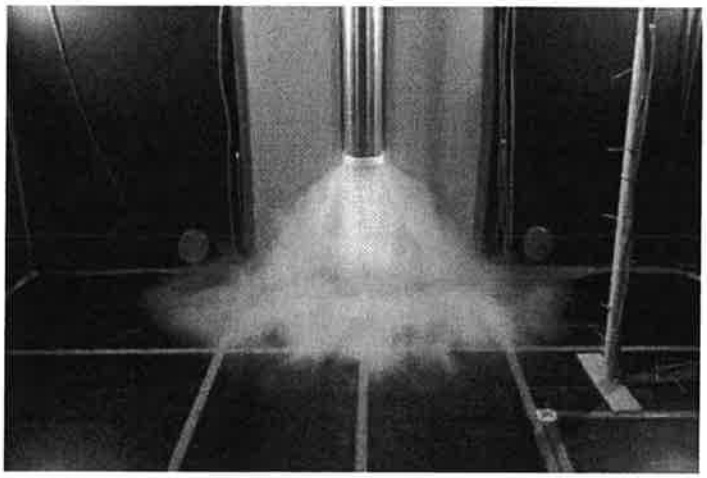


Predictions in the **whole** room
n = 5 h^{-1}
218.400 grid points



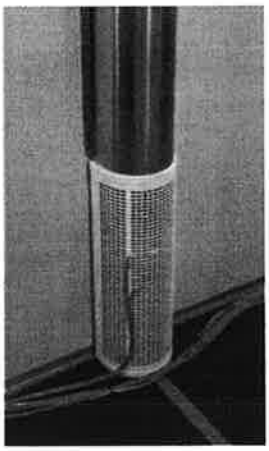
513

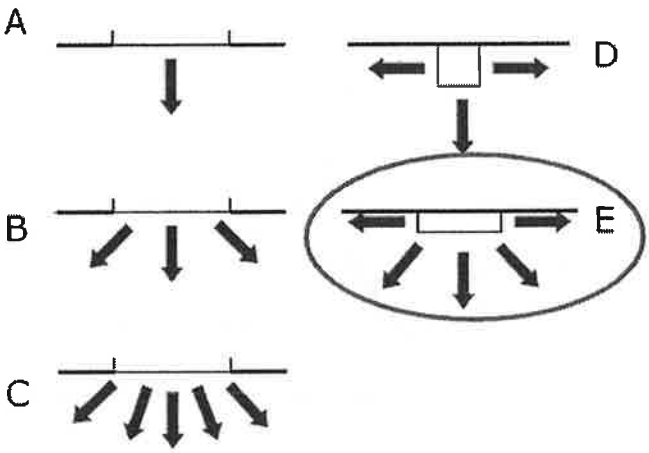
Case Study, Displacement Ventilation



514

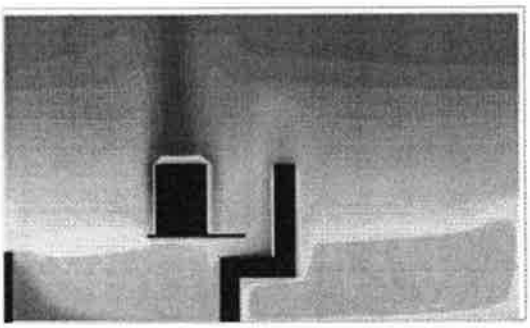
Diffuser for Displacement Ventilation

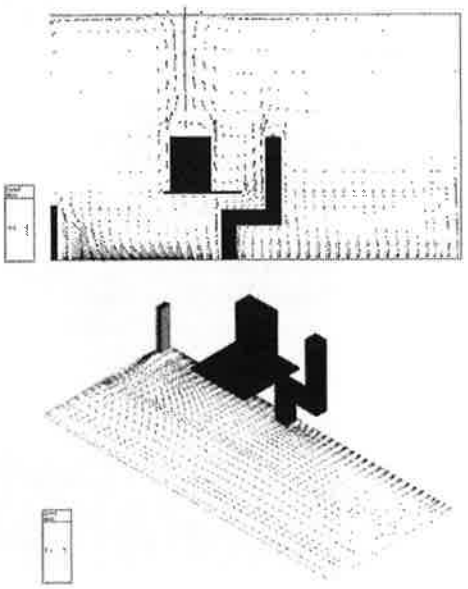




515

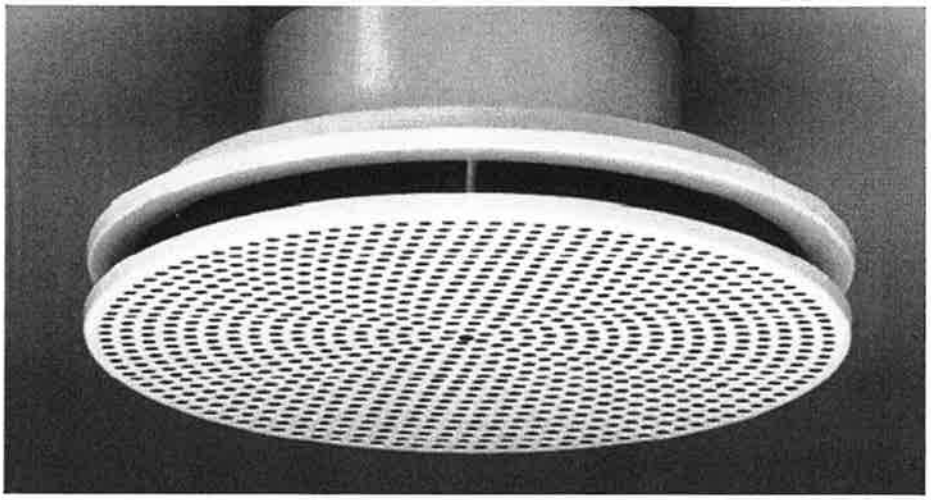
Displacement Ventilation





516

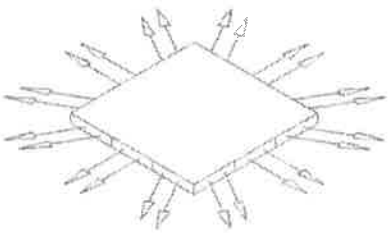
Case Study, Mixing Ventilation with Ceiling Mounted ATD

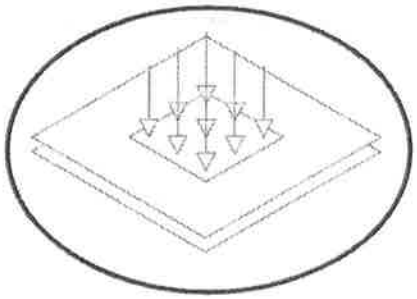


517

Mixing Ventilation with Ceiling Diffuser, Diffuser Models





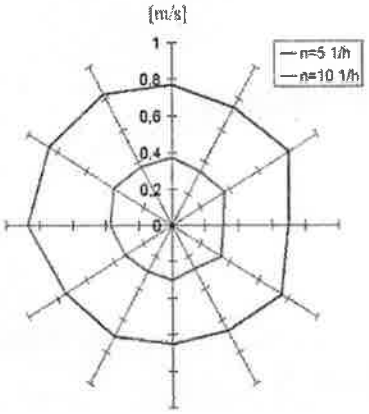


- Fixed-flow diffuser
- Diffuser with horizontal surface

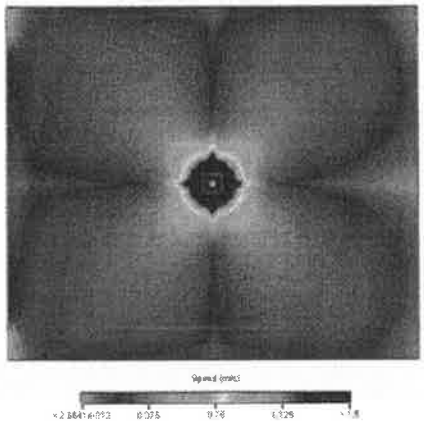
518

Diffuser Model

Measurements



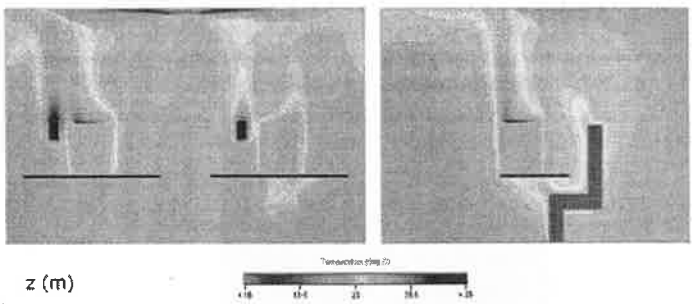
Diffuser with horizontal surface

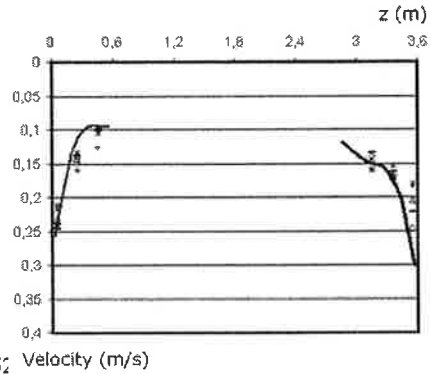


519

CFD Simulations

Temperatures



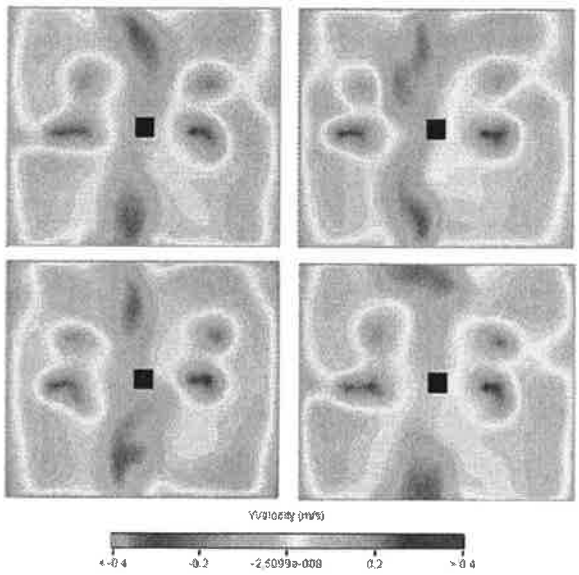


52 Velocity (m/s)

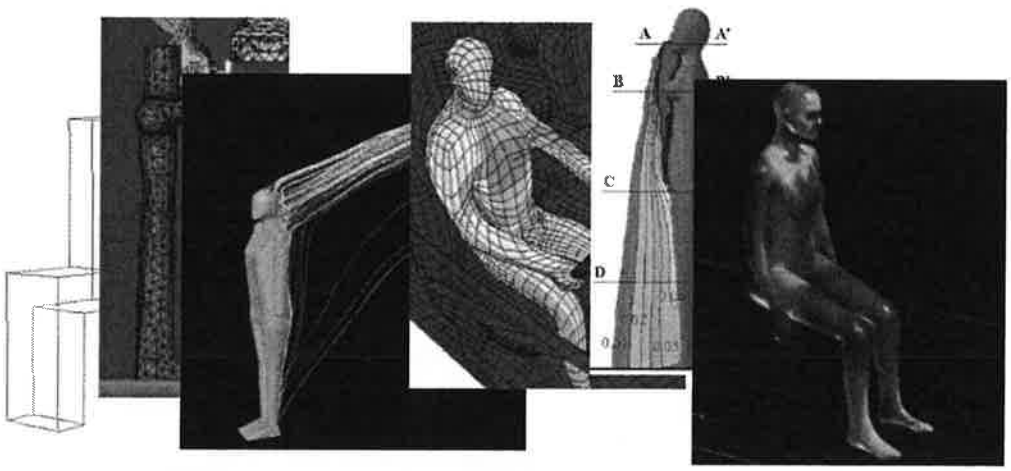
Velocities at the the height of 1.80 m close to the side walls

Simulation of Unsteady Flow

(Time Dependent Equations)



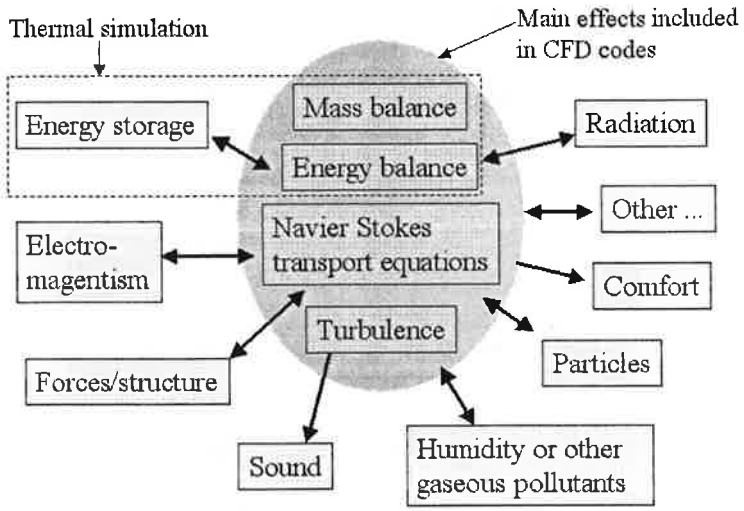
CFD Manikins



Benchmark test for different CFD manikins:

www.cfd-benchmarks.com

CFD Combined with Other Prediction Models



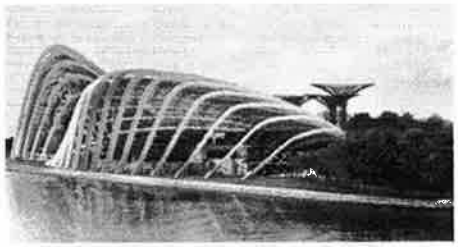
523

524

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







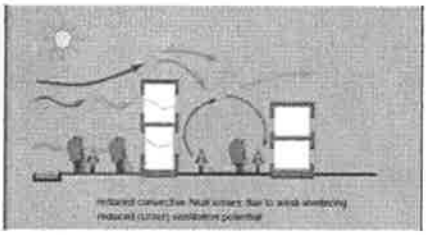
524

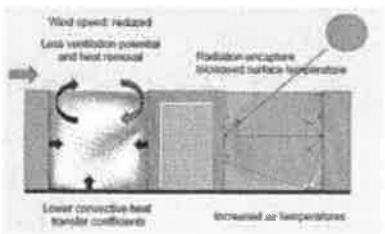
Computer Modelling of Indoor Environment

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)

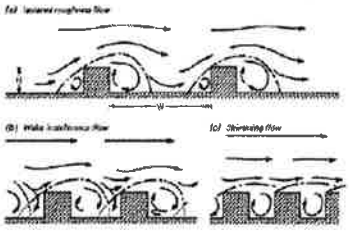
CFD in Urban Physics

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)

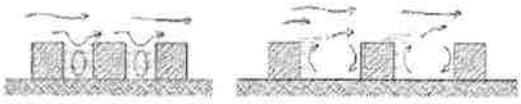
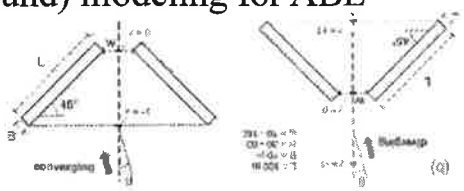


Figure 2.7 Effect of building spacing (street width) to building height on traffic pollution dispersion.

(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

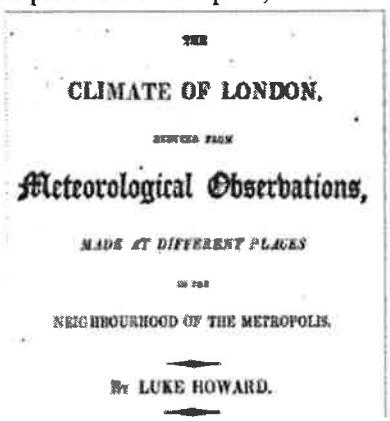


Richardson'(1922)' – Turbulence Energy Cascading

'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 OR

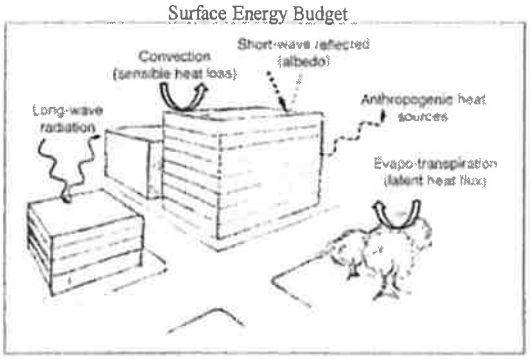
NEIGHBOURHOOD OF THE METROPOLIS.

By LUKE HOWARD.

Lisa Gartland, Heat islands: understanding and mitigating heat in urban areas, London : Earthscan, 2008

UHI – Introduction

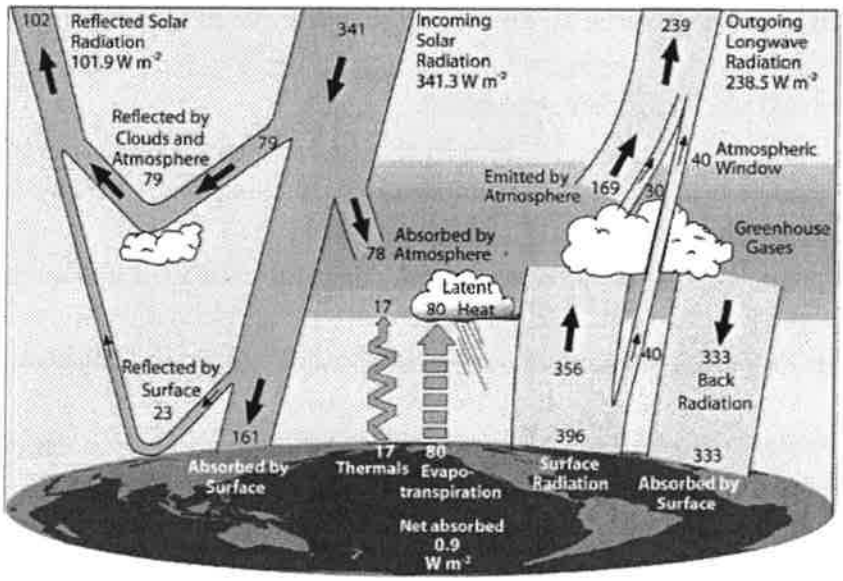
- Heat Island : characterized by consistent rise in urban atmosphere temperature, an increase in air pollutants (e.g. NO_X & SO_X) and decreased in RH
- Numerical approaches to analyze UHI
  - Mechanical model – emphasize flow characteristics
  - Energy balance model – concerned with the difference between 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

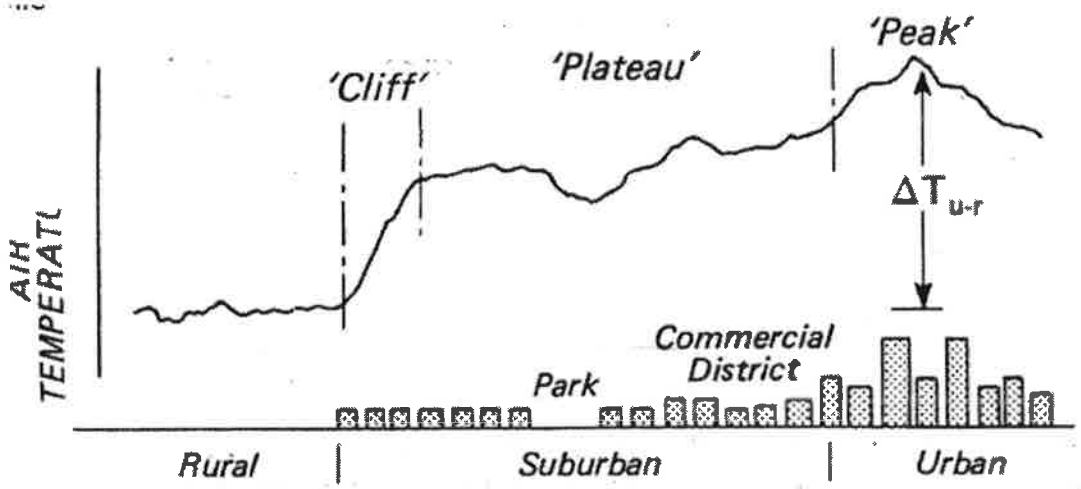
UHI – Surface Energy Budget

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



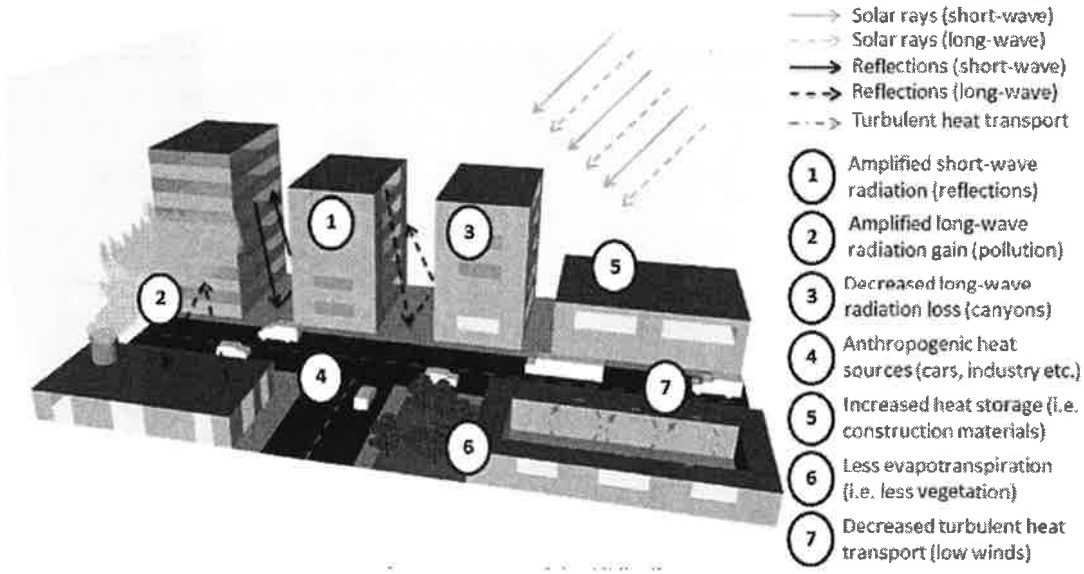
T. R. Oke, "The energetic basis of the urban heat island," *Quarterly Journal of the Royal Meteorological Society, vol 108, pp. 1-24, 1982.*

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

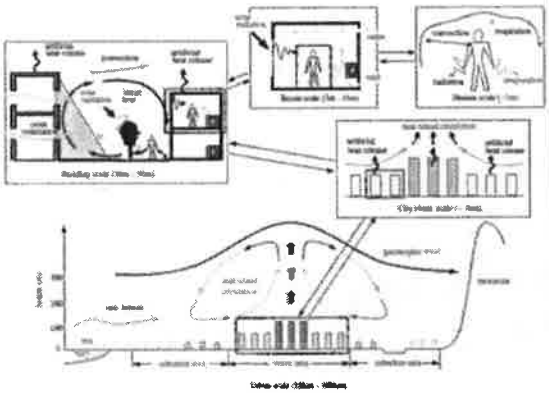
- EFM (Environmental Fluid Mechanics) – Scientific study of naturally occurring fluid flows of air and water on our planet earth, especially those affecting environmental quality of air and water
- Only concern with two bulk of fluids, air and water, under relatively narrow range of ambient pressure 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

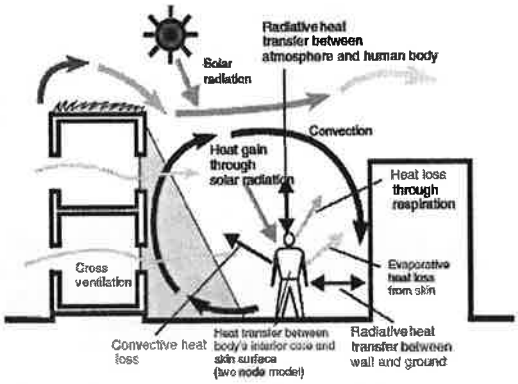
- Challenging issues – multi-scale & multi-physics

Multi-scale



Interaction of phenomena at various scales from indoor to urban

Multi-physics



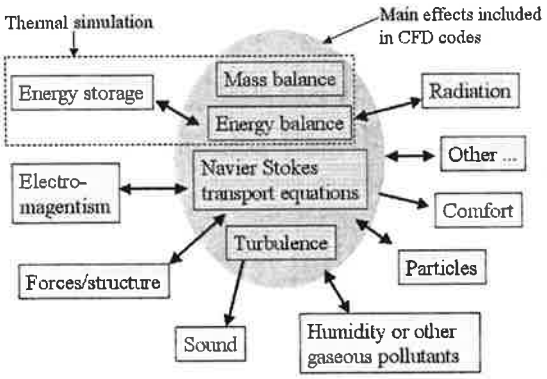
Complicated mechanism of heat exchange between the human body and the surrounding outdoor environment

Picture courtesy of Prof. Shuzo Murakami

Research Challenges

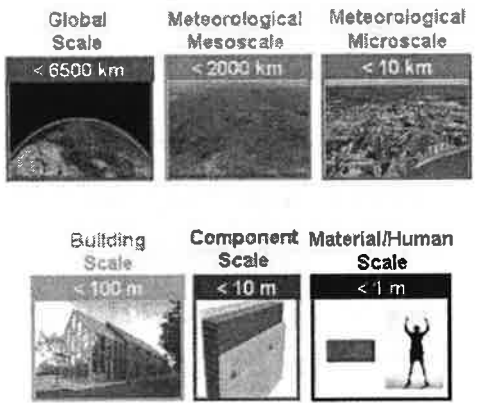
- Challenging issues – multi disciplinary

Multi-disciplinary – modeling methodology



CFD Combined with Other Prediction Models

Multi-disciplinary – urban physics

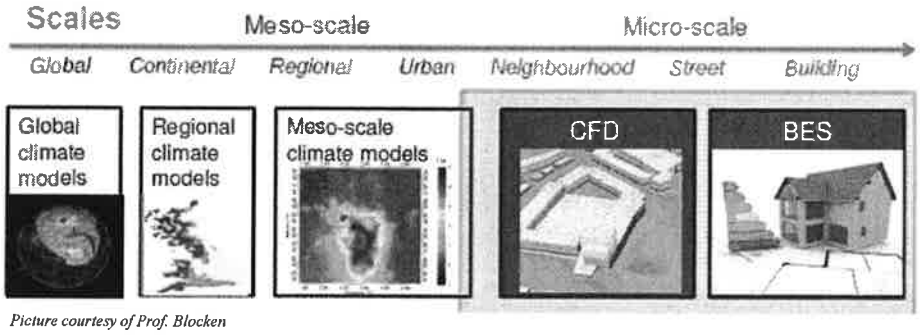


From macro-scale meteorology to human thermo physiology

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.



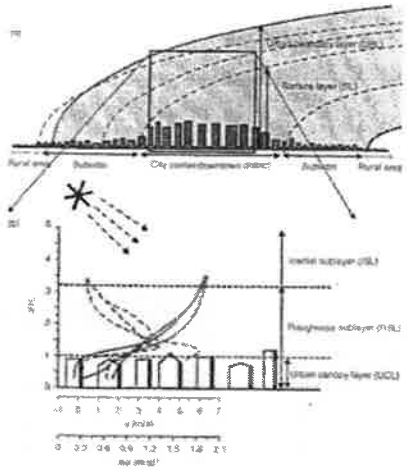
Picture courtesy of Prof. Blocken

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

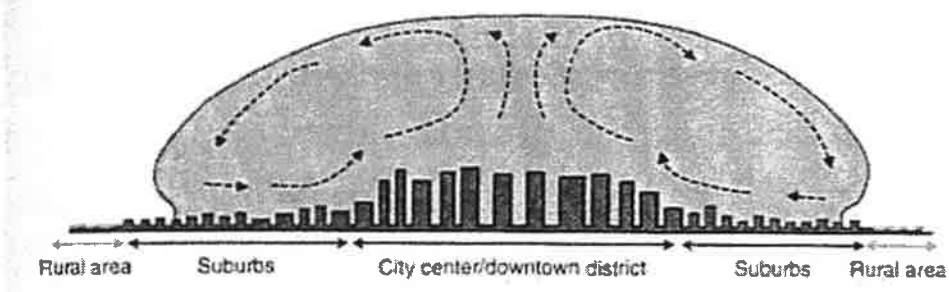
- Layered structure of ABL over urban areas for moderately strong wind speeds (>3m/s)



539 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

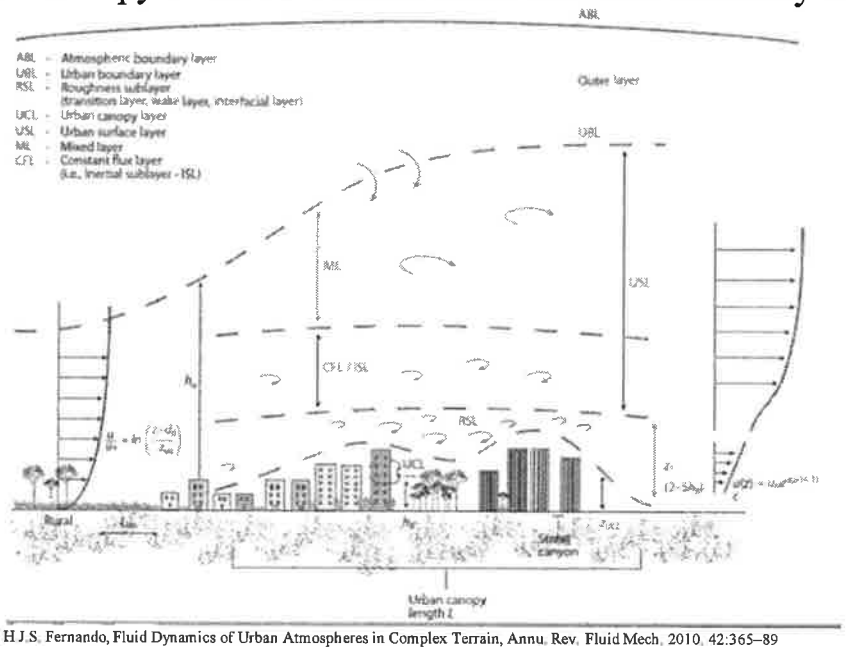
- Development of thermal driven urban circulation for weak wind (<3m/s)



540 Reference : 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

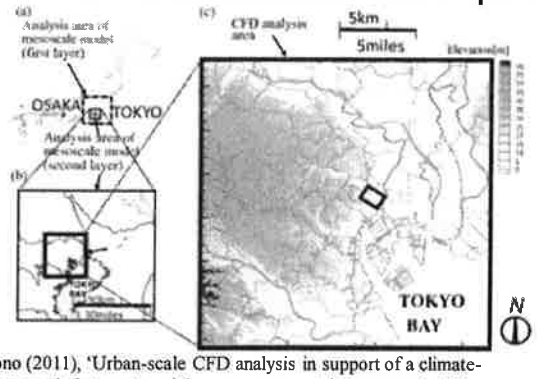
- Urban canopy effects – various zones of urban boundary layer



541 Reference : 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 [542]

- 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

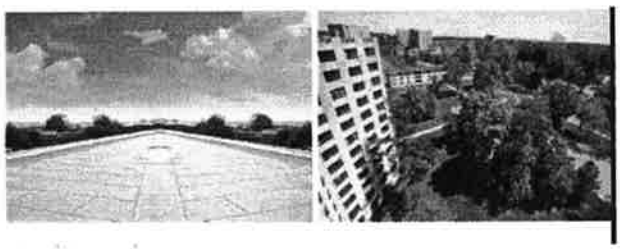


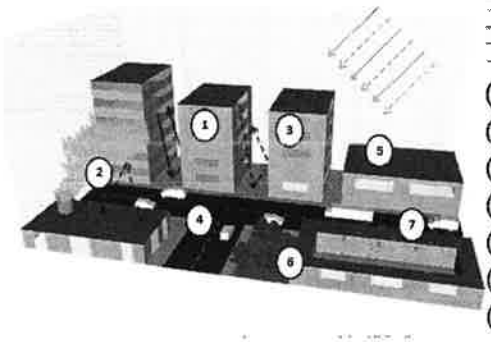
CFD analysis domain is 33km x 33km (includes the 33 wards of Tokyo, 621 km^2)

542 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

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)







Moonen, P., Defraeye, T., Dorer, V., Blocken, B., Carmeliet, J., (2012). Urban Physics: Effect of the microclimate on comfort, health and energy demand.

543

Urban Climatic Mapping Studies for Marina Bay, Singapore

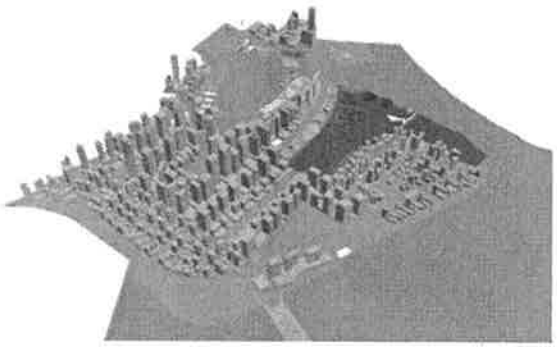


Urban Heat Island profile in Singapore

- 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 Climatic Mapping Studies for Marina Bay, Singapore

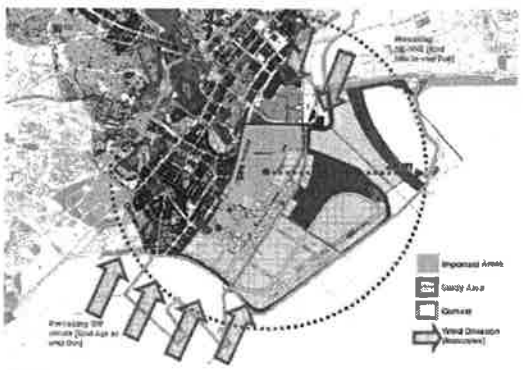
- Objectives
  a. To provide inputs for land use planning that safeguard important environmental conditions from intensive urban development
  b. To reduce heat island effects and energy consumption and improve ventilation in high density environments
  c. To create more liveable and conducive life, work and play spaces in the city



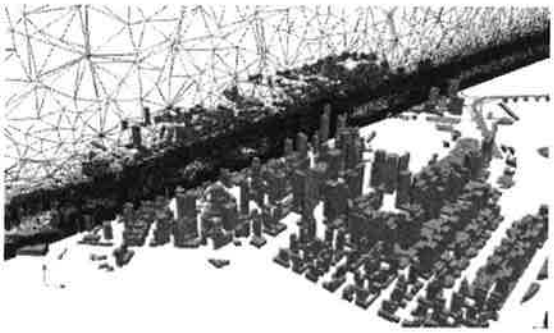
Urban Climatic Mapping Studies for Marina Bay, Singapore

- 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**)

+

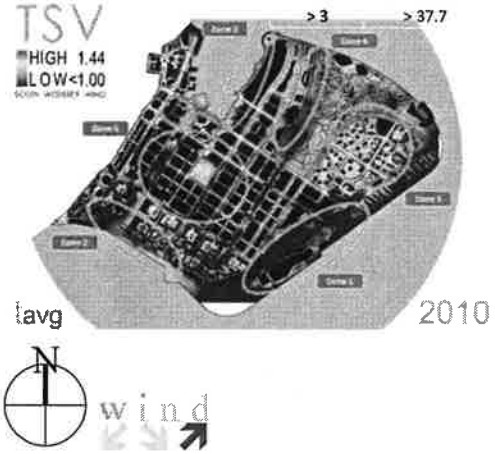


Wind Map (**CFD**)

549

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)



550

Geometrical comparison between Scenario 1 and Scenario 2

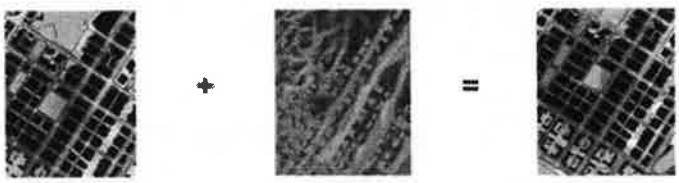
Urban Climatic Mapping Studies for Marina Bay, Singapore

- 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.
			

CFD Studies of Climate Change Impacts

- Singapore, from Garden City to City in Garden
  - Highly urbanized city: Population: 5.2Mil, Land Area: 714 km^2
  - Environmental Risks, including *Climate Change, Severe Weather*
  - High-rise Buildings ▶ Urban Wind Channels ▶ *Heat Ventilation; Biodiversity*



CFD Studies of Climate Change Impacts

- Methodology for Geospatial Risk Assessment of Wind Channels
  - Integrated GIS-based, coupled multi-scales models & data mining:
    - Weather/climate models (100 ~ 1km resolution)
    - Urban CFD modeling (1 ~ 10 meter resolution)
  - Case studies:
    - Northeast & Southwest Monsoons
    - Climate Change Effect
    - Severe Weather Impacts

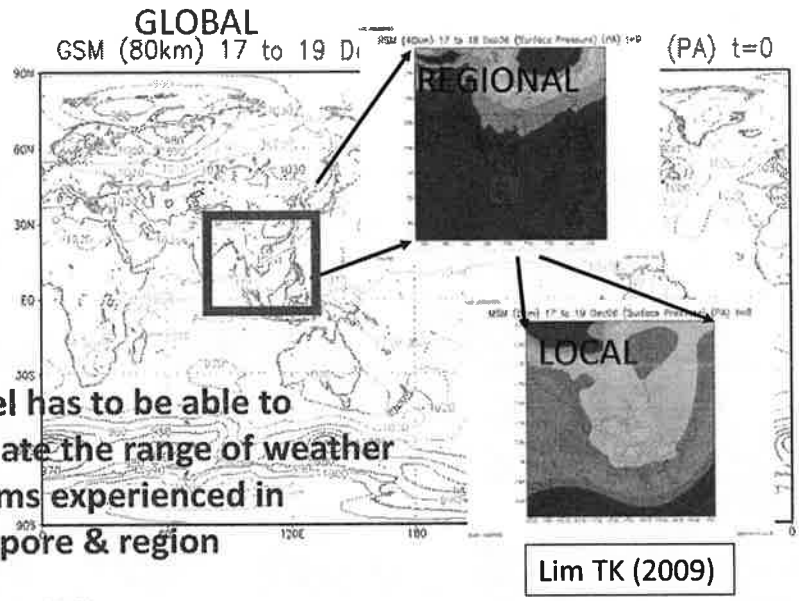
National Environment Agency

NUS National University of Singapore

A*STAR Institute of High Performance Computing

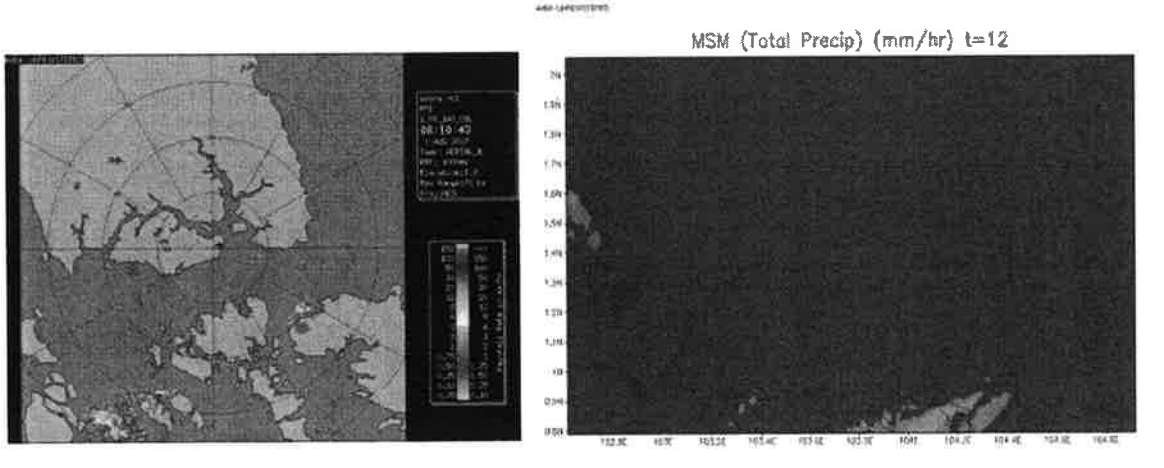
CFD Studies of Climate Change Impacts

- Weather/Climate Predictions - Global, Regional and Local Scales



CFD Studies of Climate Change Impacts

- Weather/Climate Predictions - Global, Regional and Local Scales (1km)

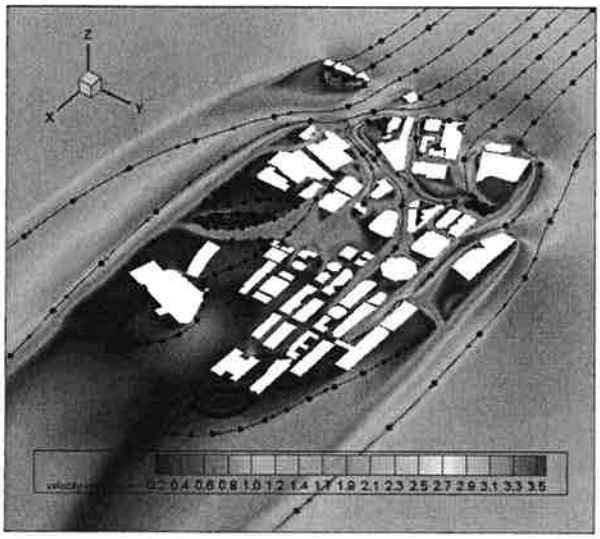


Southwest Monsoon Weather: 01 Aug 2007 (8am to 6pm)

CFD Studies of Climate Change Impacts

- Computational Study on Impacts to Urban biodiversity





CFD Studies of Climate Change Impacts

- Integrated Multi-layer GIS system



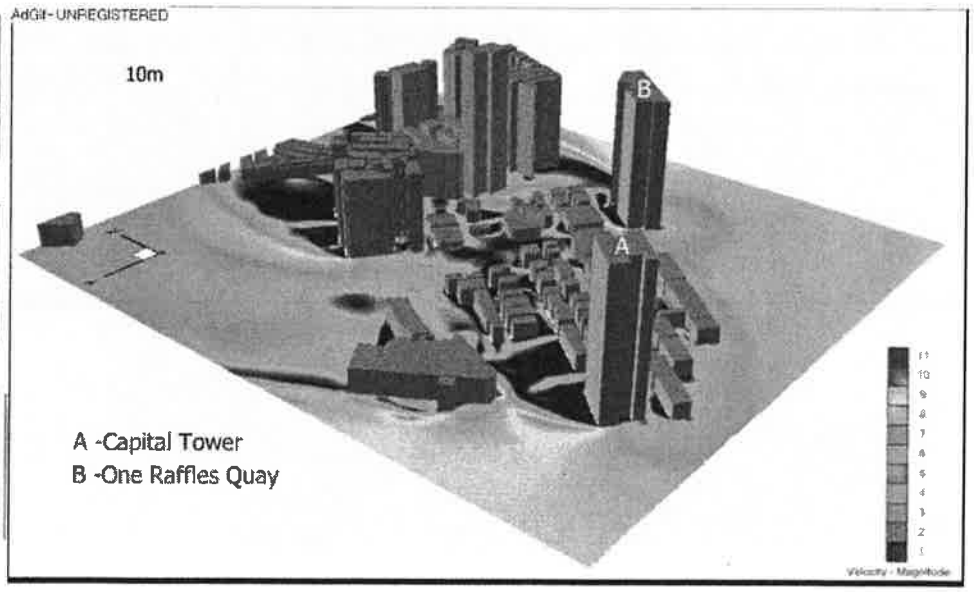
10 m

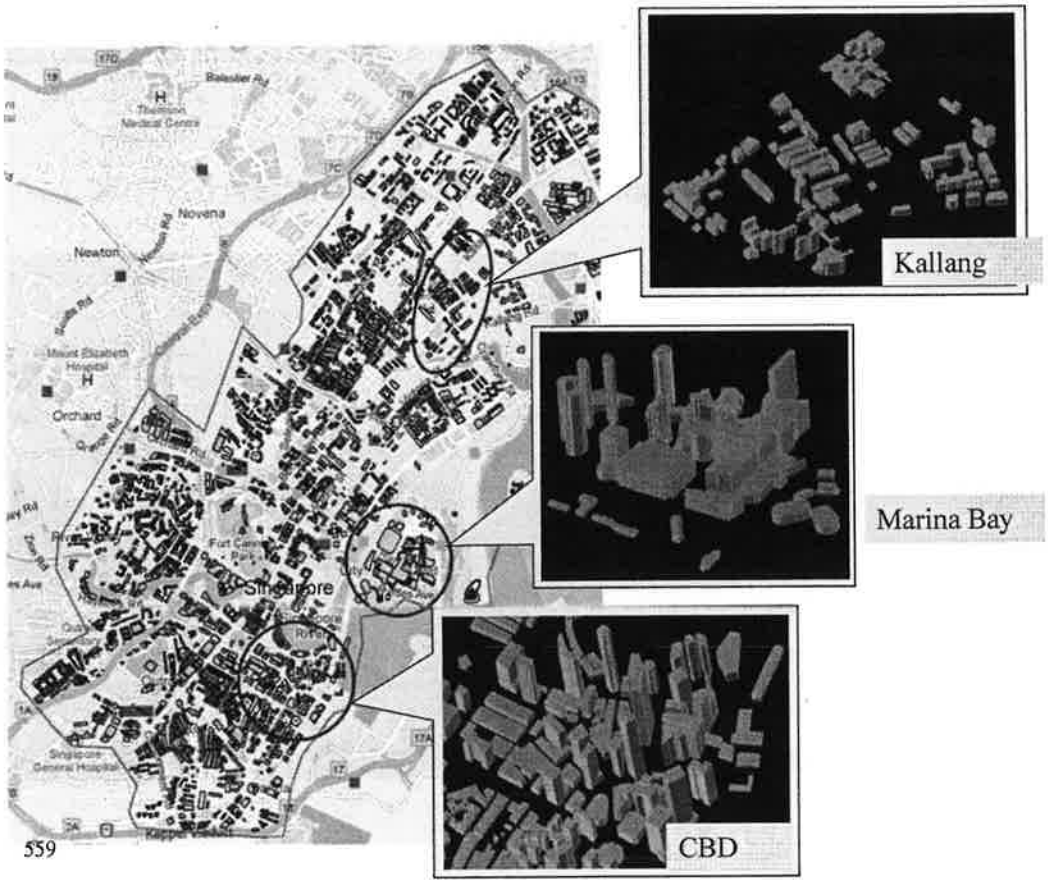


30 m



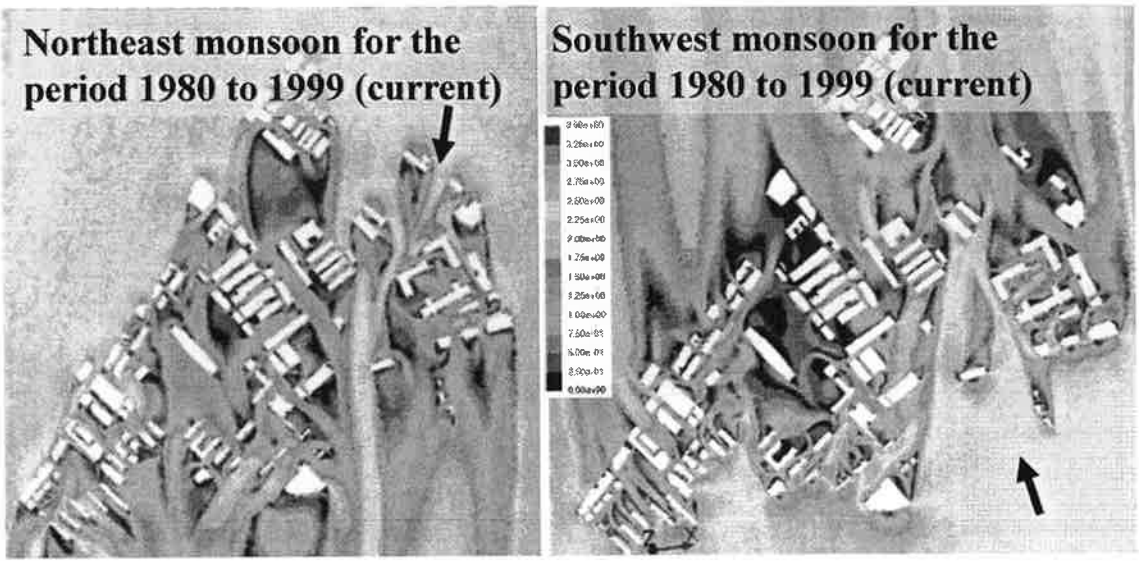
50 m





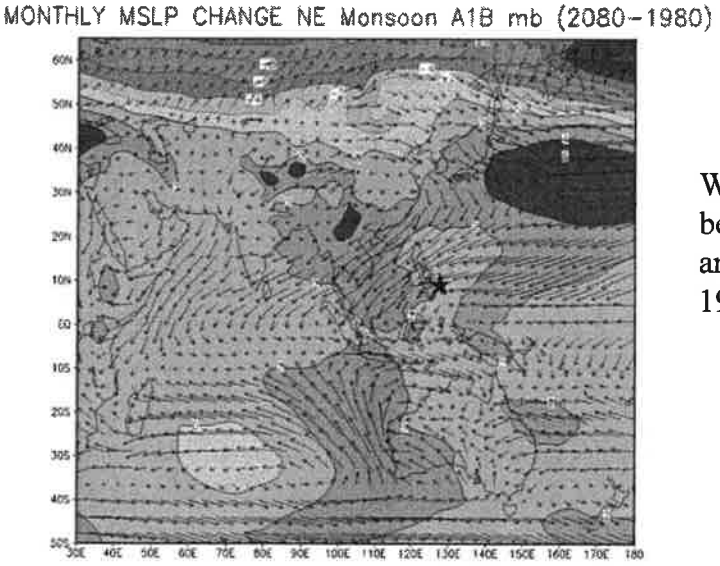
CFD Studies of Climate Change Impacts

- Northeast & Southwest Monsoons – locating wind channels at Kallang (industrial + residential)



CFD Studies of Climate Change Impacts

- Climate Change Impact on Wind Channels - Weather/Climate Predictions



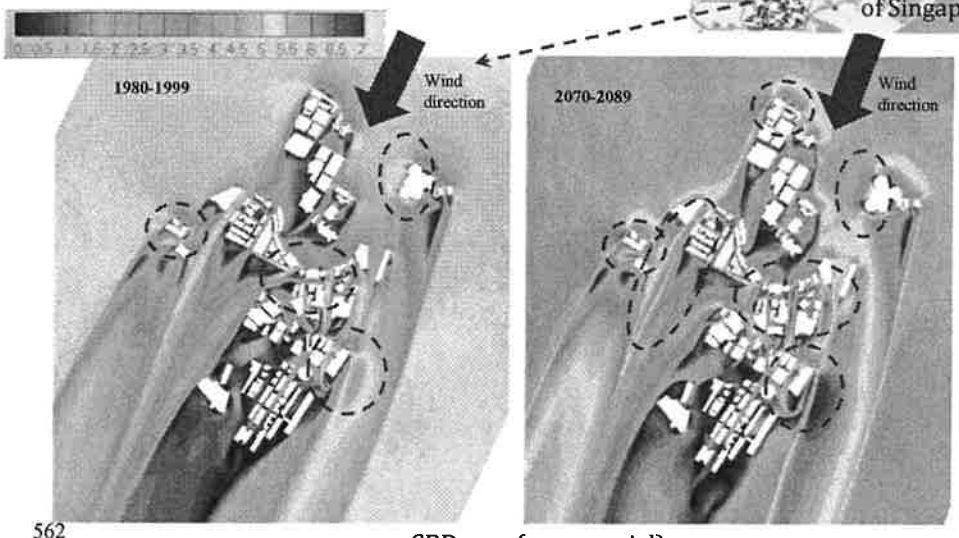
Wind Change between 2070-2090 and 1980-1999

NE Monsoon Wind (2070-2090)

CFD Studies of Climate Change Impacts

- Climate Change Impact on Wind Channel Study
  - Current climatology (1980-2000) vs projection (2070-2090)





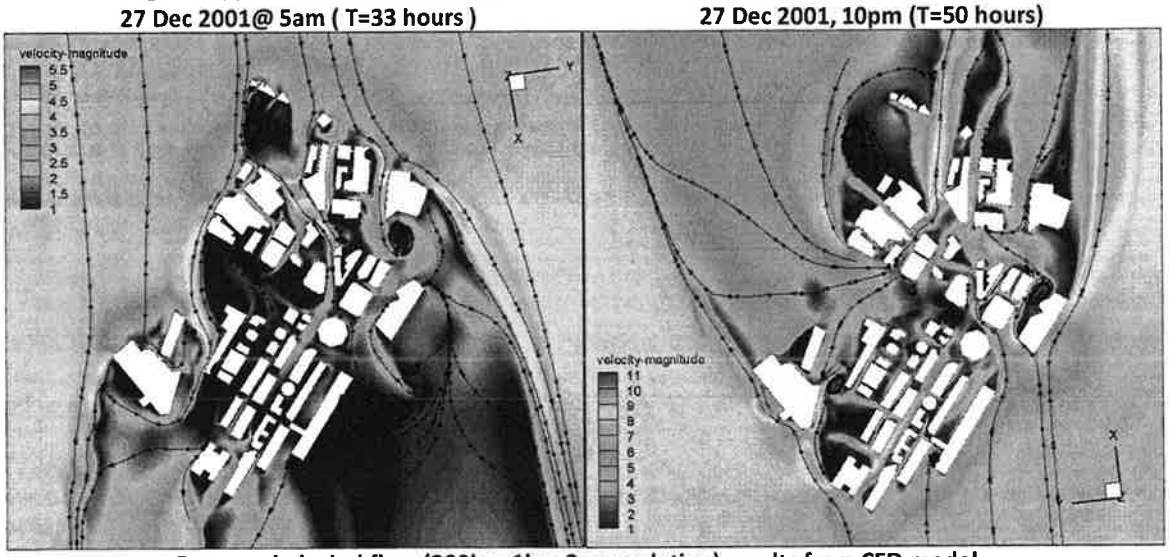
CBD area (commercial)

CFD Studies of Climate Change Impacts

- Severe Weather Impacts (Tropical Storm Vamei, Singapore, 27 Dec 2001) - Wind Flow: Before and After Storm

During the Approaching Phase of Vamei, 27 Dec 2001@ 5am (T=33 hours)

During the Receding Phase of Vamei, 27 Dec 2001, 10pm (T=50 hours)



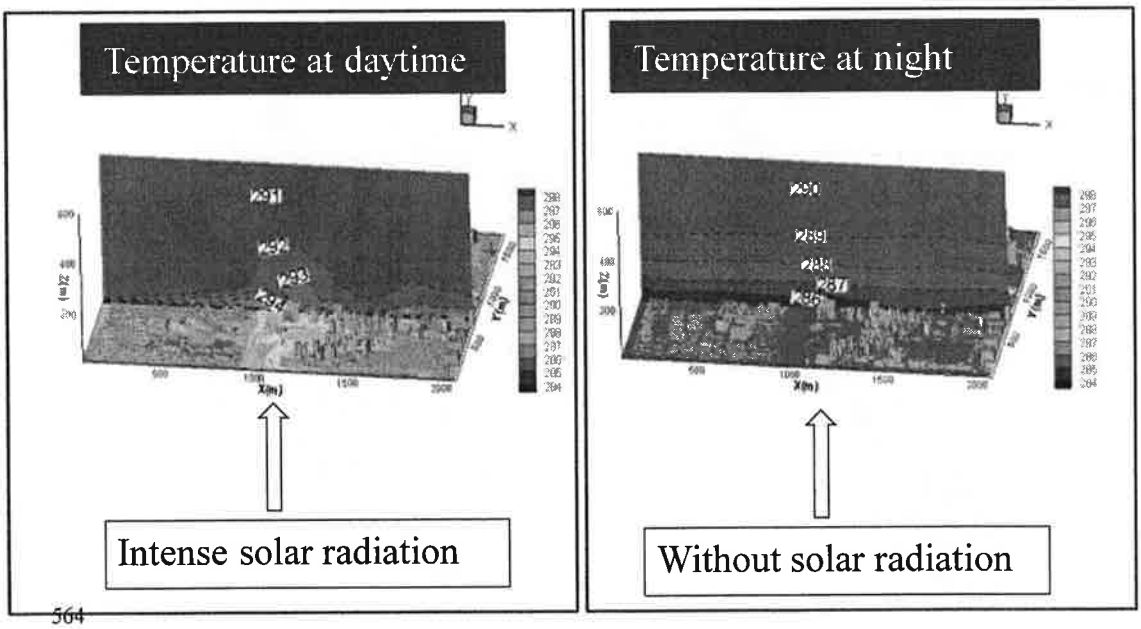
Downscaled wind flow (200km-1km-2m resolution) results from CFD model

CFD initial wind condition: based on high resolution climate model (1km)

563

Urban Heat Island – potential extension

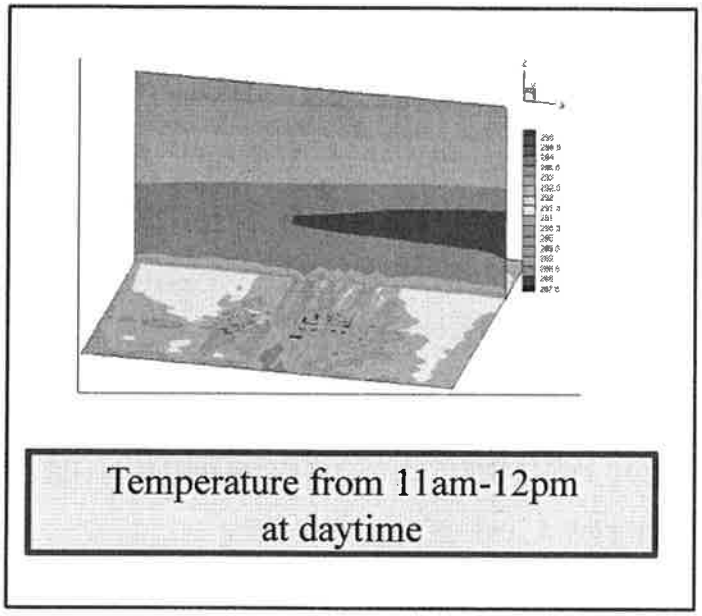
- Urban Thermal Environment with LES simulation



564

Urban Heat Island – potential extension

- Evolution of ground temperature



Temperature from 11am-12pm at daytime

CFD Studies of Climate Change Impacts

- Concluding Remarks
  - The local wind projections were conducted by employing various techniques, using available global monthly projection data (up to 2090), and local hourly historical data (1980-1999).
  - Coupling atmospheric models and CFD models provides an effective tool for urban wind risk assessment
    - of high wind speeds that may threaten tree failure, as well as understanding localised effects caused by climate change as well as severe storm events
    - of low wind regions that ventilation improvement may be required

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 (10:45am – 5:30pm)

- Studio assessment - Assignment 2 – Use CFD simulation tool (ANSYS-FLUENT) to perform ventilation simulation modelling and analysis in order to ensure good natural ventilation within the functional unit (70%)

568

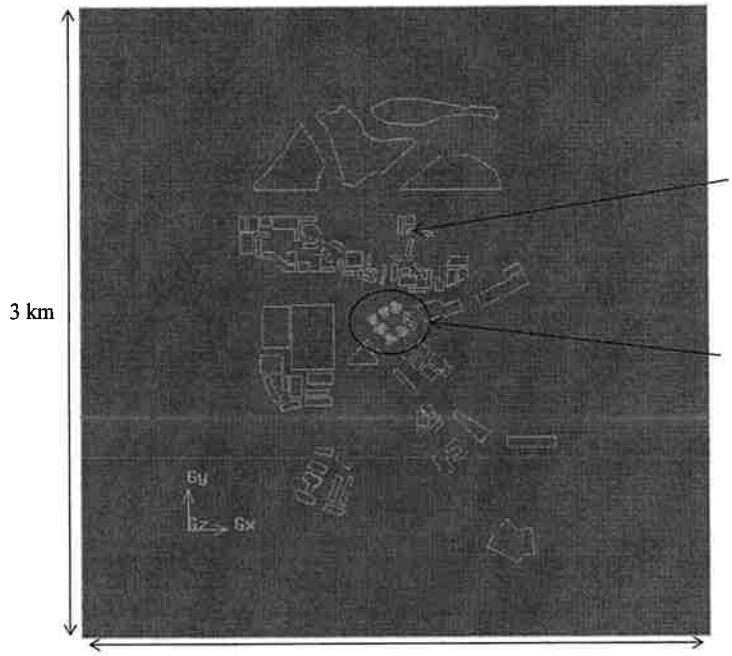
CFD simulation for Green Mark Building design

- Studio assessment for Assignment 2 (70%)
  - Simulation of air flow around building development and adjacent buildings
  - Demonstrate capability to use CFD tool
    - To capture vortices/recirculation around building premises
    - To display pressure forces exerted on the building surfaces

Practice for Unit Level Ventilation

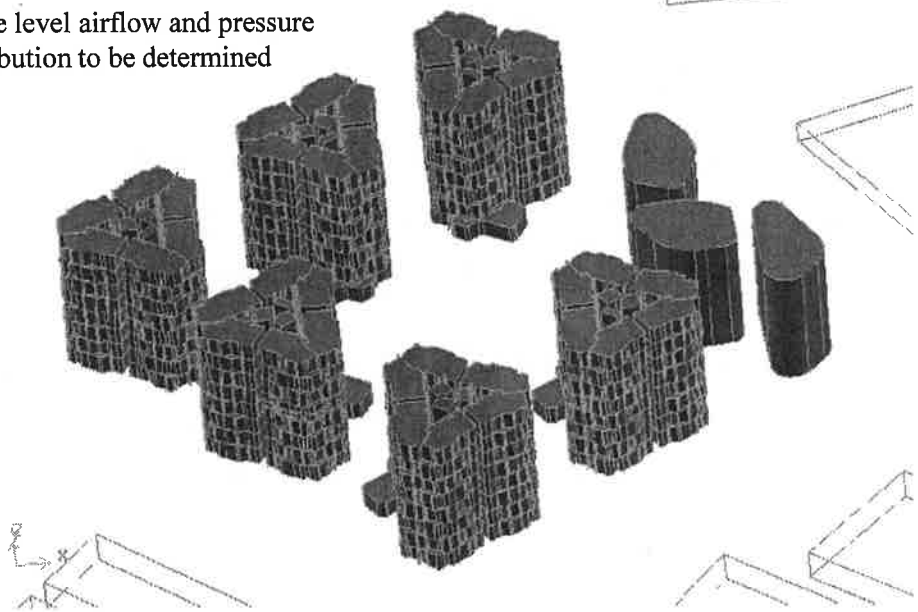
- Simulation of air flow within the building premises (for either Residential or non-Residential type)
- Non-Residential Building type include *hospitals, industrial facilities, hawker centres, sports facilities, commercial atriums and schools*
- Demonstrate capability to use CFD tool
  - To determine average velocity on various zones
  - To identify stagnation zones and propose mitigation/adaption measures

Example: CFD simulation for Green Mark Building design

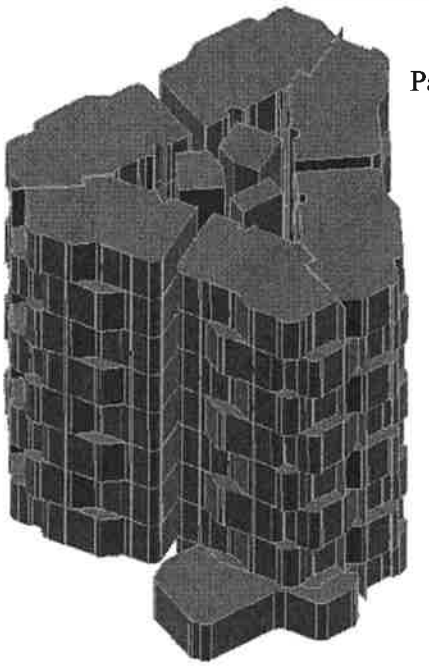


CFD simulation for Green Mark Building design

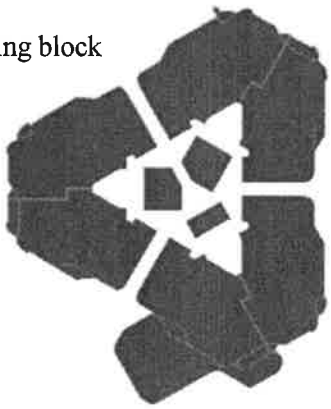
Estate level airflow and pressure distribution to be determined



CFD simulation for Green Mark Building design

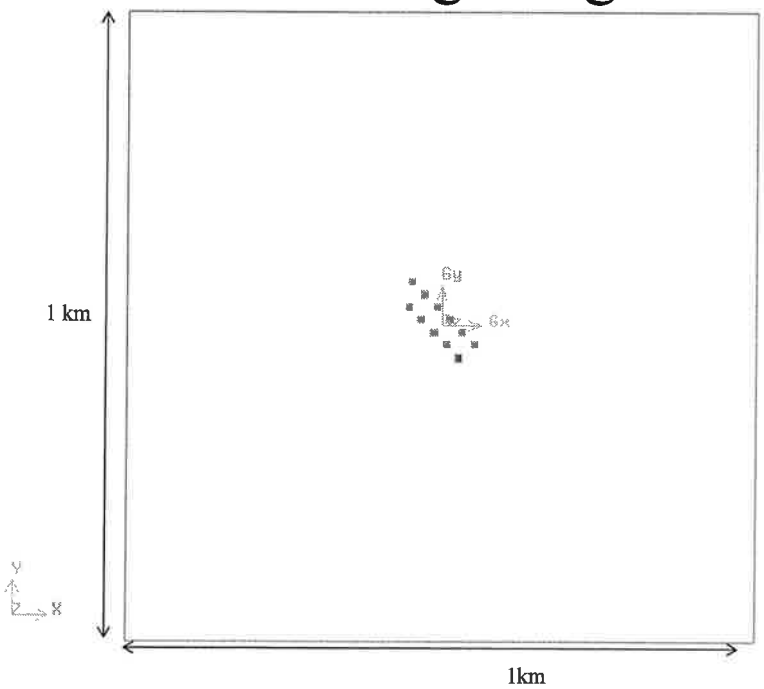


Particular building block



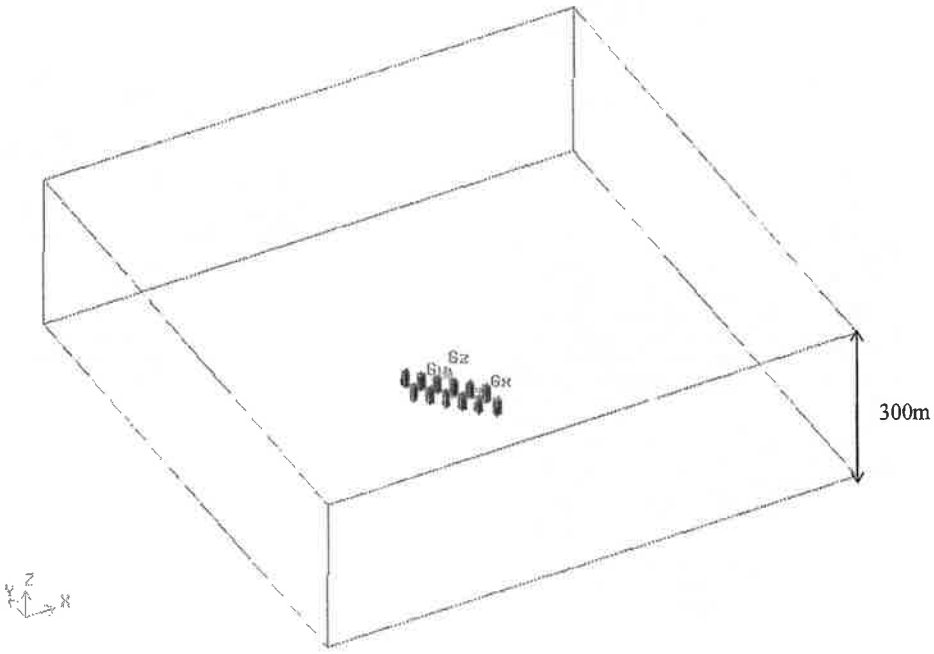
573

CFD simulation for Green Mark Building design



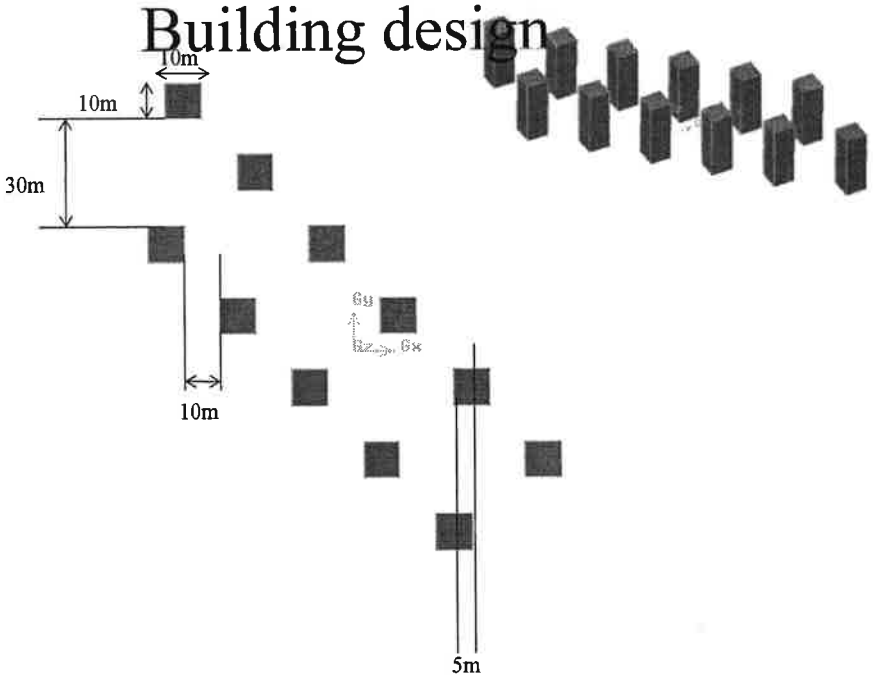
574

CFD simulation for Green Mark Building design



575

CFD simulation for Green Mark Building design



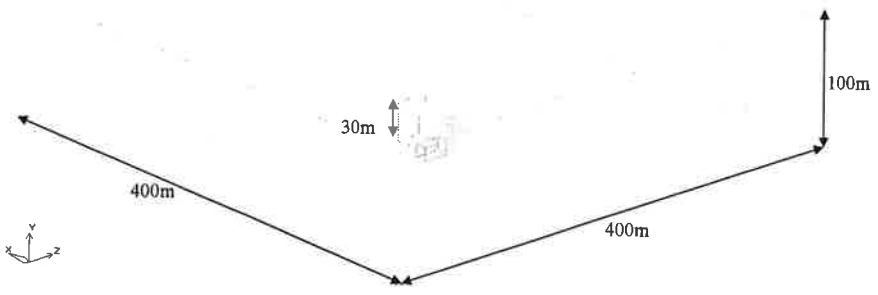
576

CFD simulation for Green Mark Building design

- Assignment for Assessment 2 (30%)
  - Simulation of air flow through typical dwelling unit
  - Demonstrate capability to simulation natural (cross) ventilation for base case design

577

CFD simulation for Green Mark Building design



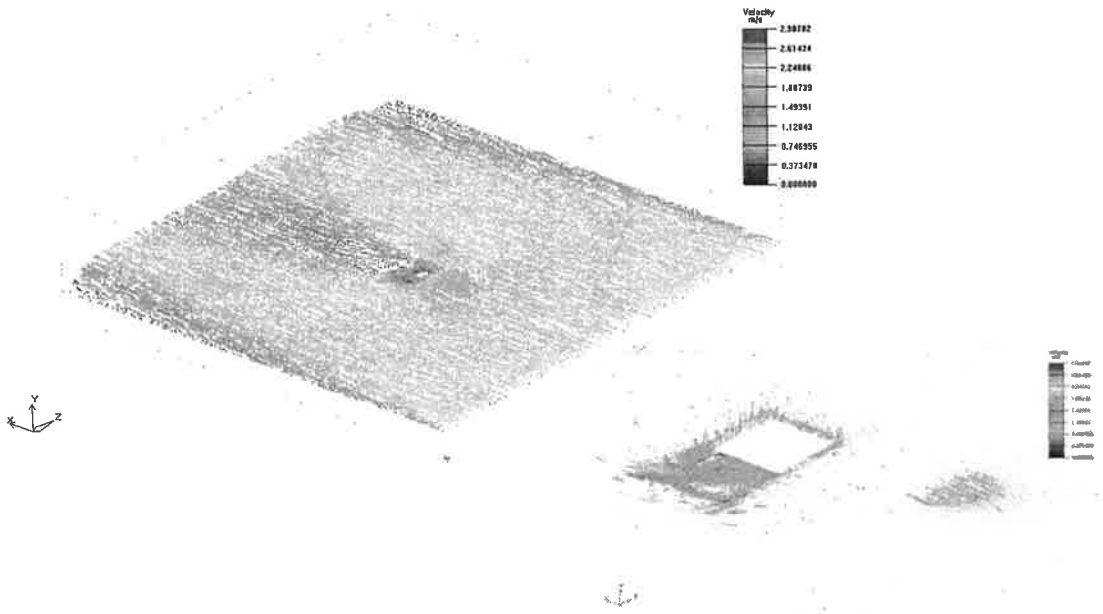
578

CFD simulation for Green Mark Building design



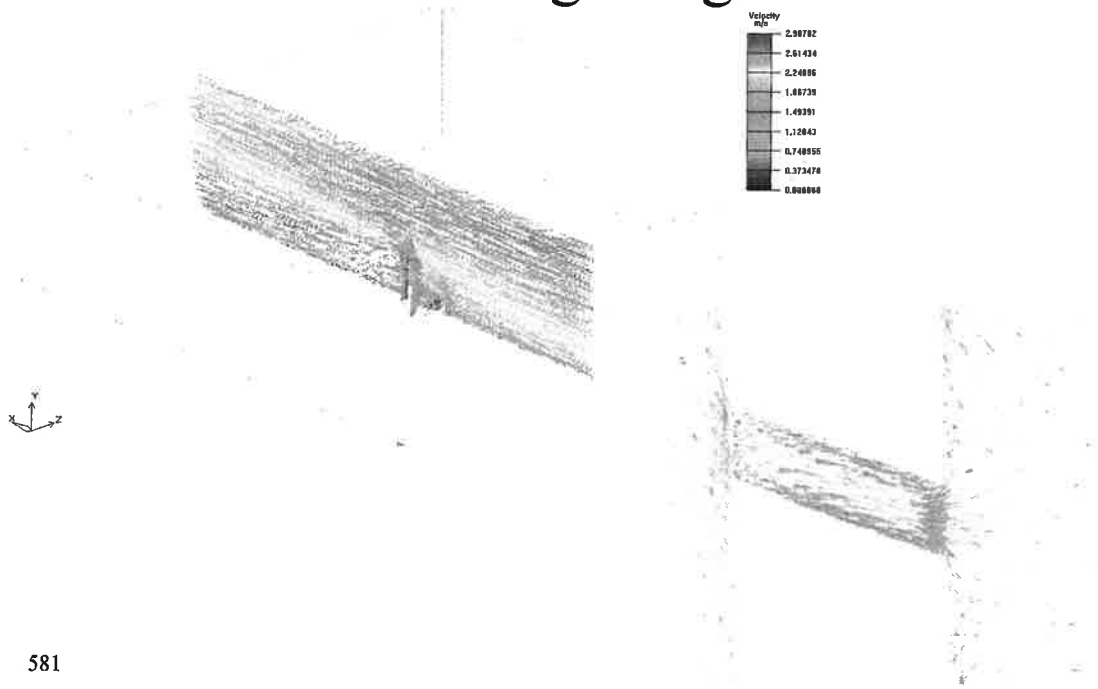
579

CFD simulation for Green Mark Building design



580

CFD simulation for Green Mark Building design



581

CFD simulation for Green Mark Building design

- Assignment for Assessment 2 (30%)
  - Simulation of air flow through typical dwelling unit
  - Demonstrate capability to simulation natural (cross) ventilation for improved design

582

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!

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

Acknowledgement

- Prof. Arun Mujumdar, McGill University
- Prof. Hazim Awbi, Reading University
- Prof. Chen Qingyan, Purdue University
- Prof. Zhiqiang (John) Zhai, University of Colorado
- CAD-IT & ANSYS-FLUENT

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 both ANSYS Fluent and in the CFD-Post 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 and CFD-Post.

Release 16.0 - © 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, or CFD-Post, and so each step will be explicitly described.

1.3. Problem Description

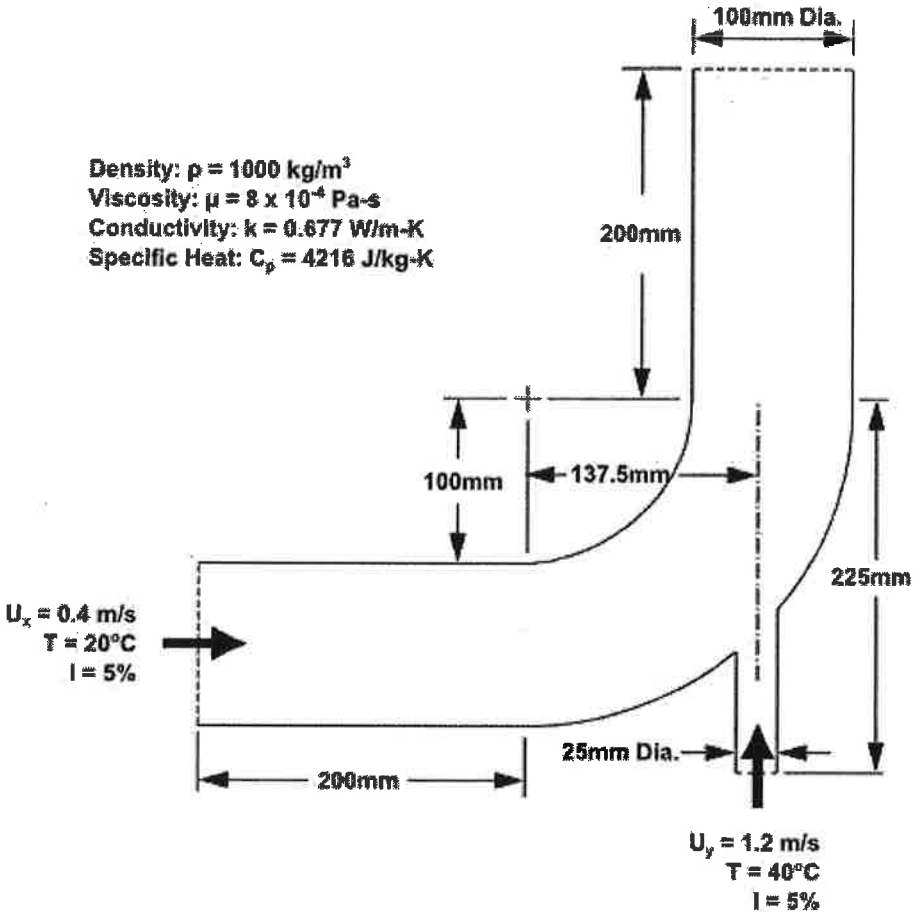
The problem to be considered is shown schematically in Figure 1.1: Problem Specification (p. 3). A cold fluid at 293.15 K flows into the pipe through a large inlet and mixes with a warmer fluid at 313.15 K that enters through a smaller inlet located at the elbow. The mixing elbow configuration is encountered 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 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.1: Problem Specification



Note

The functionality to create named selections exists in both ANSYS DesignModeler and ANSYS Meshing. For the purposes of this tutorial, named selections are created in ANSYS Meshing since the meshing application provides more comprehensive and extensive named selection functionality.

1.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

1.4.1. Preparation
1.4.2. Creating a Fluent Fluid Flow Analysis System in ANSYS Workbench
1.4.3. Creating the Geometry in ANSYS DesignModeler
1.4.4. Meshing the Geometry in the ANSYS Meshing Application
1.4.5. Setting Up the CFD Simulation in ANSYS Fluent
1.4.6. Displaying Results in ANSYS Fluent and CFD-Post
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
1.4.11. Comparing the Results of Both Systems in CFD-Post

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.4.1. Preparation

1. Set up a working folder on the computer you will be using.
2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

> **Note**
>
> If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

3. Enter the name of this tutorial into the search bar.
4. Narrow the results by using the filter on the left side of the page.
   a. Click **ANSYS Fluent** under **Product**.
   b. Click **16.0** under **Version**.
5. Select this tutorial from the list.
6. Click the **elbow-workbench_R160.zip** link to download the input and solution files.
7. Unzip `elbow-workbench_R160.zip` to your working folder. This file contains a folder, `elbow-workbench`, that holds the following items:
   - two geometry files, `elbow_geometry.agdb` and `elbow_geometry.stp`
   - an ANSYS Workbench project archive, `elbow-workbench.wbpz`

> **Tip**
>
> The Workbench project archive contains the project as it will be once you have completed all of the steps of the tutorial and is included for reference. If you want to extract the project archive, start Workbench and select the **File → Restore Archive...** menu item. You will be prompted with a dialog box to specify a location in which to extract the project and its supporting files. You may choose any convenient location.

> **Note**
>
> ANSYS Fluent tutorials are prepared using ANSYS Fluent on a Windows system. The screen shots and graphic images in the tutorials may be slightly different than the appearance on your system, depending on the operating system or graphics card.

1.4.2. 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.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the Windows **Start** menu, select **Start > All Programs > ANSYS 16.0** > **Workbench 16.0** 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**.

   Note

   Depending on which other products you have installed, the analysis systems that appear may differ from those in the figures that follow in this tutorial.

   Note

   When you first start ANSYS Workbench, the **Getting Started** pop-up window is displayed, offering assistance through the online help for using the application. You can keep the window open, or close it by clicking the 'X' icon in the upper right-hand corner. If you need to access the online help at any time, use the **Help** menu, or press the **F1** key.

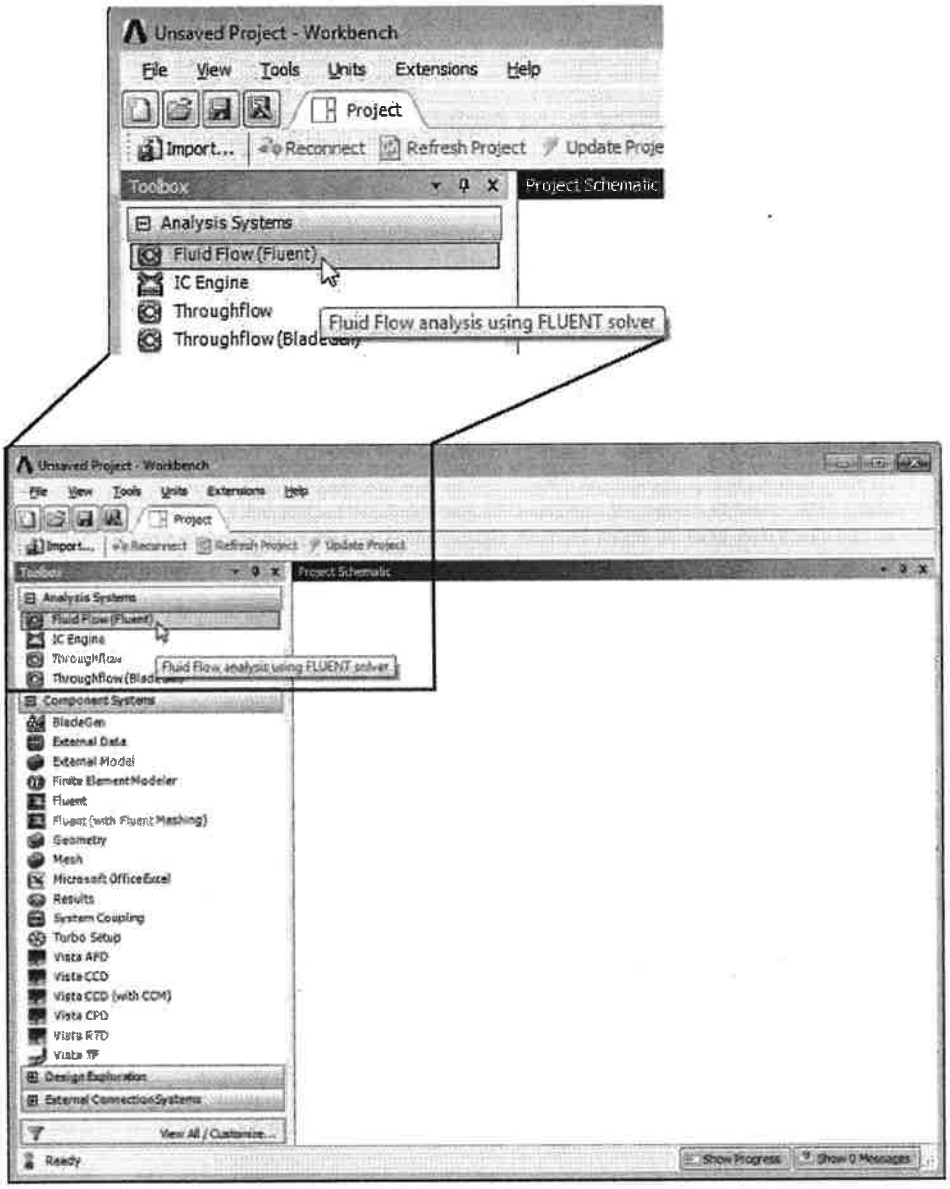
2. Create a new Fluent fluid flow analysis system by double-clicking the **Fluid Flow (Fluent)** option under **Analysis Systems** in the **Toolbox**.

   Tip

   You can also drag-and-drop the analysis system into the **Project Schematic**. A green dotted outline indicating a potential location for the new system initially appears in the **Project Schematic**. When you drag the system to one of the outlines, it turns into a red box to indicate the chosen location of the new system.

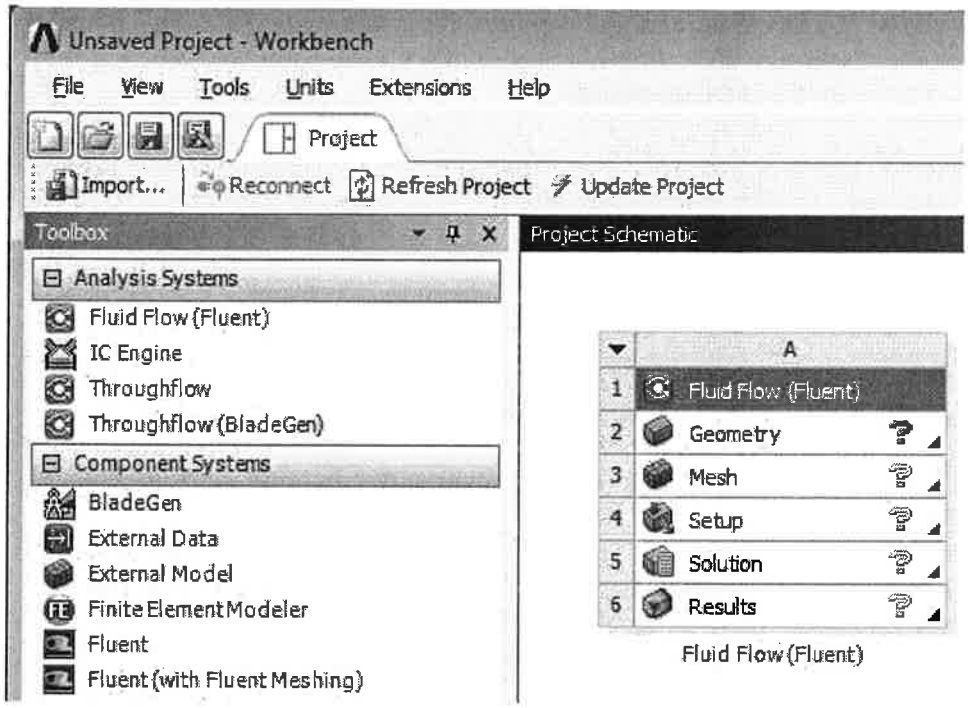
Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.2: Selecting the Fluid Flow (Fluent) Analysis System in ANSYS Workbench



Release 16.0 - © 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. Select the **Save** option under the **File** menu in ANSYS Workbench.

      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.

      Note that the fluid flow analysis system is composed of various *cells* (**Geometry**, **Mesh**, etc.) that represent the workflow for performing the analysis. ANSYS Workbench is composed of multiple data-integrated and native applications in a single, seamless project flow, where individual cells can obtain data from other cells and provide data to other cells. As a result of this constant flow of data, a cell's state can quickly change. ANSYS Workbench provides a visual indication of a cell's state at any given time via icons on the right side of each cell. Brief descriptions of the various states are provided below:

      - **Unfulfilled** ( ) indicates that required upstream data does not exist. For example, when you first create a new **Fluid Flow (Fluent)** analysis system, all cells downstream of the **Geometry** cell appear as **Unfulfilled** because you have not yet specified a geometry for the system.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Refresh Required** () indicates that upstream data has changed since the last refresh or update. For example, after you assign a geometry to the geometry cell in your new **Fluid Flow (Fluent)** analysis system, the **Mesh** cell appears as **Refresh Required** since the geometry data has not yet been passed from the **Geometry** cell to the **Mesh** cell.

- **Attention Required** () indicates that the current upstream data has been passed to the cell, however, you must take some action to proceed. For example, after you launch ANSYS Fluent from the **Setup** cell in a **Fluid Flow (Fluent)** analysis system that has a valid mesh, the **Setup** cell appears as **Attention Required** because additional data must be entered in ANSYS Fluent before you can calculate a solution.

- **Update Required** () indicates that local data has changed and the output of the cell must be re-generated. For example, after you launch ANSYS Meshing from the **Mesh** cell in a **Fluid Flow (Fluent)** analysis system that has a valid geometry, the **Mesh** cell appears as **Update Required** because the **Mesh** cell has all the data it must generate an ANSYS Fluent mesh file, but the ANSYS Fluent mesh file has not yet been generated.

- **Up To Date** () indicates that an update has been performed on the cell and no failures have occurred or that an interactive calculation has been completed successfully. For example, after ANSYS Fluent finishes performing the number of iterations that you request, the **Solution** cell appears as **Up-to-Date**.

- **Interrupted, Update Required** () indicates that you have interrupted an update (or canceled an interactive calculation that is in progress). For example, if you select the **Cancel** button in ANSYS Fluent while it is iterating, ANSYS Fluent completes the current iteration and then the **Solution** cell appears as **Interrupted, Update Required**.

- **Input Changes Pending** () indicates that the cell is locally up-to-date, but may change when next updated as a result of changes made to upstream cells. For example, if you change the **Mesh** in an **Up-to-Date Fluid Flow (Fluent)** analysis system, the **Setup** cell appears as **Refresh Required**, and the **Solution** and **Results** cells appear as **Input Changes Pending**.

- **Pending** () indicates that a batch or asynchronous solution is in progress. When a cell enters the **Pending** state, you can interact with the project to exit Workbench or work with other parts of the project. If you make changes to the project that are upstream of the updating cell, then the cell will not be in an up-to-date state when the solution completes.

For more information about cell states, see Understanding Cell States.

5. View the list of files generated by ANSYS Workbench.

   ANSYS Workbench allows you to easily view the files associated with your project using the **Files** view. To open the **Files** view, select the **Files** option under the **View** menu at the top of the ANSYS Workbench window.

   View → Files

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.4: ANSYS Workbench Files View for the Project After Adding a Fluent-Based Fluid Flow Analysis System

Files

	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	elbow-workbench.wbpj		123 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	designPoint.wbdp		25 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

In the **Files** view, you will be able to see the name and type of file, the ID of the cell that the file is associated with, the size of the file, the location of the file, and other information. For more information about the **Files** view, see Files View.

Note

The sizes of the files listed may differ slightly from those portrayed in Figure 1.4: ANSYS Workbench Files View for the Project After Adding a Fluent-Based Fluid Flow Analysis System (p. 9).

From here, you will create the geometry described in Figure 1.1: Problem Specification (p. 3), and later create a mesh and set up a fluid flow analysis for the geometry.

1.4.3. Creating the Geometry in ANSYS DesignModeler

For the geometry of your fluid flow analysis, you can create a geometry in ANSYS DesignModeler, or import the appropriate geometry file. In this step, you will create the geometry in ANSYS DesignModeler, then review the list of files generated by ANSYS Workbench.

Important

Note the **Attention Required** icon (?) within the **Geometry** cell for the system. This indicates that the cell requires data (for example, a geometry). Once the geometry is defined, the state of the cell will change accordingly. Likewise, the state of some of the remaining cells in the system will change.

Note

If you would rather not create the geometry in ANSYS DesignModeler, you can import a pre-existing geometry by right-clicking the **Geometry** cell and selecting the **Import Geometry** option from the context menu. From there, you can browse your file system to locate the `elbow_geometry.agdb` geometry file that is provided for this tutorial. If you do not have access to ANSYS DesignModeler, you can use the `elbow_geometry.stp` file instead.

To learn how to create a mesh from the geometry you imported, go to Meshing the Geometry in the ANSYS Meshing Application (p. 20).

1. Start ANSYS DesignModeler.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 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 (p. 3)) 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.

A preview of the torus geometry will appear in the graphics window. Note that this is a preview and the geometry has not been created yet. First you must specify the parameters of the torus primitive in the next step.

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.

Note

Enter only the value without the units of `mm`. They will be appended automatically because you specified the units previously.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

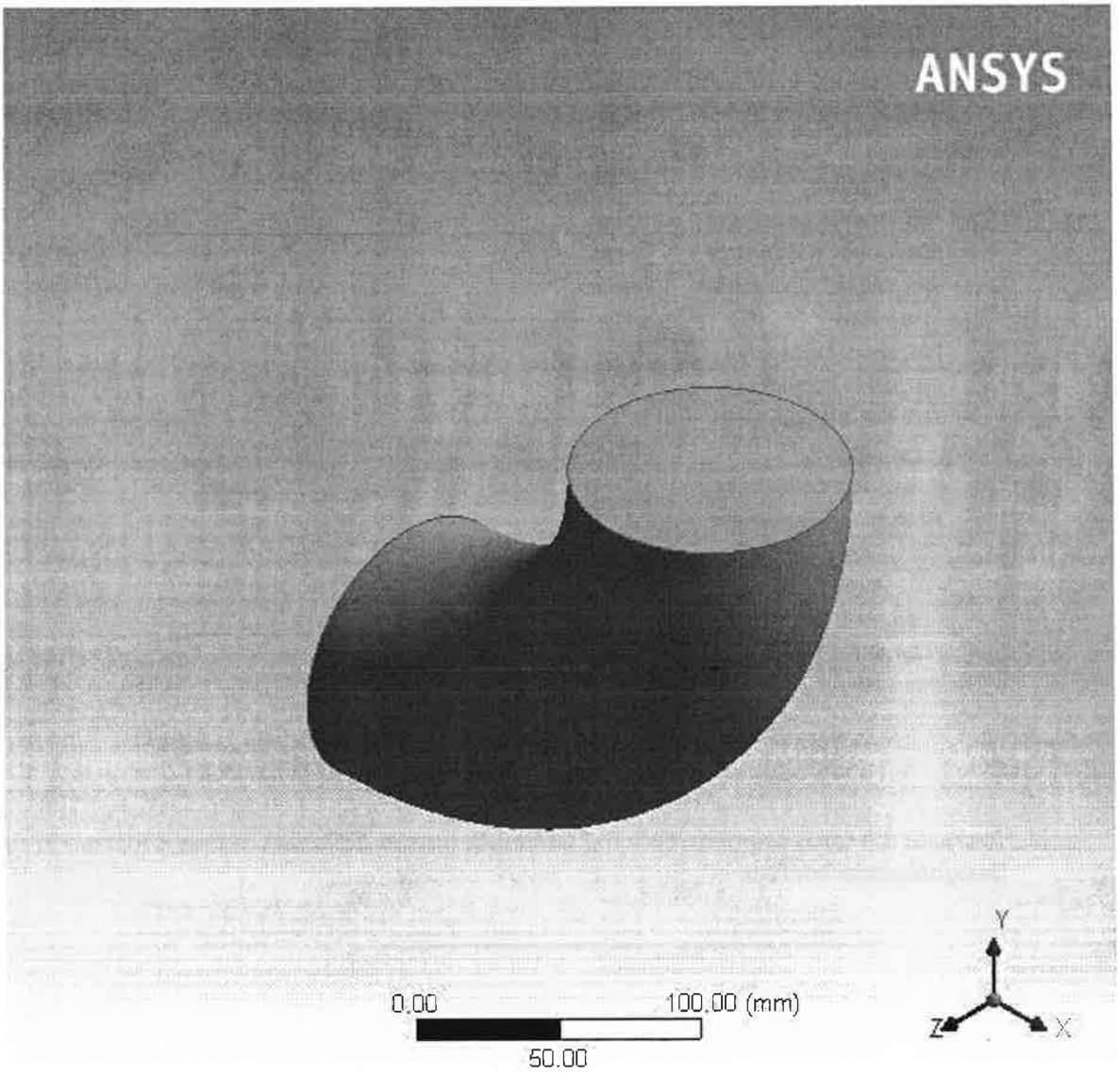
Details View

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 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



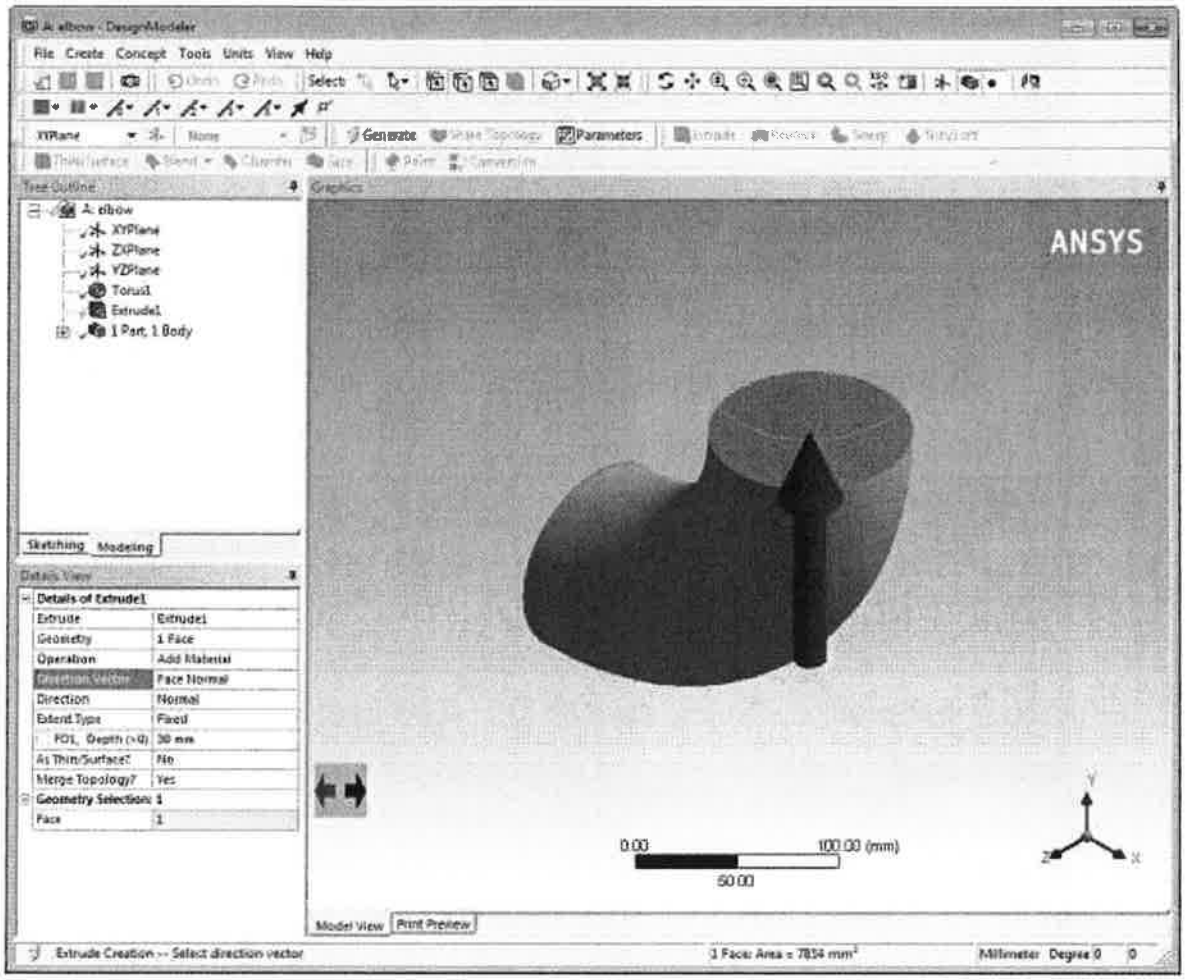
iv. Ensure that the selection filter is set to **Faces**. This is indicated by the **Faces** button appearing depressed in the toolbar and the appearance of the Face selection cursor, when you mouse over the geometry.

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.

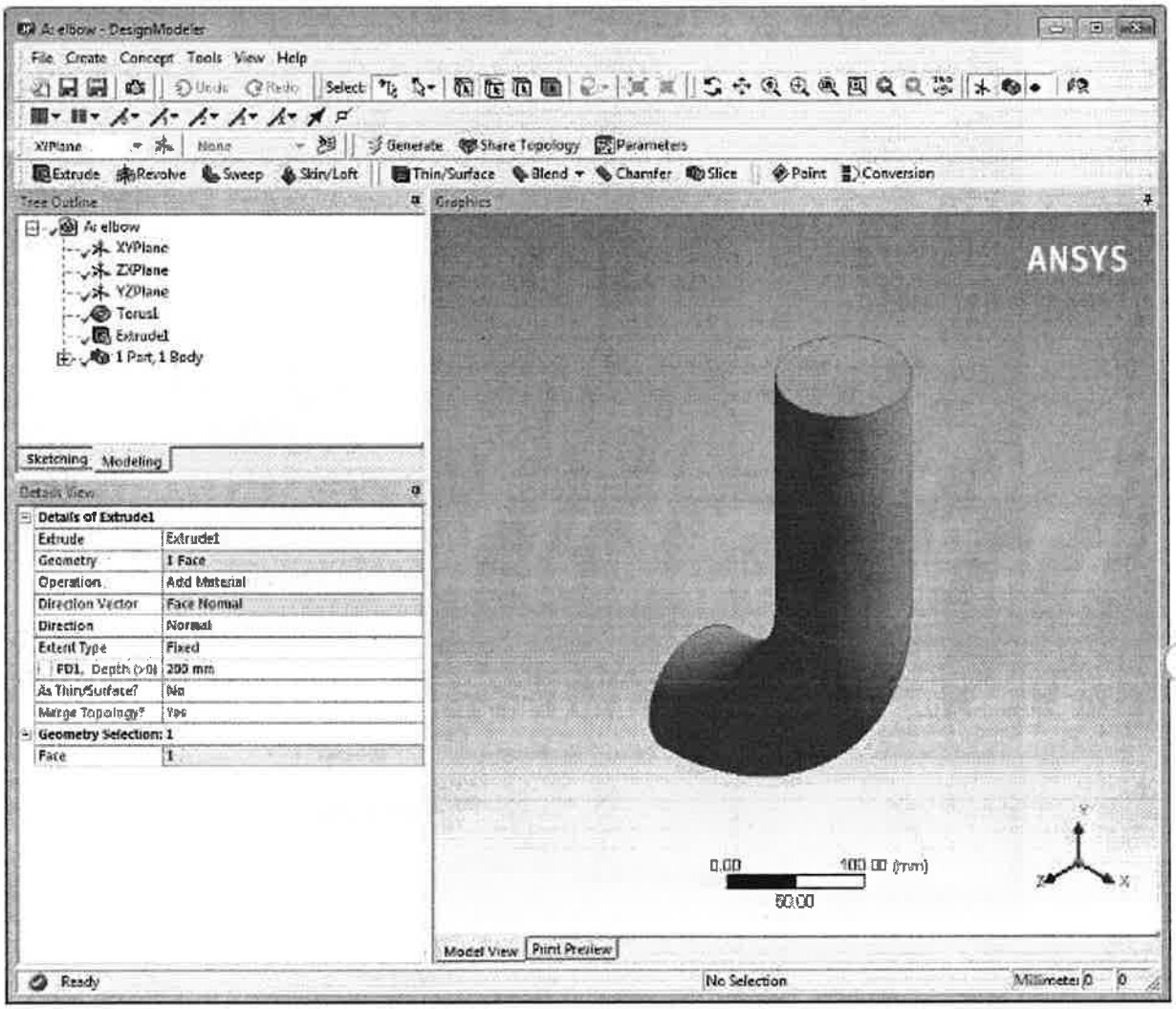
vii. Click **None (Normal)** to the right of **Direction Vector**. Again, ensure that the selection filter is set to **Faces**, select the same face on the elbow to specify that the extrusion will be normal to the face and click **Apply**.

Release 16.0 - © 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 16.0 - © 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.

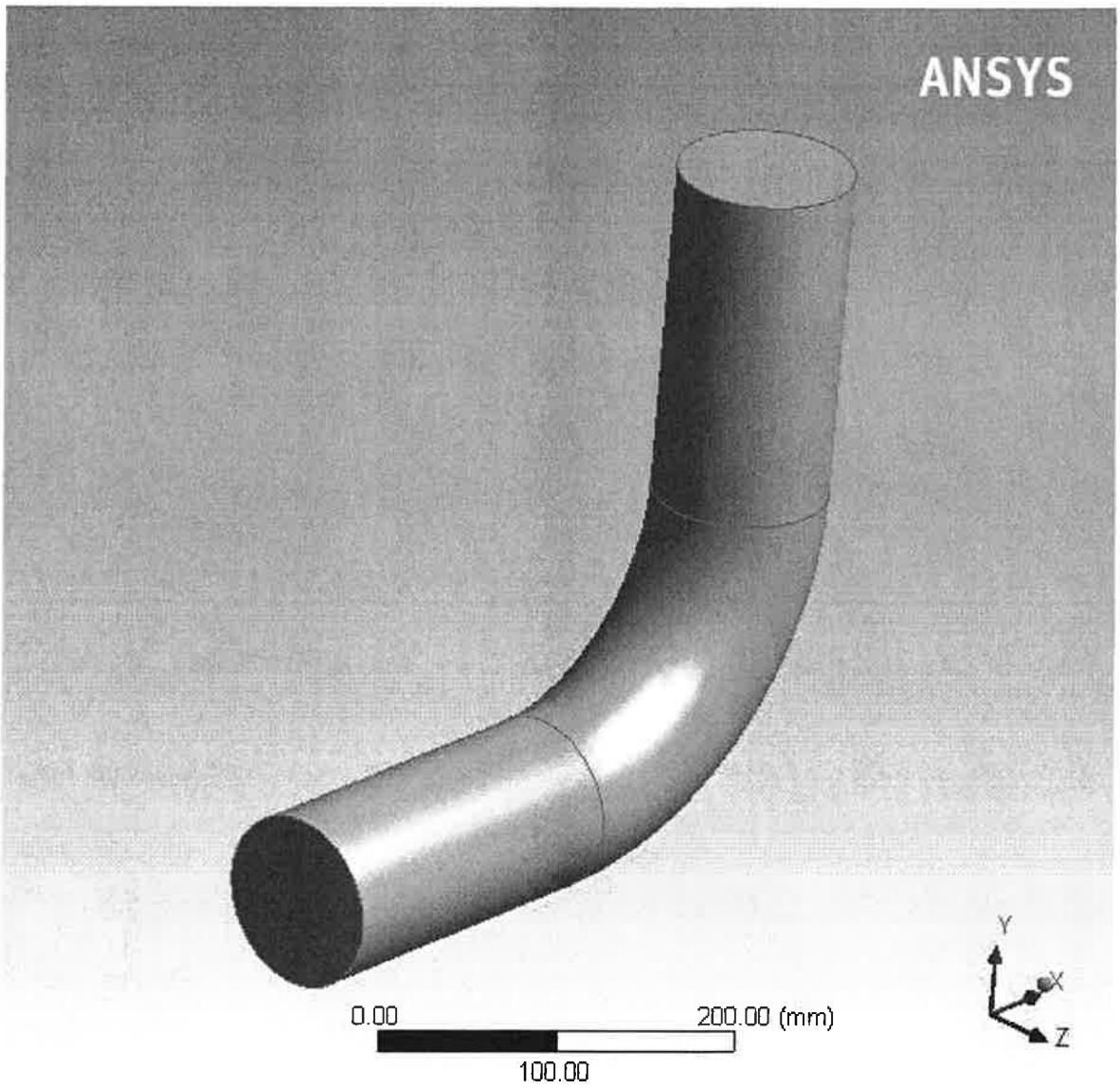
Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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 (p. 15).

Figure 1.5: Elbow Main Pipe Geometry



b. Next you will use a cylinder primitive to create the side pipe.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

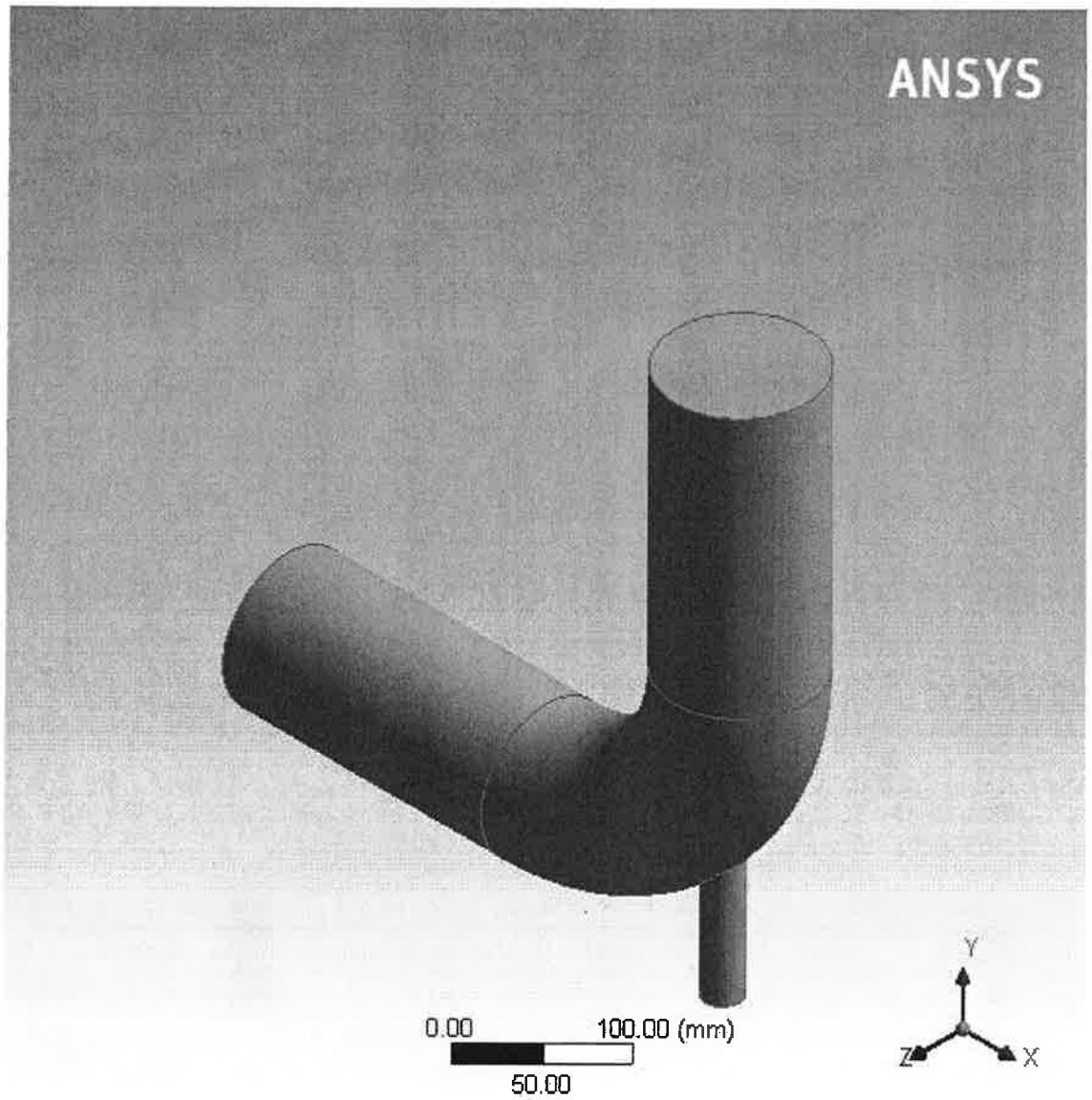
i. Choose **Create** → **Primitives** → **Cylinder** from the menubar.

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 16.0 - © 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.

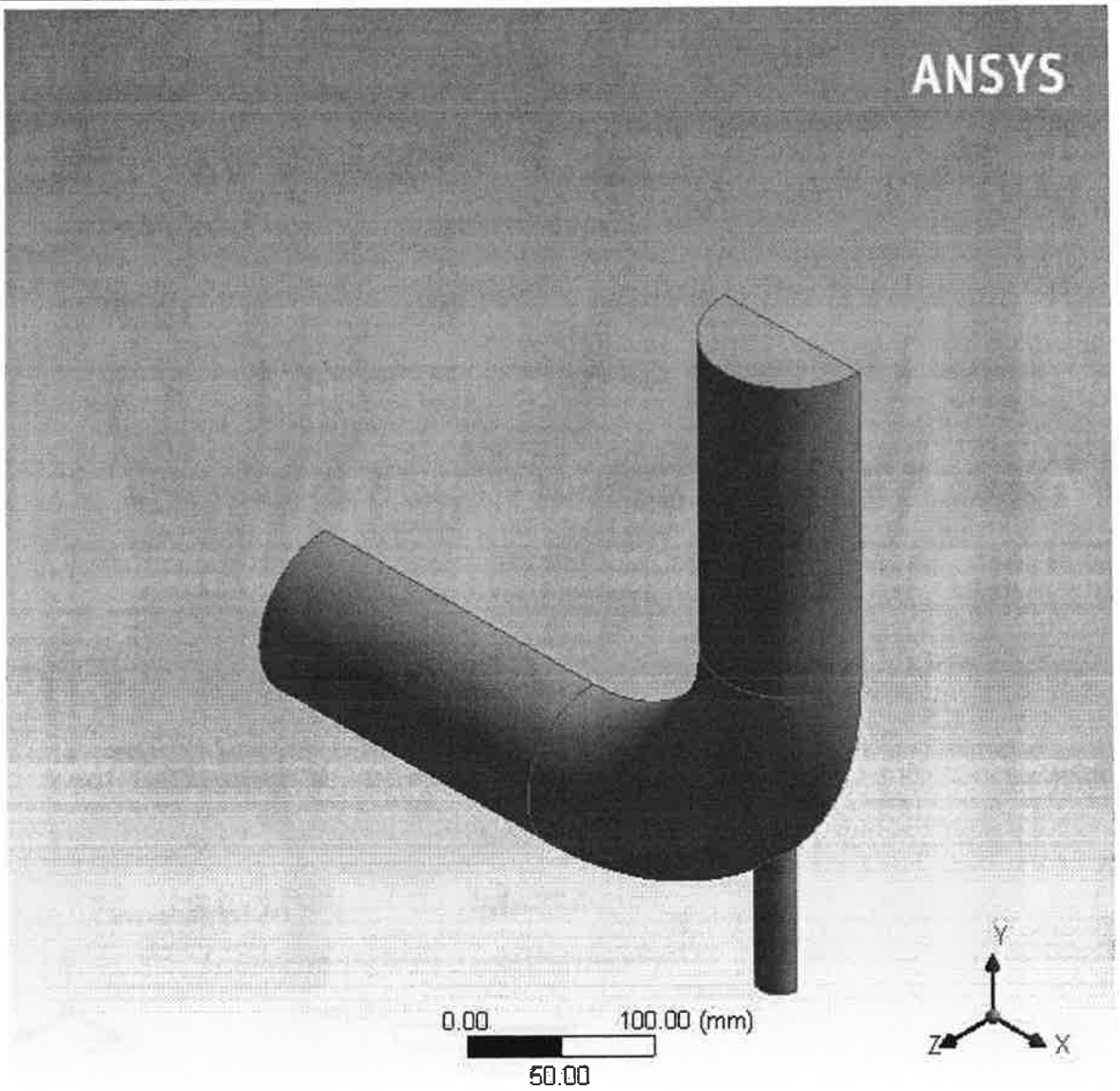
   i. Choose **Tools** → **Symmetry** from the menu bar.

   ii. Select the **XYPlane** in the **Tree Outline**.

   iii. Click **Apply** next to **Symmetry Plane 1** in the **Details** view.

   iv. Click **Generate**.

Release 16.0 - © 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 16.0 - © 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**.

Tip

In addition to the primitives you used in this tutorial, ANSYS DesignModeler offers a full suite of 2D sketching and 3D solid modeling tools for creating arbitrary geometry. Refer to ANSYS DesignModeler User's Guide for more information.

---

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. If you had imported the geometry file provided for this tutorial rather than creating the geometry yourself, the `elbow_geometry.agdb` (or the `elbow_geometry.stp`) file would be listed instead.

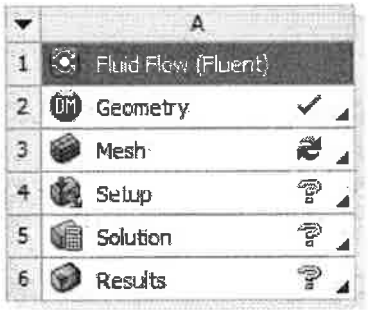
Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.4.4. 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

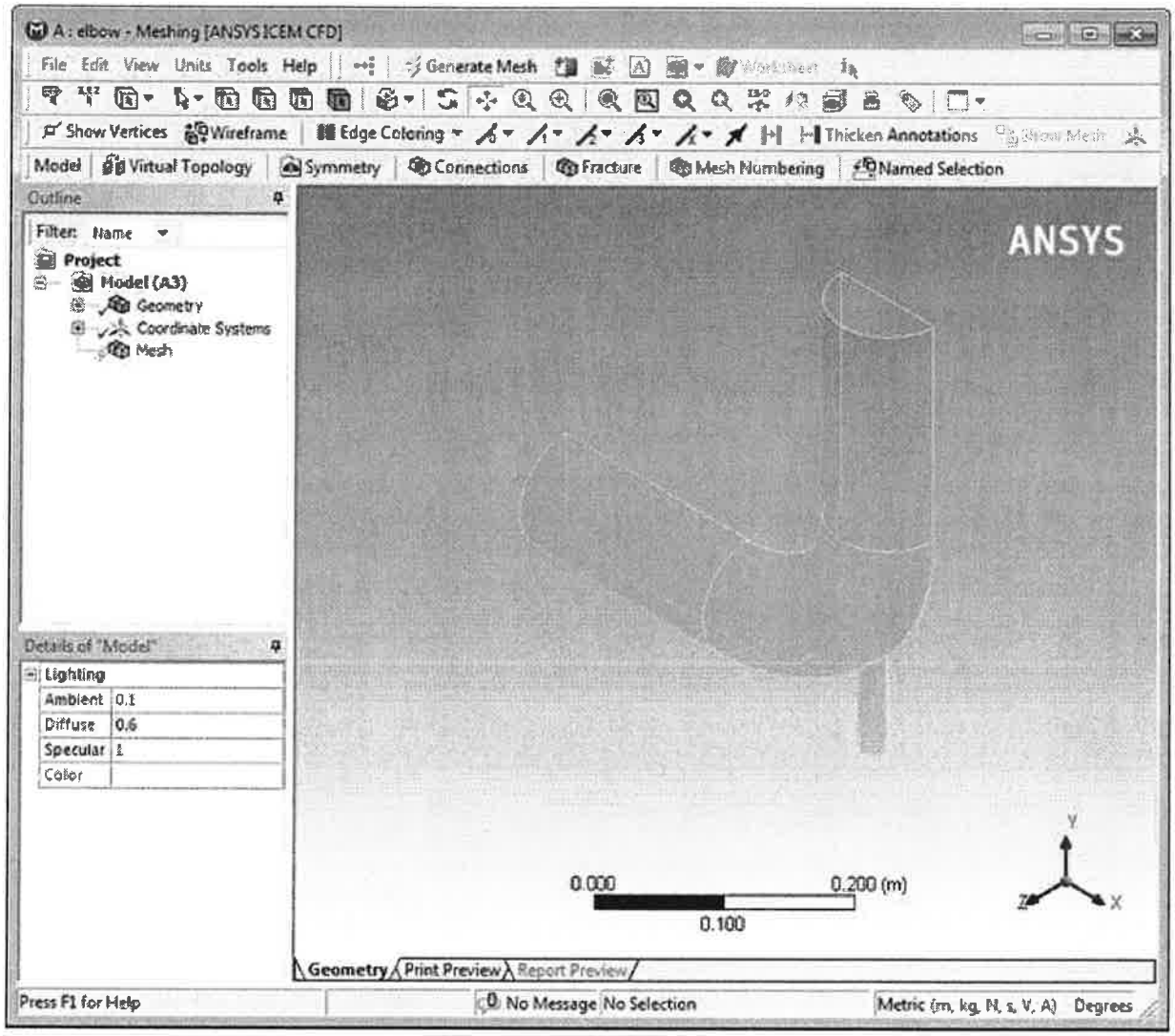
---

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 16.0 - © 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.

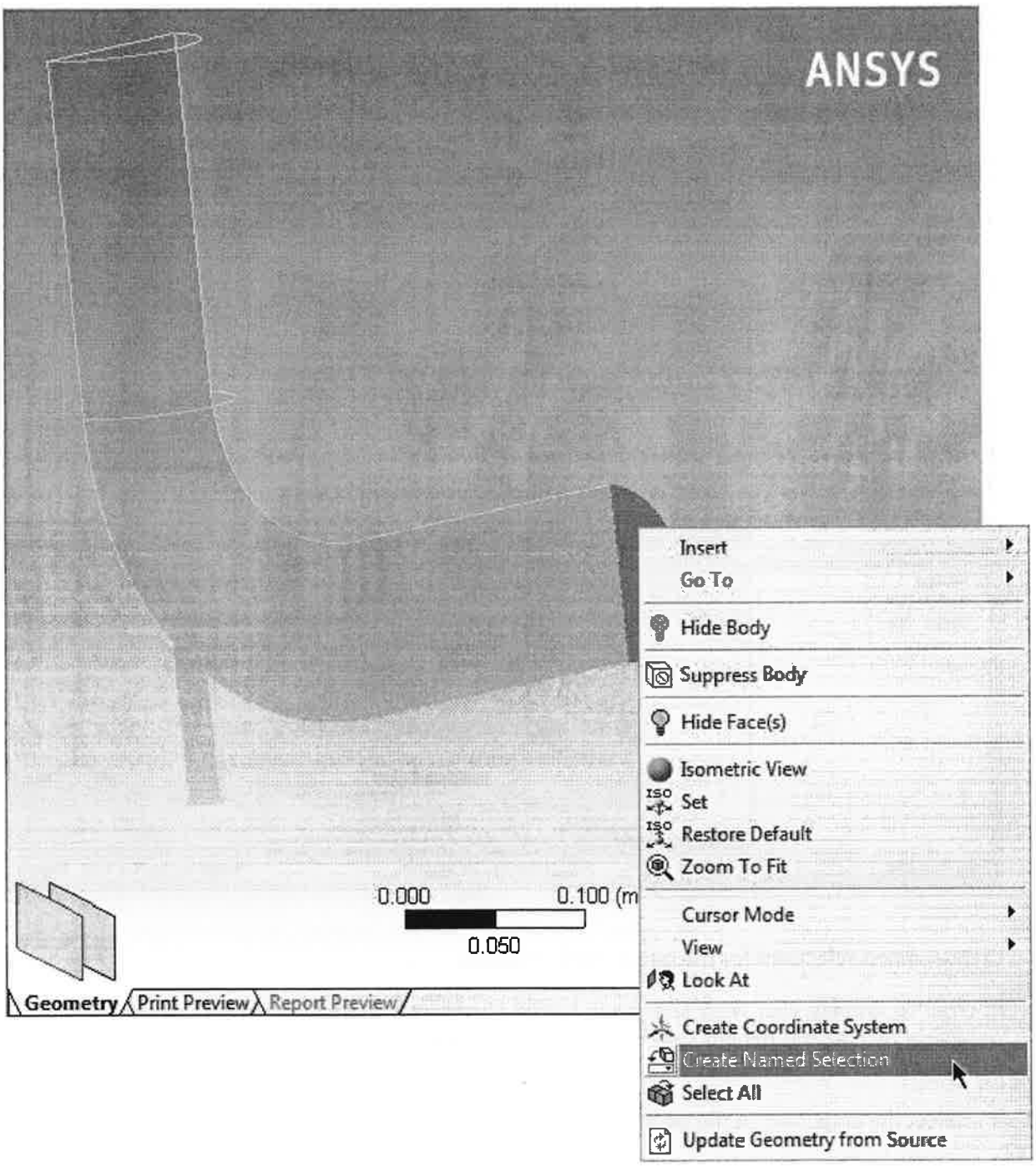
   Tip

   - 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 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

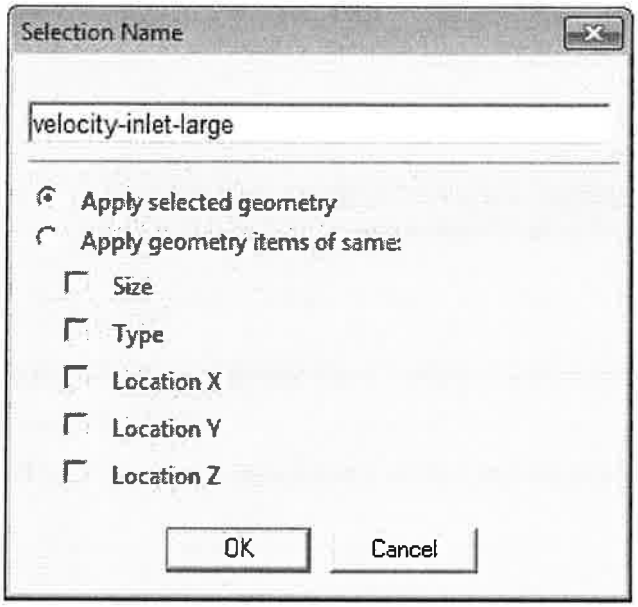
Figure 1.8: Selecting a Face to Name



This displays the **Selection Name** dialog box.

Release 16.0 - © 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`).

Important

It is important to note that by using the strings "velocity inlet" and "pressure outlet" in the named selections (with or without hyphens or underscore characters), ANSYS Fluent automatically detects and assigns the corresponding boundary types accordingly.

---

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`.

By creating a named selection called `Fluid` for the fluid body you will ensure that ANSYS Fluent automatically detects that the volume is a fluid zone and treats it accordingly.

4. Set basic meshing parameters for the ANSYS Meshing application.

For this analysis, you will adjust several meshing parameters to obtain a finer mesh.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. In the **Outline** view, select **Mesh** under **Project/Model** to display the **Details of "Mesh"** view below the **Outline** view.

Important

Note that because the ANSYS Meshing application automatically detects that you are going to perform a CFD fluid flow analysis using ANSYS Fluent, the **Physics Preference** is already set to **CFD** and the **Solver Preference** is already set to **Fluent**.

---

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 on the default value, **Coarse**, and selecting **Fine** from the drop-down list.

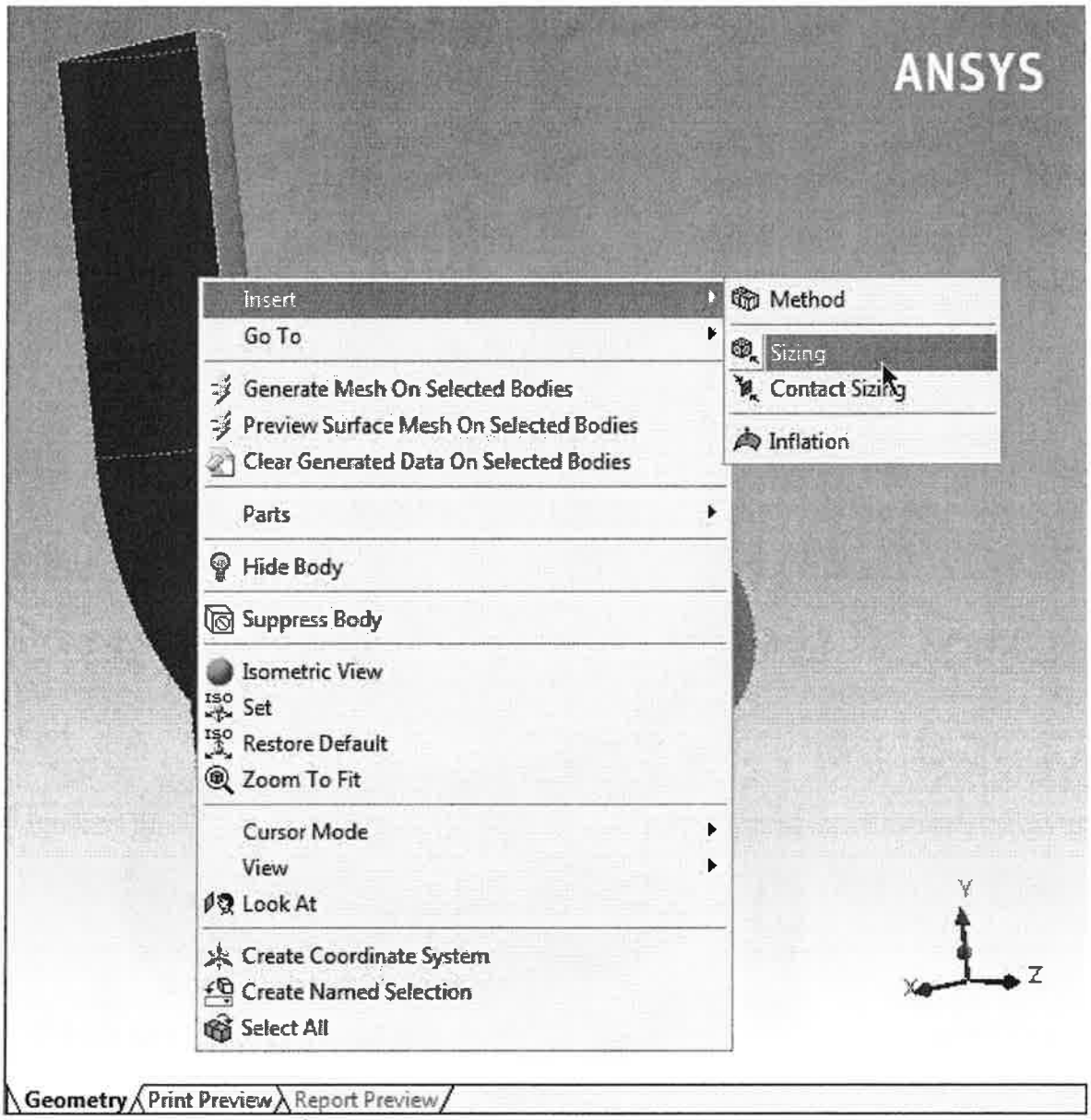
   ii. Change **Smoothing** to **High**.

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 16.0 - © 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. Click the new **Body Sizing** control in the **Outline** tree.

iv. Enter `6e-3` for **Element Size** and press **Enter**.

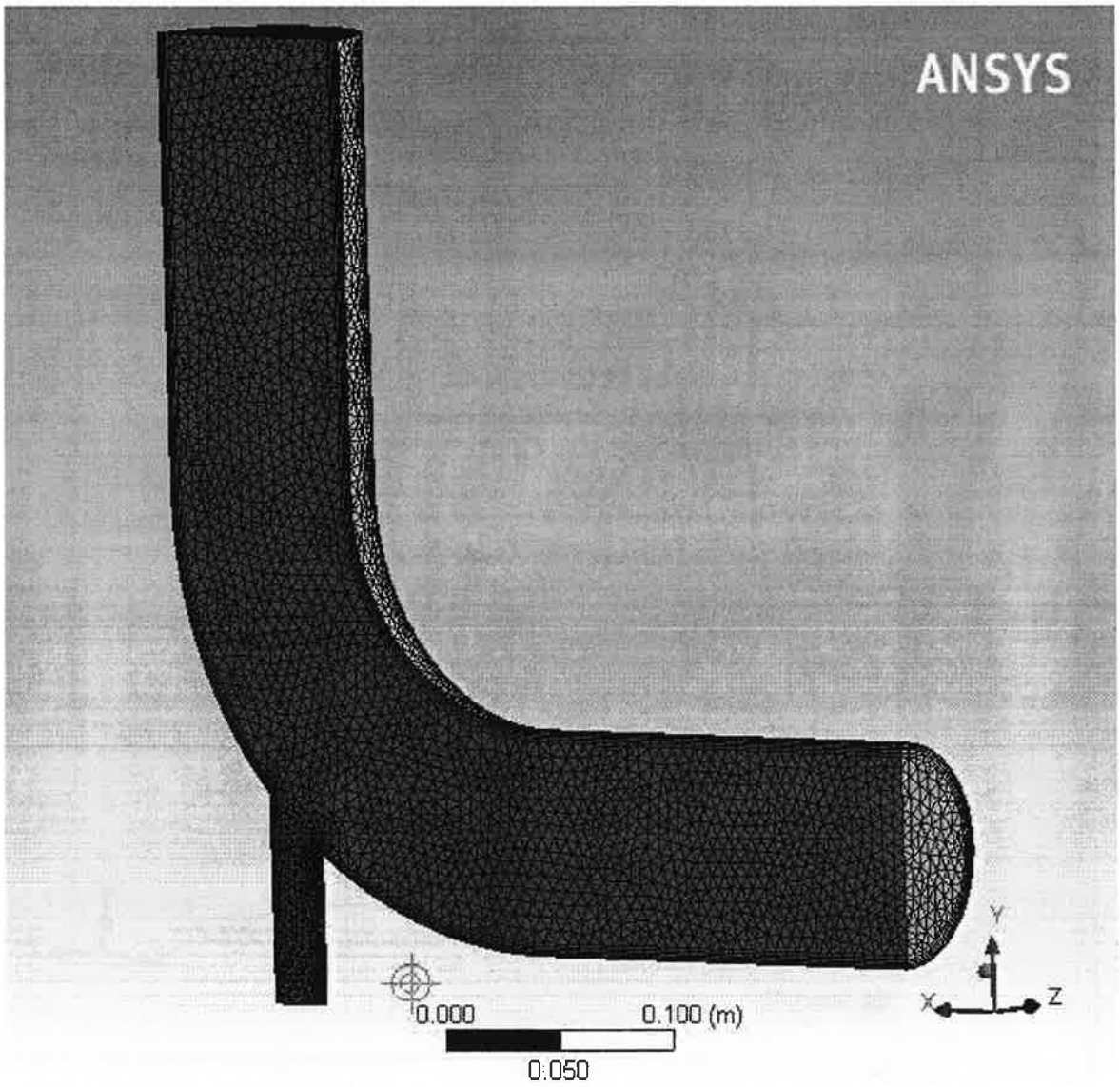
d. Click again on **Mesh** in the **Outline** view and expand the **Inflation** node in the **Details of "Mesh"** view to reveal additional inflation parameters. Change **Use Automatic Inflation** to **Program Controlled**.

5. Generate the mesh.

Right-click **Mesh** in the project **Outline** tree, and select **Update** in the context menu.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.10: The Computational Mesh for the Elbow Geometry in the ANSYS Meshing Application



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 16.0 - © 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.

1.4.5. Setting Up the CFD Simulation in ANSYS Fluent

Now that you have created a computational mesh for the elbow geometry, in this step you will set up a CFD analysis using ANSYS Fluent, then review the list of files generated by ANSYS Workbench.

1. Start ANSYS Fluent.

   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.

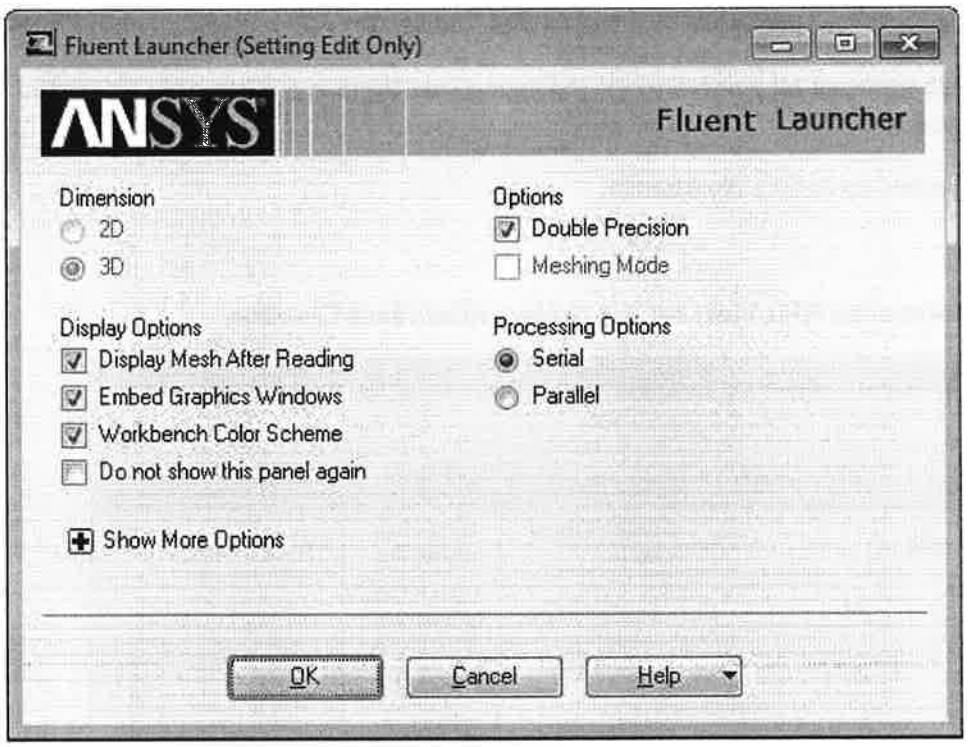
   When ANSYS Fluent is first started, the Fluent Launcher is displayed, enabling you to view and/or set certain ANSYS Fluent start-up options.

   Note

   The Fluent Launcher allows you to decide which version of ANSYS Fluent you will use, based on your geometry and on your processing capabilities.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.12: Fluent Launcher



a. Ensure that the proper options are enabled.

Important

Note that the **Dimension** setting is already filled in and cannot be changed, since ANSYS Fluent automatically sets it based on the mesh or geometry for the current system.

i. Ensure that **Serial** from the **Processing Options** list is enabled.

ii. Select **Double Precision** under **Options**.

iii. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, 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.

Note

Fluent will retain your preferences for future sessions.

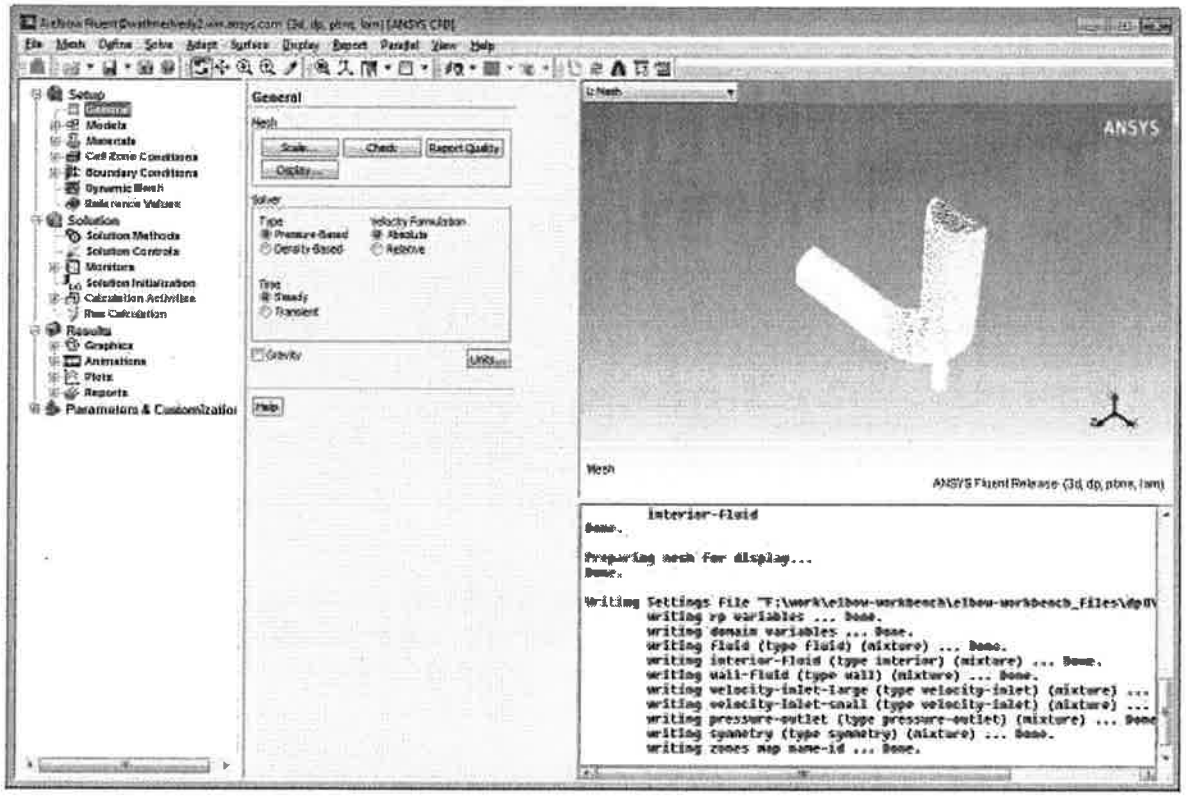
Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Click **OK** to launch ANSYS Fluent.

Note

The ANSYS Fluent settings file (`FFF.set`) is written as soon as ANSYS Fluent opens.

Figure 1.13: The ANSYS Fluent Application

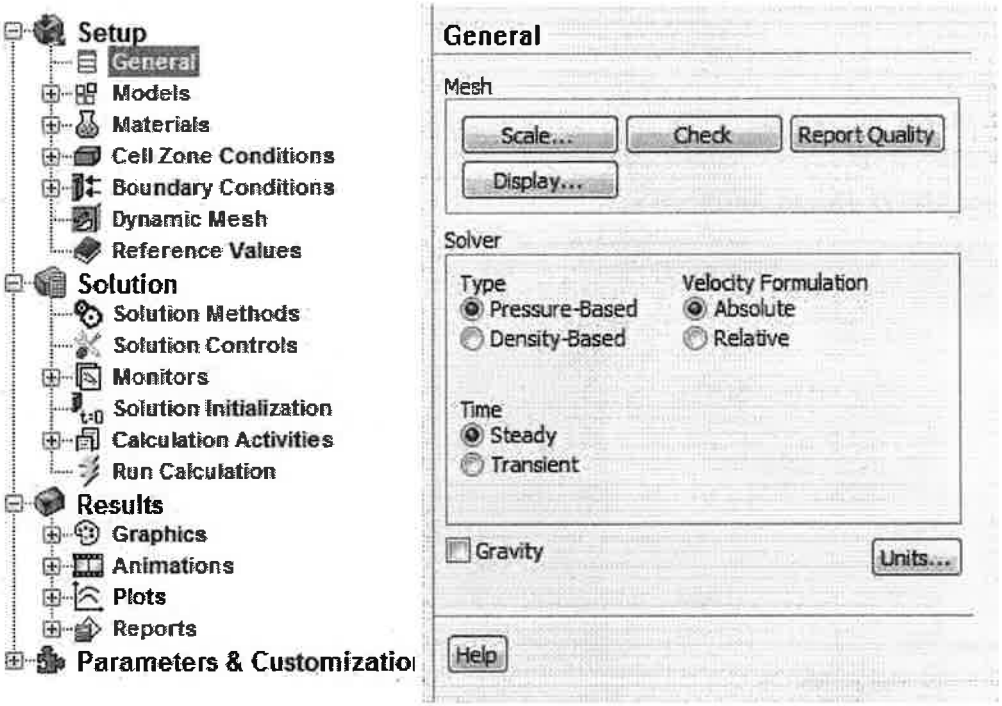


2. Set general settings for the CFD analysis.

   Click **General** in the tree to open the **General** task page where you can perform the mesh-related activities and to choose a solver.

   Setup → General

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note

Many of the common setup, solution and results commands are also accessible through the submenus that open by right-clicking the tree items under the relevant tree branches.

a. Change the units for length.

Because you want to specify and view values based on a unit of length in millimeters from within ANSYS Fluent, change the units of length within ANSYS Fluent from meters (the default) to millimeters.

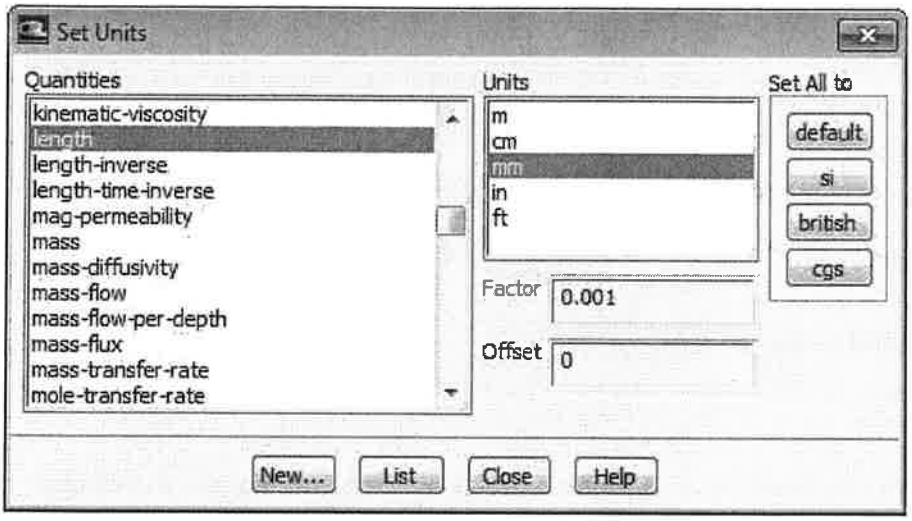
Important

Note that the ANSYS Meshing application automatically converts and exports meshes for ANSYS Fluent using meters (m) as the unit of length regardless of what units were used to create them. This is so you do not have to scale the mesh in ANSYS Fluent under ANSYS Workbench.

Setup → **General** → **Units...**

This displays the **Set Units** dialog box.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



i. Select **length** in the **Quantities** list.

ii. Select **mm** in the **Units** list.

iii. 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.

---

b. Check the mesh.

Setup → General → Check

Note

ANSYS Fluent will report the results of the mesh check in the console.

```
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.
```

Note

The minimum and maximum values may vary slightly when running on different platforms. The mesh check will list the minimum and maximum x and y values from

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the mesh in the default SI unit of meters. It will also report a number of other mesh features that are checked. Any errors in the mesh will be reported at this time. Ensure that the minimum volume is not negative as ANSYS Fluent cannot begin a calculation when this is the case.

c. Review the mesh quality.

Setup → General → Report Quality

Note

ANSYS Fluent will report the results of the mesh quality below the results of the mesh check in the console.

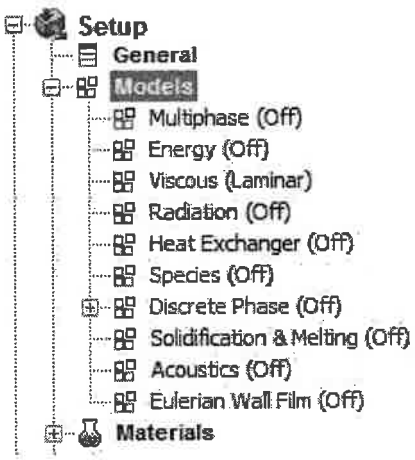
```
Mesh Quality:
Minimum Orthogonal Quality = 1.77384e-01
(Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.)
Maximum Ortho Skew = 7.60448e-01
(Ortho Skew ranges from 0 to 1, where values close to 1 correspond to low quality.)
Maximum Aspect Ratio = 1.92111e+01
```

Note

The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. Checking the quality of your mesh is, therefore, an important step in performing a robust simulation. Minimum cell orthogonality is an important indicator of mesh quality. Values for orthogonality can vary between 0 and 1 with lower values indicating poorer quality cells. In general, the minimum orthogonality should not be below 0.01 with the average value significantly larger. The high aspect ratio cells in this mesh are near the walls and are a result of the boundary layer inflation applied in the meshing step. For more information about the importance of mesh quality refer to Mesh Quality in the User's Guide.

3. Set up your models for the CFD simulation.

Setup → Models

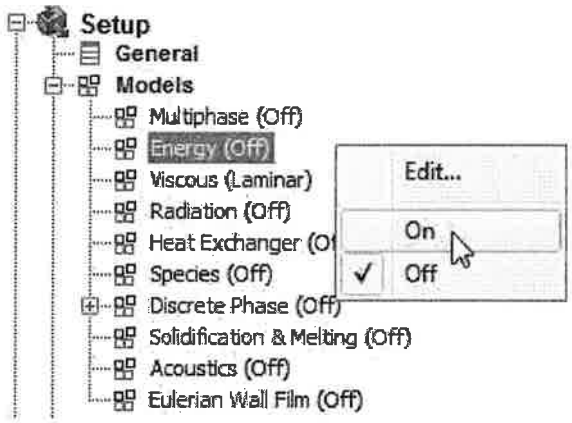


Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Enable heat transfer by activating the energy equation.

In the tree, right-click **Energy** under **Models** and from the submenu that opens, click **On**.

Setup → Models → Energy → On



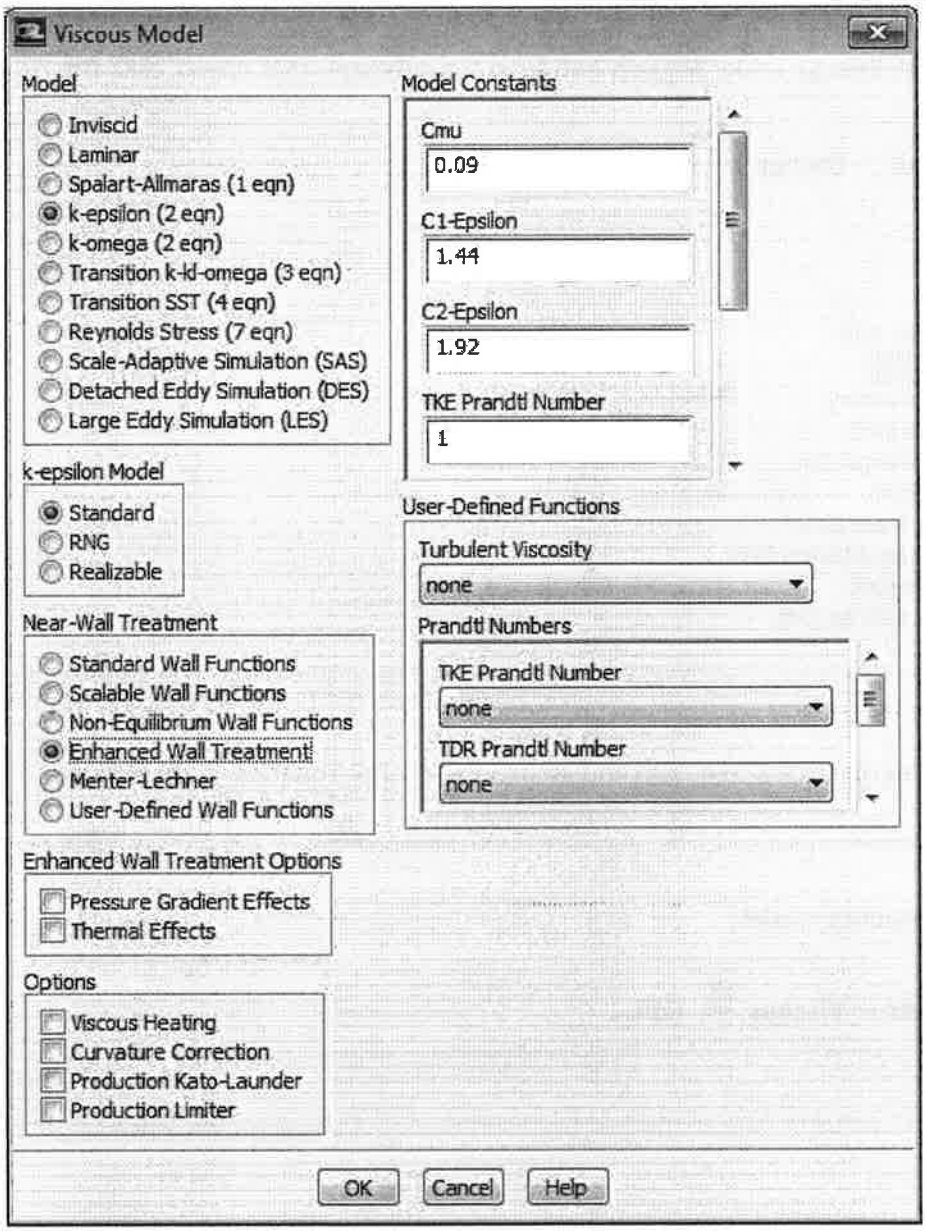
Note

You can also double-click a tree item and enable the energy equation in the **Energy** dialog box.

b. Enable the k-ε turbulence model.

Setup → Models → Viscous → Edit...

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



i. Select **k-epsilon** from the **Model** list.

Note

The **Viscous Model** dialog box will expand.

ii. Use the default **Standard** from the **k-epsilon Model** list.

iii. Select **Enhanced Wall Treatment** for the **Near-Wall Treatment**.

Note

The default Standard Wall Functions are generally applicable if the first cell center adjacent to the wall has a y+ larger than 30. In contrast, the Enhanced Wall

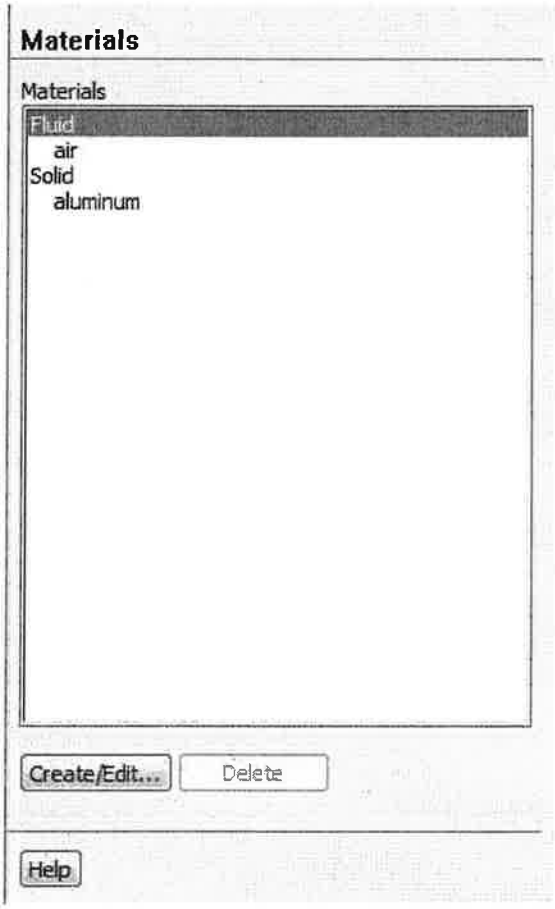
Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Treatment option provides consistent solutions for all y+ values. Enhanced Wall Treatment is recommended when using the k-epsilon model for general single-phase fluid flow problems. For more information about Near Wall Treatments in the k-epsilon model refer to Setting Up the k-ε Model in the User's Guide.

iv. Click **OK** to accept the model and close the **Viscous Model** dialog box.

4. Set up your materials for the CFD simulation.

Setup → Materials



a. Create a new material called **water** using the **Create/Edit Materials** dialog box (Figure 1.14: The Create/Edit Materials Dialog Box (p. 36)).

Setup → Materials → Fluid → Create/Edit...

i. Type `water` for **Name**.

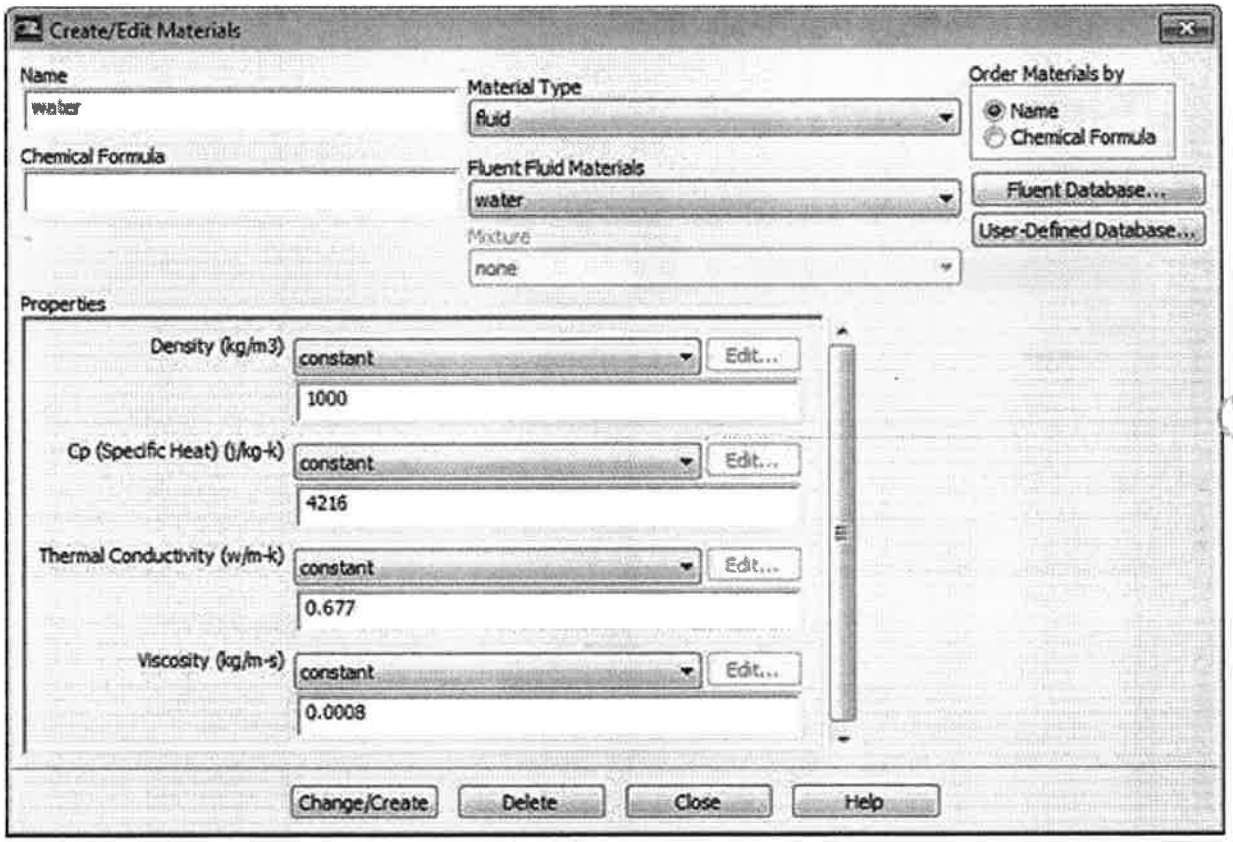
ii. Enter the following values in the **Properties** group box:

Property	Value
Density	1000 kg/m^3

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Value
c_p (Specific Heat)	4216 $J/kg-K$
Thermal Conductivity	0.677 $W/m-K$
Viscosity	8e-04 $kg/m-s$

Figure 1.14: The Create/Edit Materials Dialog Box

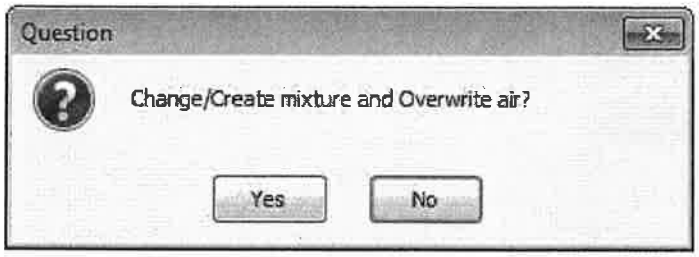


iii. Click **Change/Create**.

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 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Extra

You could have copied the material **water-liquid (h2o < l >)** from the materials database (accessed by clicking the **ANSYS Fluent Database...** button). If the properties in the database are different from those you want to use, you can edit the values in the **Properties** group box in the **Create/Edit Materials** dialog box and click **Change/Create** to update your local copy. The original copy will not be affected.

iv. Ensure that there are now two materials (water and air) defined locally by examining the **Fluent Fluid Materials** drop-down list.

Note

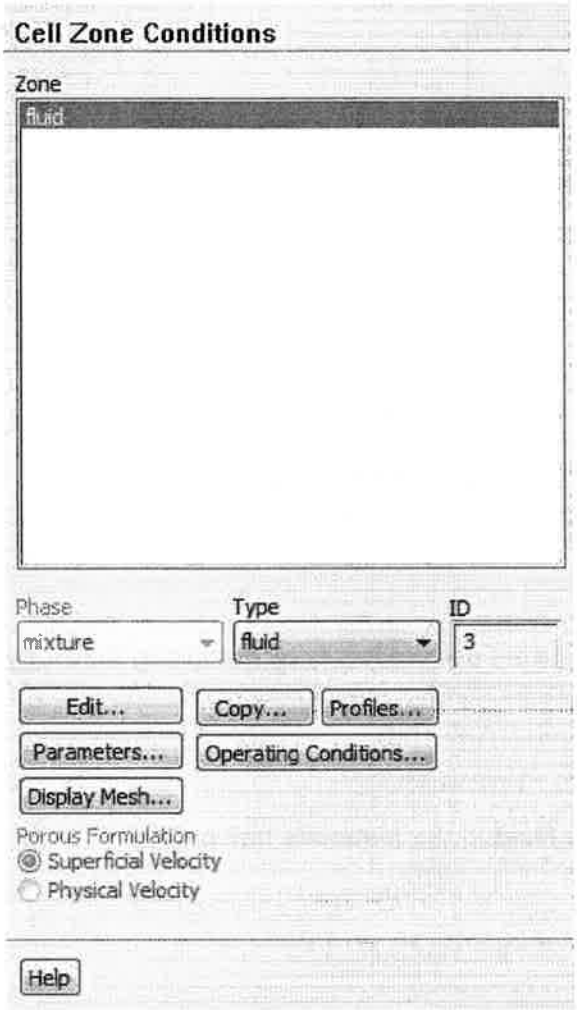
Both the materials will also be listed under **Fluid** in the **Materials** task page.

v. Close the **Create/Edit Materials** dialog box.

5. Set up the cell zone conditions for the CFD simulation.

Setup → Cell Zone Conditions

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



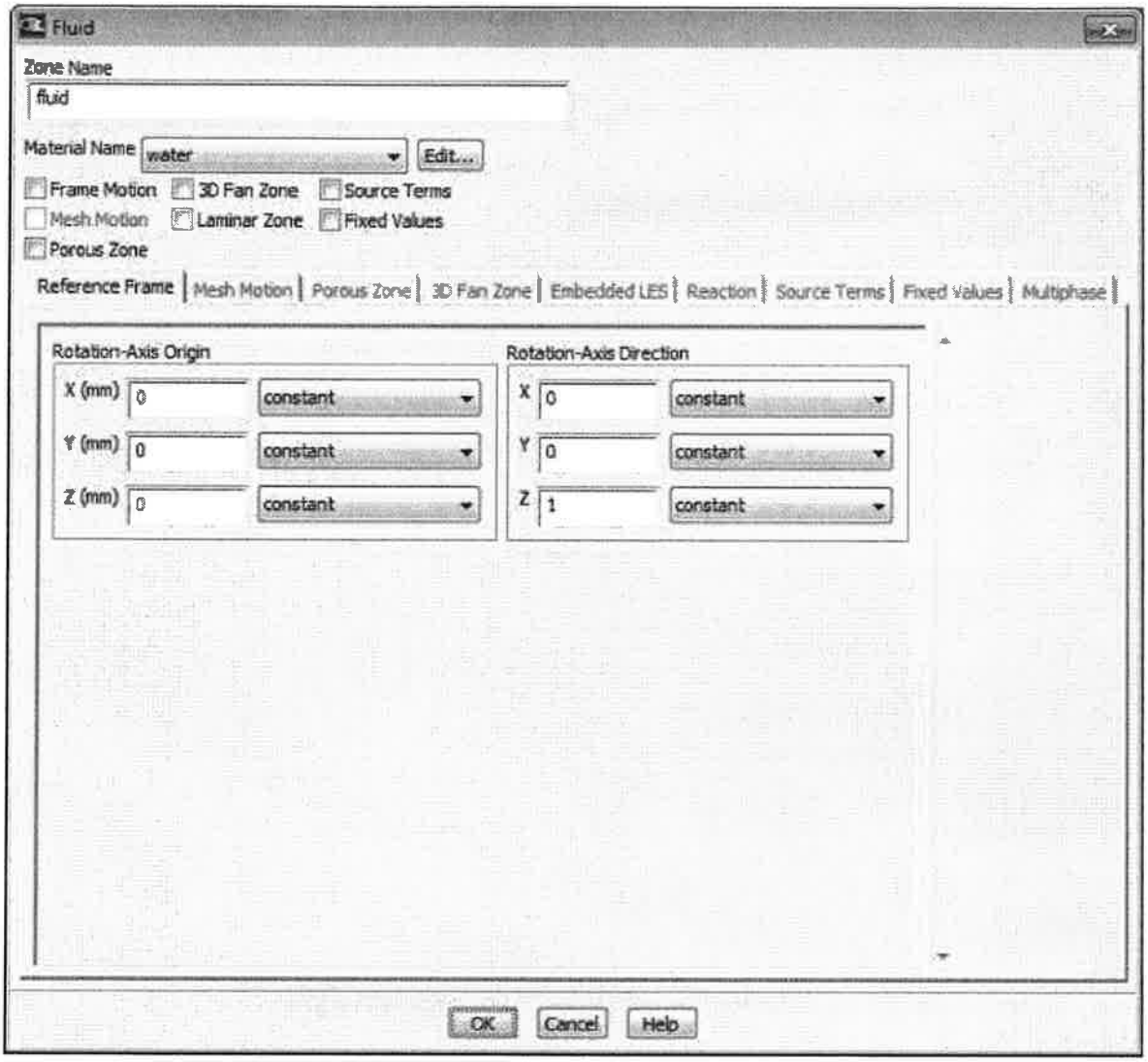
a. Set the cell zone conditions for the fluid zone.

   i. In the **Cell Zone Conditions** task page, in the **Zone** list, select **fluid** and click **Edit...** to open the **Fluid** dialog box.

Note

You can also double-click a list or tree item in order to open the corresponding dialog box.

Release 16.0 - © 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.

6. Set up the boundary conditions for the CFD analysis.

Setup → Boundary Conditions

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



a. Set the boundary conditions at the cold inlet (**velocity-inlet-large**).

Setup → **Boundary Conditions** → **velocity-inlet-large** → **Edit...**

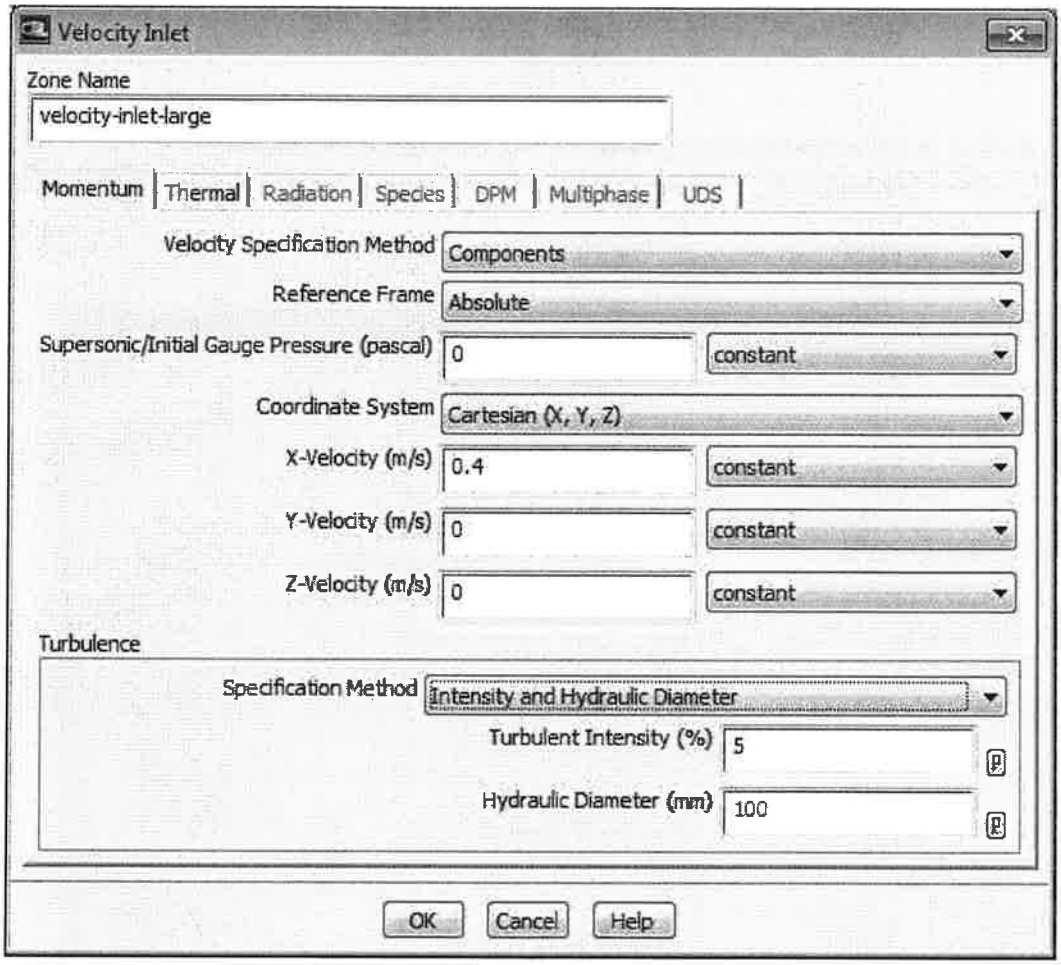
This opens the **Velocity Inlet** dialog box.

Tip

If you are unsure of which inlet zone corresponds to the cold inlet, you can use the mouse to probe for mesh information in the graphics window. If you click the right mouse button with the pointer on any node in the mesh, information about the associated zone will be displayed in the ANSYS Fluent console, including the name of the zone. The zone you probed will be automatically selected from the **Zone** selection list in the **Boundary Conditions** task page.

Alternatively, you can click the probe button ( ) in the graphics toolbar and click the left mouse button on any node. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly. The information will be displayed in the console.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



i. Select **Components** from the **Velocity Specification Method** drop-down list.

Note

The **Velocity Inlet** dialog box will expand.

---

ii. Enter `0.4` m/s for **X-Velocity**.

iii. Retain the default value of `0` m/s for both **Y-Velocity** and **Z-Velocity**.

iv. In the **Turbulence** group box, 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

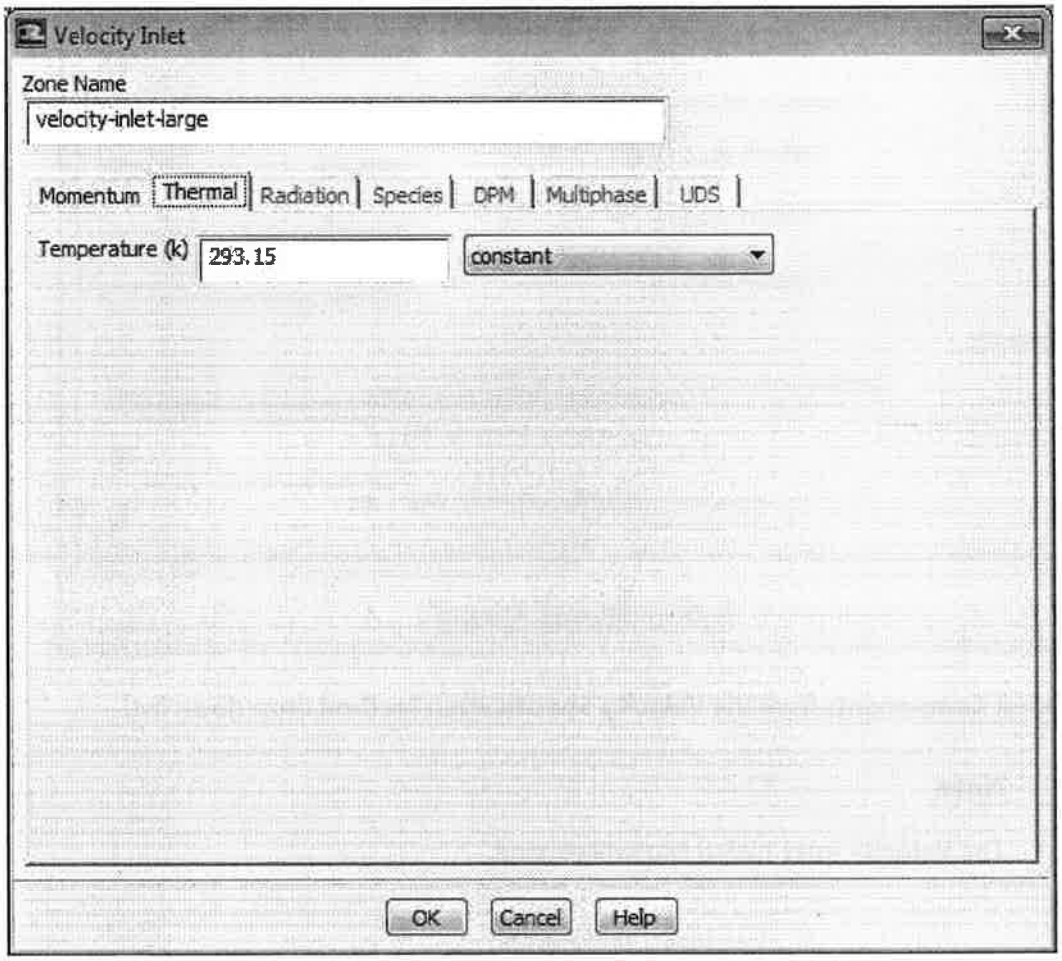
The hydraulic diameter D_h is defined as:

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$D_h = \frac{4A}{P_w}$$

where A is the cross-sectional area and P_w is the wetted perimeter.

vii. Click the **Thermal** tab.



viii. Enter `293.15` K for **Temperature**.

ix. Click **OK** to close the **Velocity Inlet** dialog box.

b. In a similar manner, set the boundary conditions at the hot inlet (**velocity-inlet-small**), using the values in the following table:

Setup → Boundary Conditions → velocity-inlet-small → Edit...

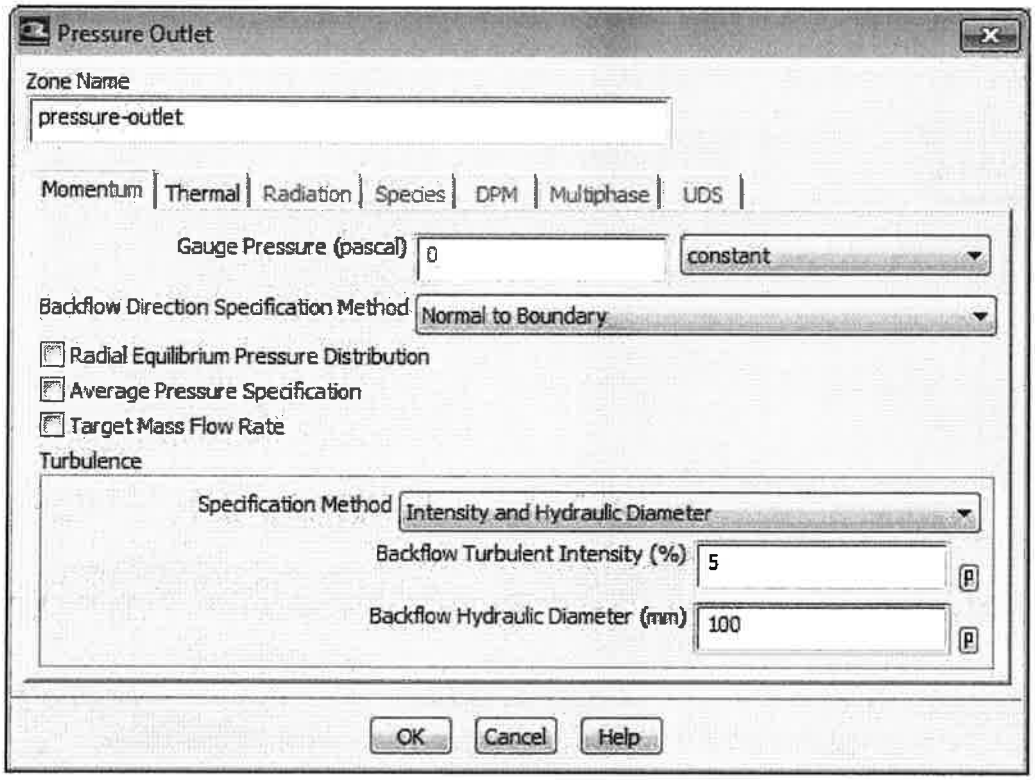
Velocity Specification Method	Components
X-Velocity	`0` m/s
Y-Velocity	`1.2` m/s
Z-Velocity	`0` m/s

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Velocity Specification Method	Components
Specification Method	Intensity & Hydraulic Diameter
Turbulent Intensity	5%
Hydraulic Diameter	25 mm
Temperature	313.15 K

c. Set the boundary conditions at the outlet (**pressure-outlet**), as shown in the **Pressure Outlet** dialog box.

Setup → Boundary Conditions → pressure-outlet → Edit...



Note

ANSYS Fluent will use the backflow conditions only if the fluid is flowing into the computational domain through the outlet. Since backflow might occur at some point during the solution procedure, you should set reasonable backflow conditions to prevent convergence from being adversely affected.

7. Set up solution parameters for the CFD simulation.

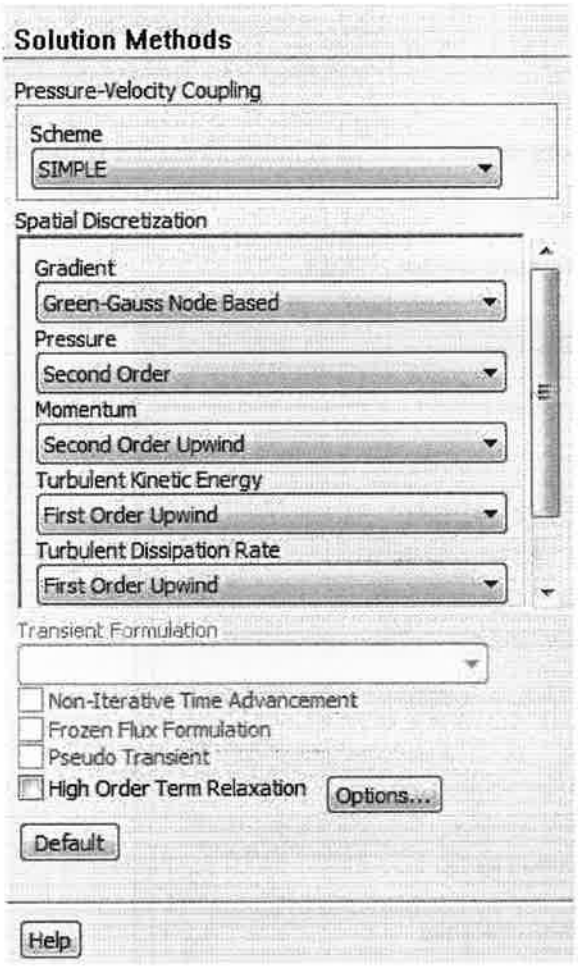
Note

In the steps that follow, you will set up and run the calculation using the task pages listed under the **Solution** branch in the tree.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Change the Gradient method.

Solution → Solution Methods

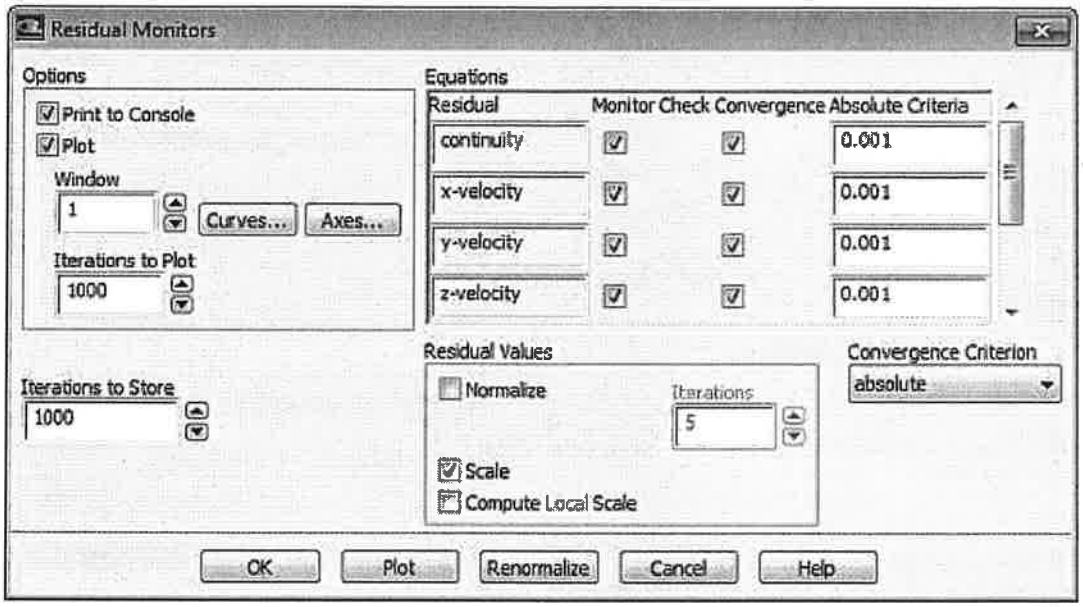


In the **Spatial Discretization** section of the **Solution Methods** pane, change the **Gradient** to **Green-Gauss Node Based**. This gradient method is suggested for tetrahedral meshes.

b. Examine the convergence criteria for the equation residuals.

Solution → Monitors → Residuals Edit...

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



i. Ensure that **Plot** is enabled in the **Options** group box.

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.

Note

By default, all variables will be monitored and checked by ANSYS Fluent as a means to determine the convergence of the solution.

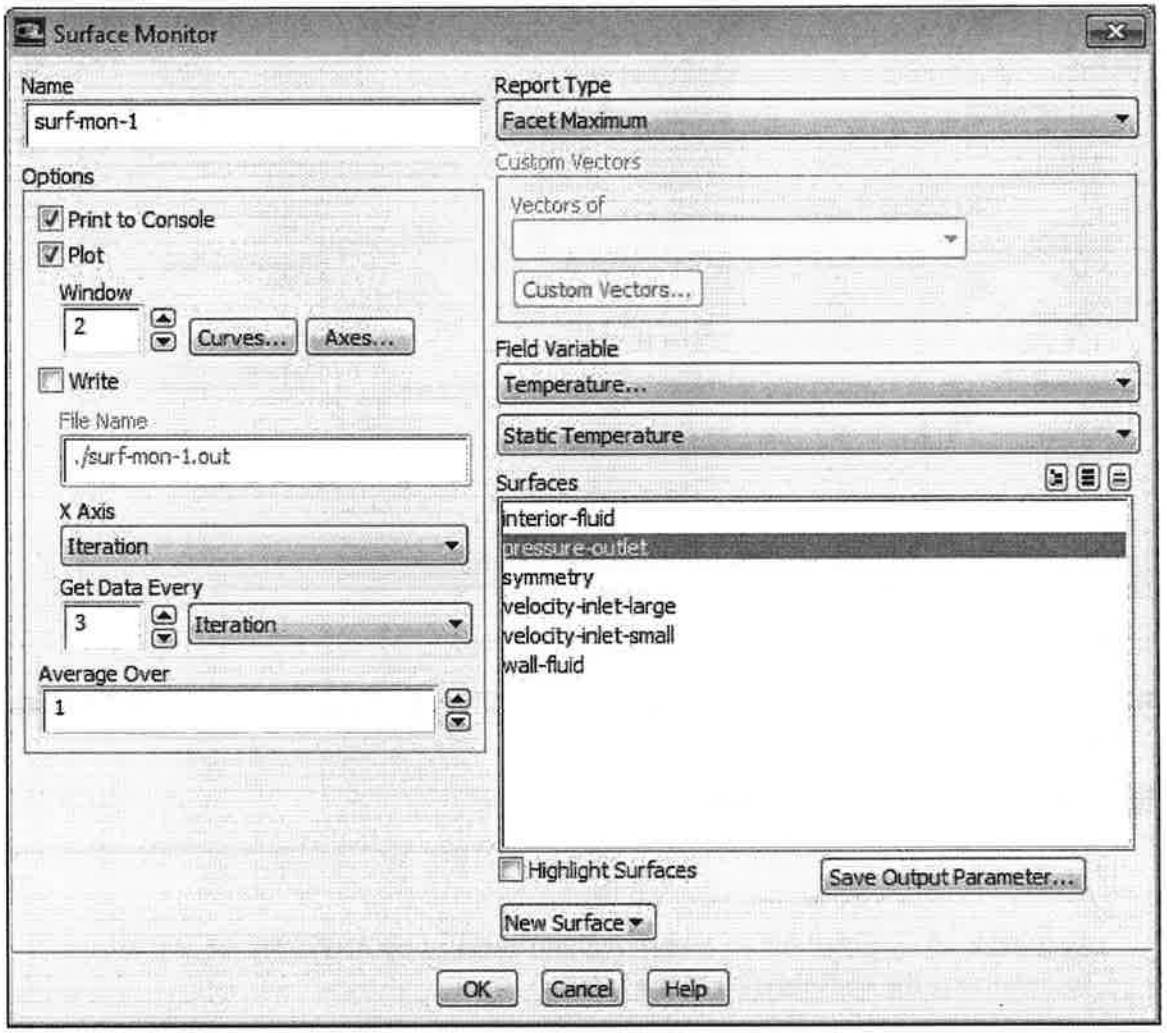
---

c. Create a surface monitor at the outlet (**pressure-outlet**)

It is good practice to monitor physical solution quantities in addition to equation residuals when assessing convergence.

Solution → Monitors → Surface **New...**

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



i. Retain the default entry of **surf-mon-1** for the **Name** of the surface monitor.

ii. Enable the **Plot** option for **surf-mon-1**.

iii. Set **Get Data Every** to 3 by clicking the up-arrow button.

This setting instructs ANSYS Fluent to update the plot of the surface monitor and write data to a file after every 3 iterations during the solution.

iv. Select **Facet Maximum** from the **Report Type** drop-down list.

v. Select **Temperature...** and **Static Temperature** from the **Field Variable** drop-down lists.

vi. Select **pressure-outlet** from the **Surfaces** selection list.

vii. Click **OK** to save the surface monitor settings and close the **Surface Monitor** dialog box.

*The name and report type of the surface monitor you created will be displayed under the **Surface** tree item.*

d. Initialize the flow field.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → Solution Initialization

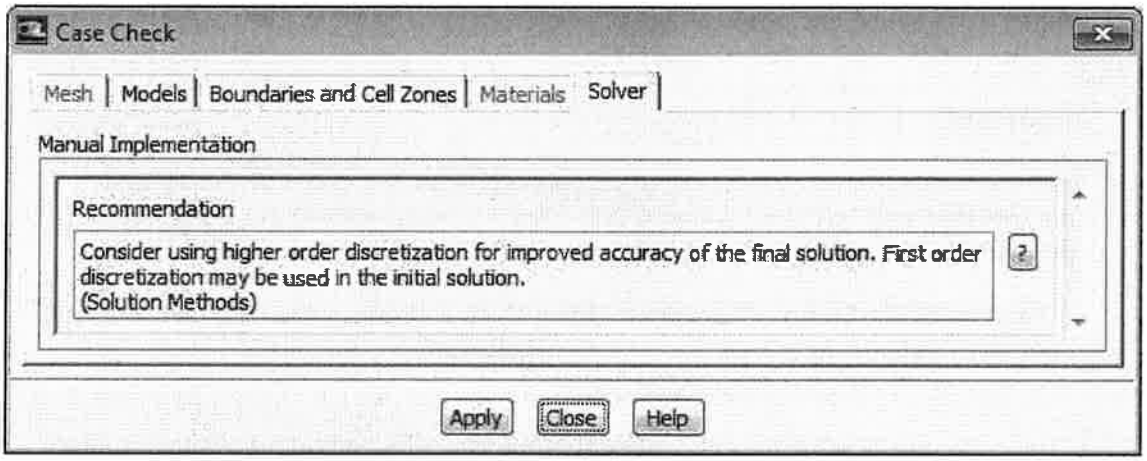


i. Keep the default of **Hybrid Initialization** from the **Initialization Methods** group box.

ii. Click **Initialize**.

e. Check to see if the case conforms to best practices.

Solution → Run Calculation → Check Case



i. Click the **Models** and **Solver** tabs and examine the **Recommendation** in each. These recommendations can be ignored for this tutorial. The issues they raise will be addressed in later tutorials.

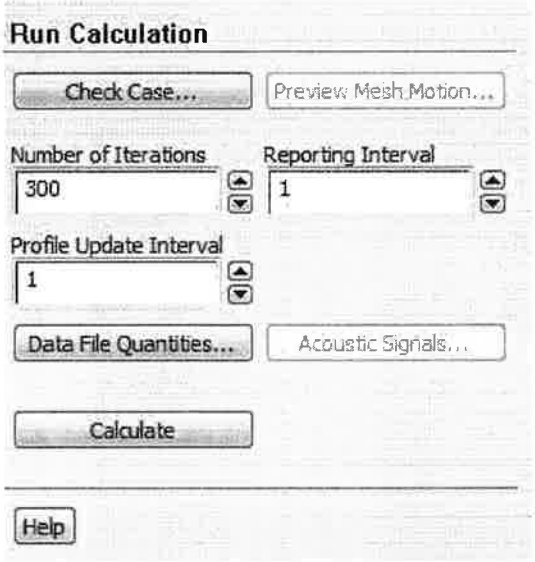
ii. Close the **Case Check** dialog box.

8. Calculate a solution.

a. Start the calculation by requesting 300 iterations.

Solution → Run Calculation

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



i. Enter `300` for **Number of Iterations**.

ii. Click **Calculate**.

Important

Note that the ANSYS Fluent settings file (`FFF.set`) is updated in the **Files** view of the ANSYS Workbench before the calculation begins.

Important

Note that while the program is calculating the solution, the states of the **Setup** and **Solution** cells in the fluid flow ANSYS Fluent analysis system in ANSYS Workbench are changing. For example:

- The state of the **Setup** cell becomes **Up-to-Date** and the state of the **Solution** cell becomes **Refresh Required** after the **Run Calculation** task page is visited and the number of iterations is specified.
- The state of the **Solution** cell is **Update Required** while iterations are taking place.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The state of the **Solution** cell is **Up-to-Date** when the specified number of iterations are complete (or if convergence is reached).

Note

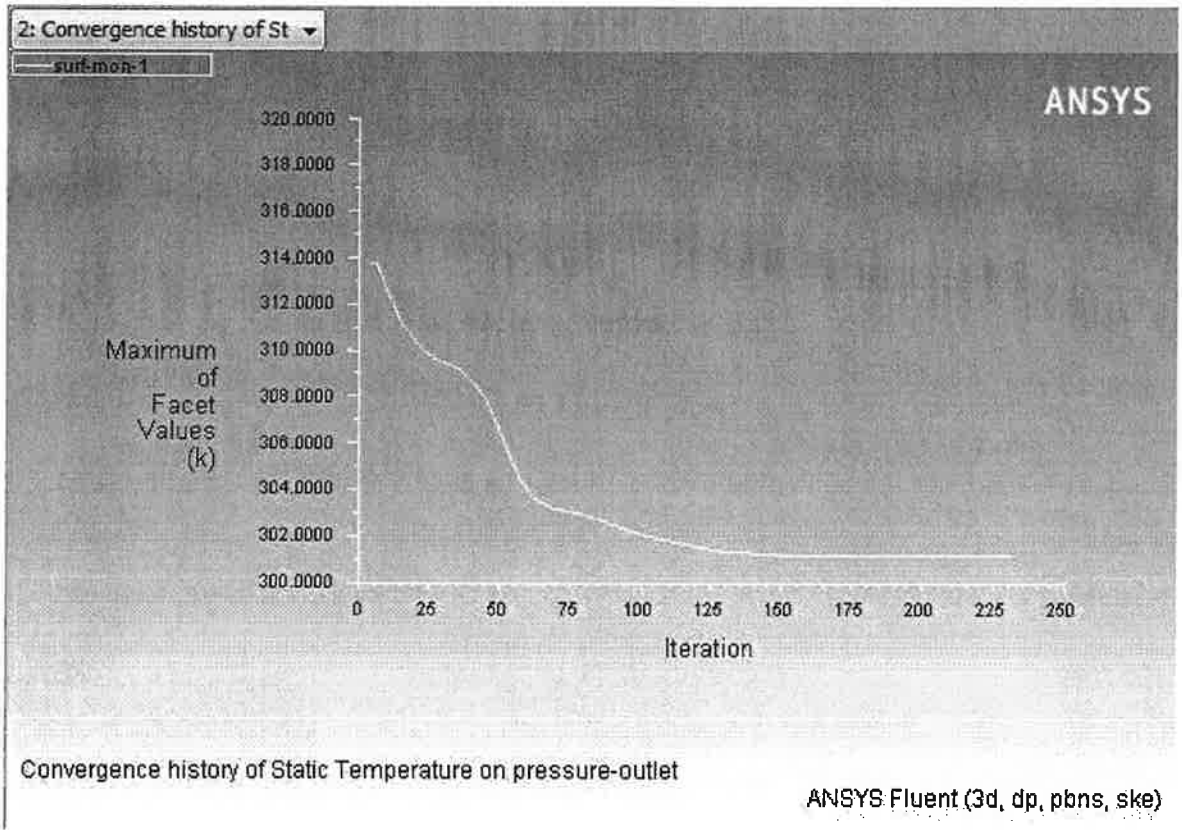
As the calculation progresses, the surface monitor history will be plotted in the graphics window (Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet (p. 49)).

Note

The solution will be stopped by ANSYS Fluent when the residuals reach their specified values or after 300 iterations. The exact number of iterations will vary depending on the platform being used. An **Information** dialog box will open to alert you that the calculation is complete. Click **OK** in the **Information** dialog box to proceed.

Because the residual values vary slightly by platform, the plot that appears on your screen may not be exactly the same as the one shown here.

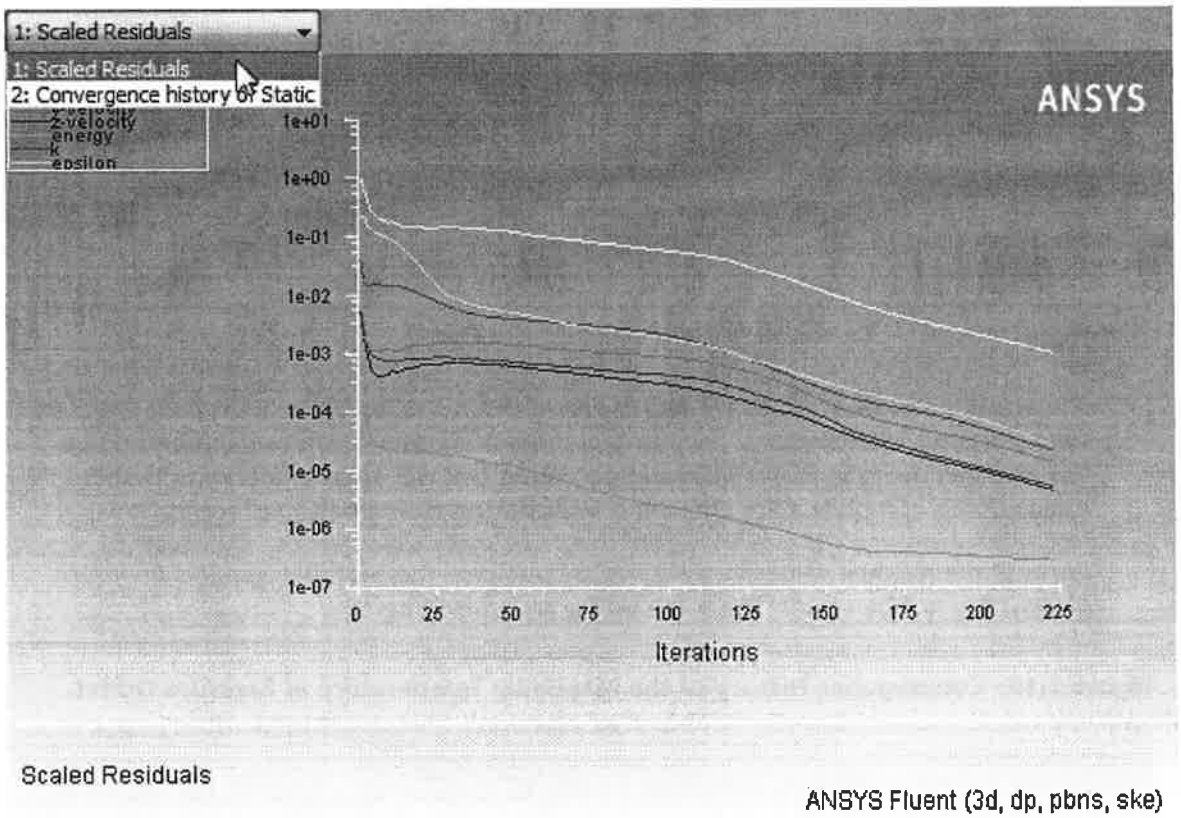
Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet



You can display the residuals history (Figure 1.16: Residuals for the Converged Solution (p. 50)), by selecting it from the graphics window drop-down list.

Release 16.0 - © 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 (p. 50) and Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet (p. 49)).

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:

- 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

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

may show that the residuals have stagnated and do not change with further iterations. This could also be considered as convergence.

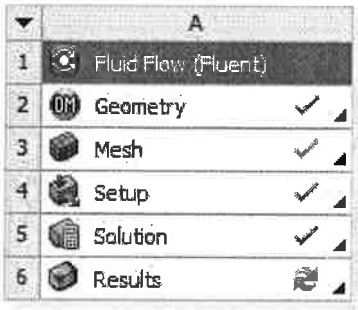
- The overall mass, momentum, energy, and scalar balances are obtained.

  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.

9. View the list of files generated by ANSYS Workbench.

   View → Files

   Note that the status of the **Solution** cell is now up-to-date.



elbow

1.4.6. Displaying Results in ANSYS Fluent and CFD-Post

In this step, you will display the results of the simulation in ANSYS Fluent, display the results in CFD-Post, then review the list of files generated by ANSYS Workbench.

1. Display results in ANSYS Fluent.

   With ANSYS Fluent still running, you can perform a simple evaluation of the velocity and temperature contours on the symmetry plane. Later, you will use CFD-Post (from within ANSYS Workbench) to perform the same evaluation.

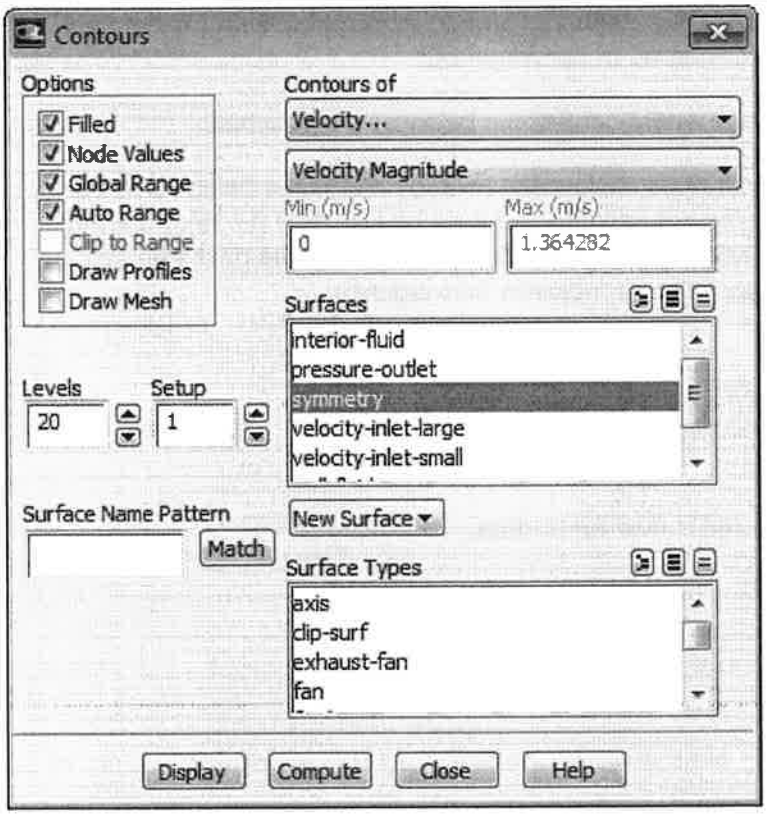
   a. Display filled contours of velocity magnitude on the symmetry plane (Figure 1.17: Velocity Distribution Along Symmetry Plane (p. 53)).

      Results → Graphics → Contours → Edit...

      Note

      You can also double-click a tree item in order to open the corresponding dialog box.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



i. In the **Contours** dialog box, enable **Filled** in the **Options** group box.

ii. Ensure that **Node Values** is enabled in the **Options** group box.

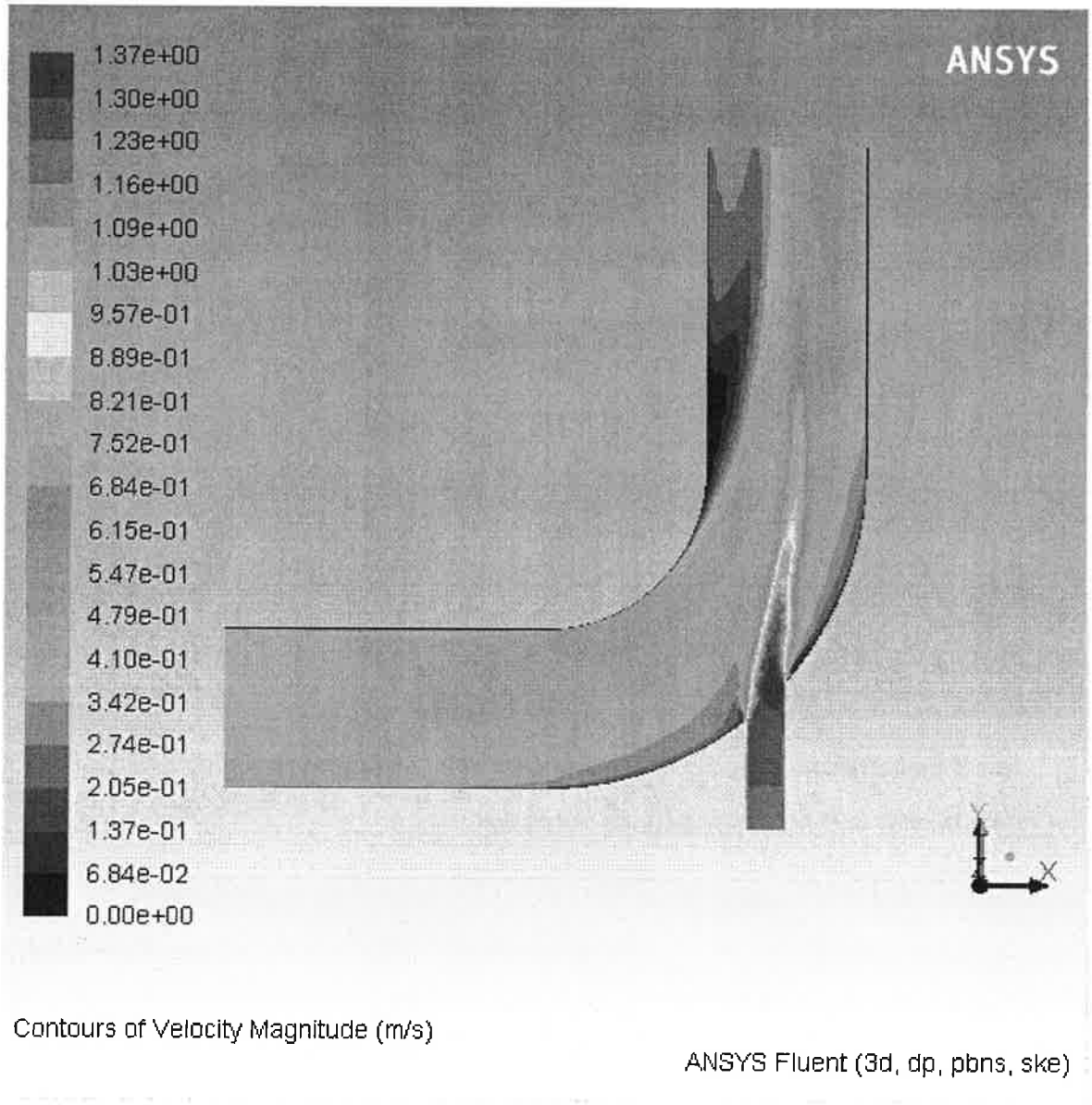
iii. Select **Velocity...** and **Velocity Magnitude** from the **Contours of** drop-down lists.

iv. Select **symmetry** from the **Surfaces** selection list.

v. Click **Display** to display the contours in the active graphics window.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

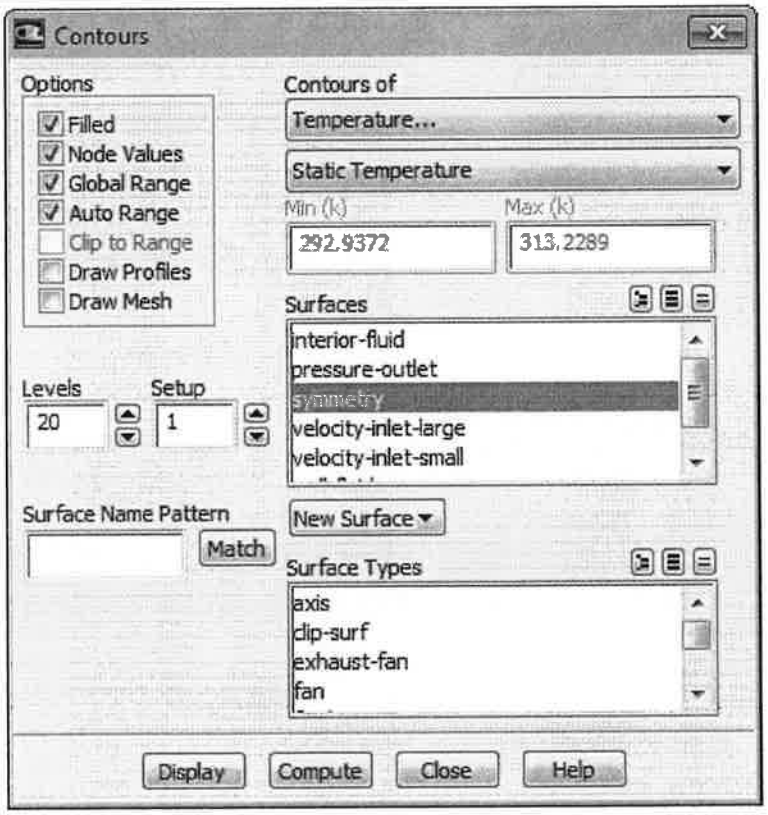
Figure 1.17: Velocity Distribution Along Symmetry Plane



b. Display filled contours of temperature on the symmetry plane (Figure 1.18: Temperature Distribution Along Symmetry Plane (p. 55)).

Results → Graphics → Contours Edit...

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

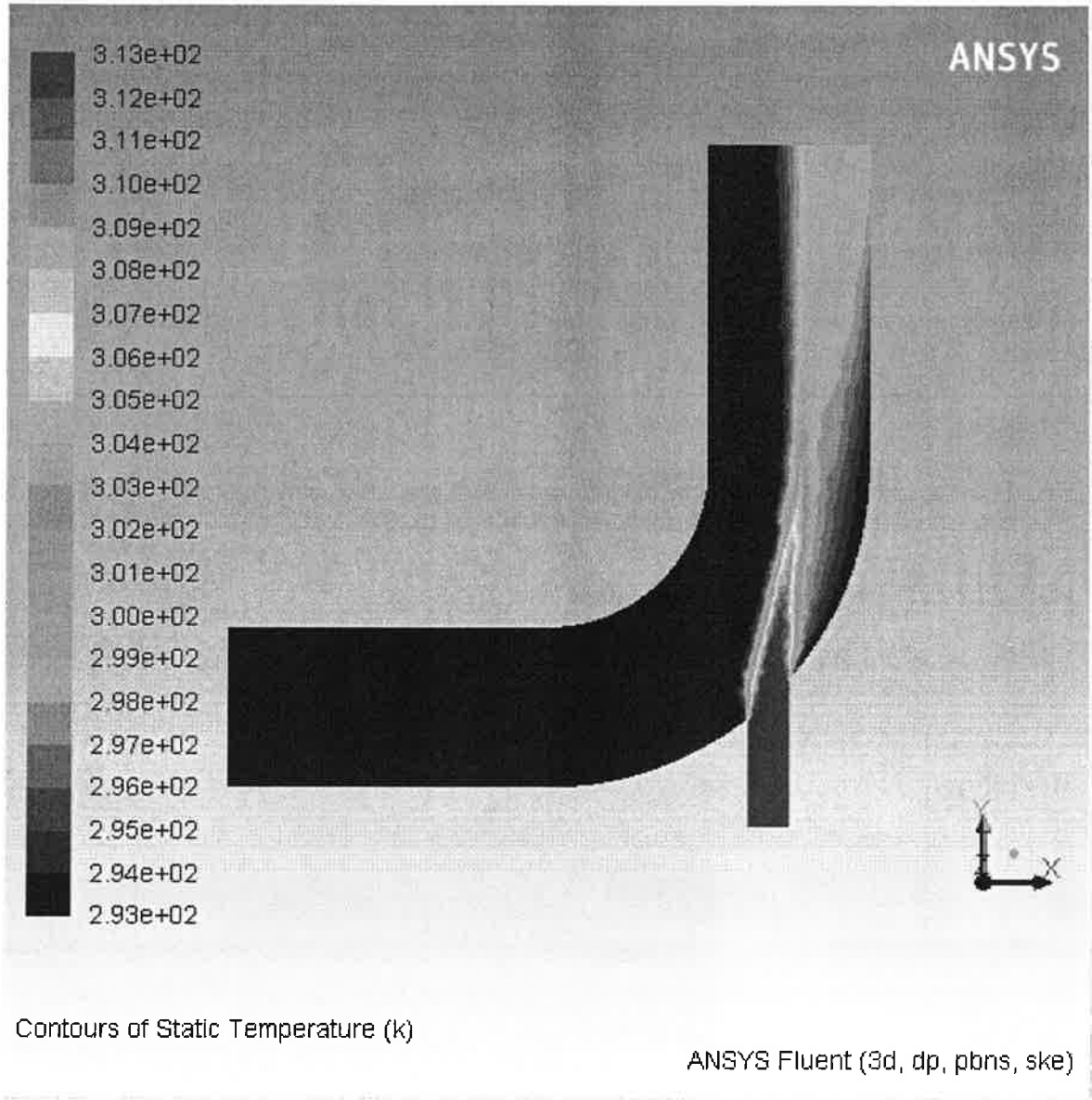


i. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

ii. Click **Display** and close the **Contours** dialog box.

Release 16.0 - © 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

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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.

2. Display results in CFD-Post.

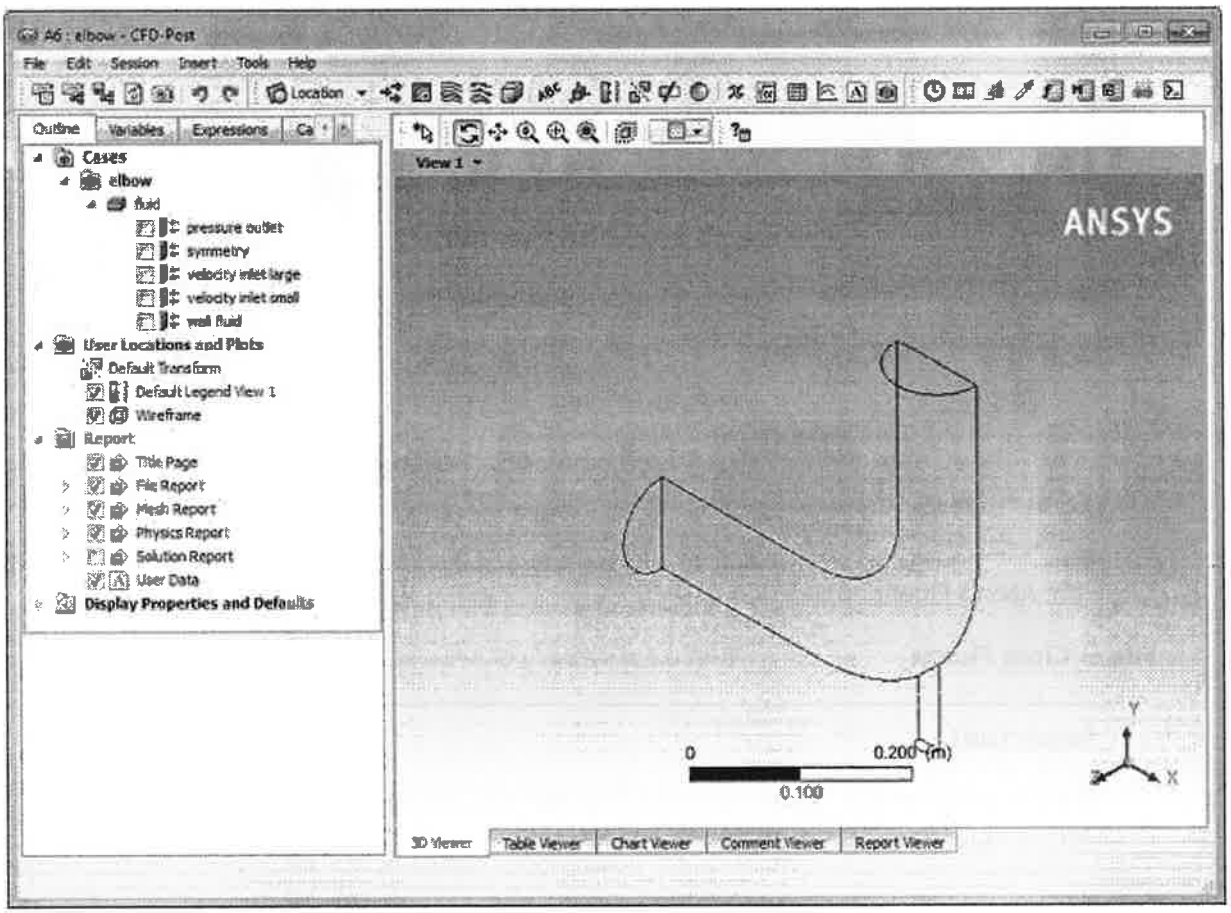
   a. Start CFD-Post.

   In the ANSYS Workbench **Project Schematic**, double-click the **Results** cell in the `elbow` fluid flow analysis system (cell `A6`). This displays the CFD-Post application. You can also right-click the **Results** cell to display the context menu where you can select the **Edit...** option.

   Note

   The elbow geometry is already loaded and is displayed in outline mode. ANSYS Fluent case and data files are also automatically loaded into CFD-Post.

Figure 1.19: The Elbow Geometry Loaded into CFD-Post



   b. Reorient the display.

   Click the blue Z axis on the axis triad in the bottom right hand corner of the graphics display to orient the display so that the view is of the front of the elbow geometry.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Ensure that *Highlighting* () is disabled.

d. Display filled contours of velocity magnitude on the symmetry plane (Figure 1.20: Velocity Distribution Along Symmetry Plane (p. 58)).

   i. Insert a contour object using the **Insert** menu item at the top of the CFD-Post window.

      Insert → Contour

      This displays the **Insert Contour** dialog box.

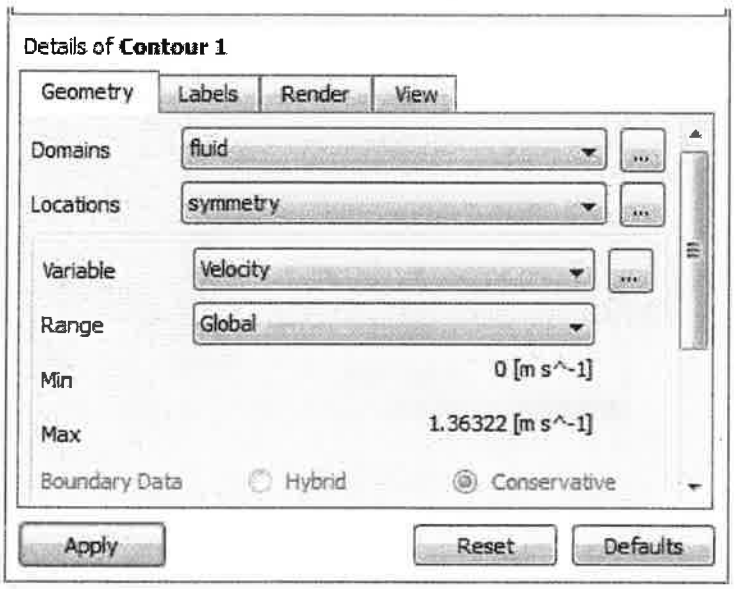
   ii. Keep the default name of the contour (`Contour 1`) and click **OK** to close the dialog box. This displays the **Details of Contour 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a contour object.

   iii. In the **Geometry** tab, from the **Domains** drop-down list, select **fluid**.

   iv. Select **symmetry** in the **Locations** list.

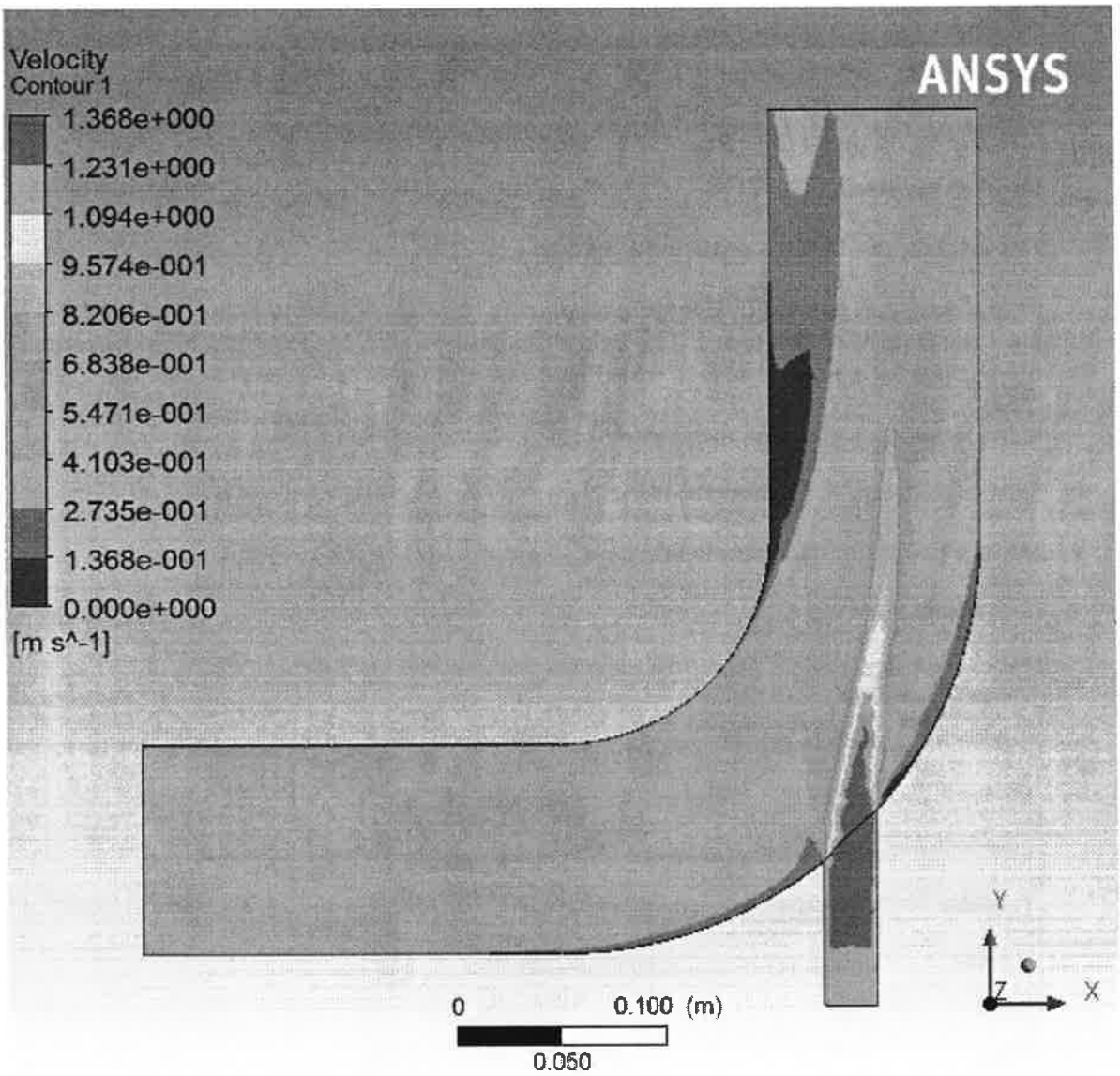
   v. Select **Velocity** in the **Variable** list.

   vi. Click **Apply**.



Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.20: Velocity Distribution Along Symmetry Plane



e. Display filled contours of temperature on the symmetry plane (Figure 1.21: Temperature Distribution Along Symmetry Plane (p. 59)).

i. Under **User Locations and Plots**, clear the check box beside the **Contour 1** object to disable the **Contour 1** object and hide the first contour display.

ii. Insert a contour object.

Insert → Contour

This displays the **Insert Contour** dialog box.

iii. Keep the default name of the contour (`Contour 2`) and click **OK** to close the dialog box. This displays the **Details of Contour 2** view below the **Outline** view.

iv. In the **Geometry** tab, from the **Domains** drop-down list, select **fluid**.

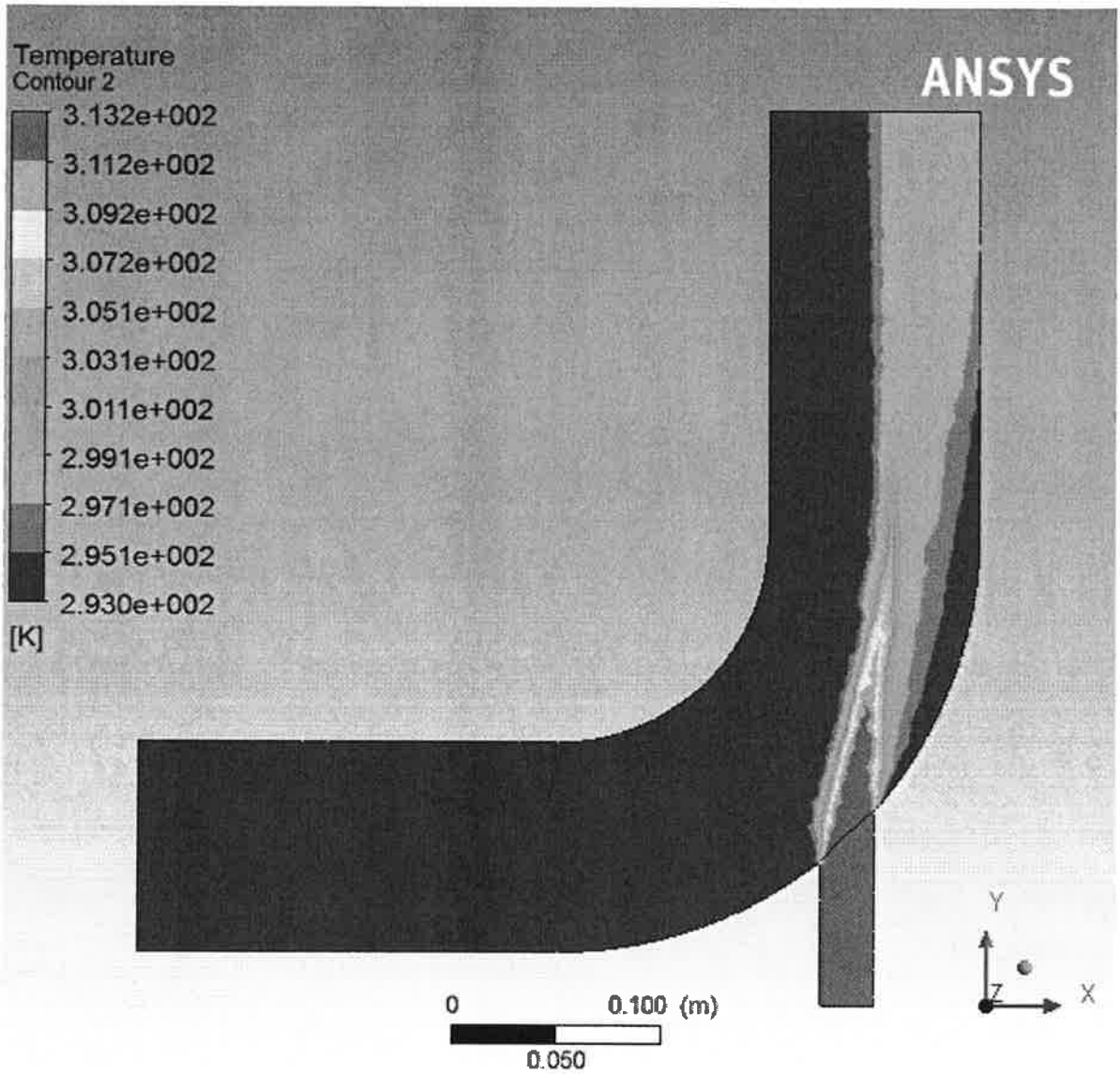
Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

v. Select **symmetry** in the **Locations** list.

vi. Select **Temperature** in the **Variable** list.

vii. Click **Apply**.

Figure 1.21: Temperature Distribution Along Symmetry Plane



3. Close the CFD-Post application by selecting **File** → **Close ANSYS CFD-Post** or by clicking the 'X' in the top right corner of the window.

Important

Note that the CFD-Post state files are automatically saved when you exit CFD-Post and return to ANSYS Workbench.

Release 16.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Save the `elbow-workbench` project in ANSYS Workbench.

5. View the list of files generated by ANSYS Workbench.

   View → Files

   Note the addition of the CFD-Post state file (`elbow.cst`) to the list of files. For more information about CFD-Post (and the files associated with it), see the CFD-Post documentation.

Release 16.0 - © 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 across a residential building – Importing Geometry

Abstract

This tutorial simply demonstrates the procedure of CFD simulation of natural ventilation across units of a residential building, compliant with the CFD methodology outlined in BCA Green Mark Code.

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).

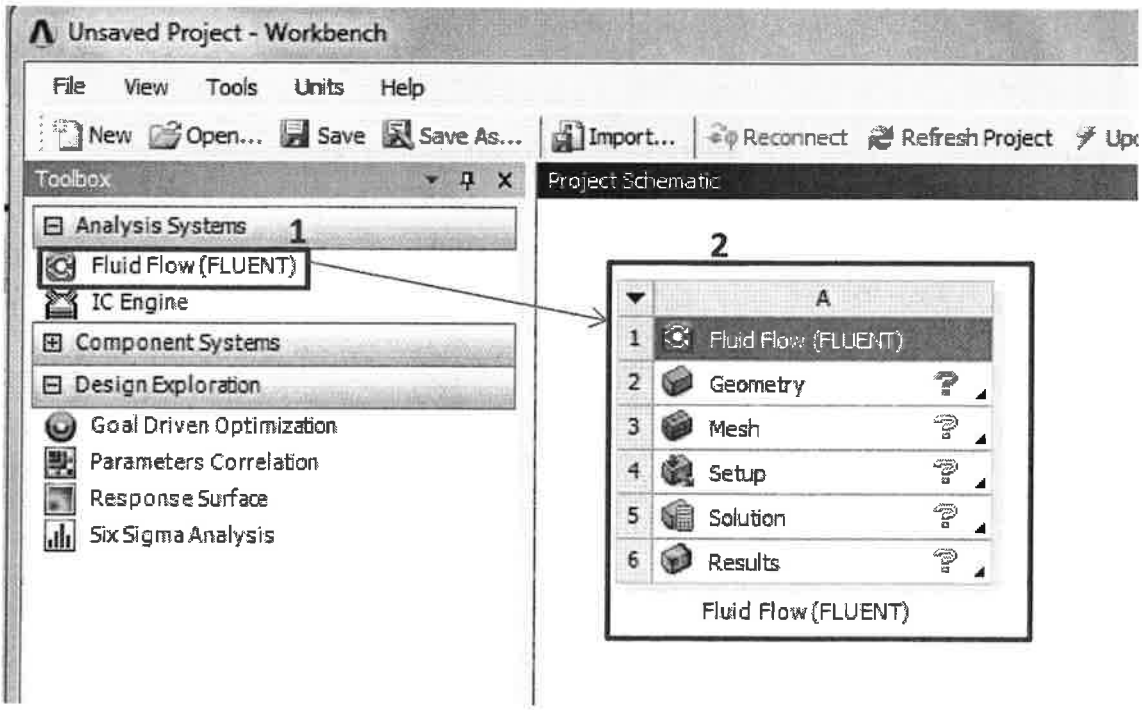
Please note that the tutorial is drafted using ANSYS Workbench 14.0.

Pre-requisite:

- Familiar with the workbench working environment (complete Tutorial 1 using workbench)
- Two external geometry files:
  - one-surrounding-building.igs – for surrounding block
  - target-building-name.agdb – for the building of concern
- One user define function:
  - profile-log-north.c – log-law wind profile (north wind)

Launch ANSYS Workbench

Create a new project for Fluid Flow (FLUENT) Analysis, by simply holding and dragging the icon for "Fluid Flow (FLUENT)" under Toolbox to the right into Project Schematic.

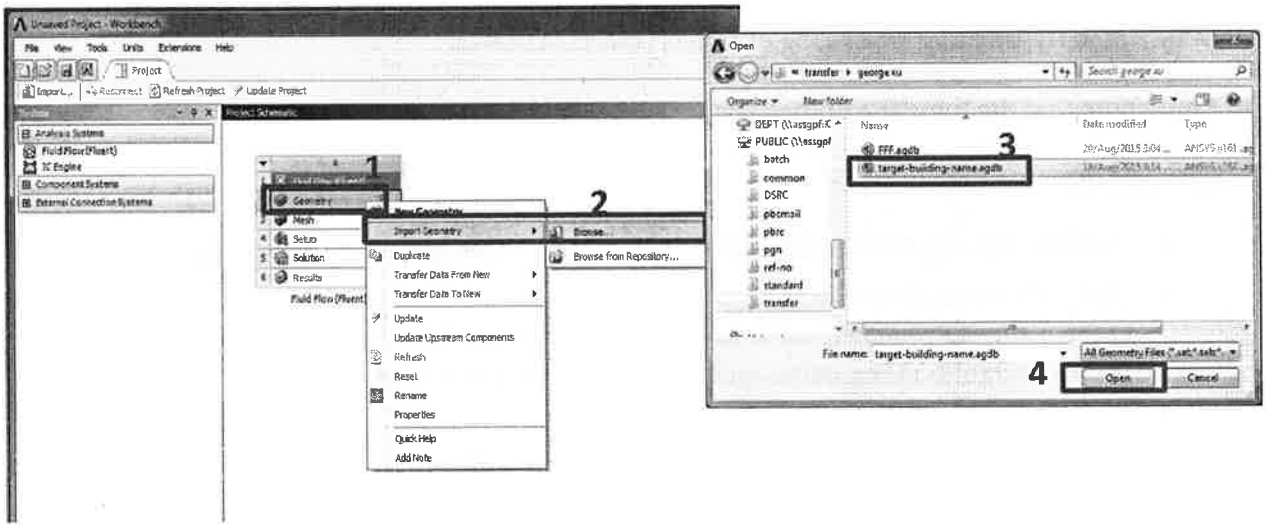


1. Geometry Creation in Design Modeler

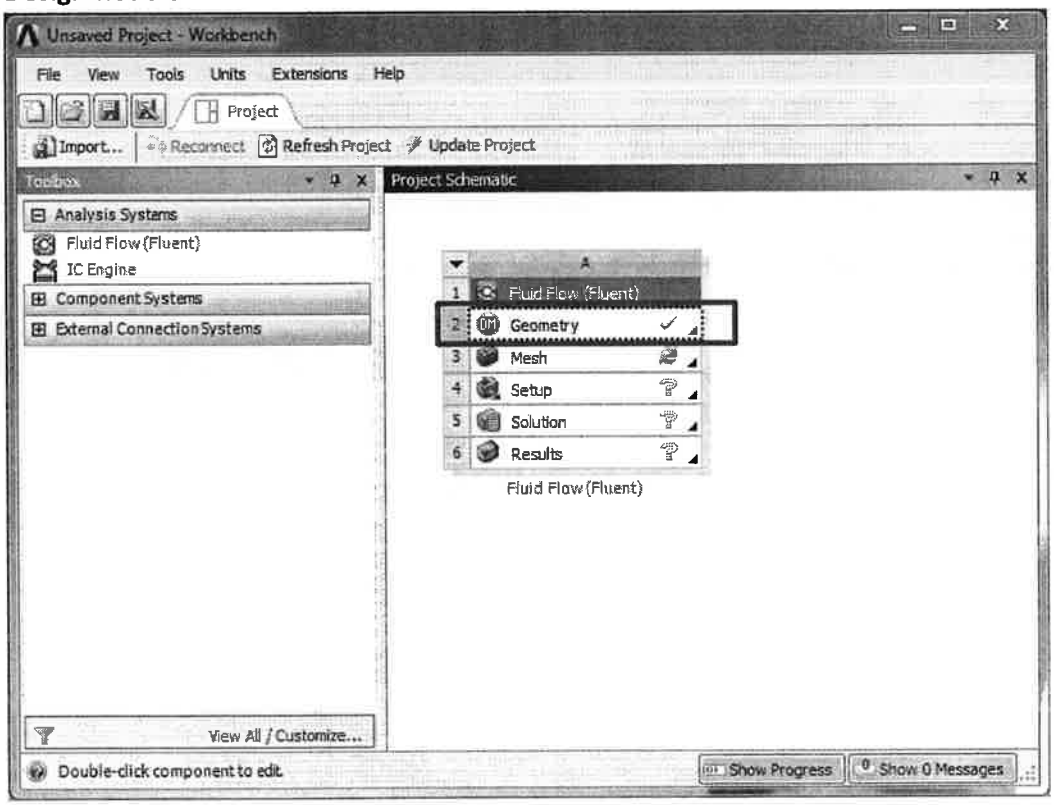
1.1 Creation of the building of interest

1.1.1 Import the building of interest (target-building-name.agdb).

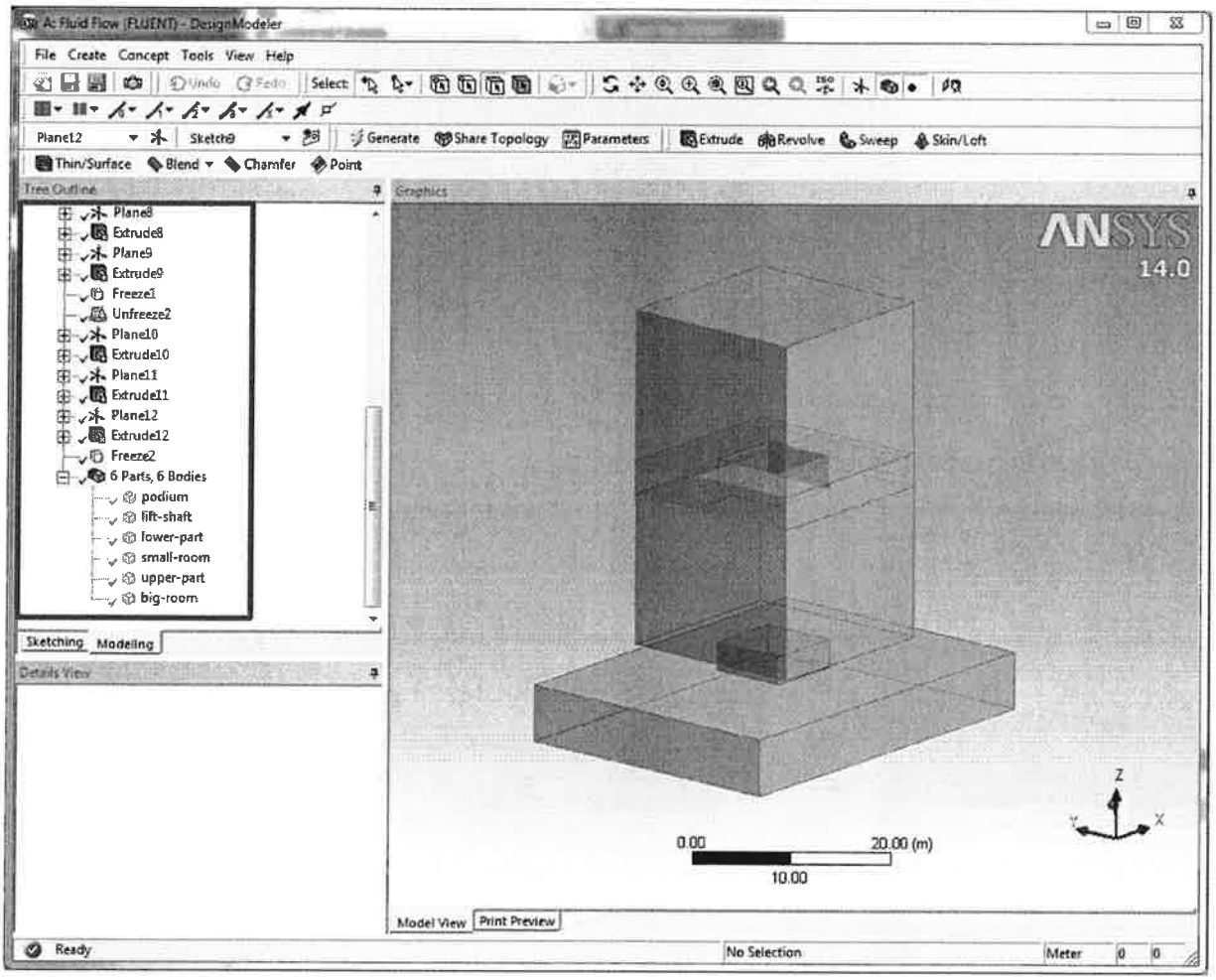
Import the building of interest (target-building-name.agdb).



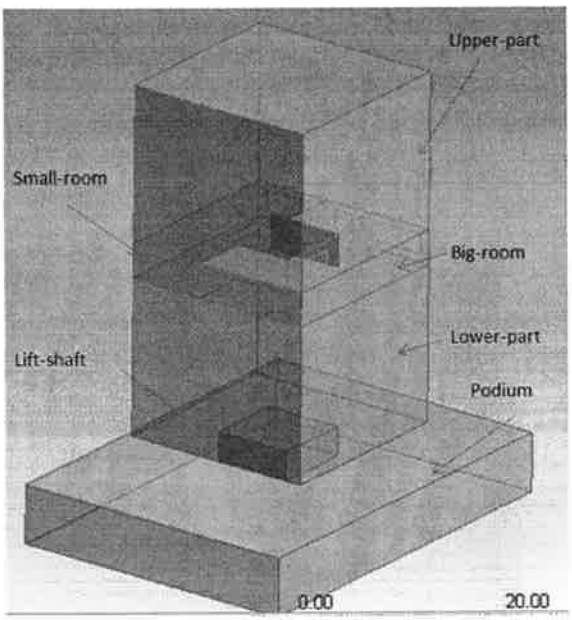
Then the "√" sign should follow the **Geometry** icon within the Project Schematic for Fluid Flow (Fluent), as shown in the picture below. Double click the **Geometry** icon to launch **DesignModeler**.



Your **DesignModeler** window should look like what is shown in the following picture: A residential building with two units with different sizings and design layouts at mid-height level should be loaded into **DesignModeler**.

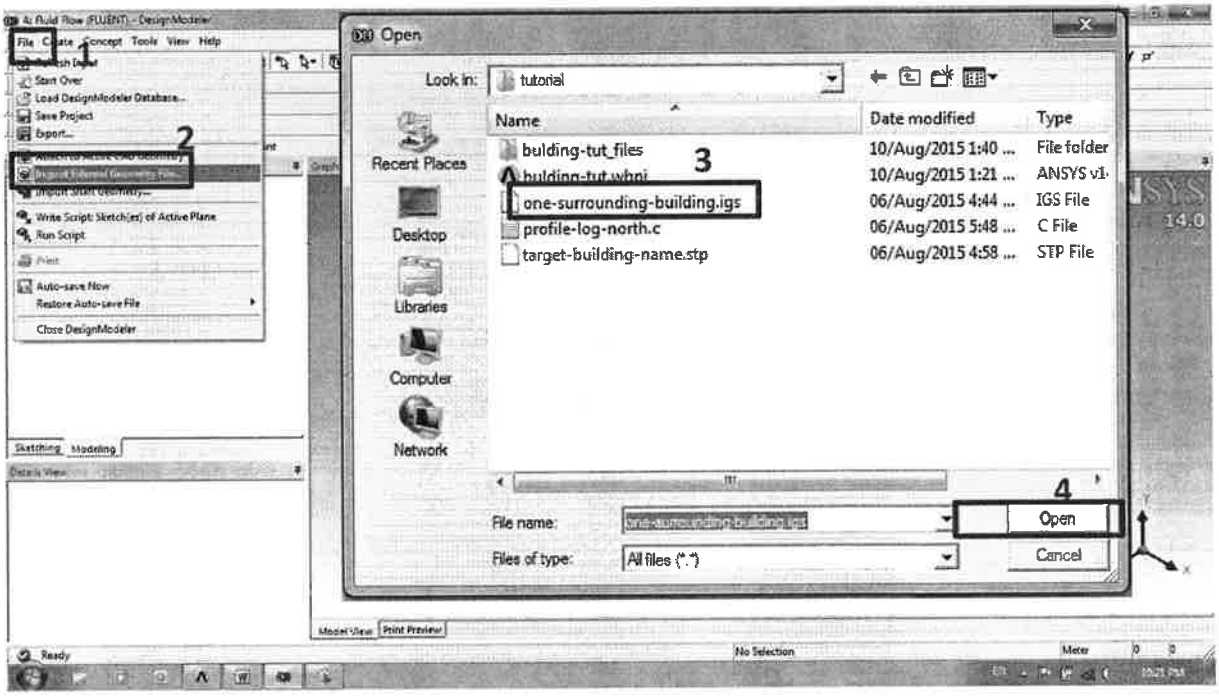


Note that the development consists of the following six bodies:

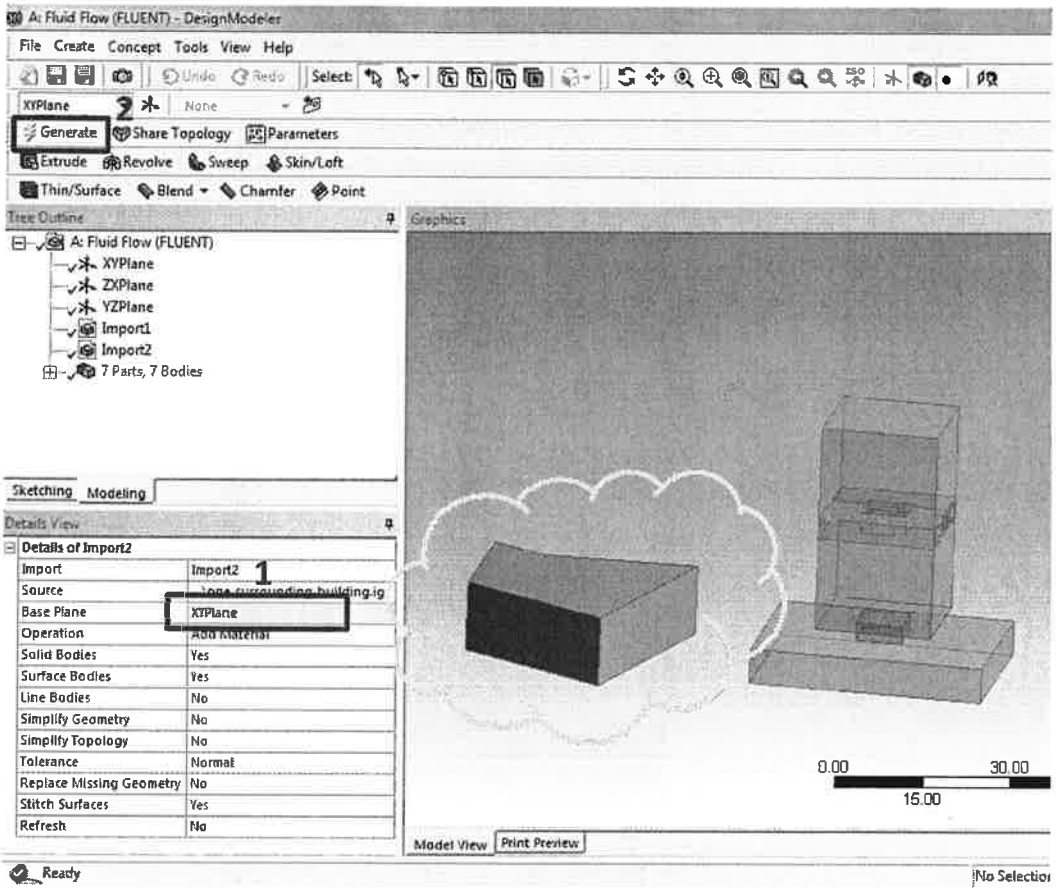


1.2 Creation of the surrounding building

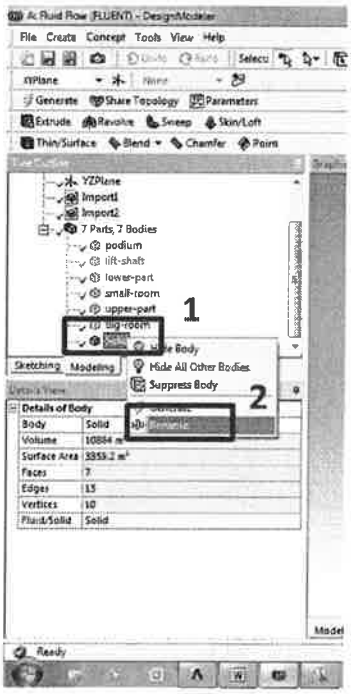
We can continue to create the surrounding building by importing file "one-surrounding-building.igs" in **DesignModeler**.

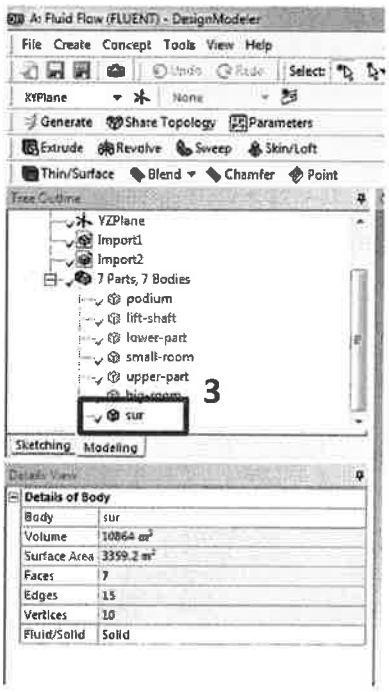


Generate the surrounding building in **DesignModeler**.



Note that the surrounding is assumed to be located ahead of the development right on the northern side. Then rename the surrounding building to be "sur".



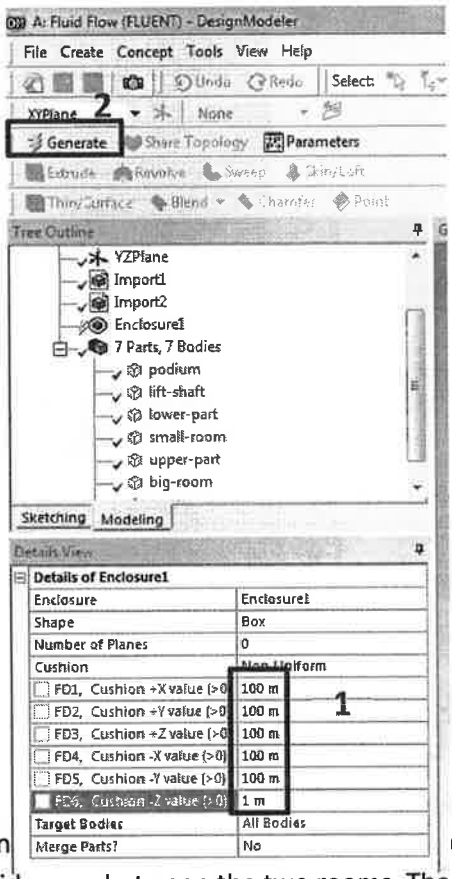


1.3 Creation of the enclosure as the computational domain

In this tutorial, the prevailing wind from the north intends to be simulated. A simple and relatively small computational domain composed of the development and its surrounding building will be created. It should be noted that the computational domain is not large enough as anticipated in GM code, due to the limited computational resources available in BCAA. Please refer to the BCA GM code about CFD methodology for details about the domain creation.

1.3.1 Create enclosure

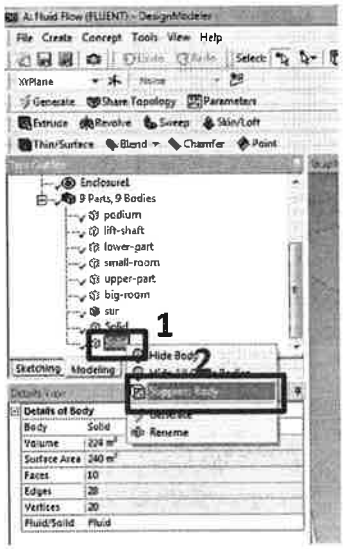
Choose *Tools* > *Enclosure*



Then two bodies will be created: one is the en ent and the surrounding building, and the other is the solid space between the two rooms. The latter is assumed to be occupied by lift shaft and thus is not needed in simulation.

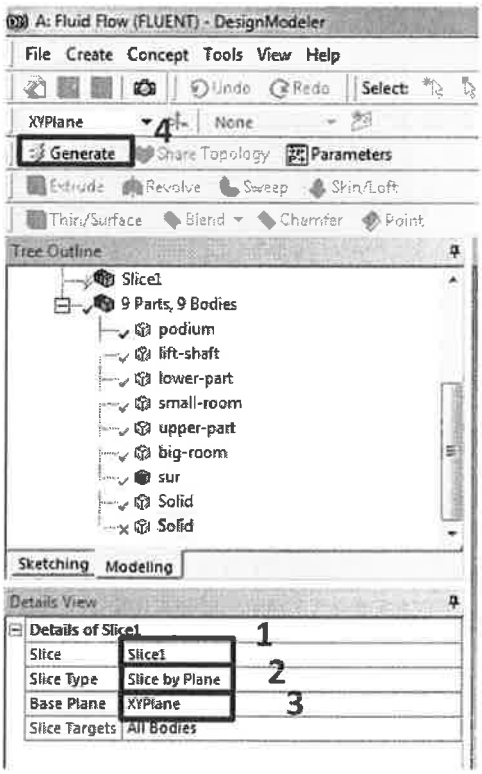
1.3.2 Suppress the extra body located between the rooms

Select the solid between the space of two rooms, and then right click the mouse button and choose ***suppress Body***.

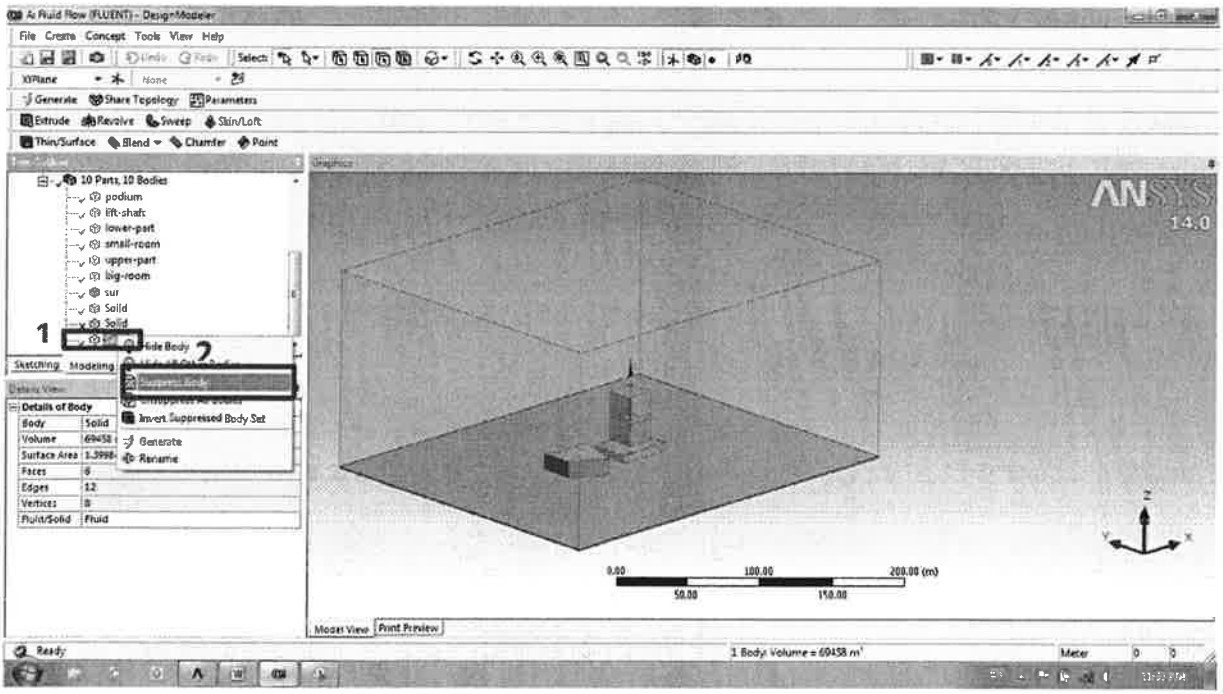


1.3.3 Trim the extra part (Z < 0) of the enclosure

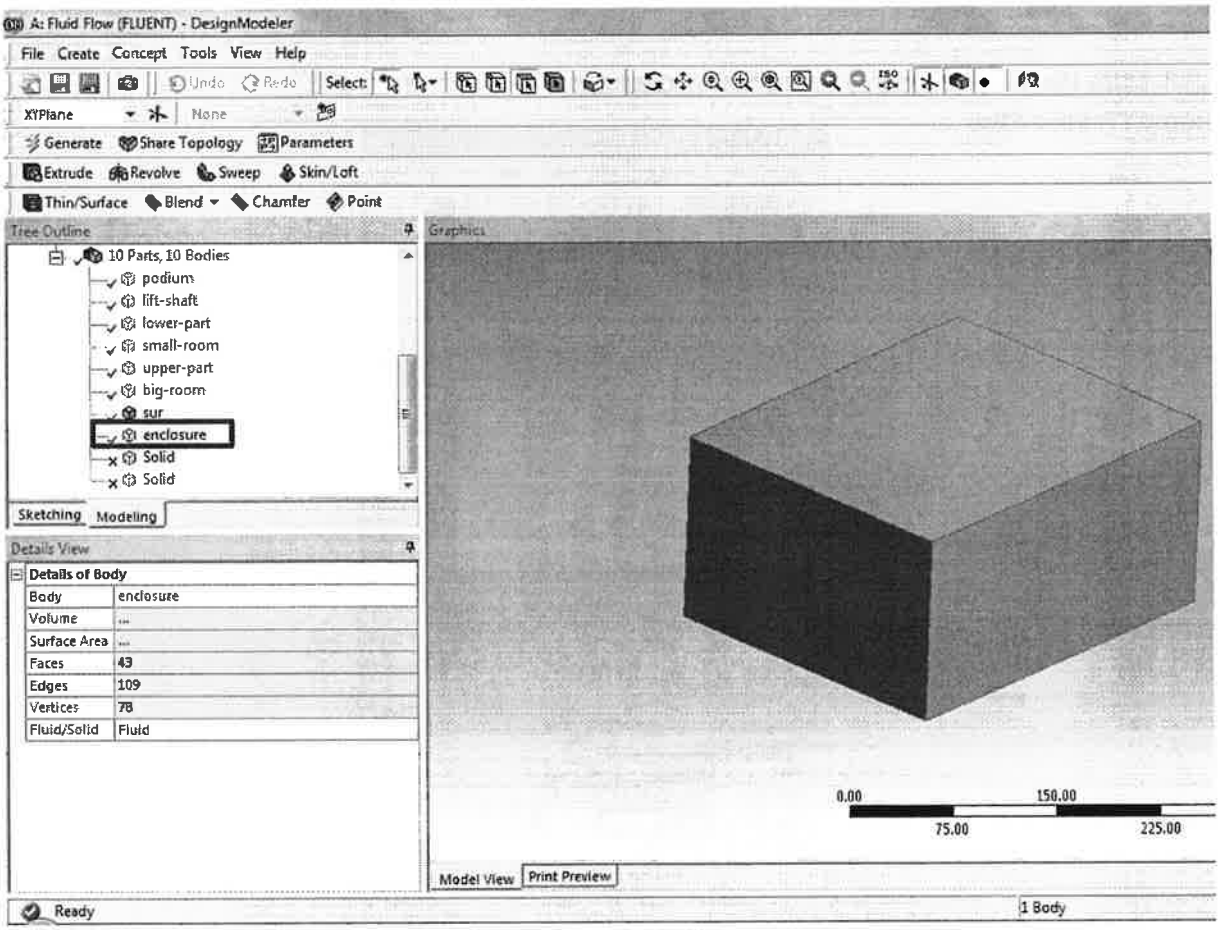
Choose ***Create > Slice***



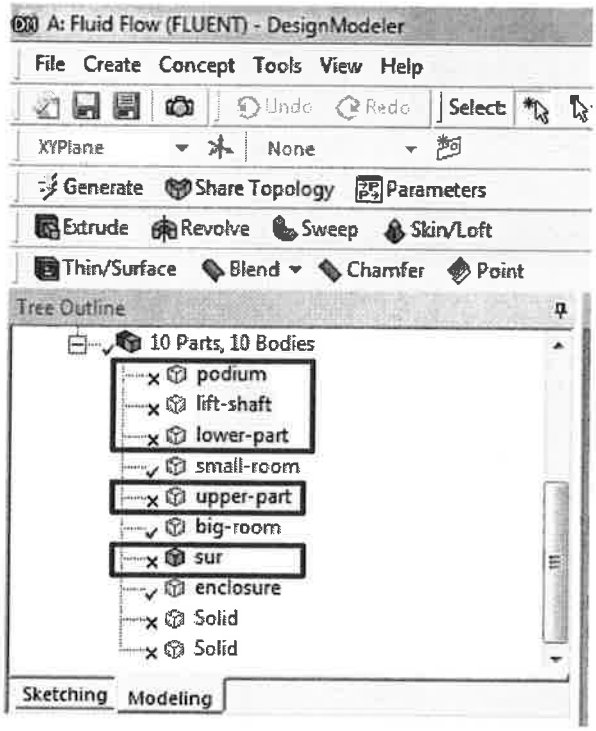
Then ***suppress*** the lower part of the enclosure.



Rename the remaining of the enclosure ("solid") to be "enclosure".

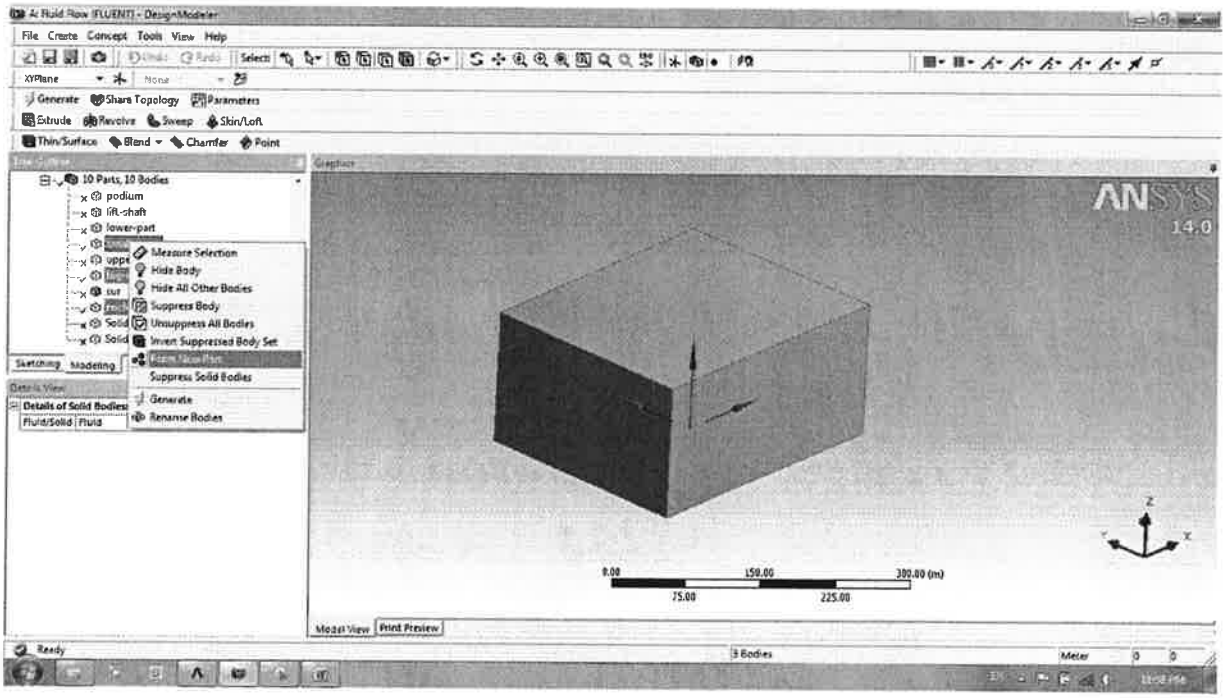


Suppress all bodies, except for "small-room", "big-room" and "enclosure". Holding CTRL key, left click respective bodies and then right click the mouse to choose ***Suppress Body*** option.

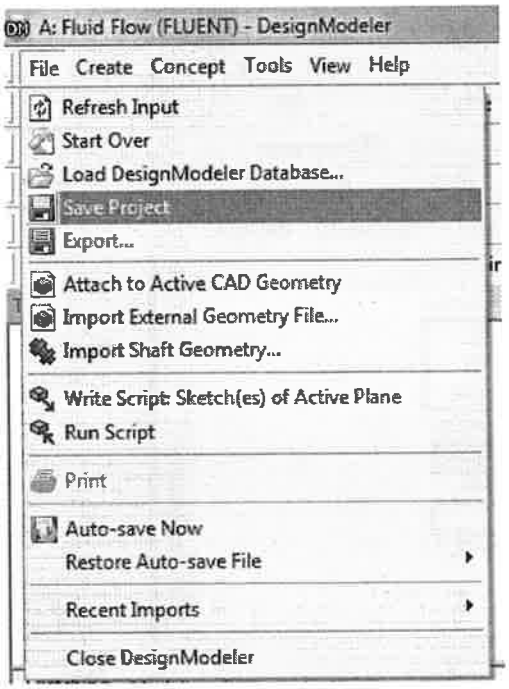


Form a single part with multiple bodies:

Holding the CTRL key and left click "small-room", "big-room" and "enclosure", and then right click the mouse to choose ***Form New Part*** option.



Save the project:

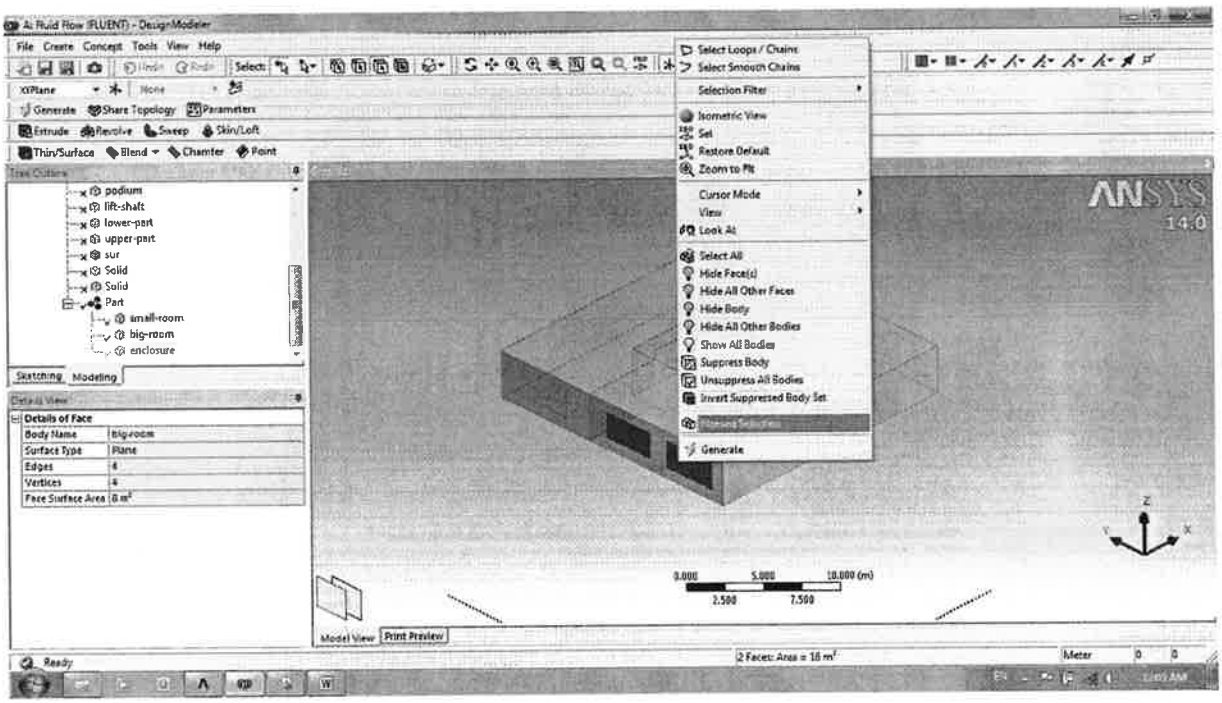


1.4 Creation of Name Selections

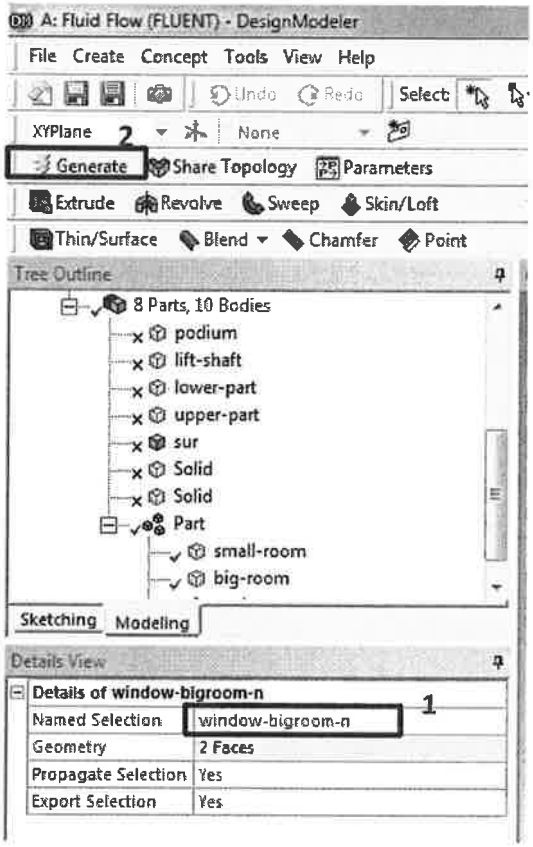
Hide body "enclosure" and fit the view for two rooms.

Define windows on the north of the big-room.

Holding the CTRL key, left click the two openings on –X side of big-room body and then right click the mouse button to choose ***Named Selection***.



Then define the selected faces as "window-bigroom-n" and generate the name selection.



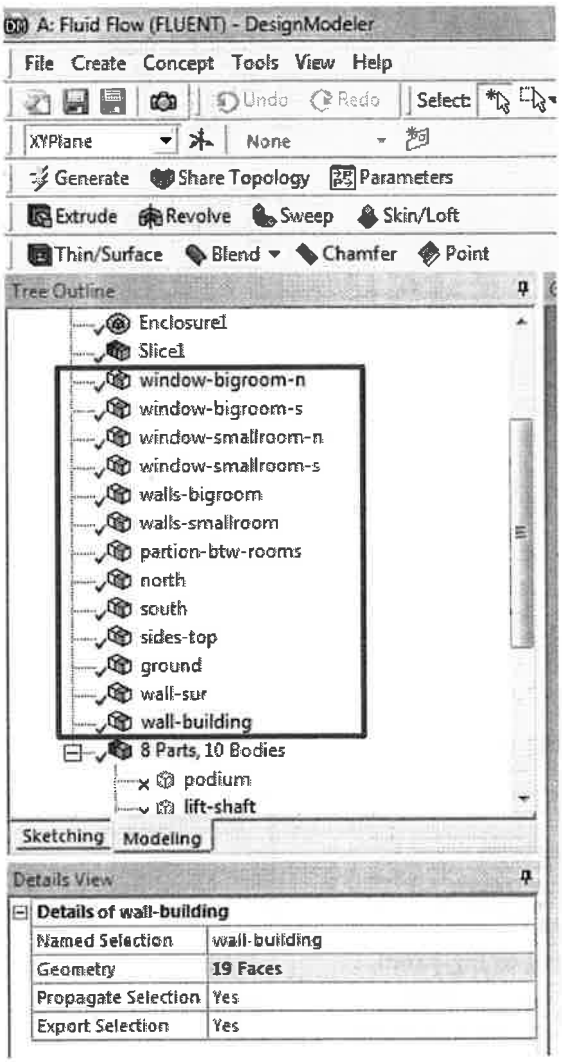
Similarly, define the other Name Selections as listed in the table below.

Name Selection	**Location**	Remark
window-bigroom-s	Two openings on +X side of big-room	
window-smallroom-n	Single opening on -X side of small-room	
window-smallroom-s	Single opening on+X side of small-room	
walls-bigroom	All vertical walls, floor and ceiling of big-room, except for the shared faces with small-room	Need to rotate the model
walls-smallroom	All vertical walls, floor and ceiling of small-room, except for the shared faces with big-room	
partition-btw-rooms	The two faces of small-room, which are shared with the body of big-room	Further hide the body of big-room

Furthermore, show the body of enclosure while hiding the other two bodies, then define the Name Selections as follows:

Name Selection	**Location**	Remark
north	The vertical face on -X side of enclosure	
south	The vertical face on +X side of enclosure	
ground	The horizontal face on the -Z side of enclosure	
sides-top	All other external bounding faces of enclosure	
wall-sur		Hide the external bounding faces and then use box selection to choose the faces for sur
wall-building		Hide the shared faces with rooms and then use box selection to choose all faces for the development

Then the feature trees with Name Selections will look like this

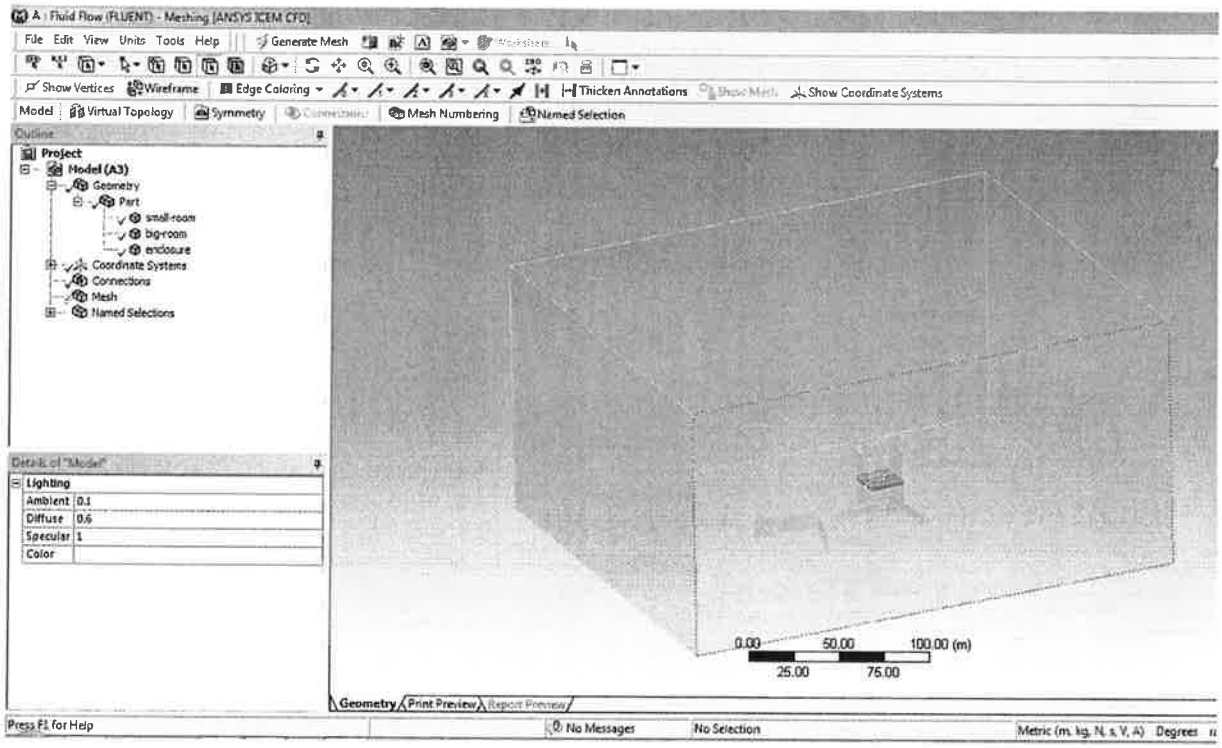


Save the project again.

Then exit Design Modeler : ***File > Close DesignModeler***

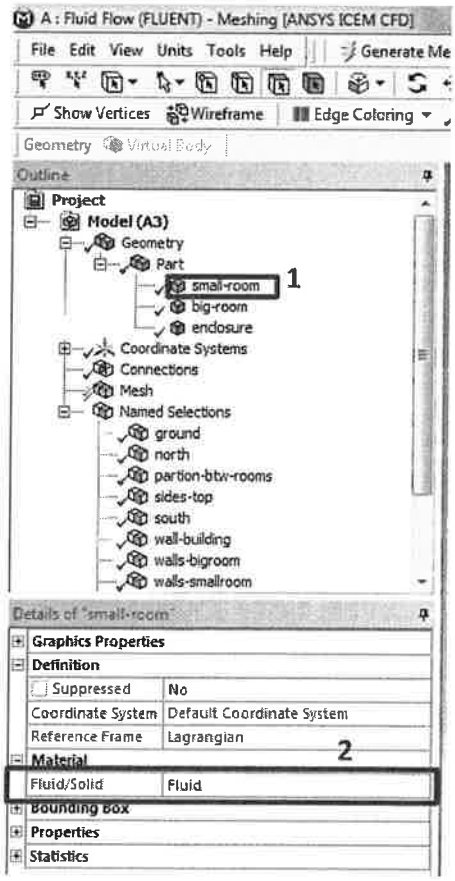
2. Mesh Generation

Double click Mesh icon to launch ANSYS Meshing.



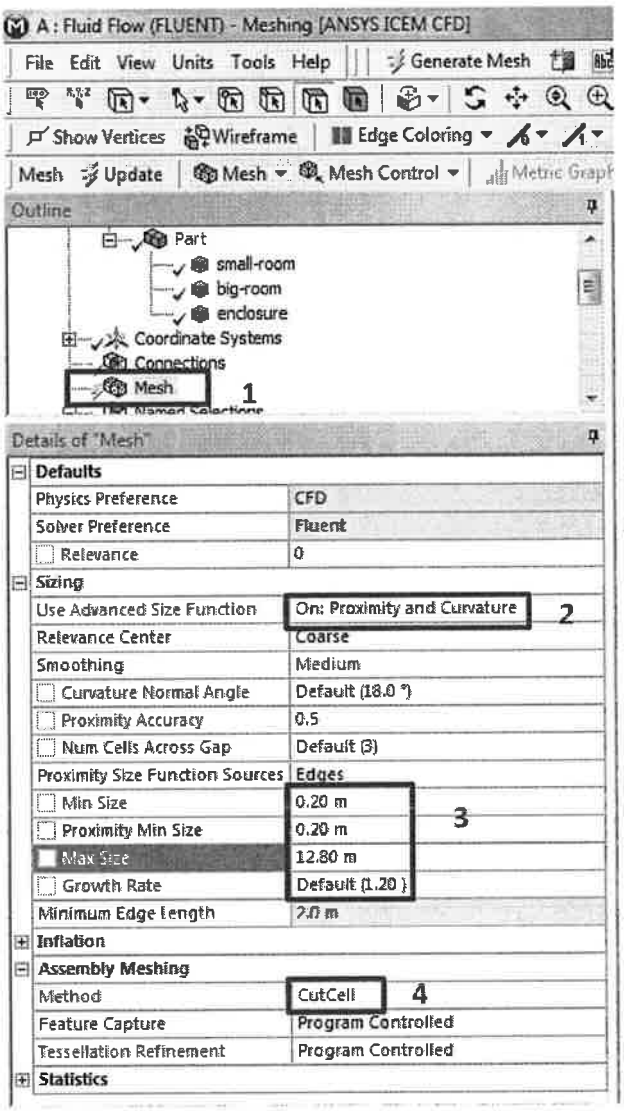
2.1 Update the material type of bodies

Change **material type** to be fluid for small-room. Repeat the process for the other two bodies.



2.2 Assign global mesh settings

Choose **Cutcell** method for assembly meshing, for the sake of high-quality hexa-dominated elements and reduced number of elements.

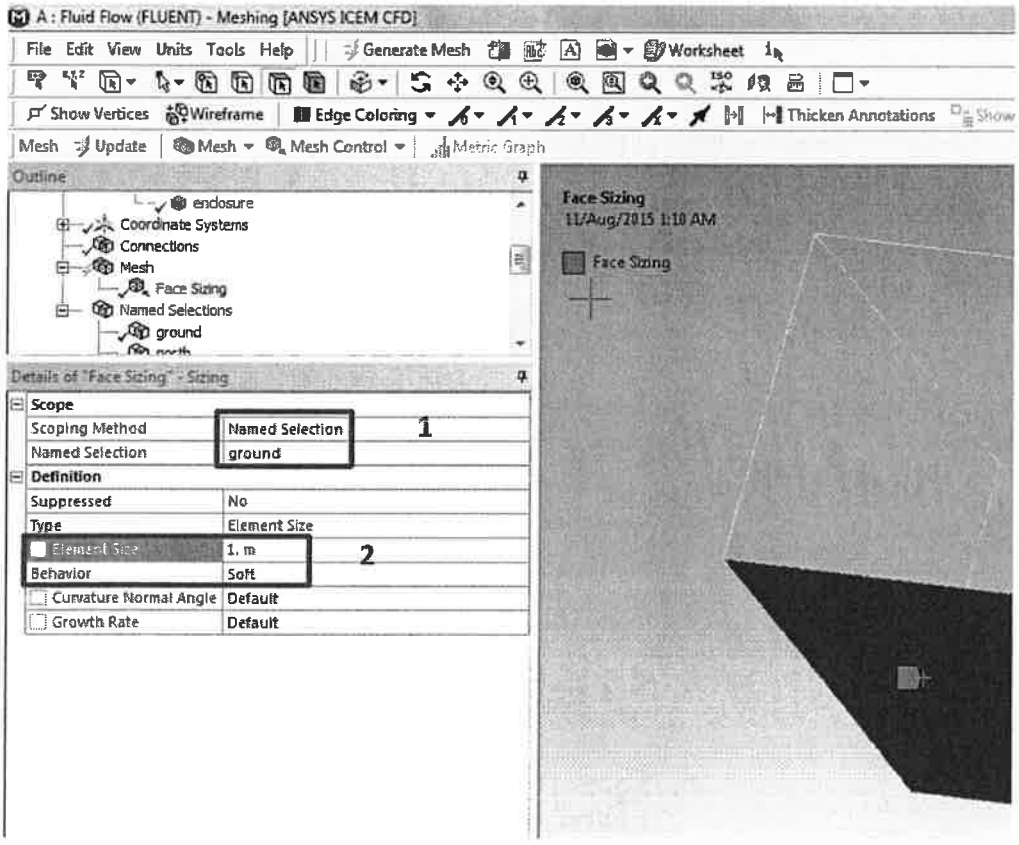


2.3 Define local size functions

Create the first size function: Right click ***Mesh***, choose ***Insert, Sizing***

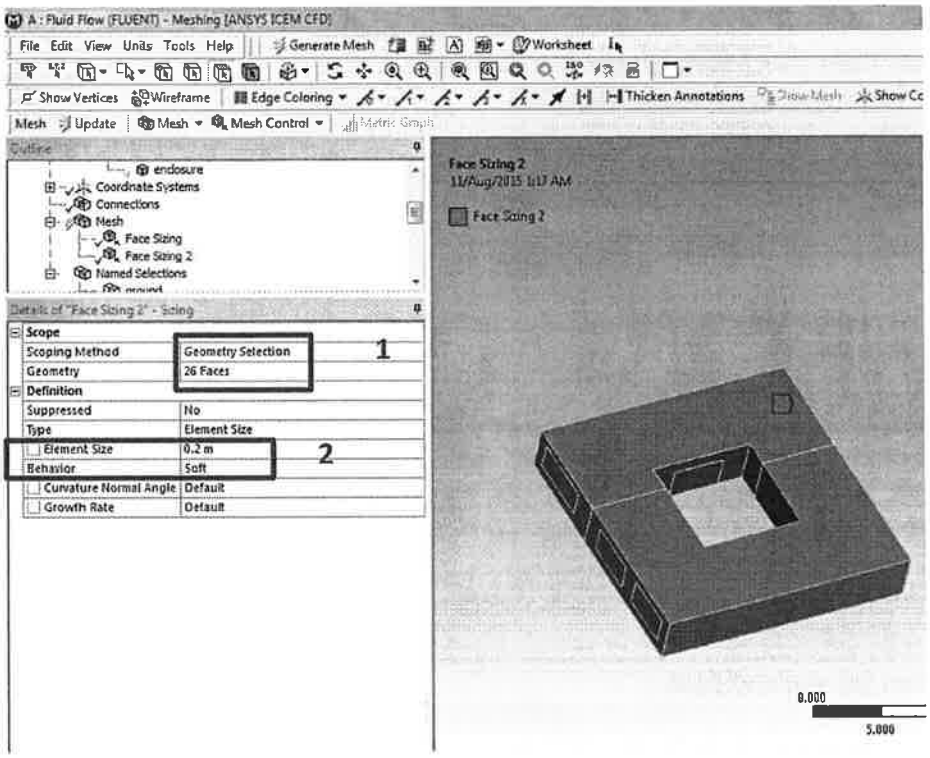


2.3.1 Set soft size function for ground face



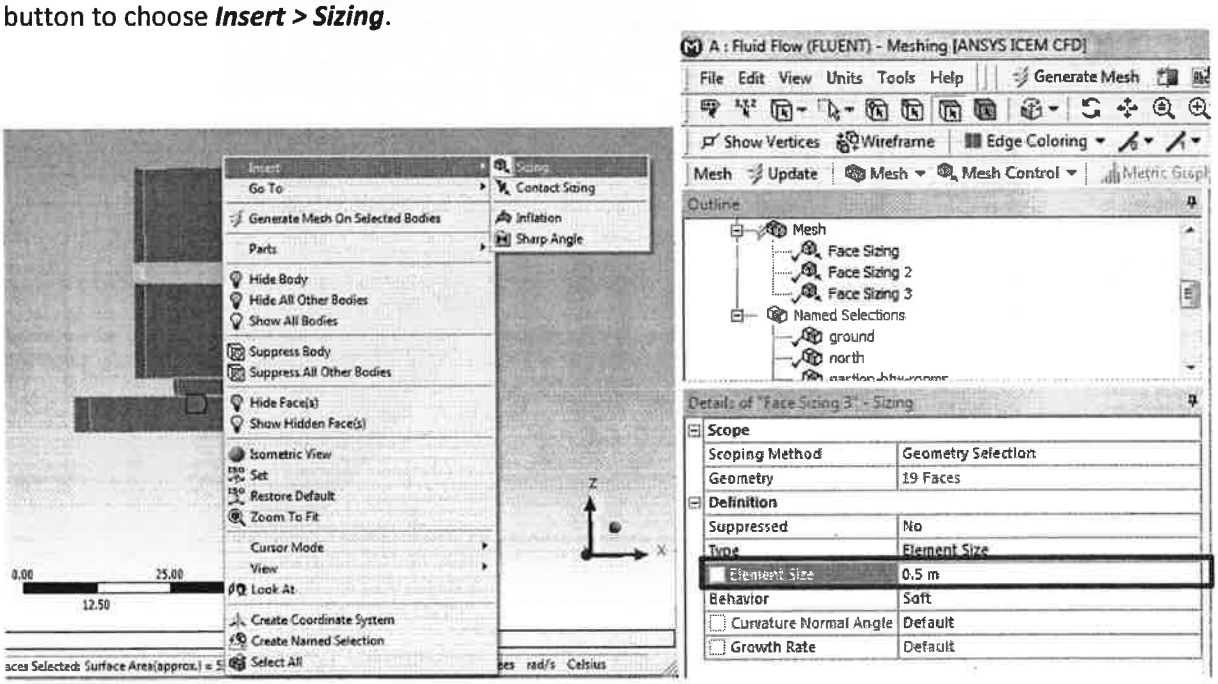
2.3.2 Set size function for faces of rooms

Hide the body of Enclosure. Then use **box selection** to choose all faces for two rooms and then set the size function accordingly.



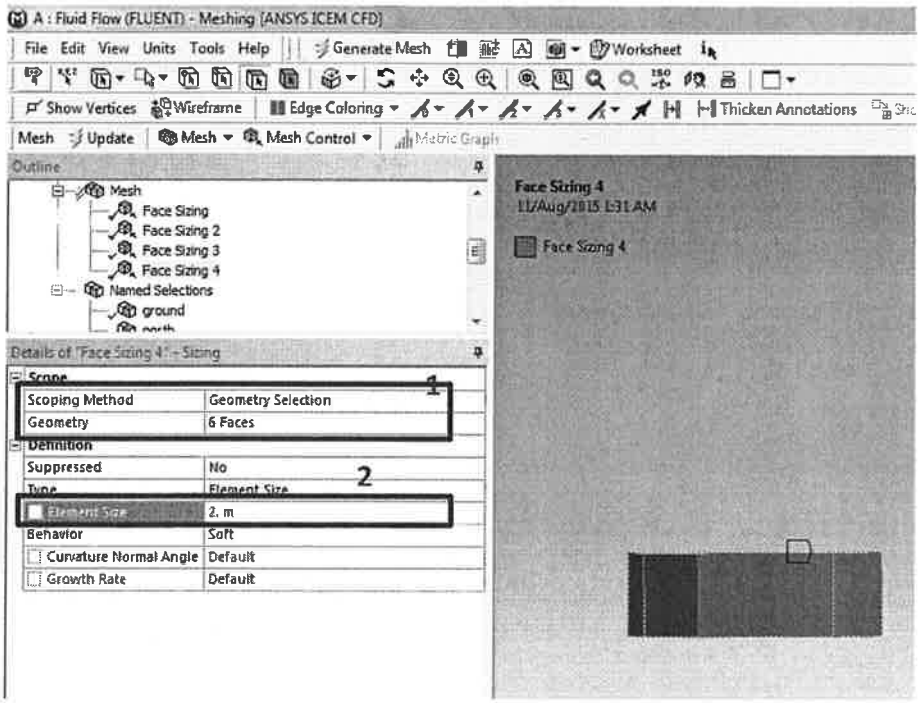
2.3.3 Size function for the development

Hide both bodies for rooms and ***show*** the body of Enclosure only.
Hide all external faces of enclosure body.
Hide shared faces with rooms using box selection.
Then use ***box selection*** to choose all remaining faces for the development. Right click the mouse button to choose ***Insert > Sizing***.



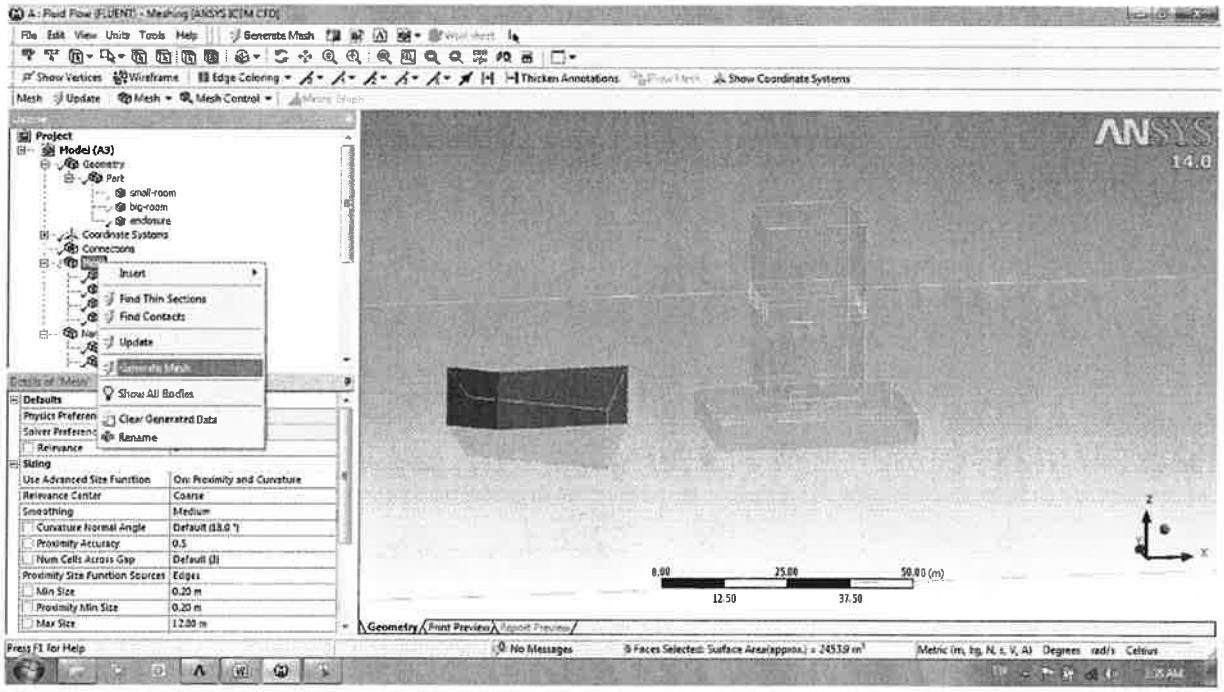
2.3.4 Size function for the surrounding building

Right click "Face Sizing 3" to choose ***Duplicate*** to create "Face Sizing 4". Update the faces by selecting the faces for sur body using box selection and then click ***Apply***. Set the element size to be no more than 2m in this tutorial.

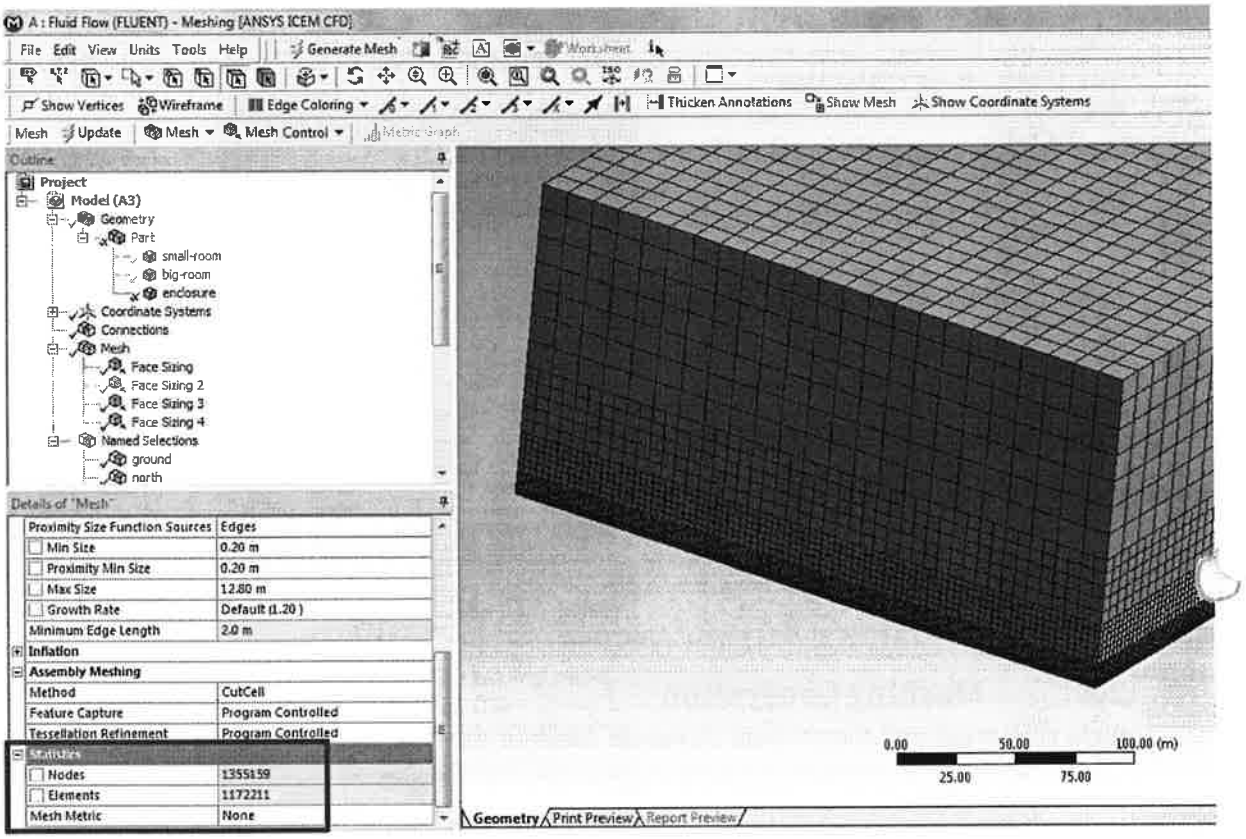


2.4 Meshing Generation

Right click ***Mesh*** and then choose ***Generate Mesh*** to start the mesh creation.



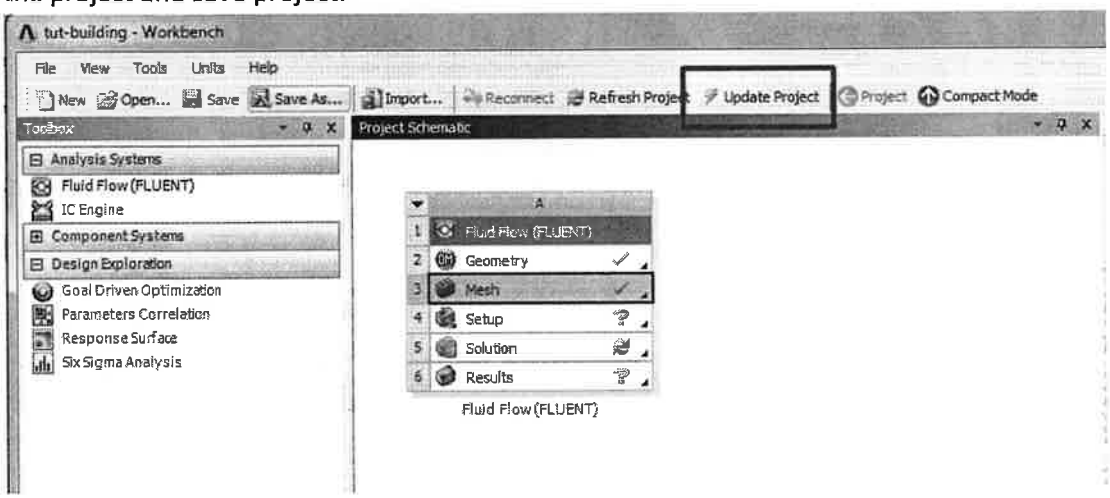
2.5 Check mesh



In total there will be around 1m hexa-dominated elements to be generated to spatially discretized the computational domain.

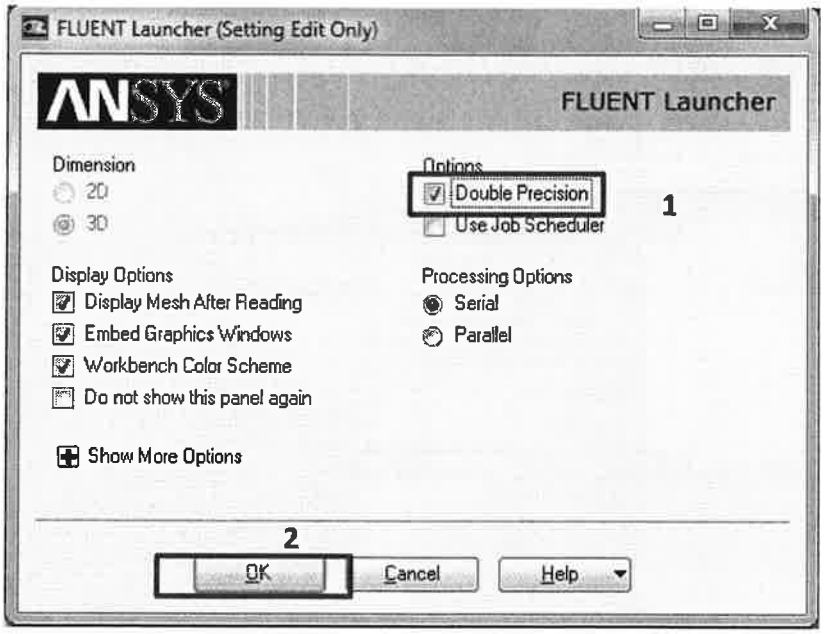
Exit ANSYS Meshing.

Update the project and save project.



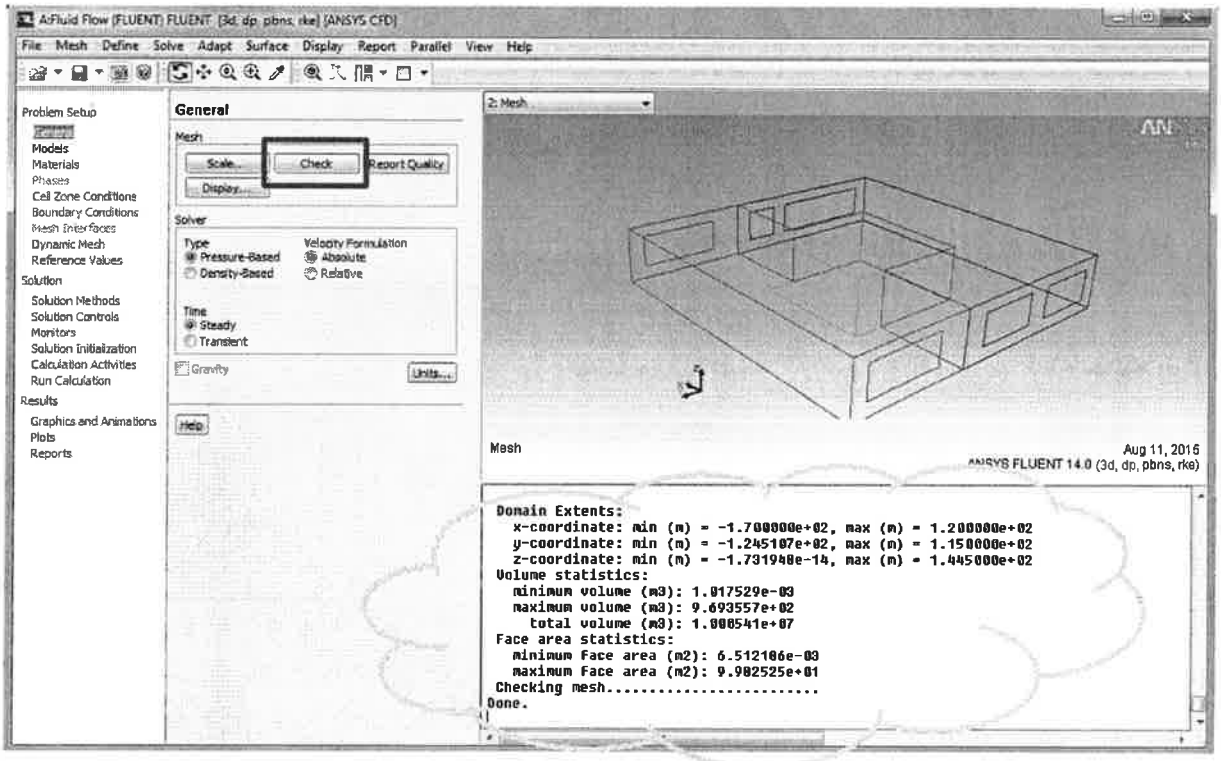
3 Numerical settings and computation

Double click Setup icon to launch ANSYS FLUENT. Turn on Double Precision option.



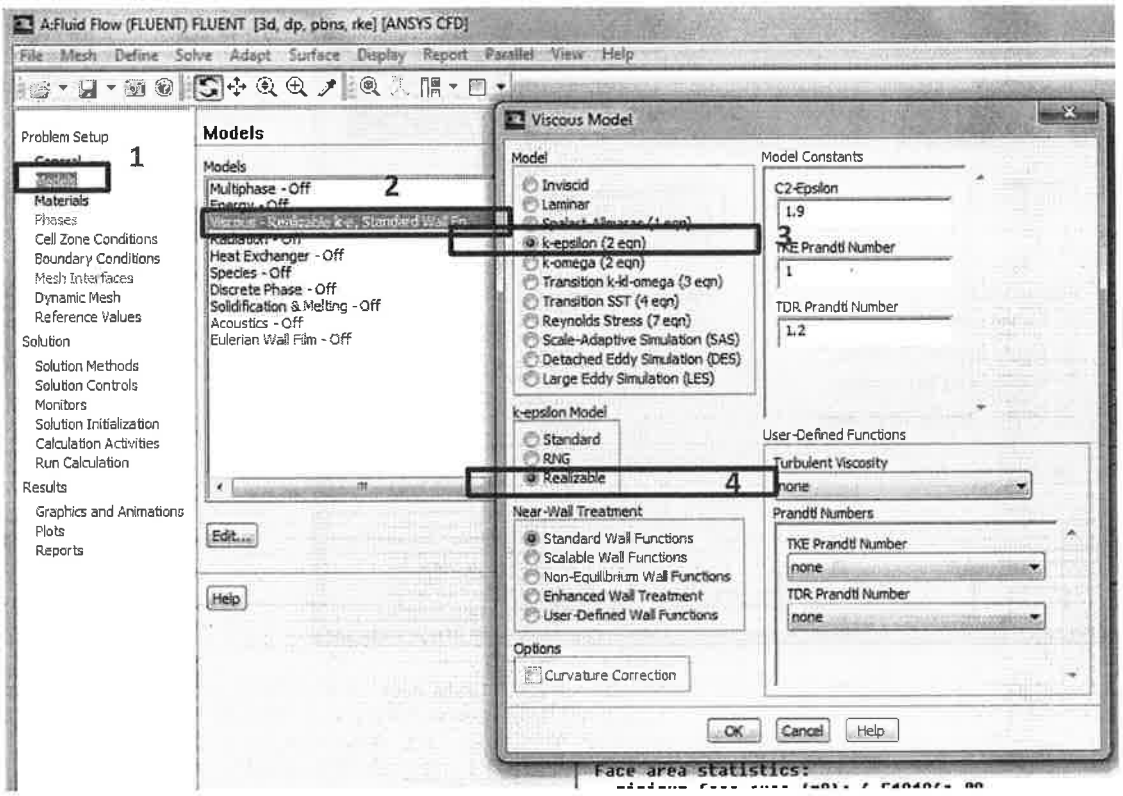
3.1 Check mesh

Make sure there is no error message about the meshing.



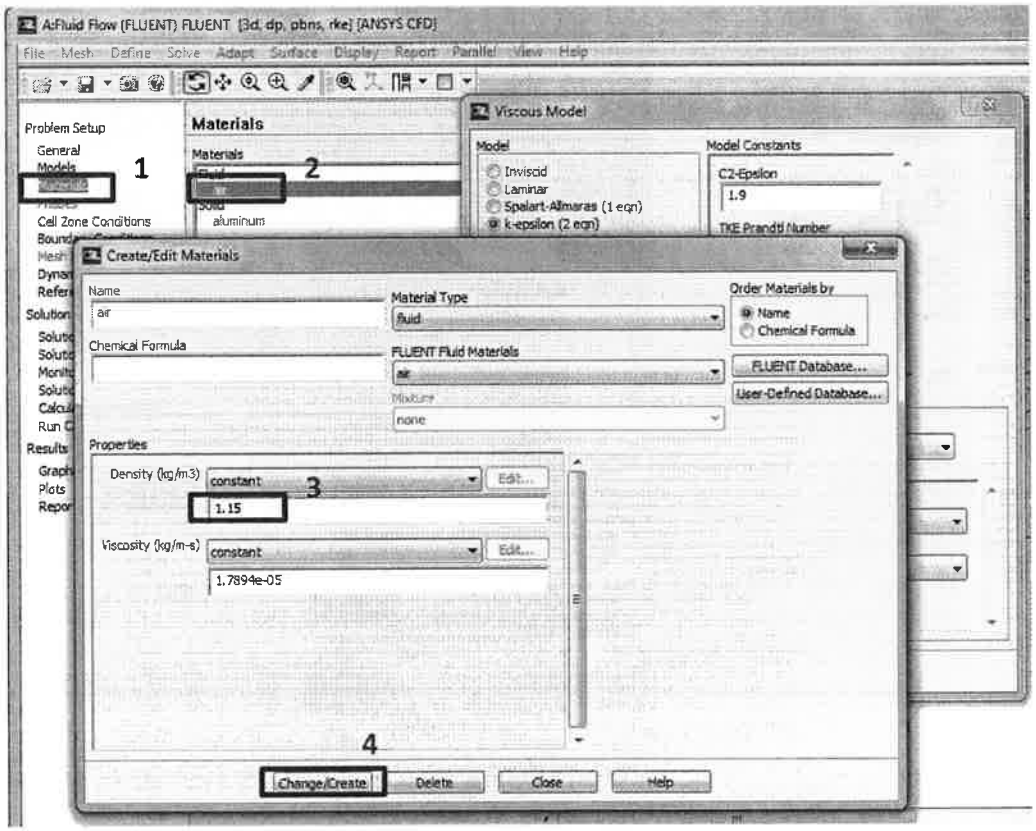
3.2 Model setting

Choose Realizable k-e turbulence model

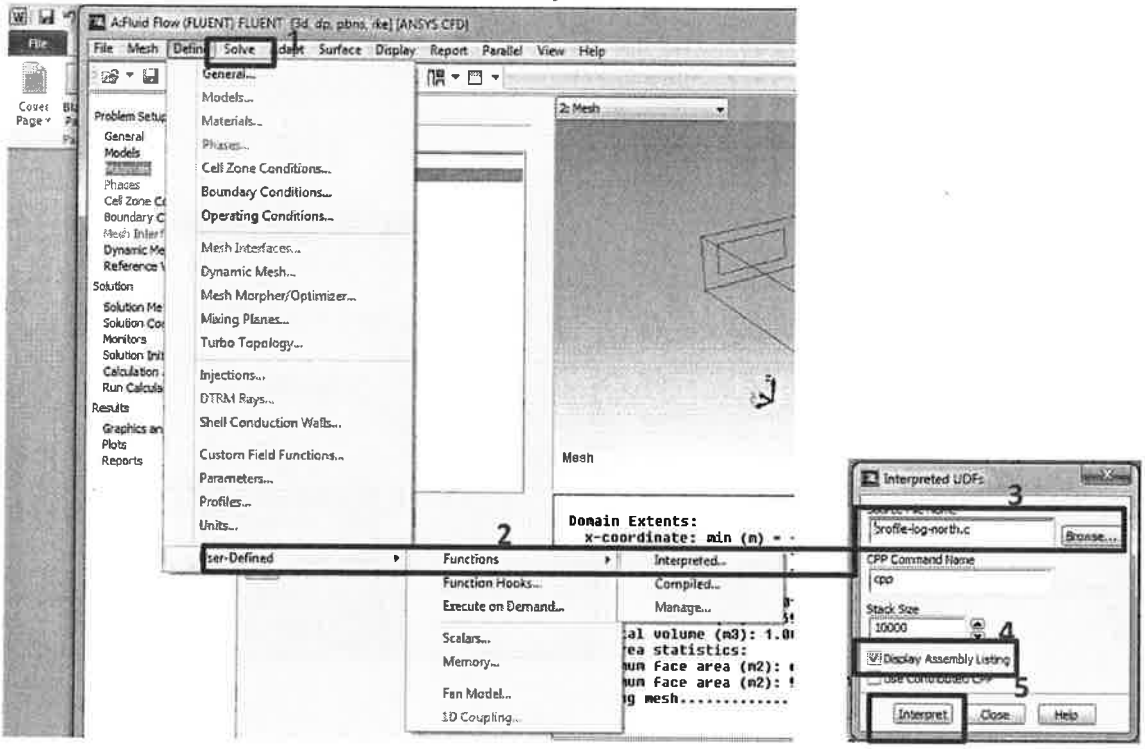


3.3 Material setting

Change air density to be 1.15 kg/m^3, in accordance with 33.1°C in temperature (Isothermal condition).

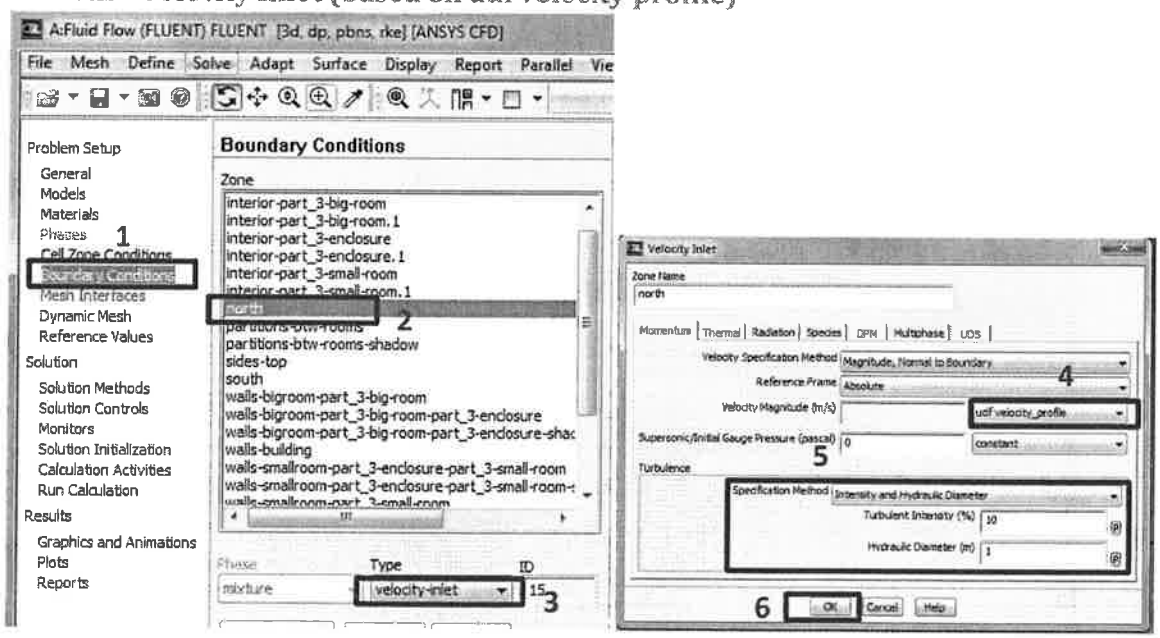


3.4 User Define Function for boundary condition

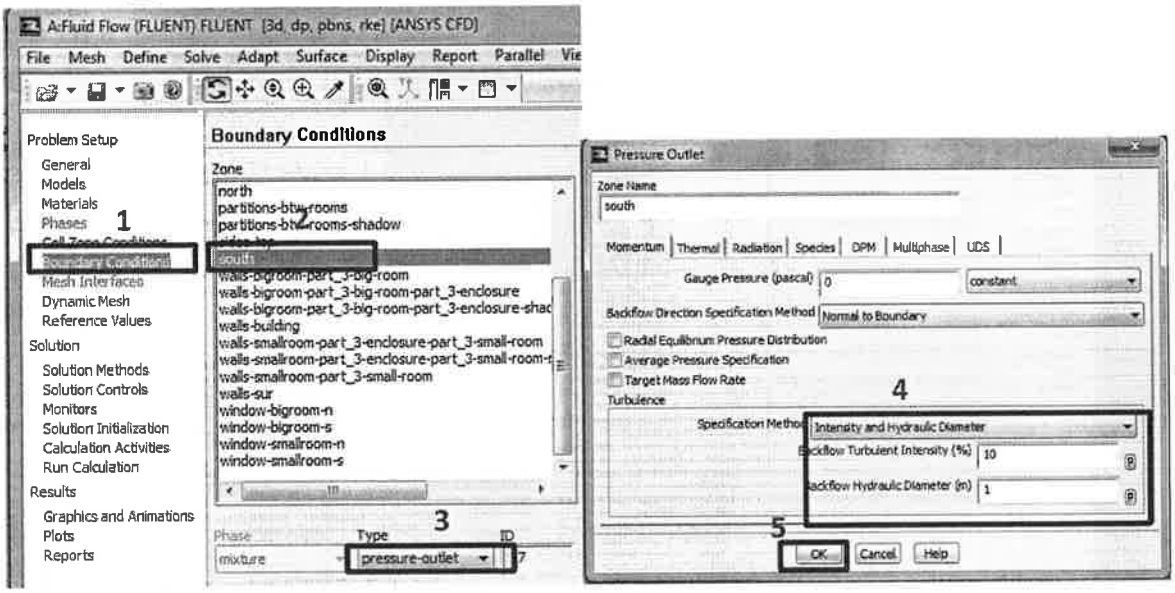


3.5 Definition of boundary conditions

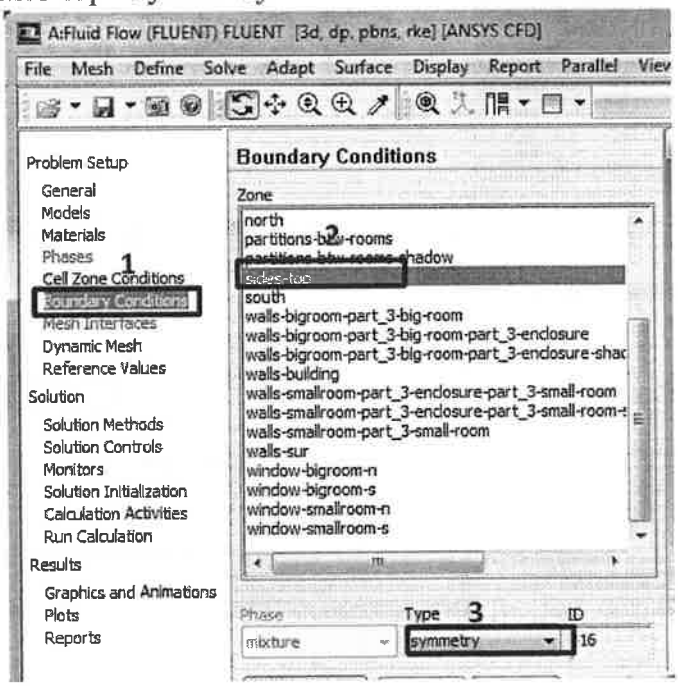
3.5.1 North – Velocity Inlet (based on udf velocity profile)



3.5.2 South – Pressure Outlet

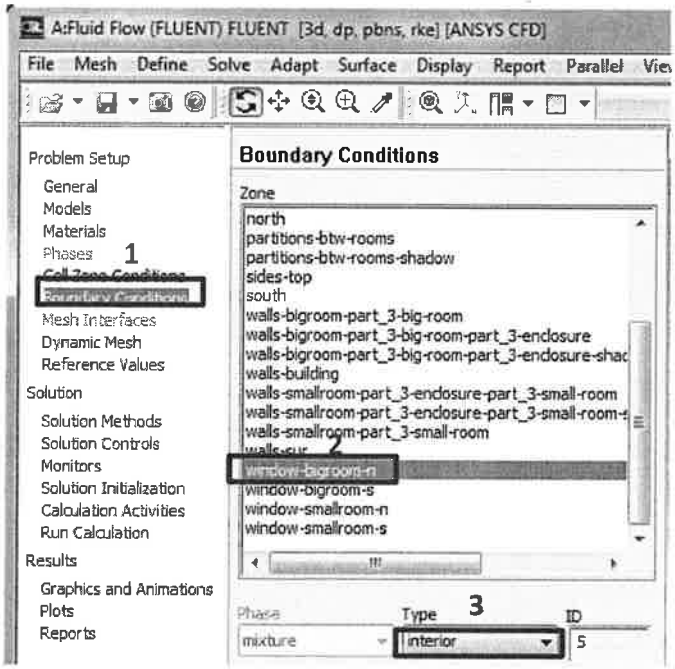


3.5.3 Sides-top – Symmetry



3.5.4 window-* - Interior

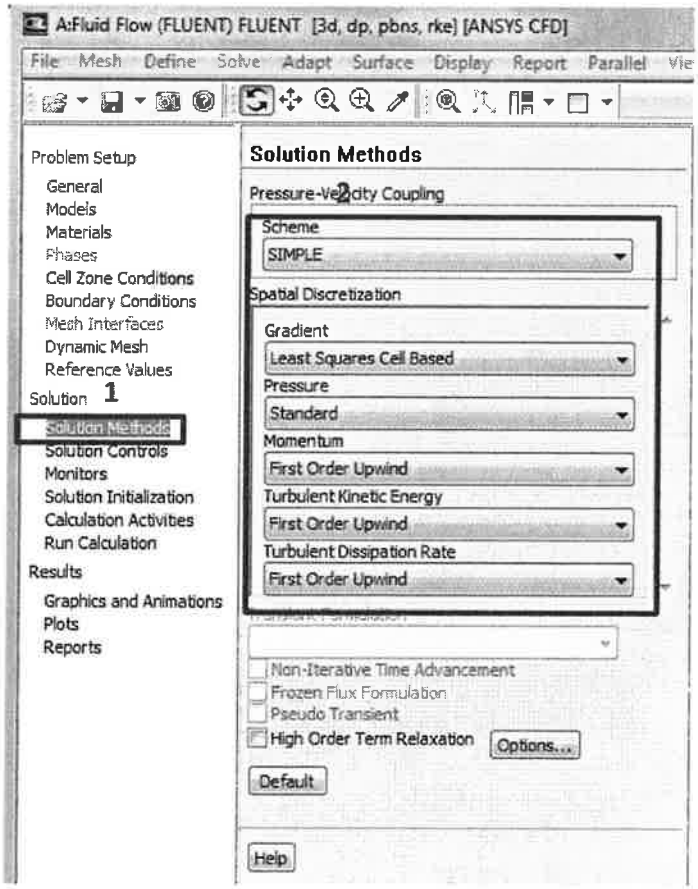
Change boundary type for all window-* Name Selections as Interior.



Repeat the changes for Window-bigroom-s, window-smallroom-n, and window-smallroom-s.

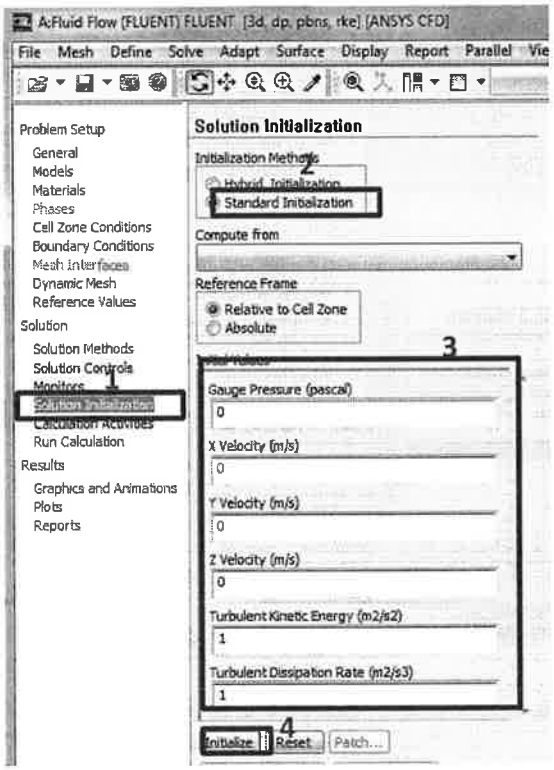
3.6 Solution method

Choose SIMPLE procedure and first-order discretization scheme for all quantities.



3.7 Initialization

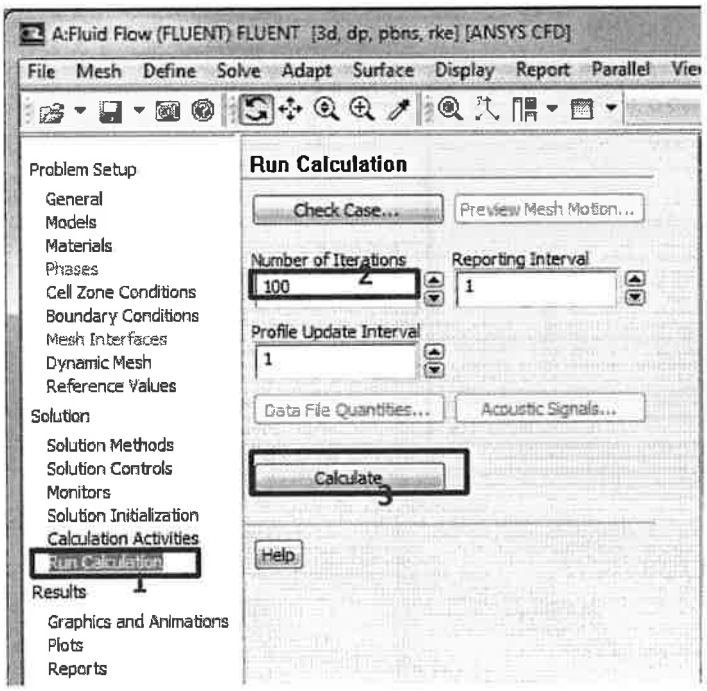
Standard initialization



3.8 Save Project

3.9 Run Calculation

Due to time constraint, only 100 iterations will be performed for the tutorial. Note that 100 iterations are not sufficient to reach convergence. Student can run more if time permits.

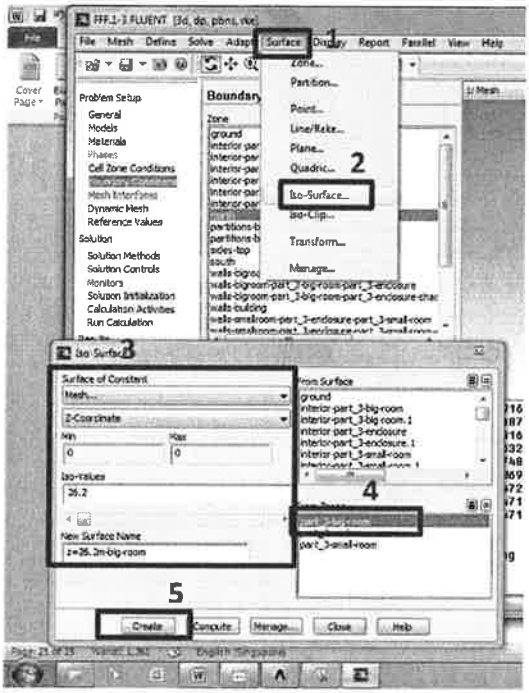


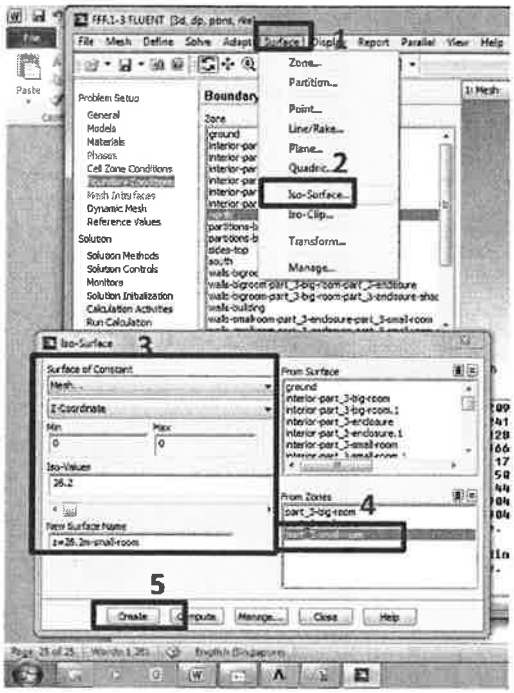
Once the simulation is completed, save project again.

4 Post processing simulation results using FLUENT

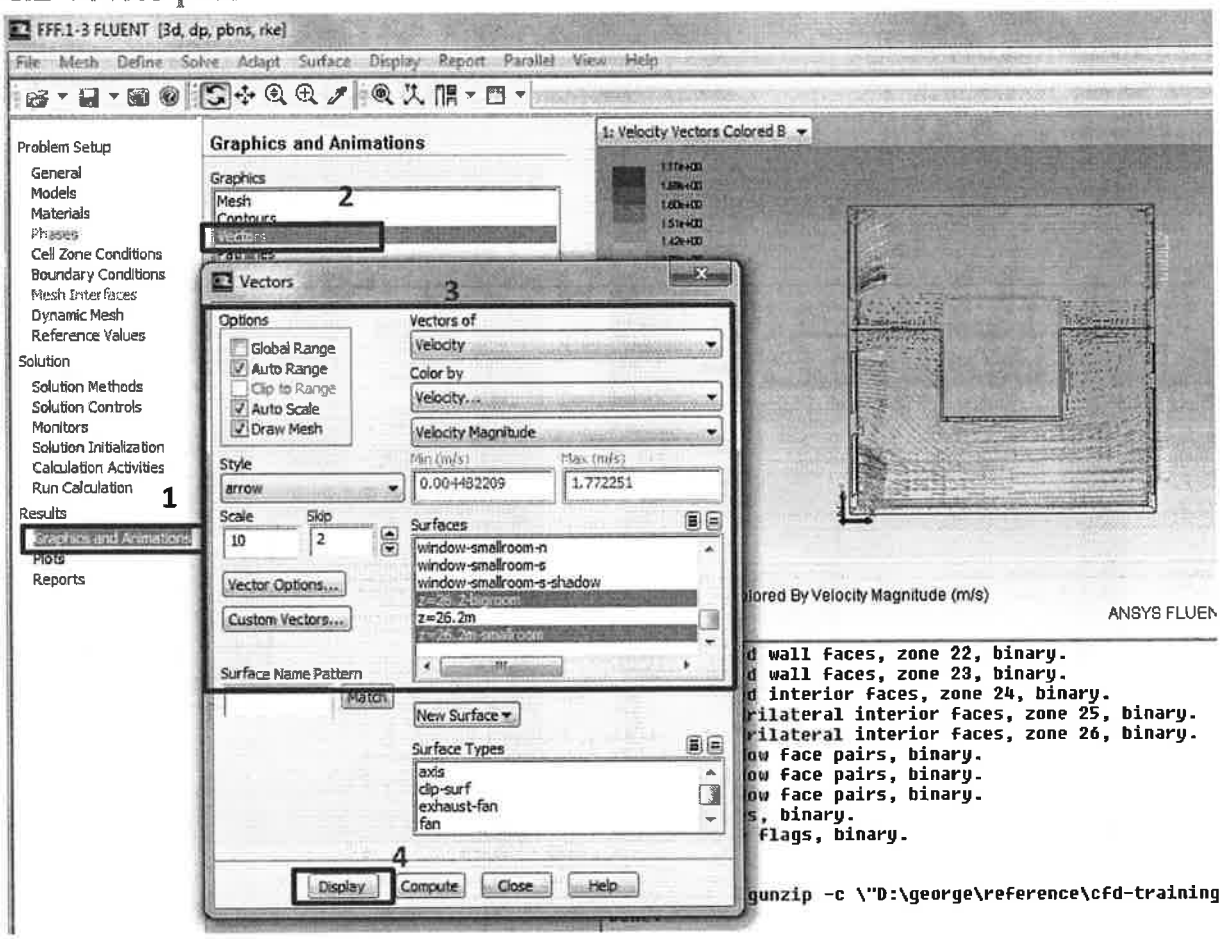
4.1 Definition of cutting planes

Z=26.2m @1.2m above the floor; two separate faces (named "z=26.2m-bigroom" and "z=26.2m-smallroom", respectively) for the two rooms.

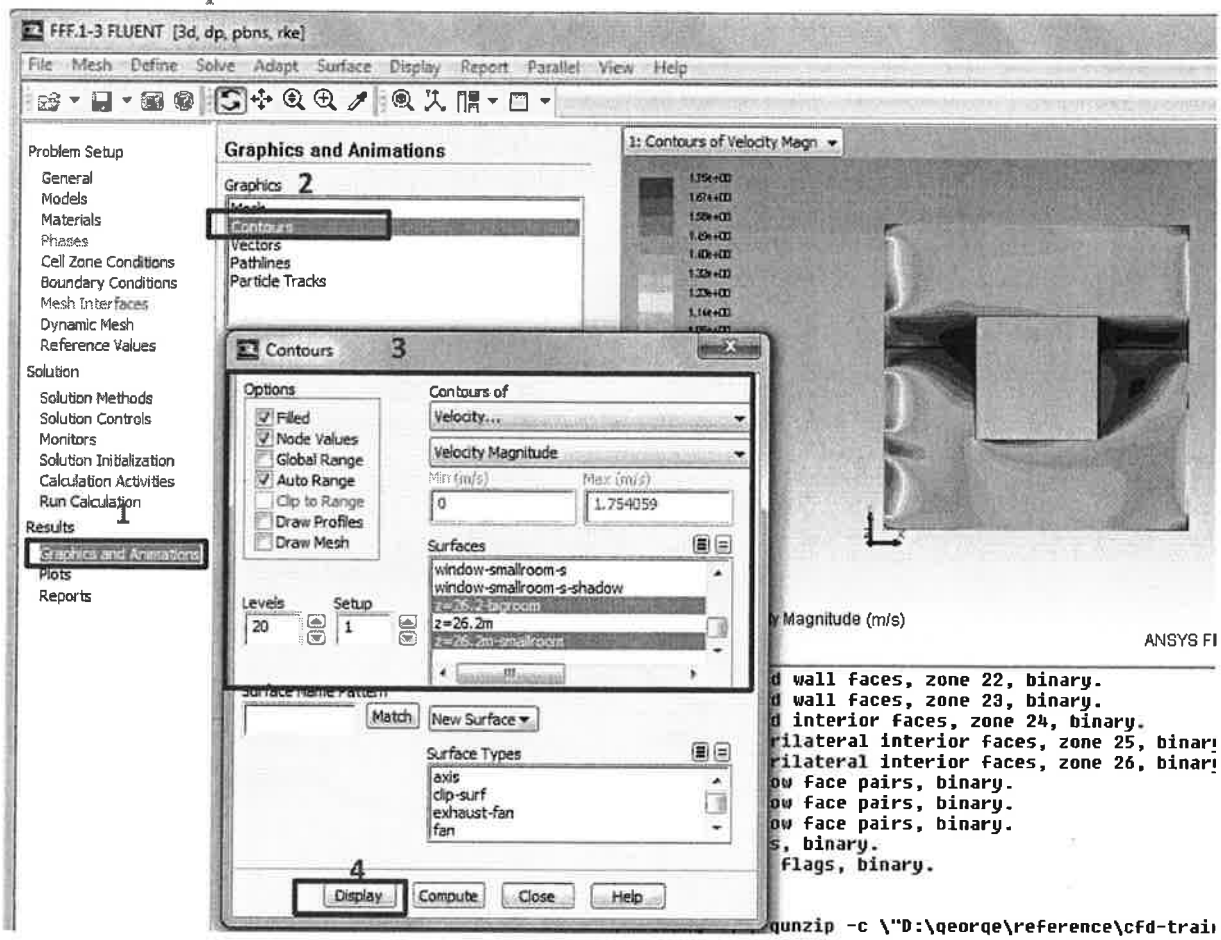




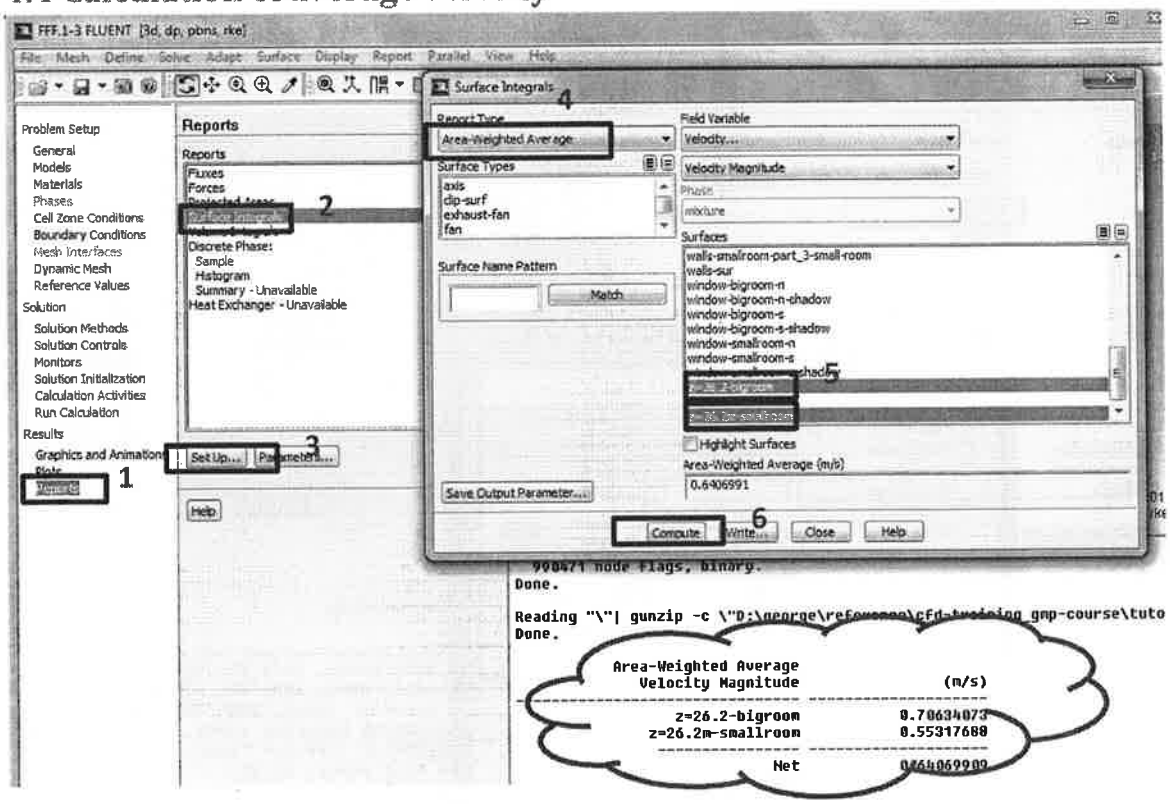
4.2 Vector plot



4.3 Contour plot

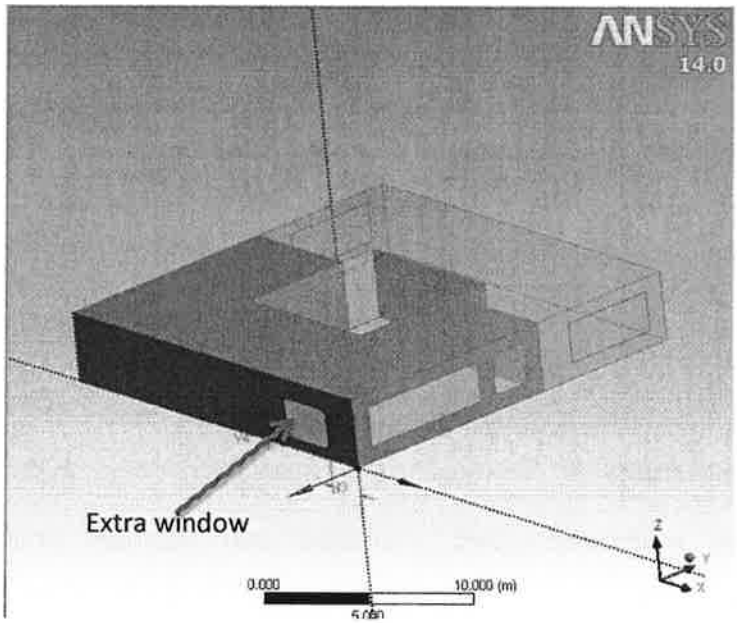


4.4 Calculation of average velocity



Appendix

Revised model: Add one more window for the "big-room" body

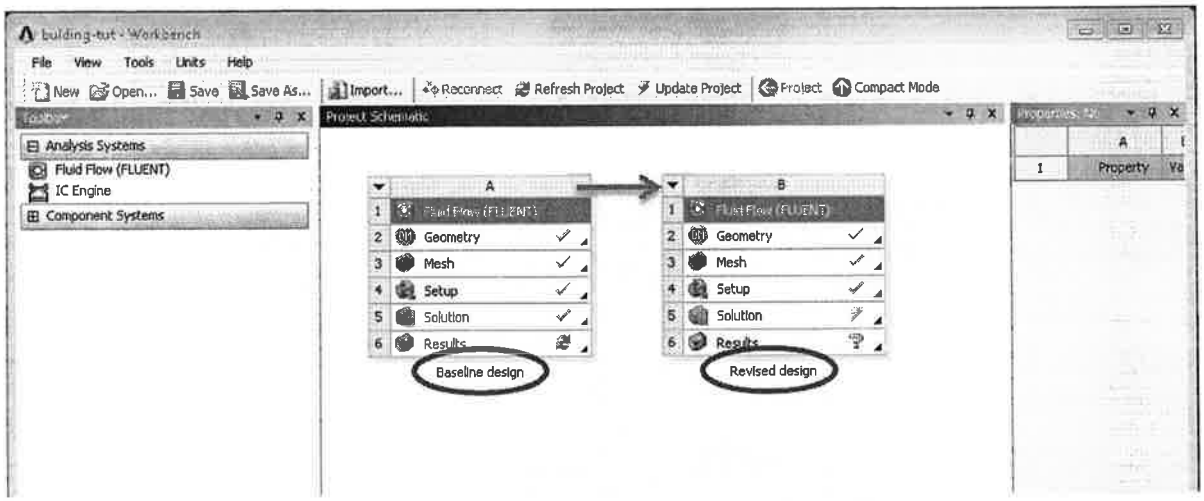


Please note that Design Modeler adopts parametric design approach in geometry creation, thus the "first-come, first serve" principle is applicable throughout the process in feature creation. Therefore, in order to add the window for Body "big-room", the operations should be inserted before the operation to import the surrounding buildings.

A.1 Duplicate the original scheme in Workbench

Duplicate the model in Workbench by ***right clicking*** the original scheme and then choosing ***duplicate*** option. ***Rename*** the original Schematic as "Baseline design" and the new one as "Revised design", simply double clicking the respective regions. Then the new case will be updated with the inclusion of a new window.

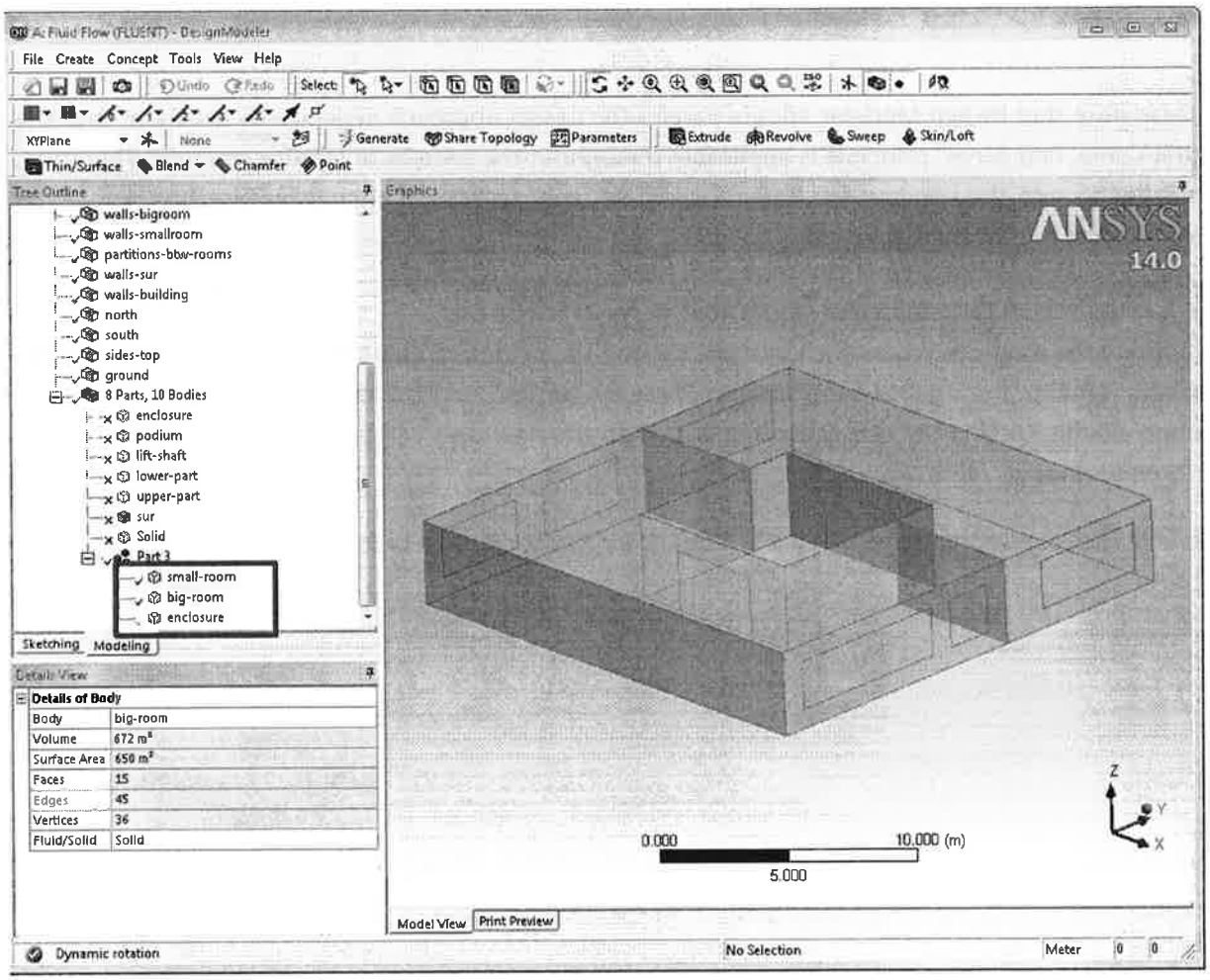
Double click the Geometry icon in the schematic for Revised design to launch Design Modeler.



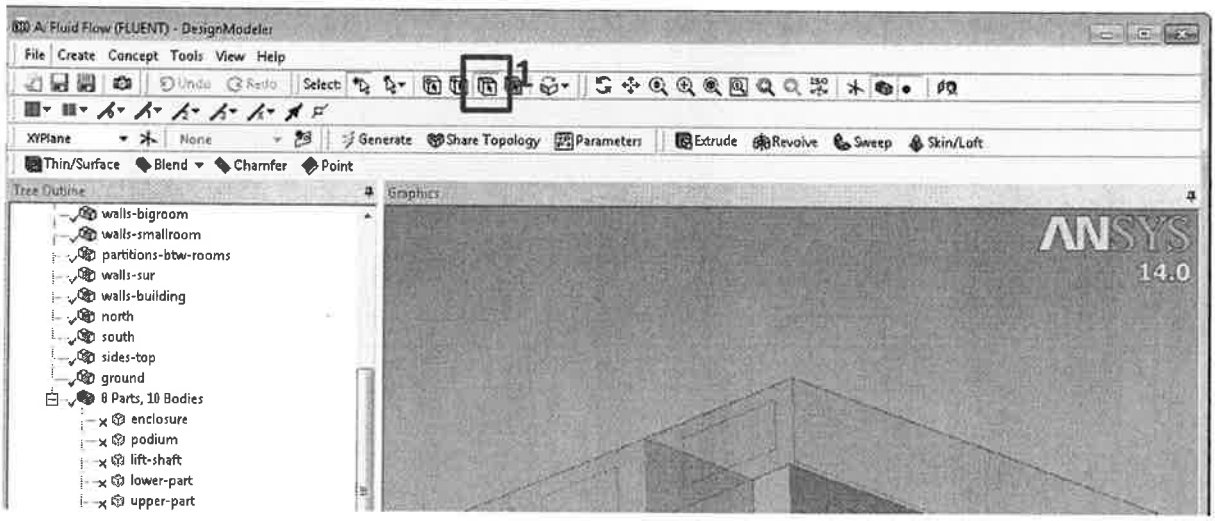
A.2 Creation of a new window for Body "big-room"

- Re-orient the model display

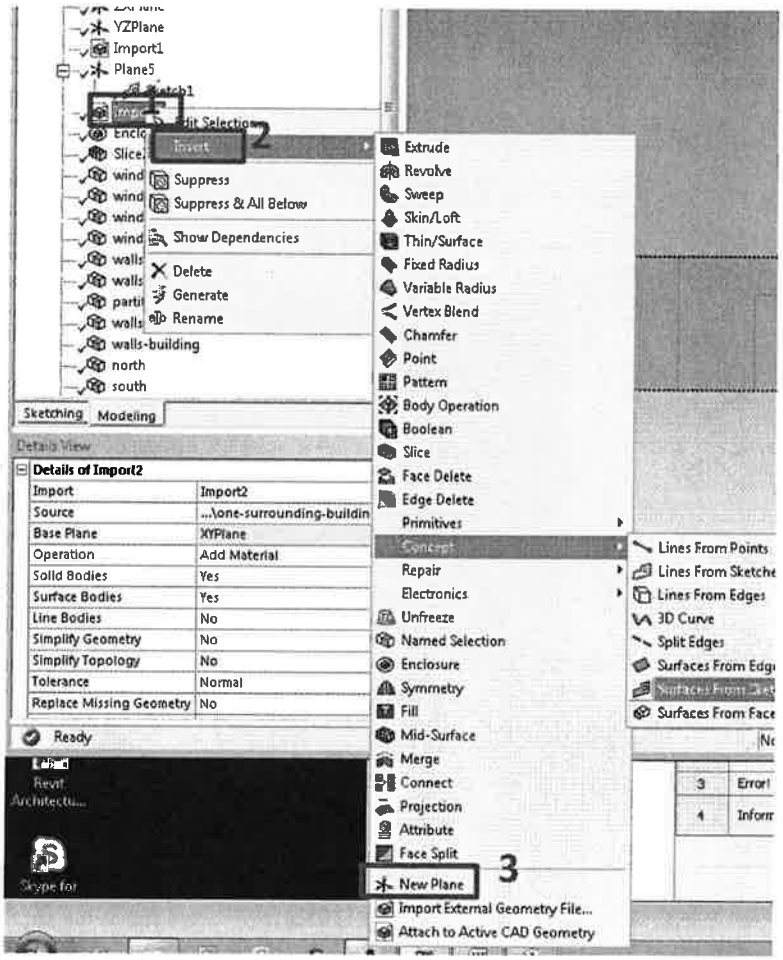
Hide body "enclosure" while keep bodies "small-room" and "big-room" **unhidden** and all solid bodies **suppressed**.



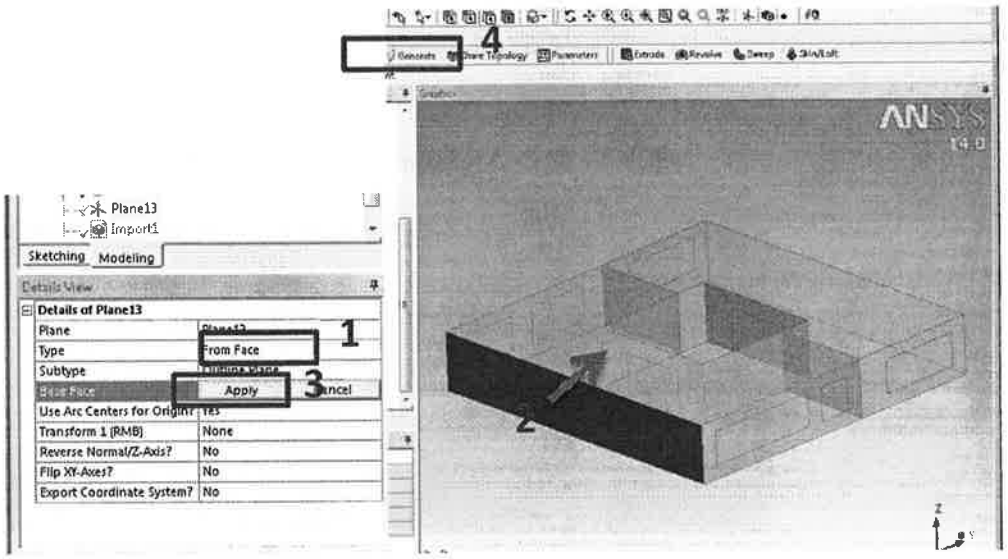
- Choose ***Face-selection*** filter.



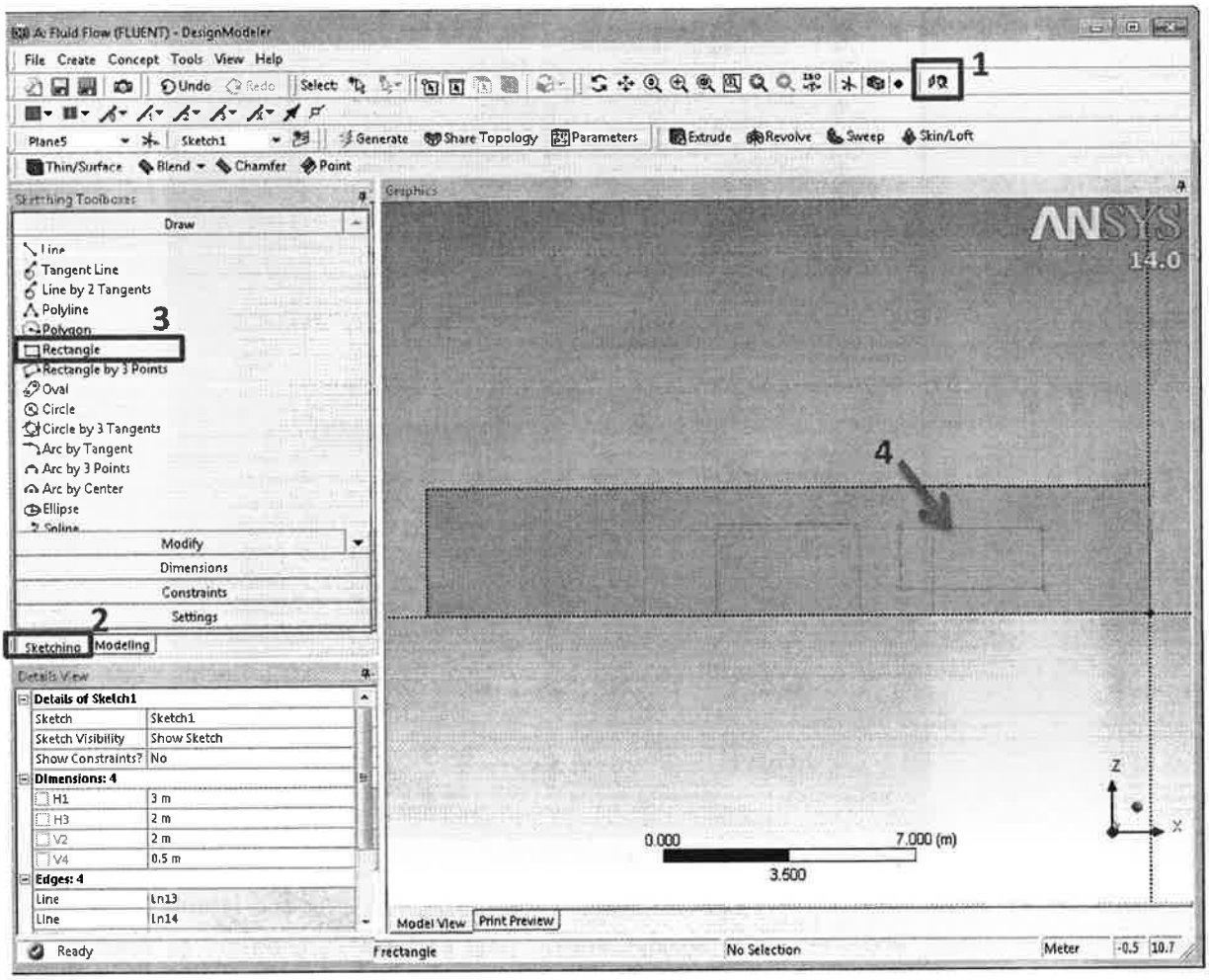
- Under **Tree Outline** side window, *right-click* **Import1** feature icon in the **Tree Outline** side window, then choose *insert -> New Plane*.



- Under **Detail View** side window, choose "**From Face**" Type, *left-click* the lateral face in the Model View screen, and then click "**Apply**" button next to Base Face option to finalize the selection. Then click "**Generate**" button to create the working plane.



- Select new plane feature (for example, "Plane13") in **Tree Outline** side window and then click "**Look at Face**" button to view the model from the angle normal to the face. Switch from Modeling mode to **Sketching** mode. Choose a **Rectangle** sketch in the **Sketching Toolboxes** side window and then draw a rectangle on the working plane in Model View window.



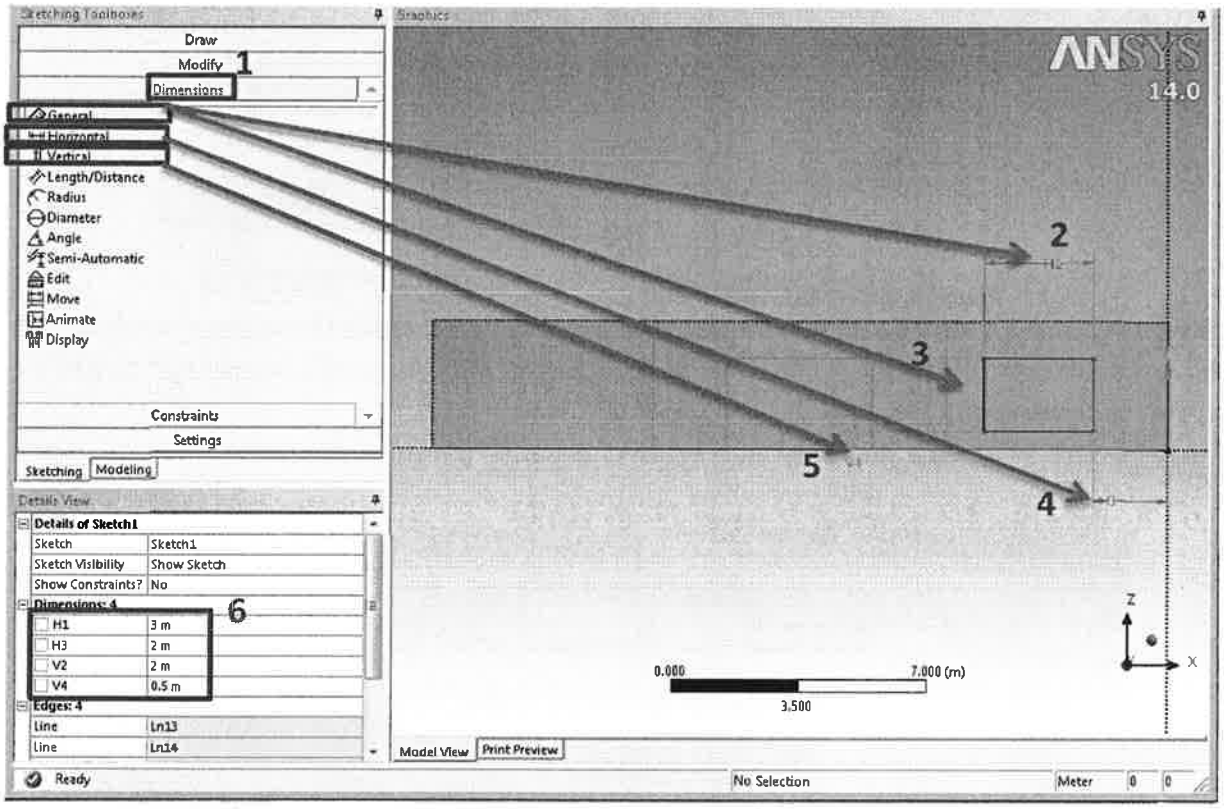
- Set dimensions for the new window.

Use "**general**" option for dimension H1 and V2, "**Horizontal**" for H3 and "**Vertical**" option for V4. For "**general**" dimension, just simply left-click the side of the rectangle, and drag the mouse upwards (for H1) or to the left (for V2) and then click the left mouse button.

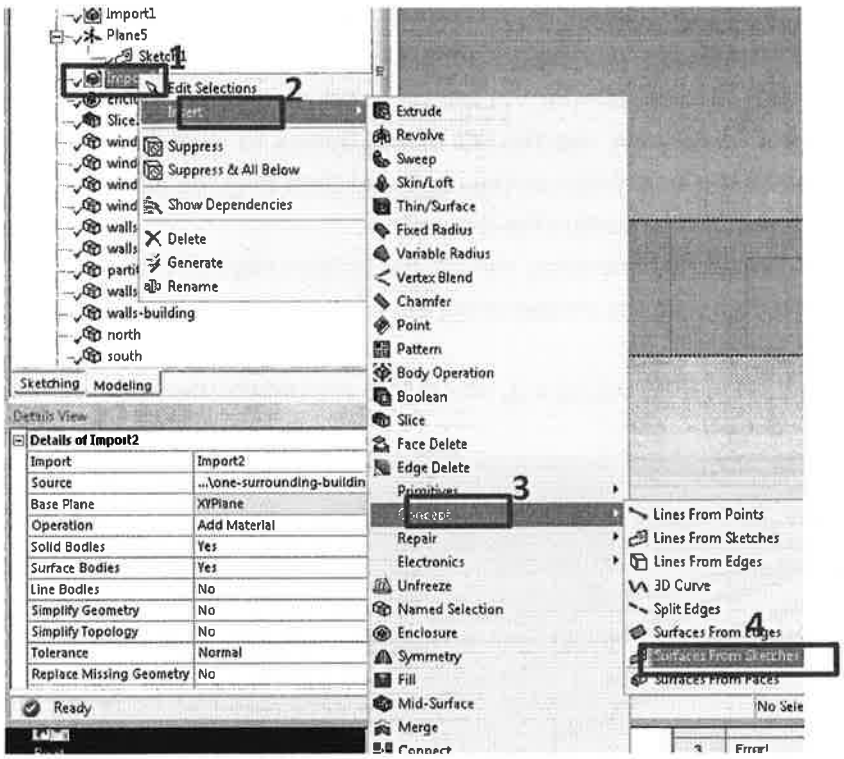
For "**Horizontal**" dimension, use the left mouse button to click the right edge of the rectangle first, followed by the local y-axis on the right, and then drag the mouse downwards a bit to click the left mouse button to position the dimension.

Similarly, for "**Vertical**" dimension, choose the bottom edge of the rectangle and local x-axis in sequence, and then drag the mouse to the left a bit to click the left mouse button to position the dimension.

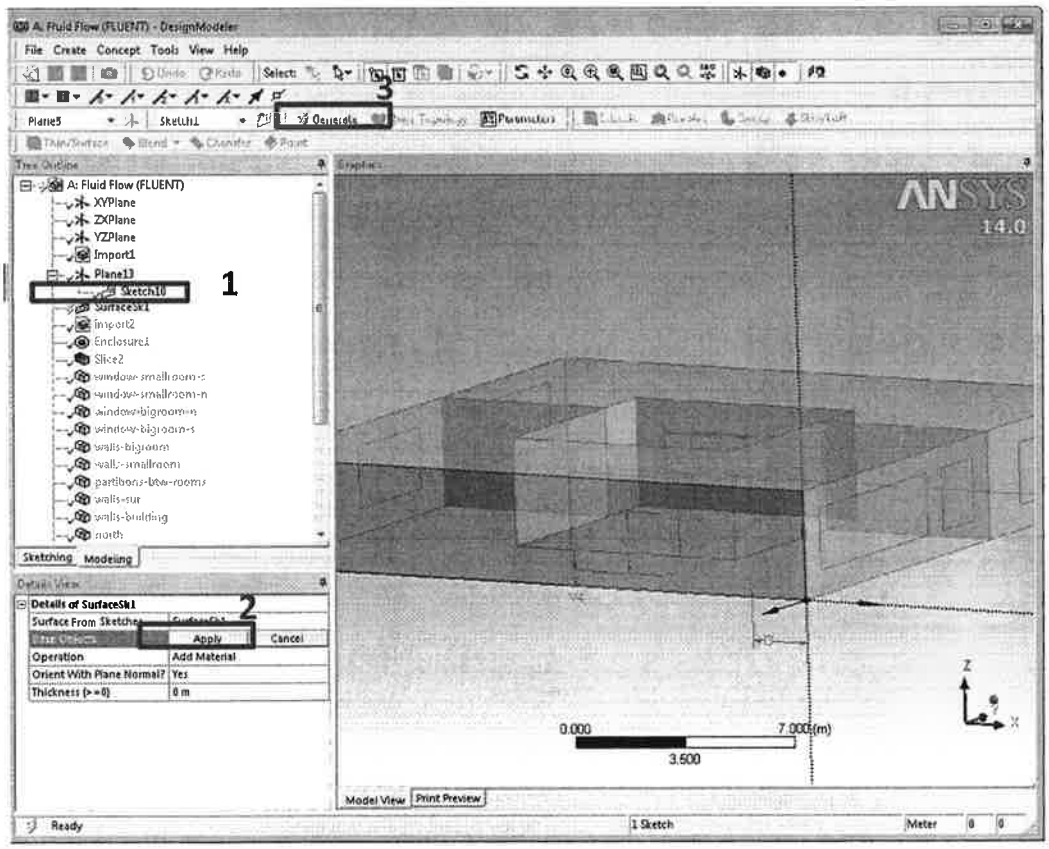
Set the values (H1 = 3, H3 = 2, V2 = 2, V4 = 0.5) in the **Details View** side window.



- Right click the feature "Import1" and choose *Insert ->Concept ->Surfaces from Sketches* to create a surface body.



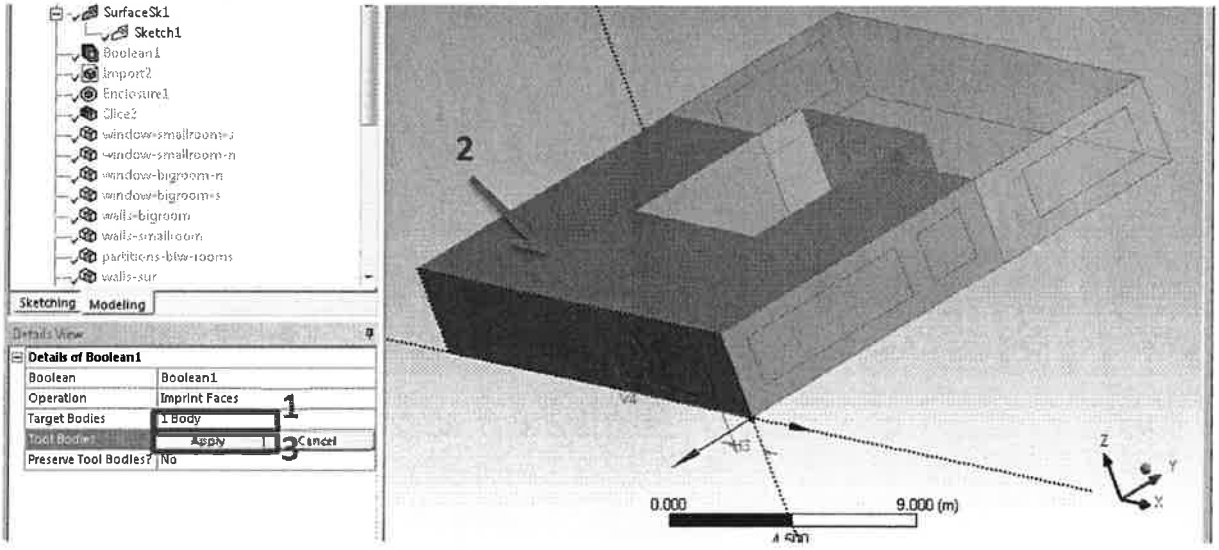
- **Left-click** the newly created sketch (for example, "**Sketch10**) within Plane13 feature in **Tree Outline** side window, then click "**Apply**" button to finalize the selection. Then click "**Generate**" button to create the surface body.



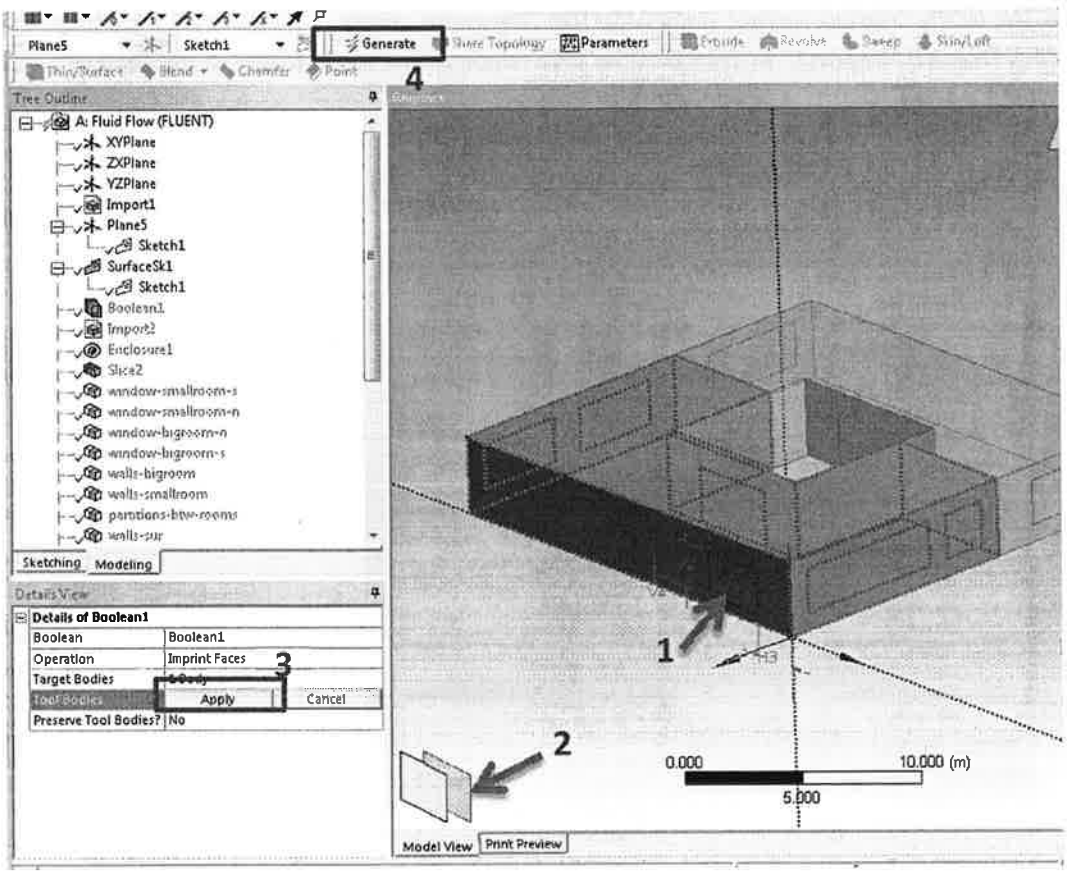
- Imprint the newly created surface body onto the "big-room" body.
  Right-click Feature "Import1" in **Tree Outline** side window to choose ***Insert -> Boolean***.



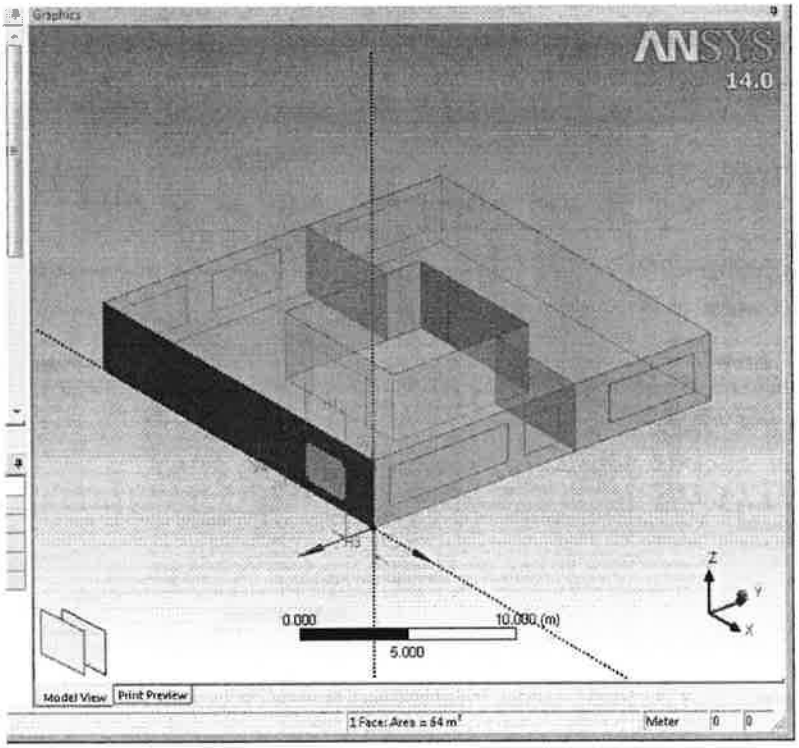
- Choose "**Imprint Faces**" Operation within the **Details View** side window at the left bottom first. left-click the "big-room" body in **Model View** screen and then click the "Not Selected" area on the right of "Target Bodies" in the **Details View** side window, and then click "**Apply**" button to select the body.



- Similarly, select the surface body as the Tool Bodies for the Boolean operation. Note that use the selection tool appearing at the left bottom of Model View Screen to choose the surface body by left-clicking the second tiny plane until it is in red. Click "**Generate**" button to complete the Boolean operation to split the newly created surface from its base face.

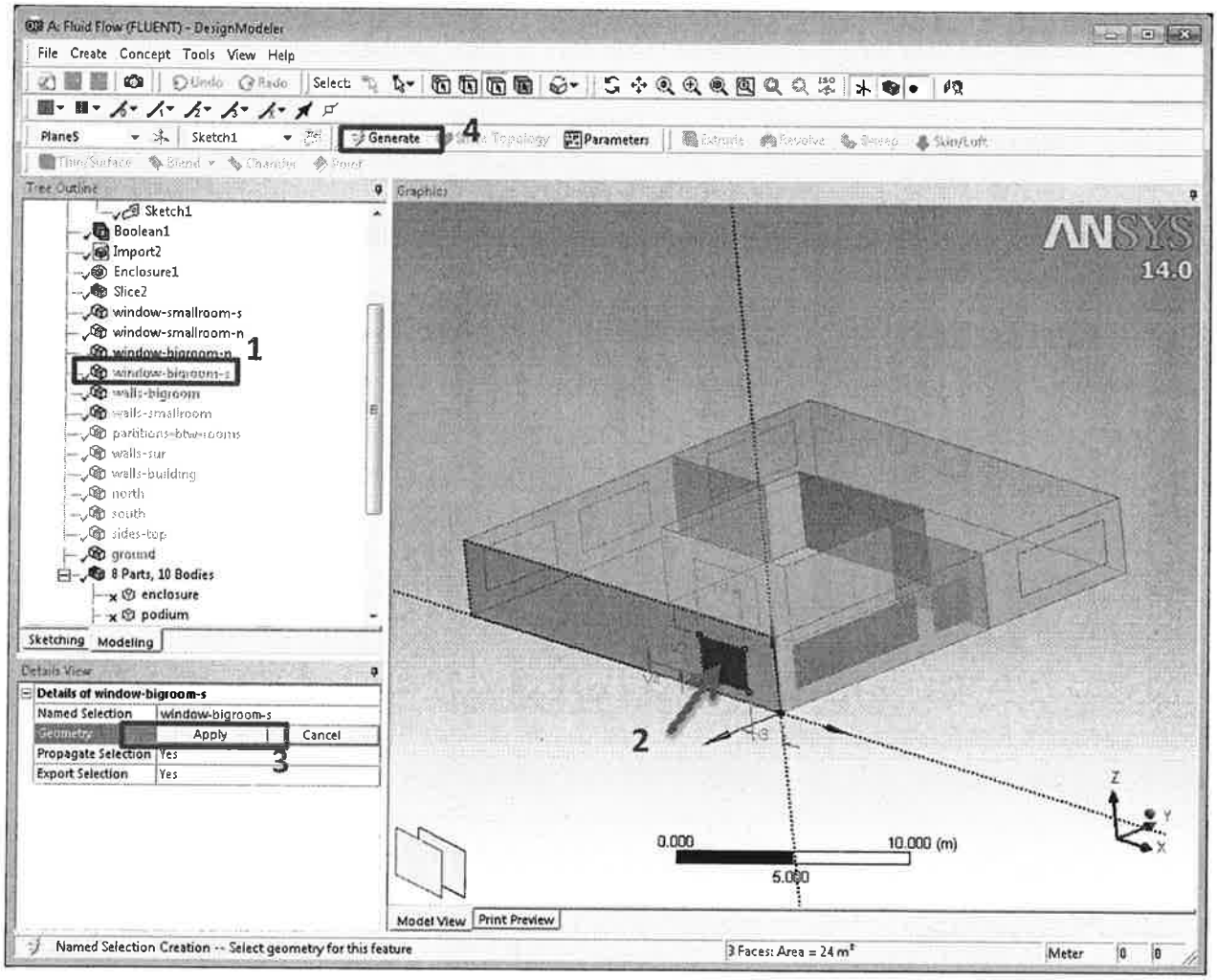


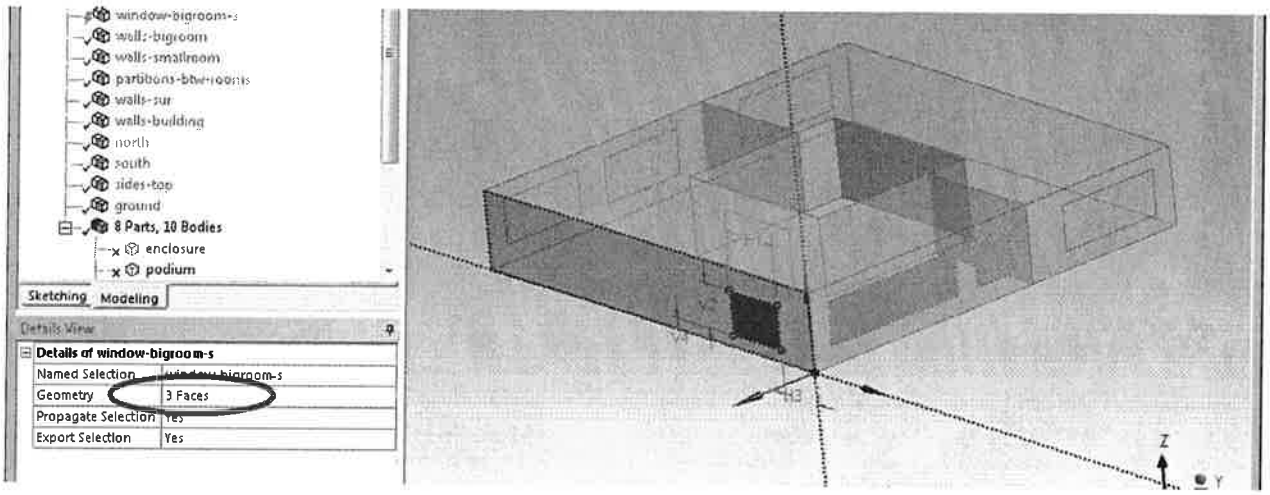
The final model should be the same as what is shown in the picture below.



A.3 Update the name selections

- Add the new window in the group of "window-bigroom-s" by right-clicking Feature "window-bigroom-s" in Tree Outline side Window. Hold *CTRL* key and then left click the newly created surface that reprsents the new window, and then *click* "**Apply**" Key to finalize the selection. ***Make sure that in total 3 faces are selected.*** Then, click "**Generate**" button to update the re-grouping.

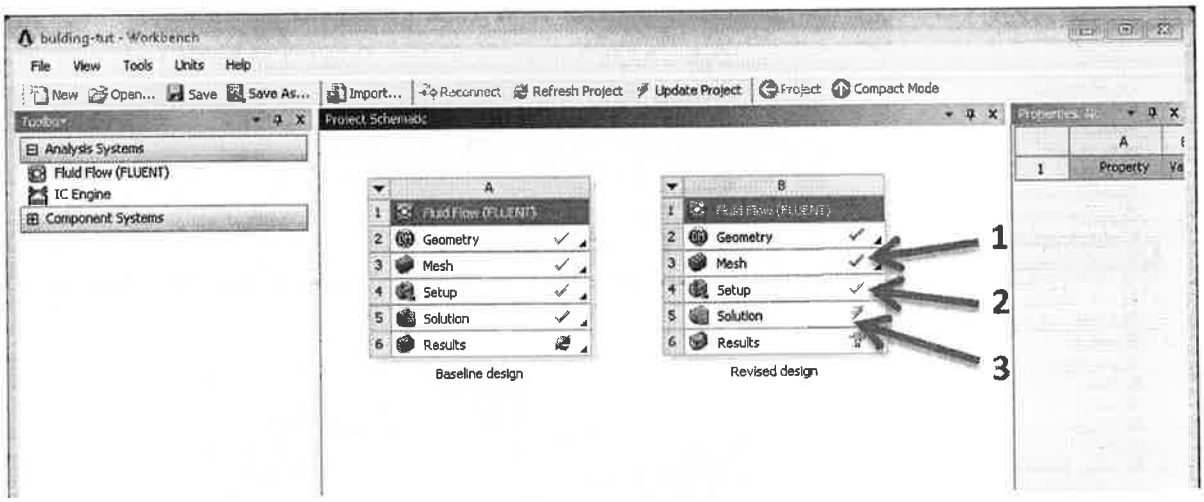




A.4 Save the project and then close Design Modeler.

A.5, update the Mesh, Setup and Solution, in sequence in the Workbench environment.

All the settings adopted in the model for baseline design will be automatically applied for the revised design. Please note that it will take some time to re-create the mesh, re-impose settings in FLUENT and re-simulate the case with the inclusion of one more window for Body "big-room".



Tutorial 3 Simulation of natural ventilation across a residential building – Model Building from Primitive Geometry

Abstract

This tutorial simply demonstrates the procedure of CFD simulation of natural ventilation across units of a residential building, compliant with the CFD methodology outlined in BCA Green Mark Code.

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).

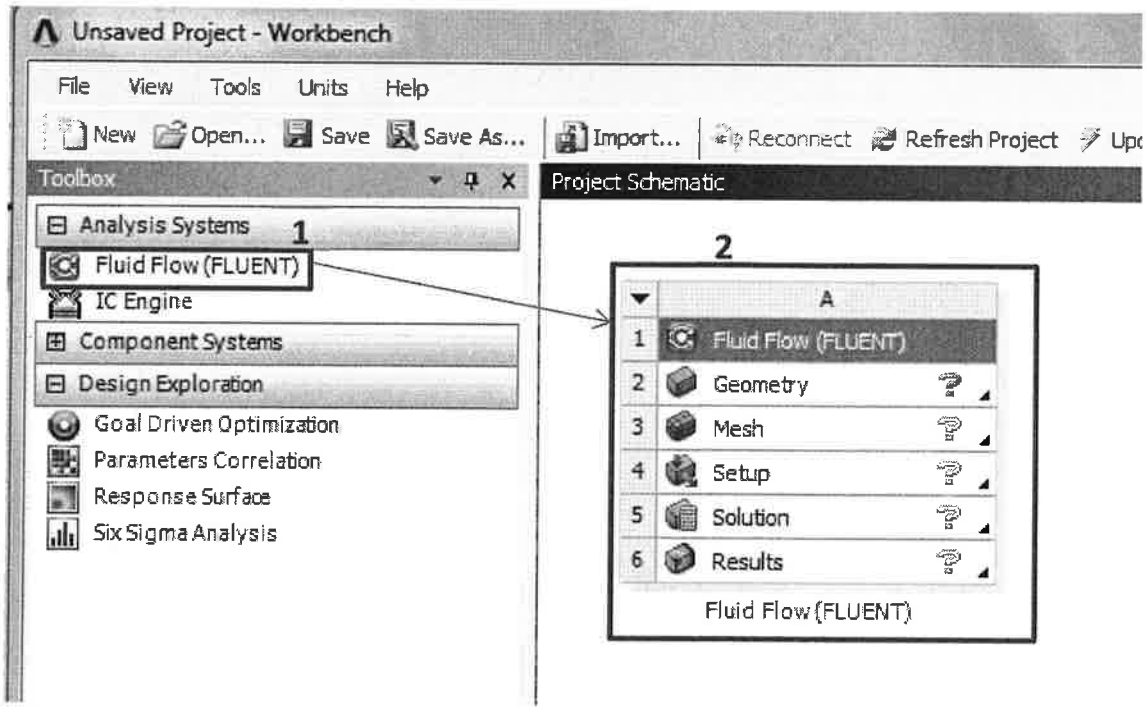
Please note that the tutorial is drafted using ANSYS Workbench 14.0.

Pre-requisite:

- Familiar with the workbench working environment (complete Tutorial 1 using workbench)
- One user define function:
  - profile-log-north.c – log-law wind profile (north wind)

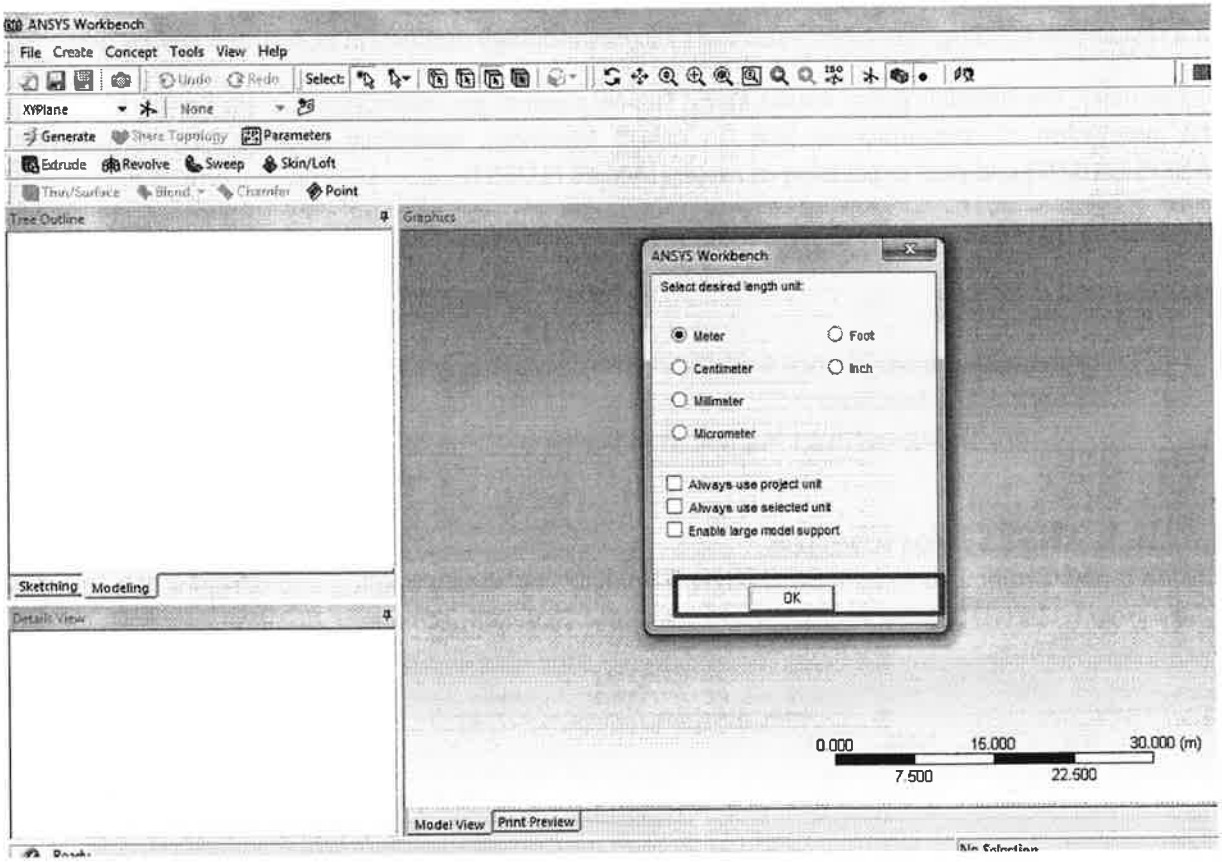
Launch ANSYS Workbench

Create a new project for Fluid Flow (FLUENT) Analysis, by simply holding and dragging the icon for "Fluid Flow (FLUENT)" under Toolbox to the right into Project Schematic.



1. Geometry Creation in Design Modeler

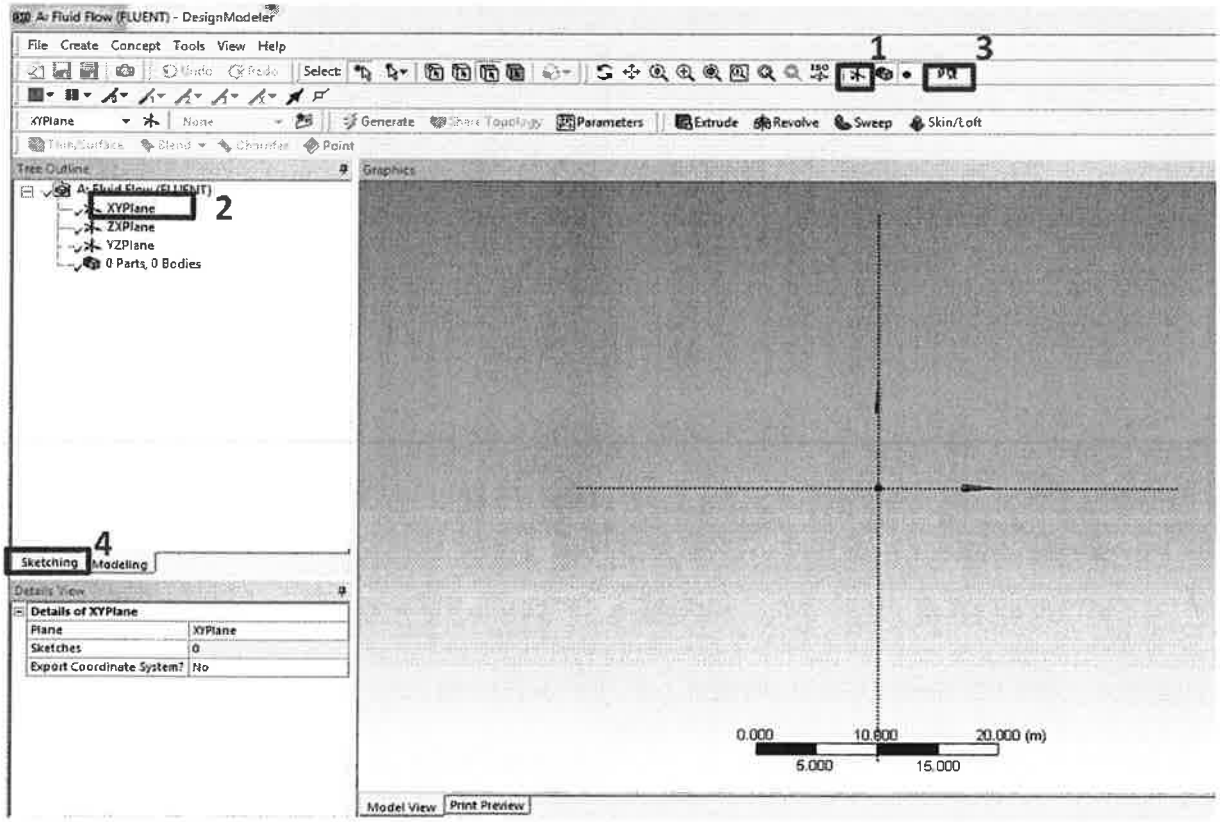
Double click Geometry icon to launch Design Modeler.
Choose the default SI unit system (meter).



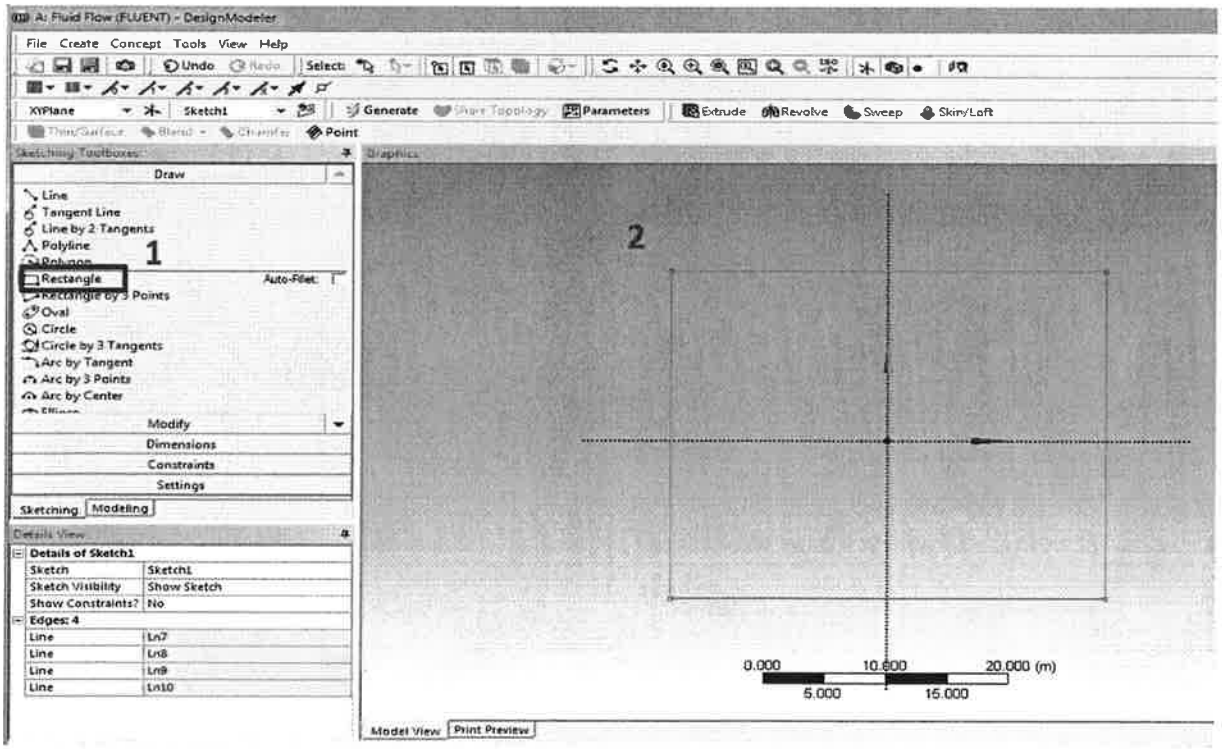
1.1 Creation of the building model of interest

1.1.1 Generate the body of "podium"

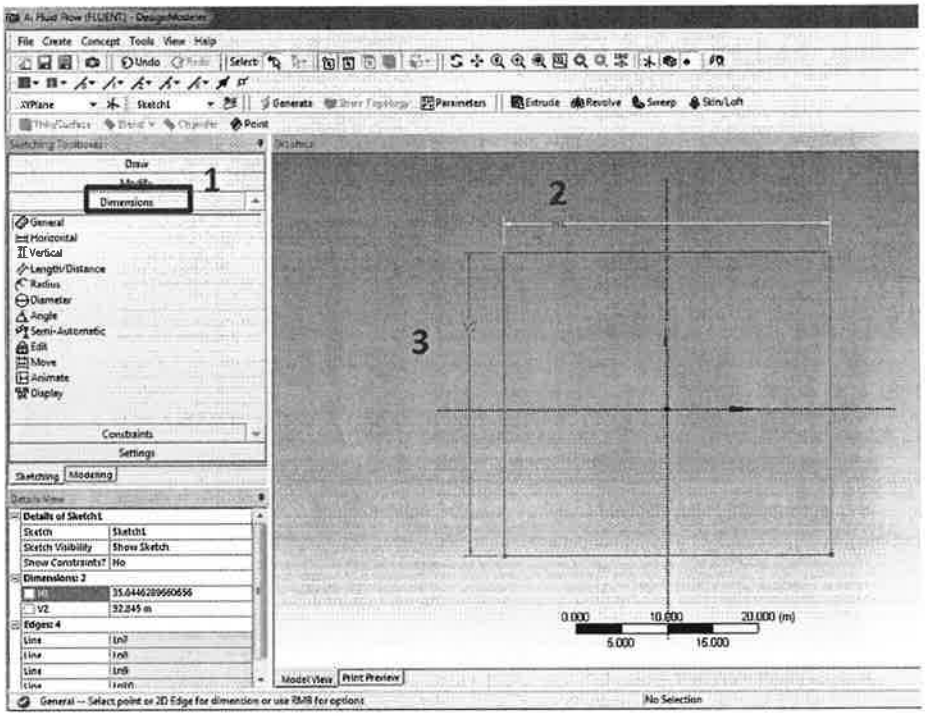
Left click on the **Display Plane** icon to view the XYZ planes. Click **XYPlane**, use the **Look At** icon to orient the plane in the normal direction. Switch to sketch mode using the **Sketching** tab to draw **Sketch1** on the XYPlane.



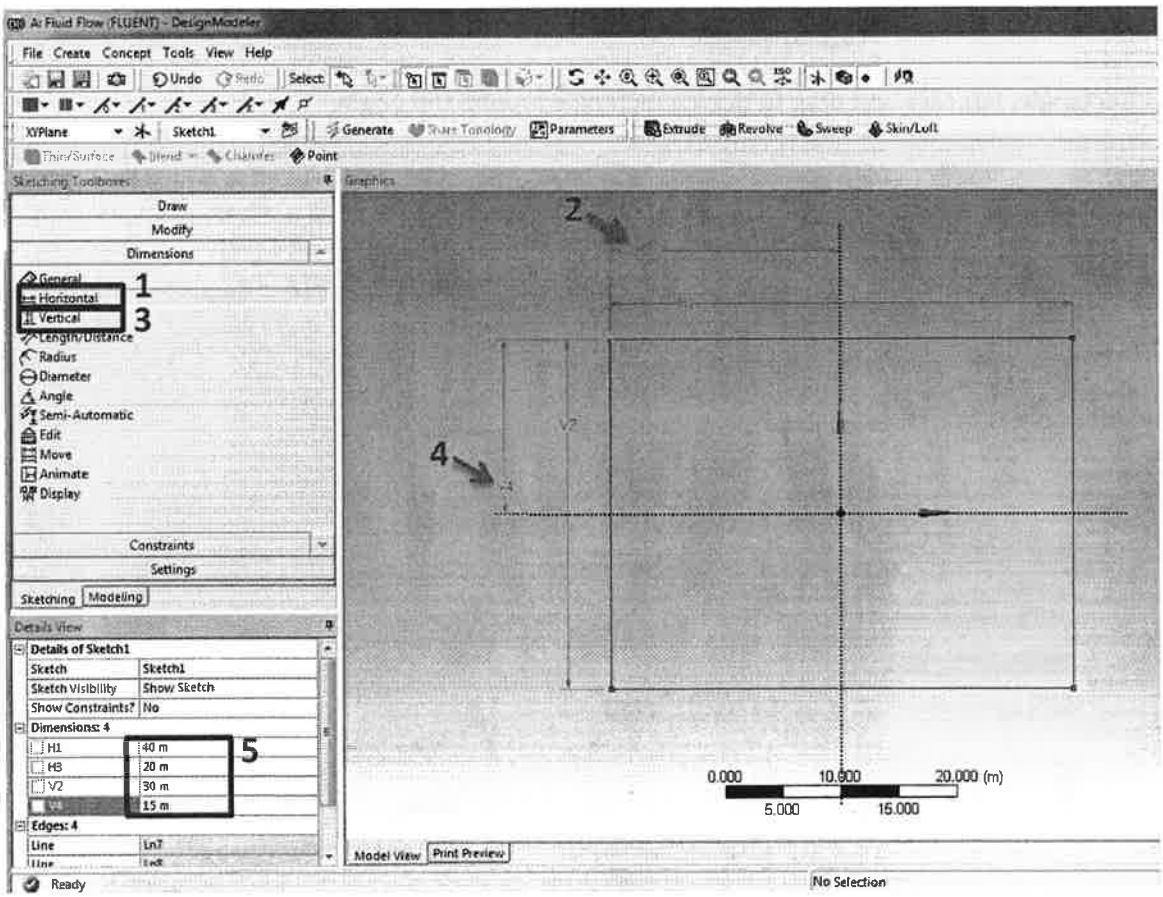
Select **Rectangle**, left click and drag to draw a rectangle around the origin.



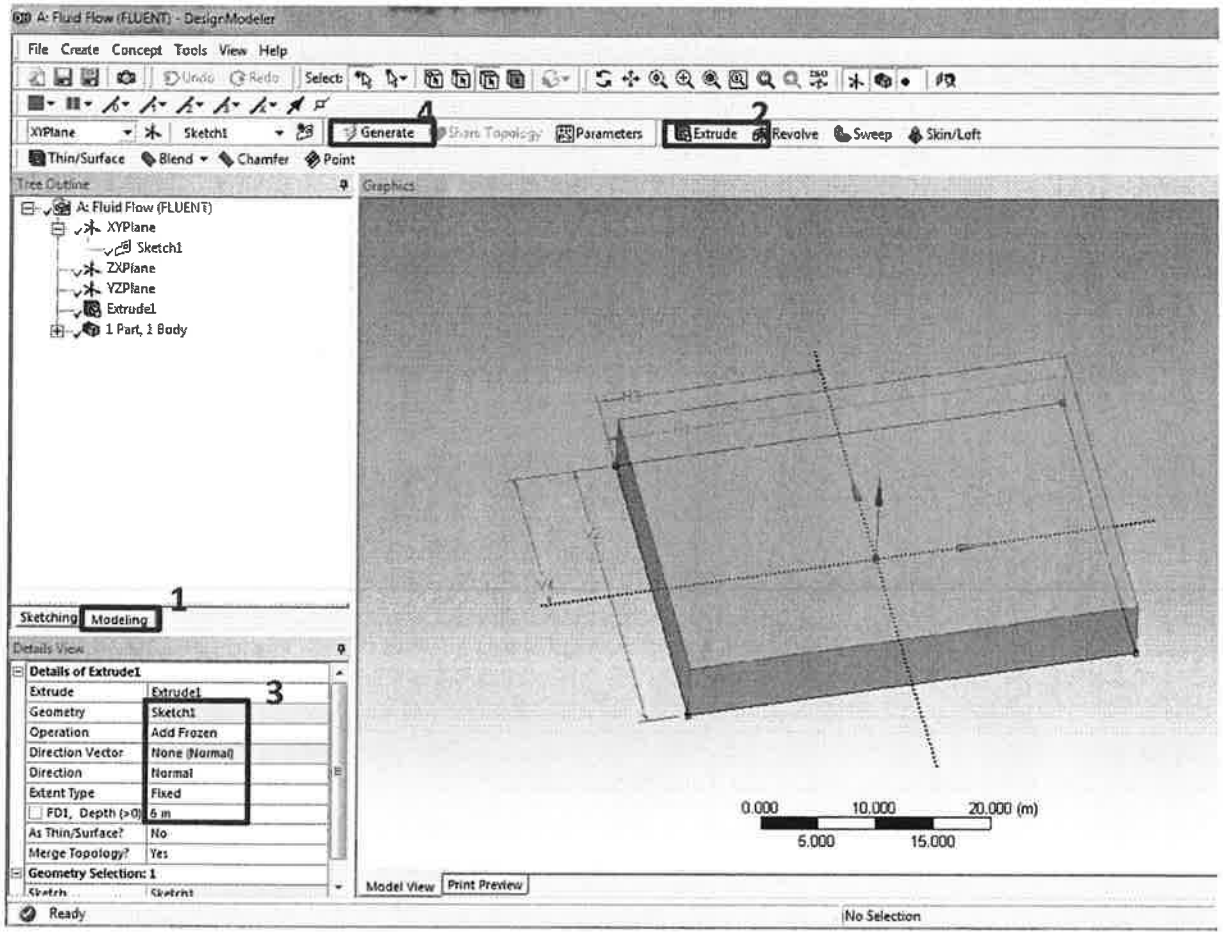
Click on **Dimensions** in Sketching. Left Click on the top edge of the rectangle, drag and release to obtain the dimension of H1. Similarly, obtain the dimension of V2.



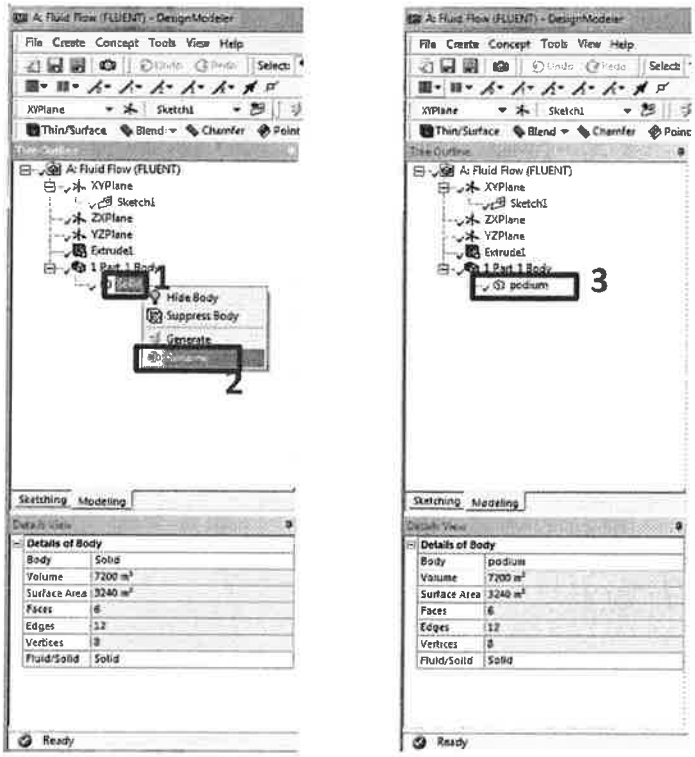
Select **Horizontal**, left click on the left edge of the rectangle and then click on the y axis, drag and release to obtain H3. Similar, select **Vertical**, left click on the top edge and then the x axis, drag and release to obtain V4. Input the dimensions as listed.



Switch to **Modeling** mode, click **Extrude**, input the details for Extrude, and then click **Generate**.

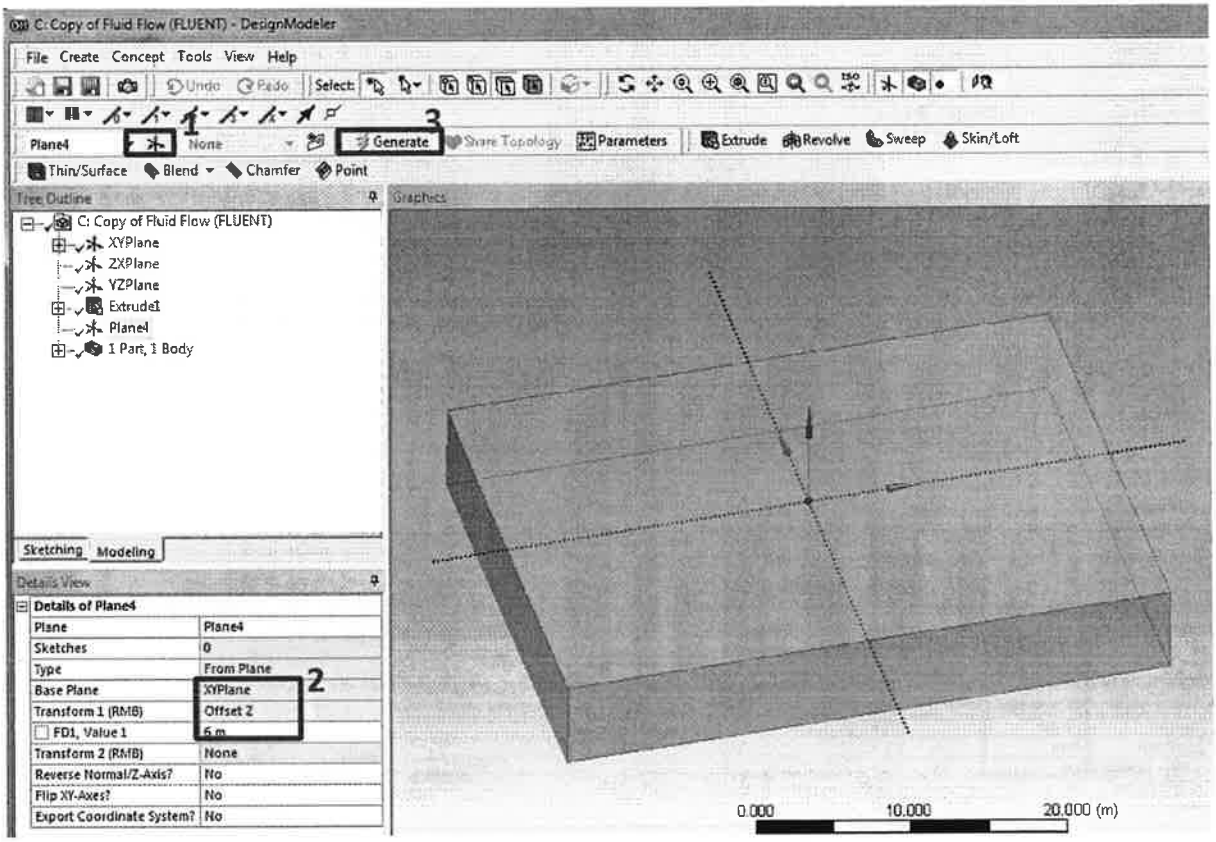


Rename the "solid" to "**podium**".

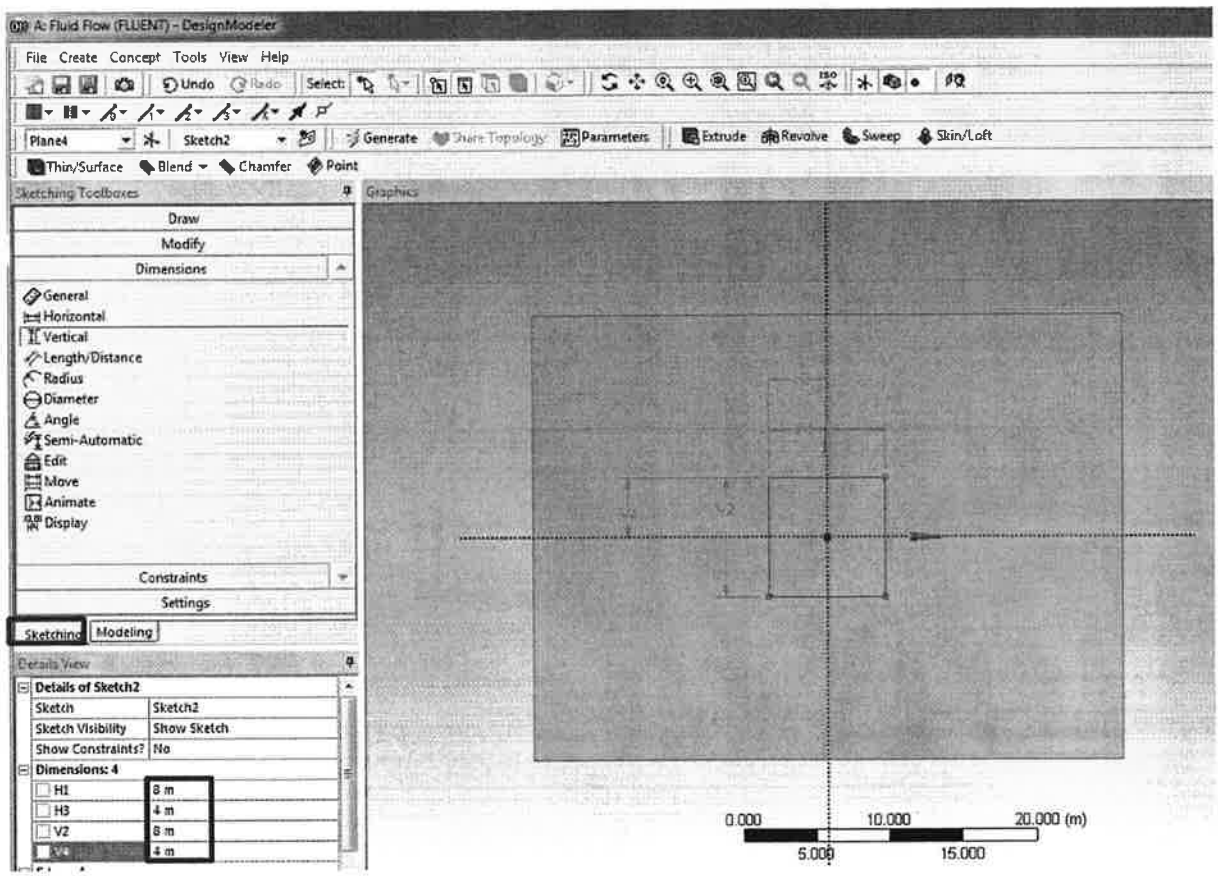


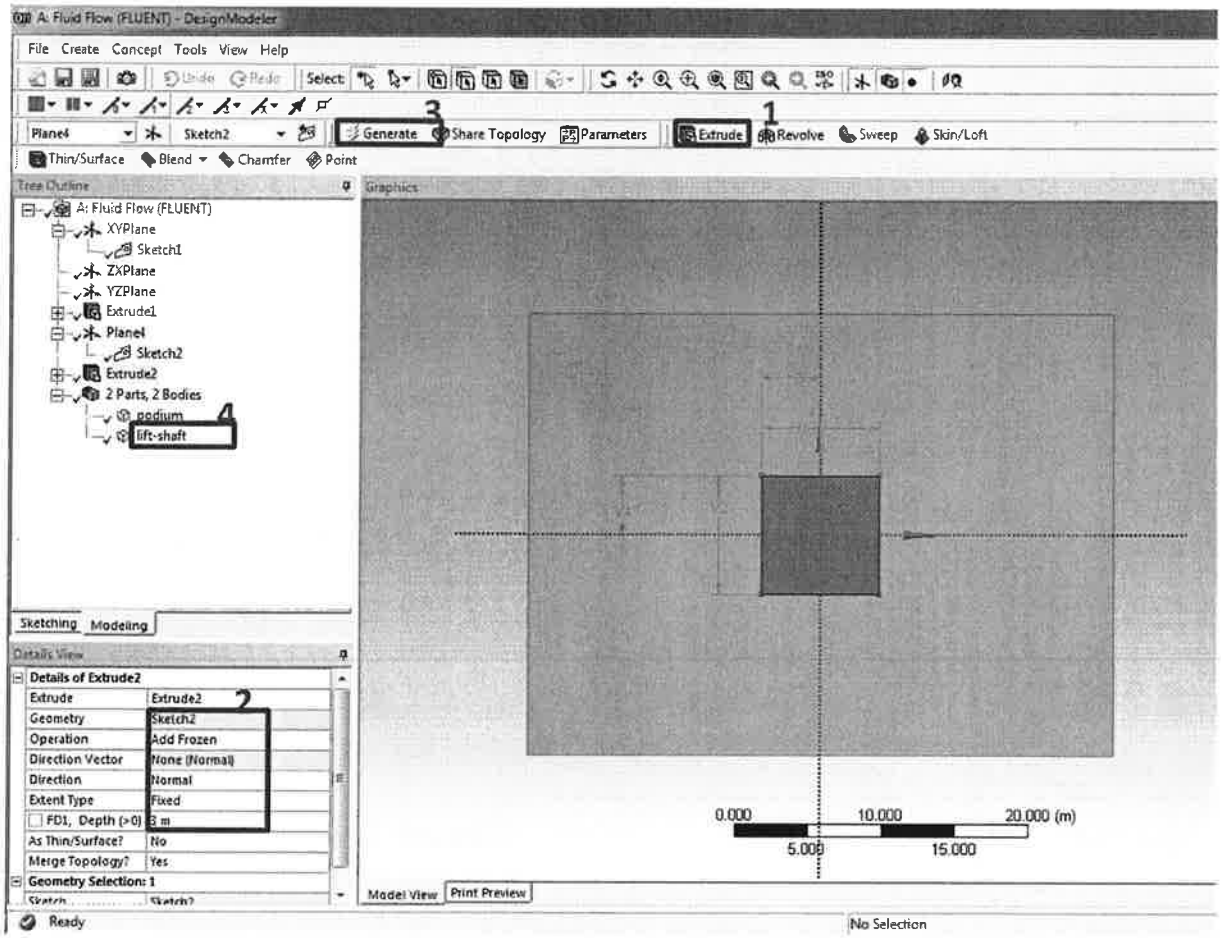
1.1.2 Generate the body of "lift-shaft"

Click on the **New Plane** icon, generate Plane4 from XYPlane by **Offset Z** with 6m.



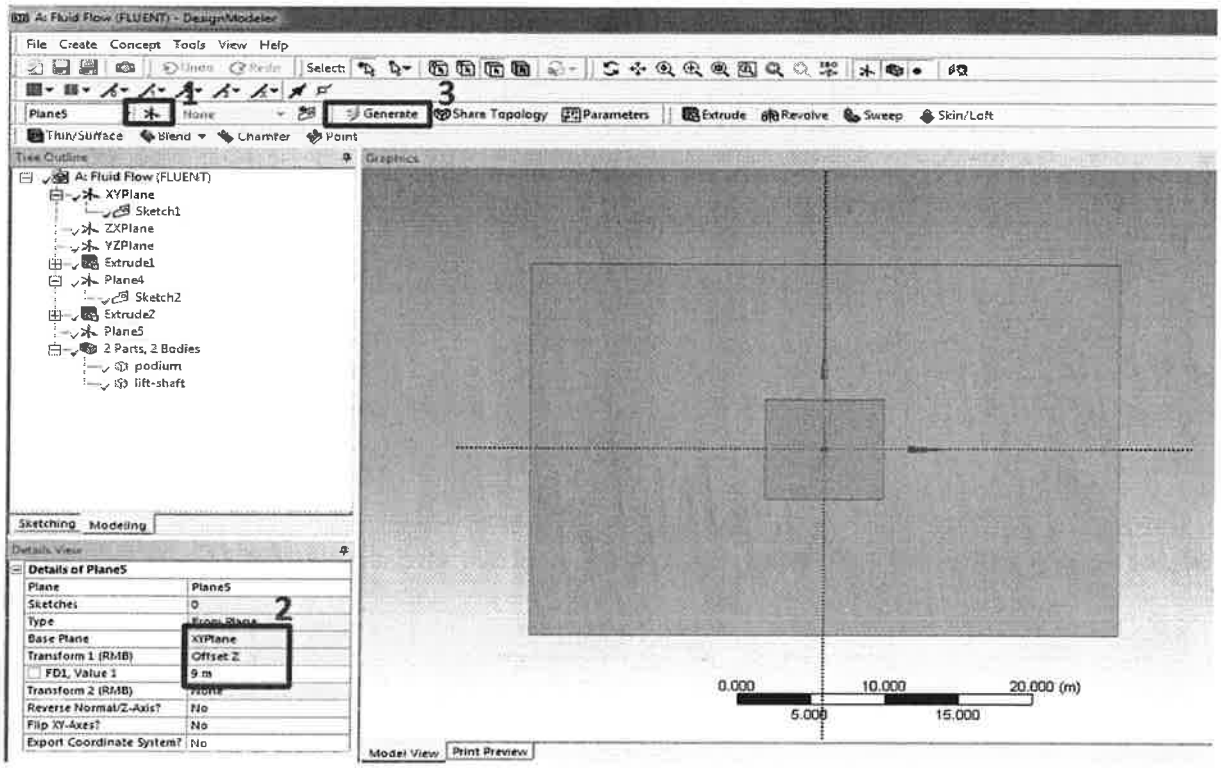
Follow the same operating procedure as for **Podium** generation to create Sketch2 on the **Plane4** based on the dimensions highlighted in the picture below. Rename the newly created body as "lift-shaft".

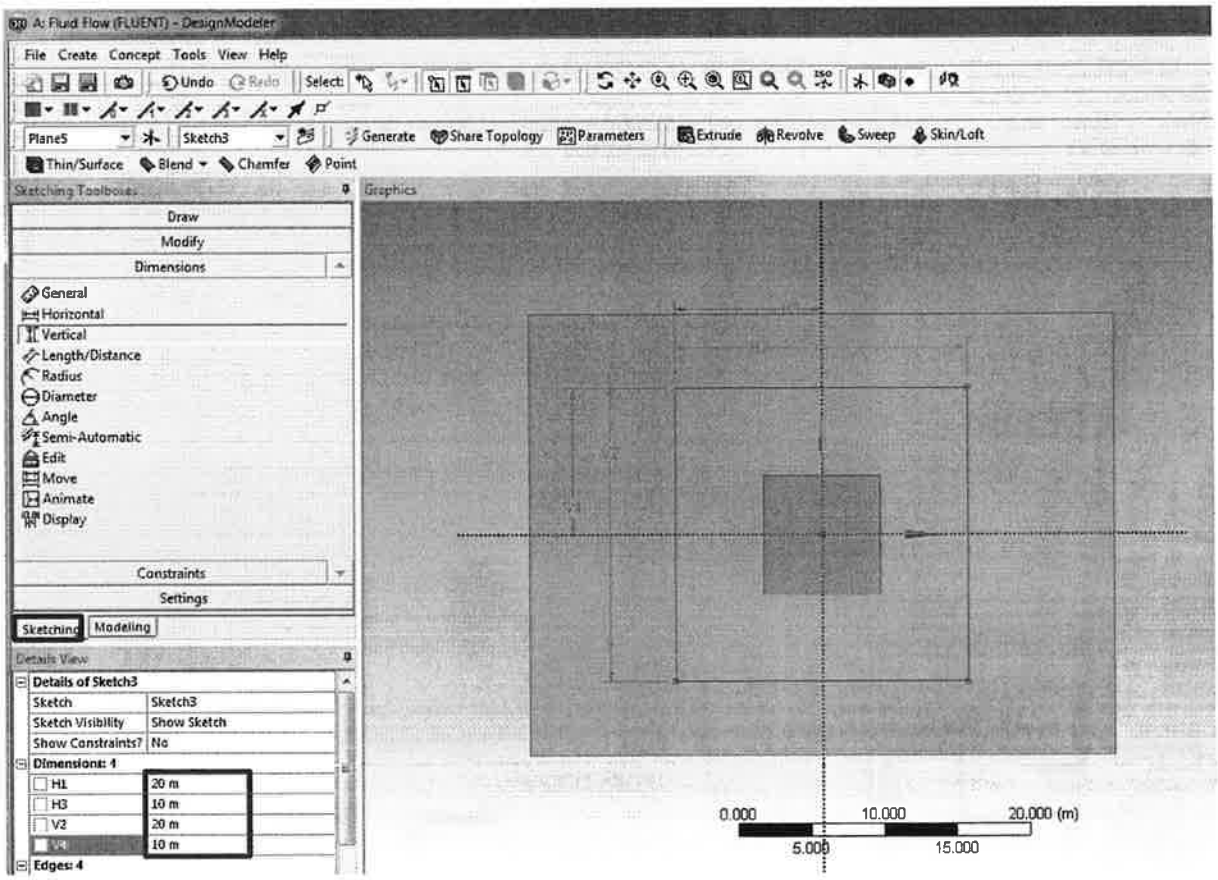


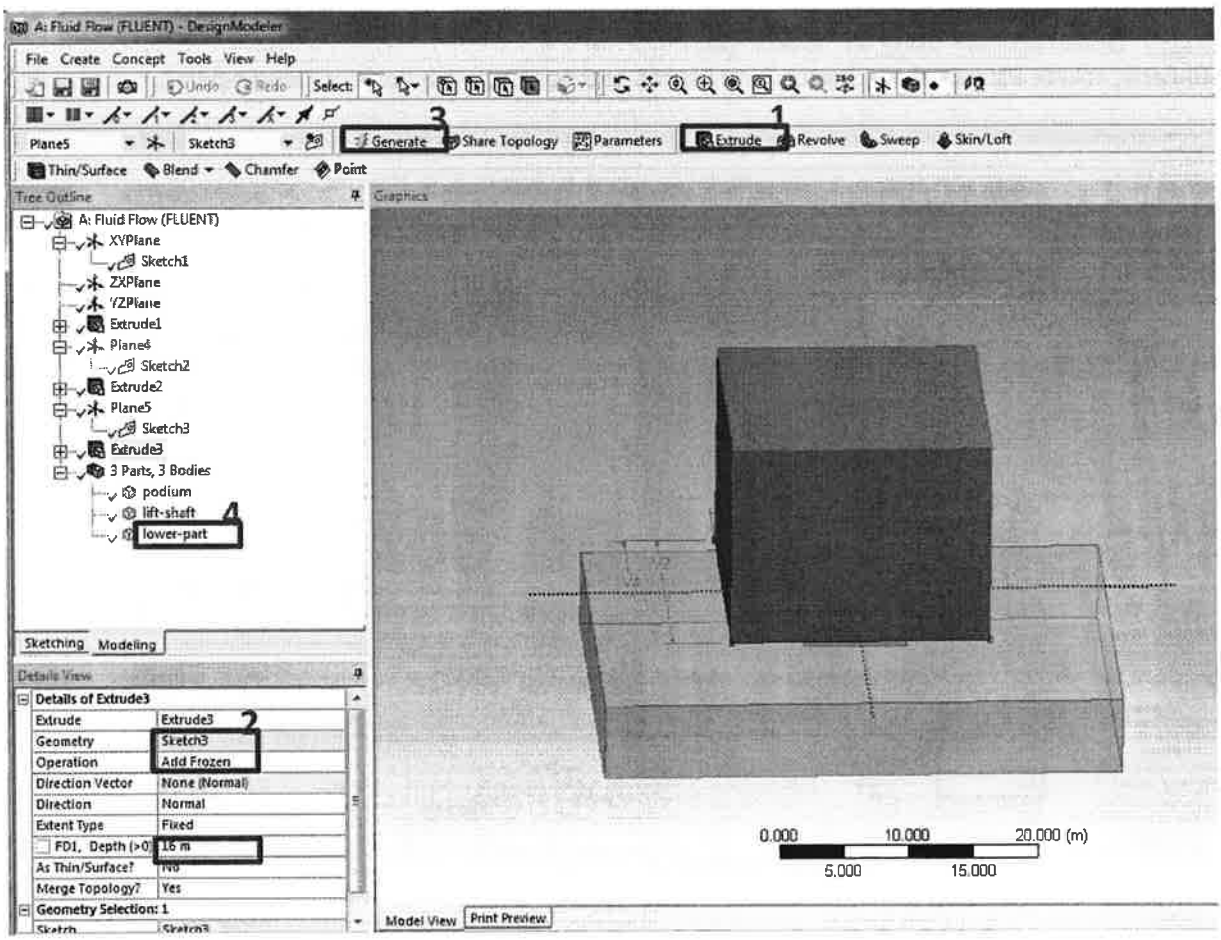


1.1.3 Generate the body of "lower-part"

Similarly, create Plane5, draw Sketch 3 on Plane5 and generate the body of "lower-part".

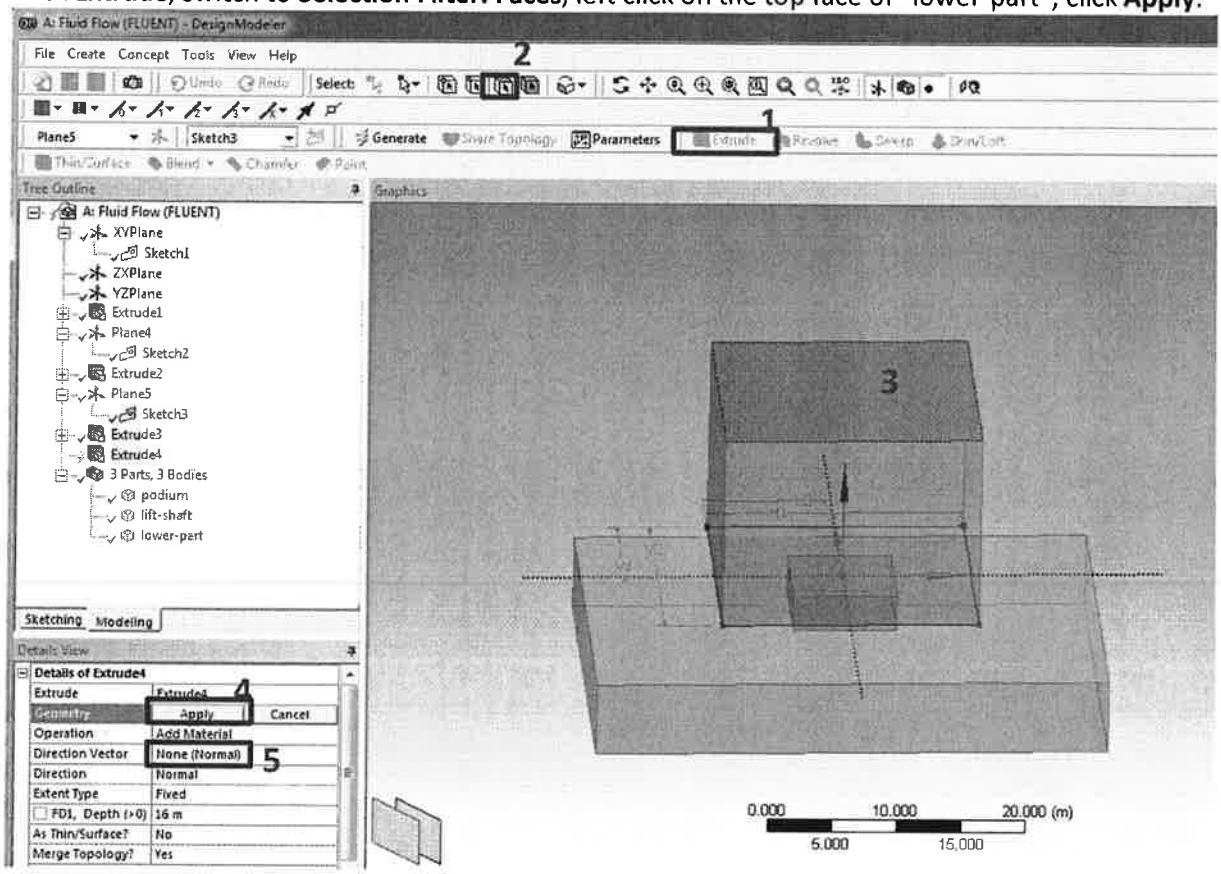



A: Fluid Flow (FLUENT) - DesignModeler
File Create Concept Tools View Help
Undo Redo Select:
Plane5 Sketch3 Generate Share Topology Parameters Extrude Revolve Sweep Skin/Loft
Thin/Surface Blend Chamfer Point
Sketching Toolboxes
Draw
Modify
Dimensions
General
Horizontal
Vertical
Length/Distance
Radius
Diameter
Angle
Semi-Automatic
Edit
Move
Animate
Display
Constraints
Settings
Sketching Modeling
Details View
Details of Sketch3
Sketch Sketch3
Sketch Visibility Show Sketch
Show Constraints? No
Dimensions: 1
H1 20 m
H3 10 m
V2 20 m
V4 10 m
Edges: 4
Graphics
0.000 10.000 20.000 (m)
5.000 15.000

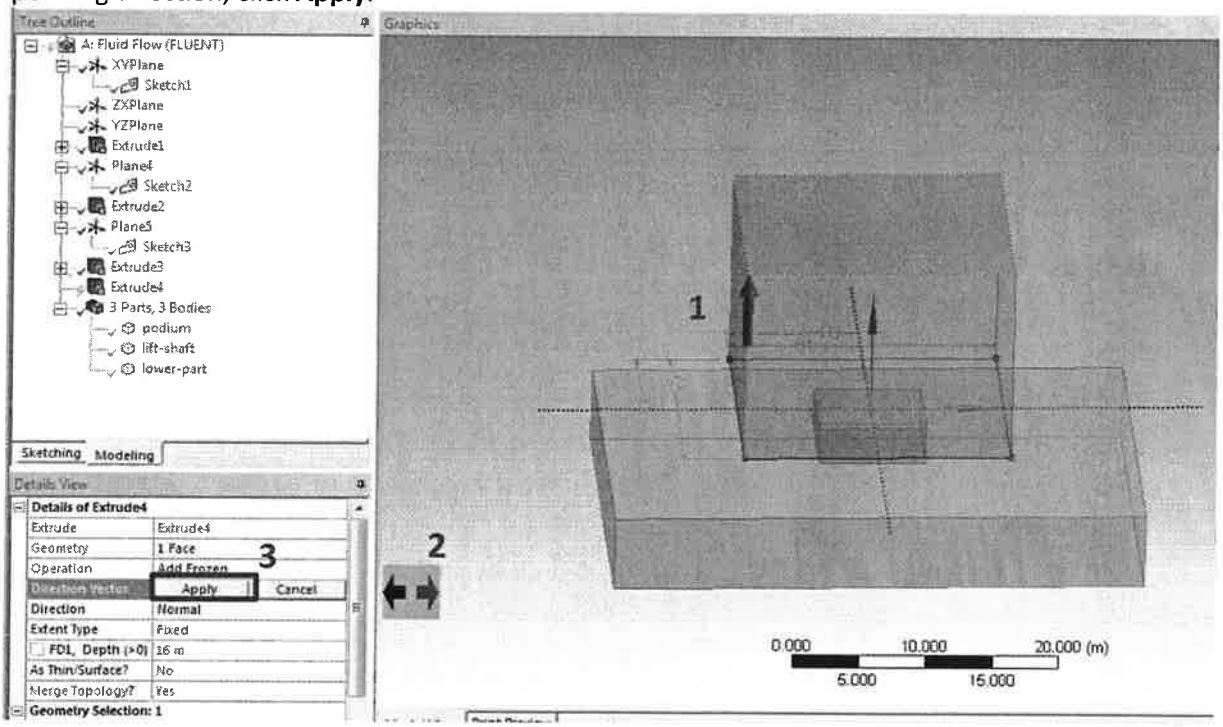

A: Fluid Flow (FLUENT) - DesignModeler
File Create Concept Tools View Help
Undo Redo Select:
3
1
Plane5 Sketch3 Generate Share Topology Parameters Extrude Revolve Sweep Skin/Loft
Thin/Surface Blend Chamfer Point
Tree Outline
A: Fluid Flow (FLUENT)
XYPlane
Sketch1
ZXPlane
YZPlane
Extrude1
Plane4
Sketch2
Extrude2
Plane5
Sketch3
Extrude3
3 Parts, 3 Bodies
podium
lift-shaft
lower-part
4
Sketching Modeling
Details View
Details of Extrude3
Extrude Extrude3
Geometry Sketch3
Operation Add Frozen
Direction Vector None (Normal)
Direction Normal
Extent Type Fixed
FD1, Depth (>0) 16 m
As Thin/Surface? No
Merge Topology? Yes
Geometry Selection: 1
2
Graphics
0.000 10.000 20.000 (m)
5.000 15.000
Model View Print Preview

1.1.4 Generate the bodies namely "small-room", "big-room" and "upper-part".

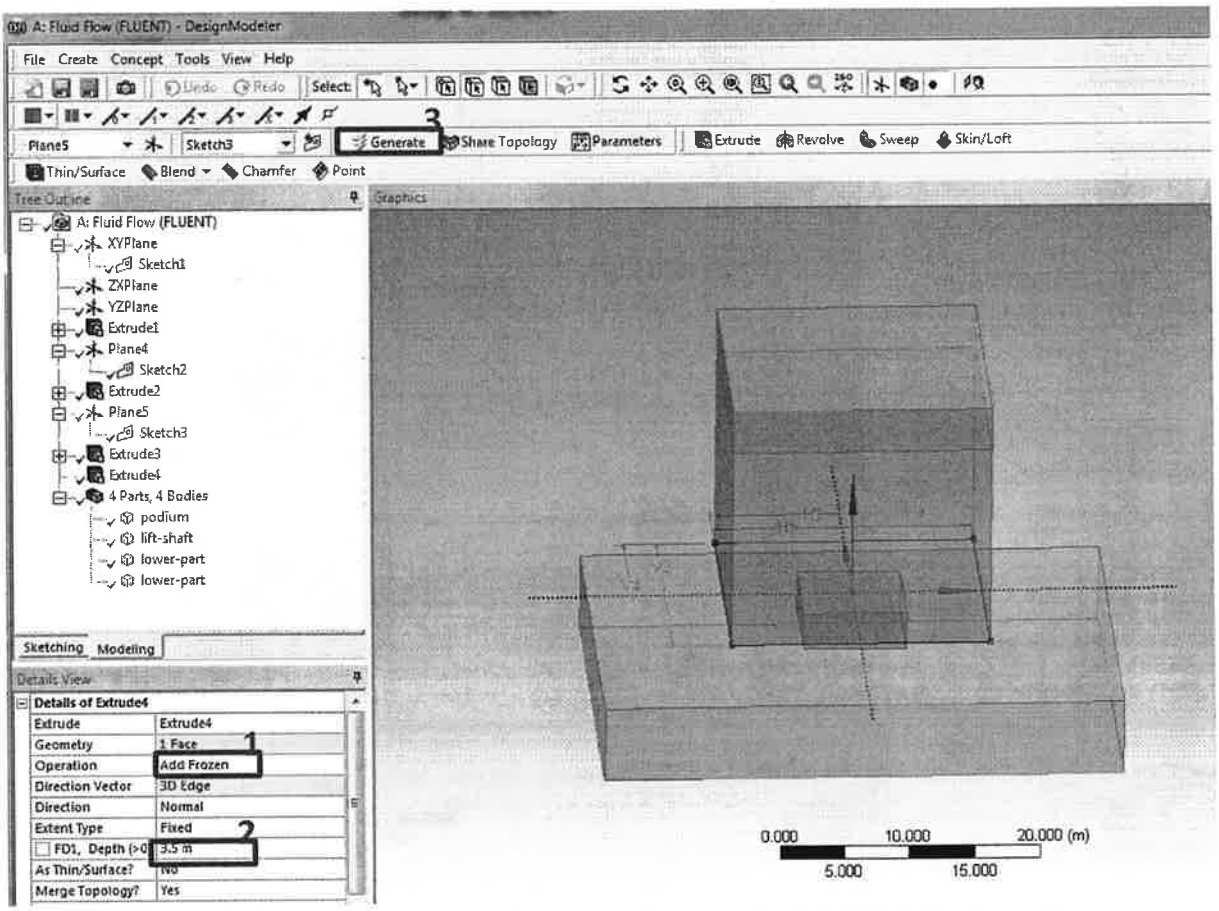
Click **Extrude**, switch to **Selection Filter: Faces**, left click on the top face of "lower-part", click **Apply**.



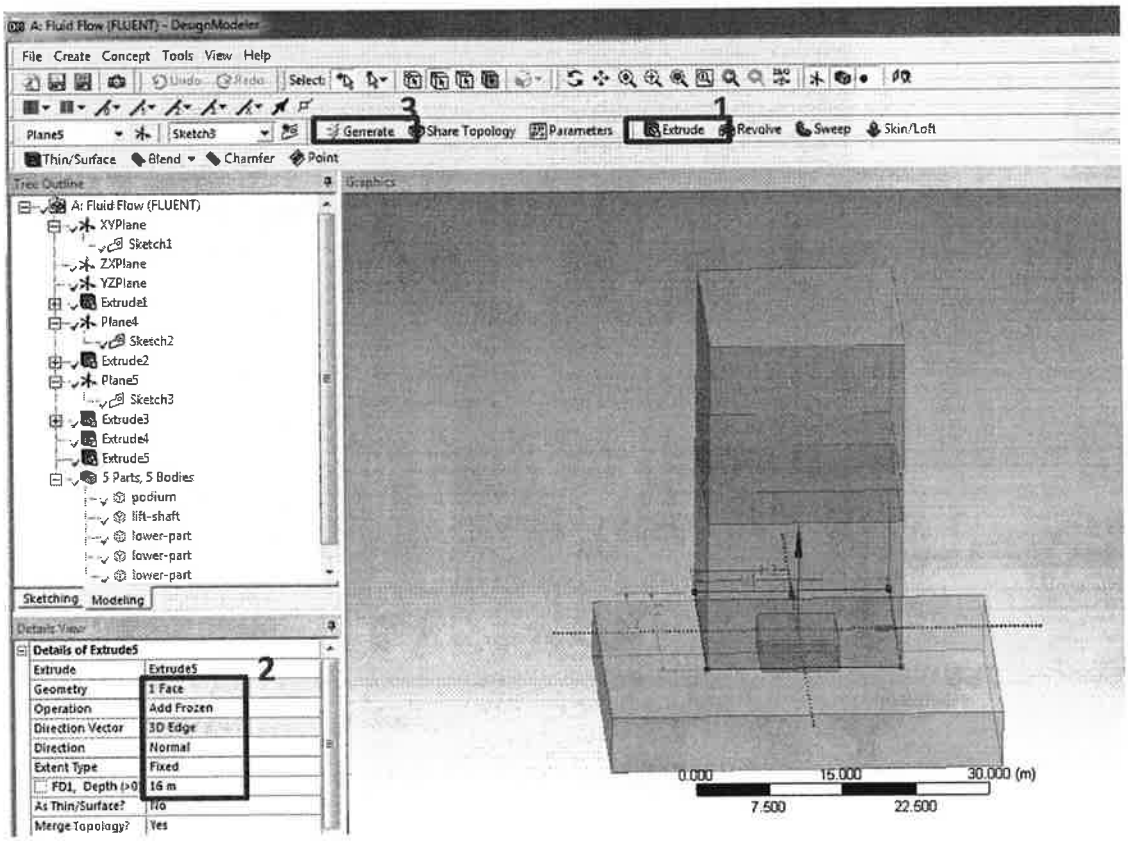
Click on the option of **Direction Vector**, select a vertical edge on the model and choose the upward pointing direction, click **Apply**.



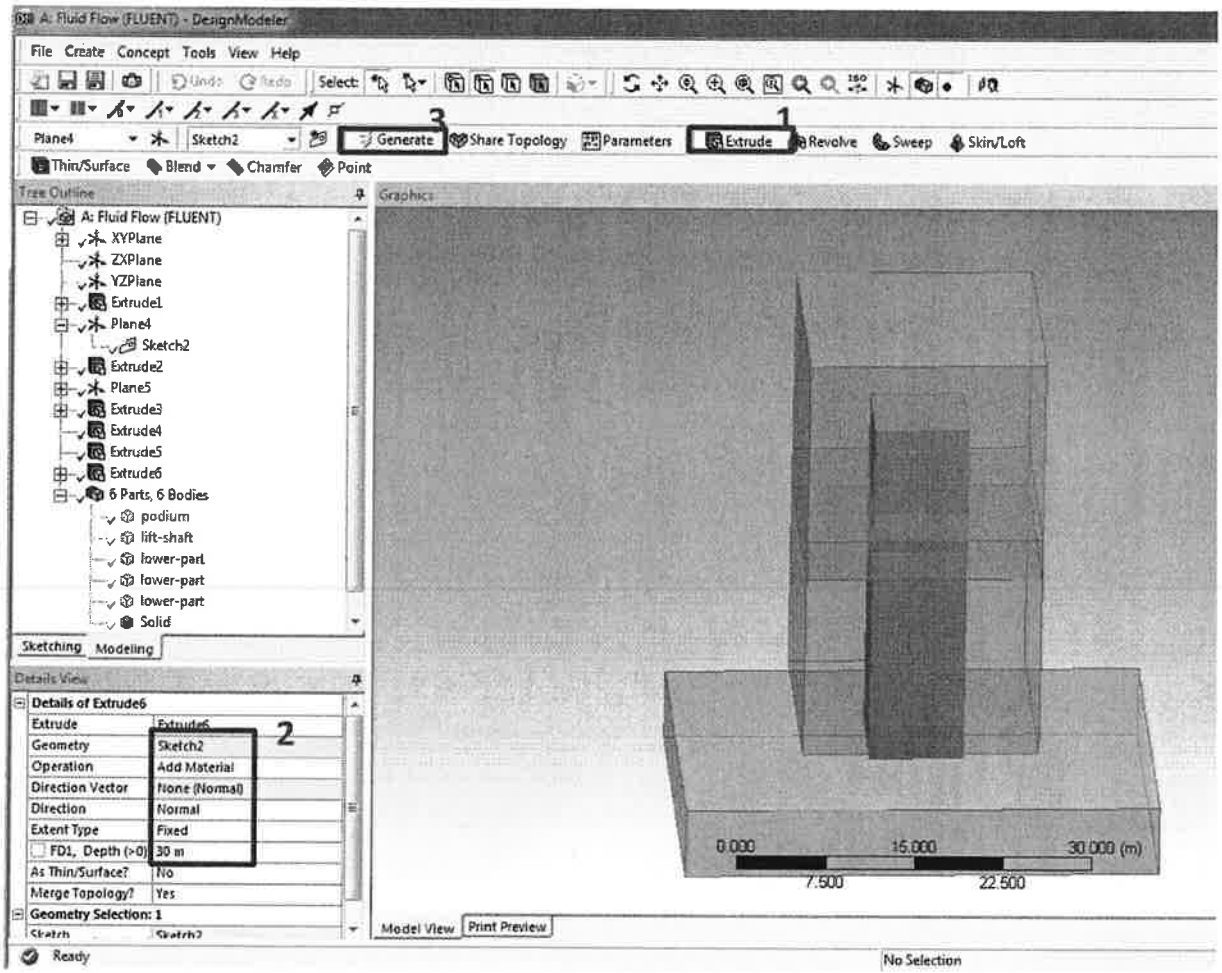
Generate a body with a height of 3.5m.



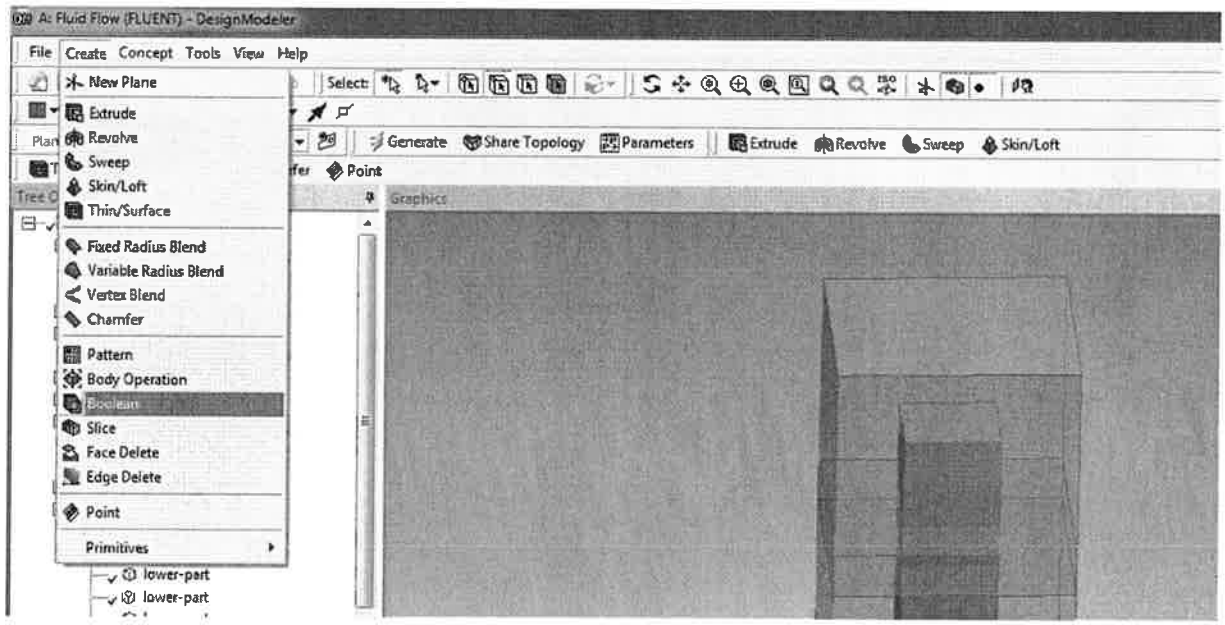
Follow the same steps to generate the body "**upper-part**" of 16m height.



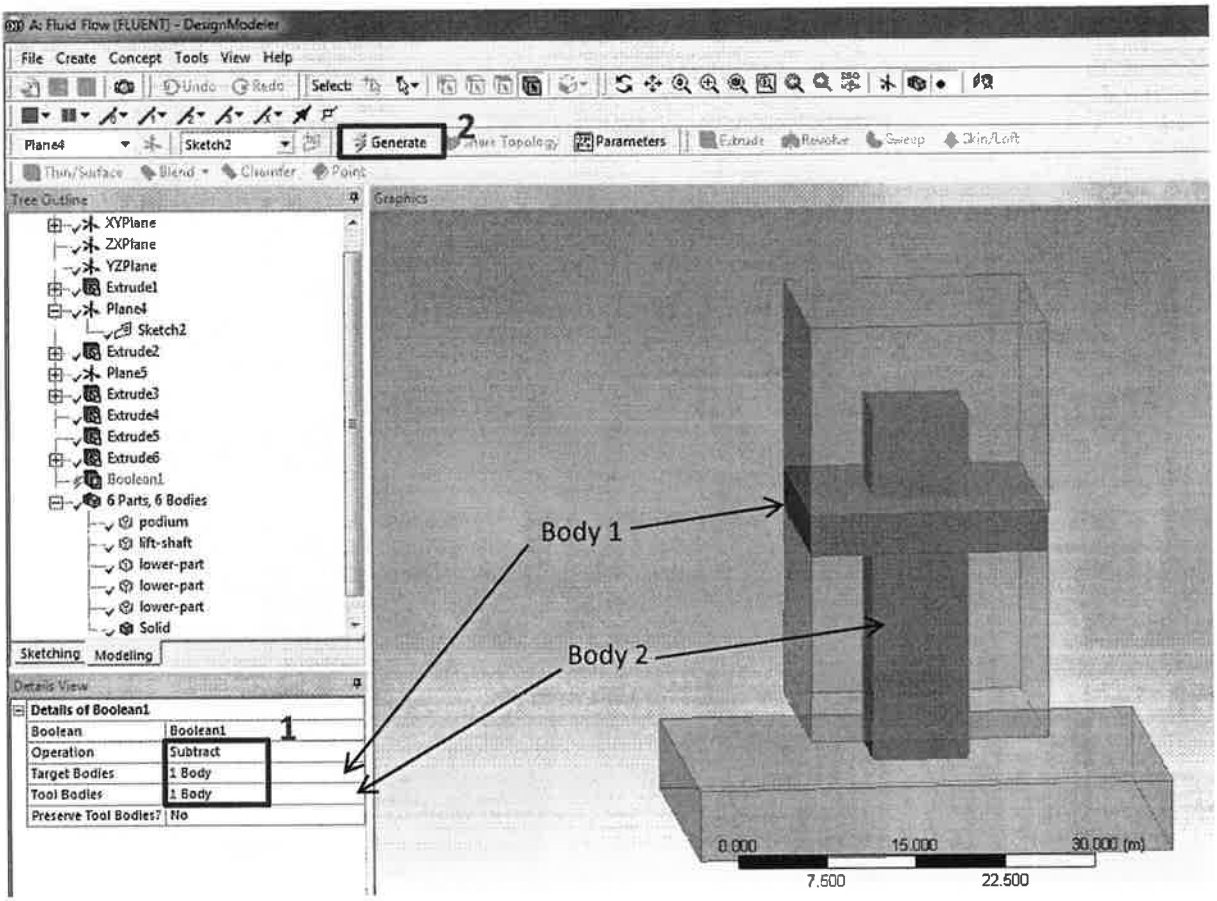
Generate a body from Sketch2.



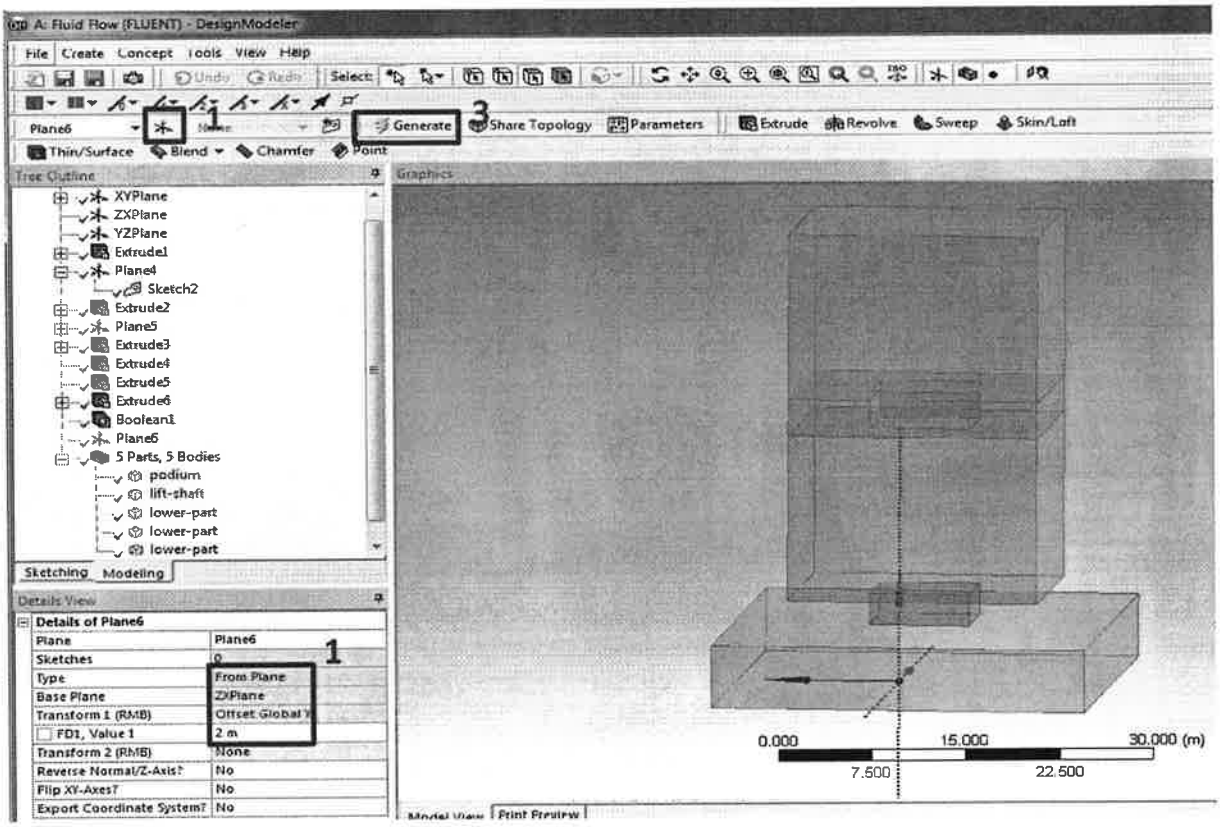
Click **Boolean** from the **Create** menu.



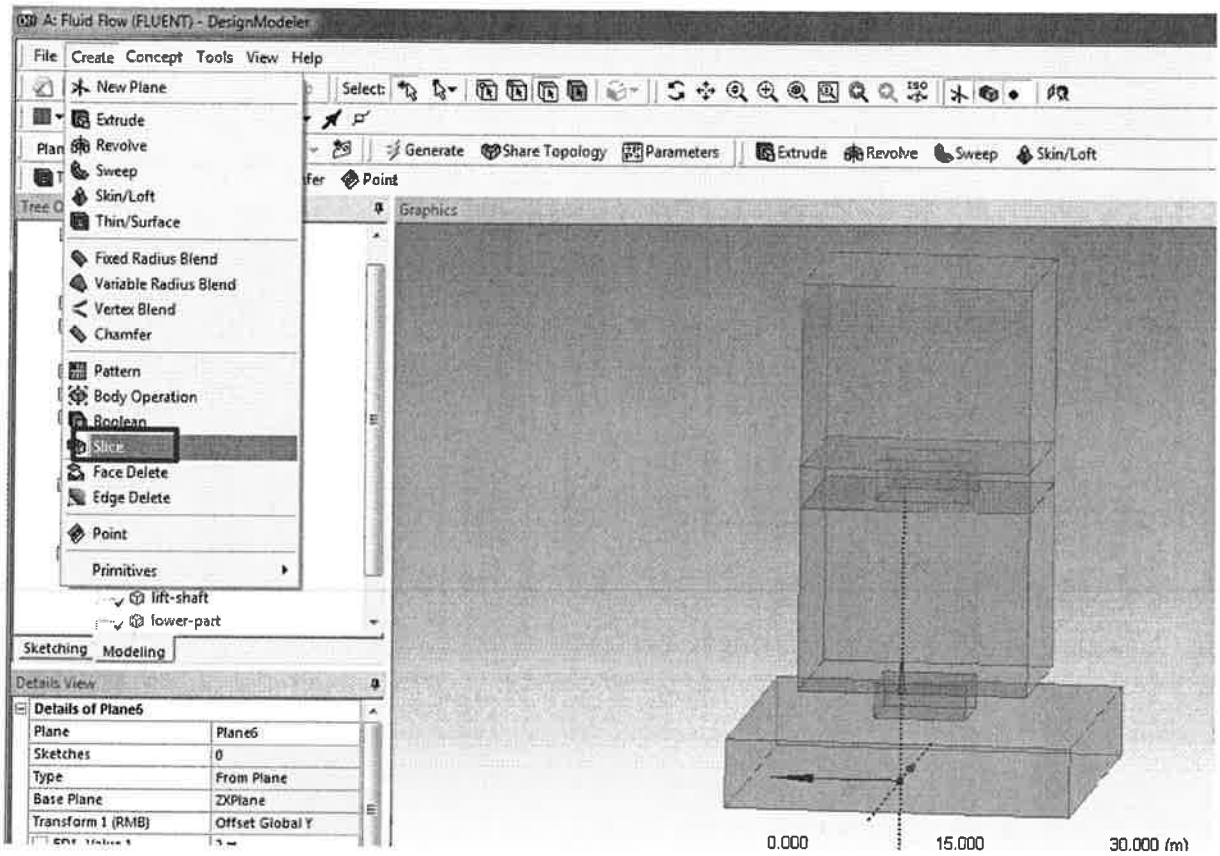
In the **Boolean** operation, subtract Body 1 by Body 2.

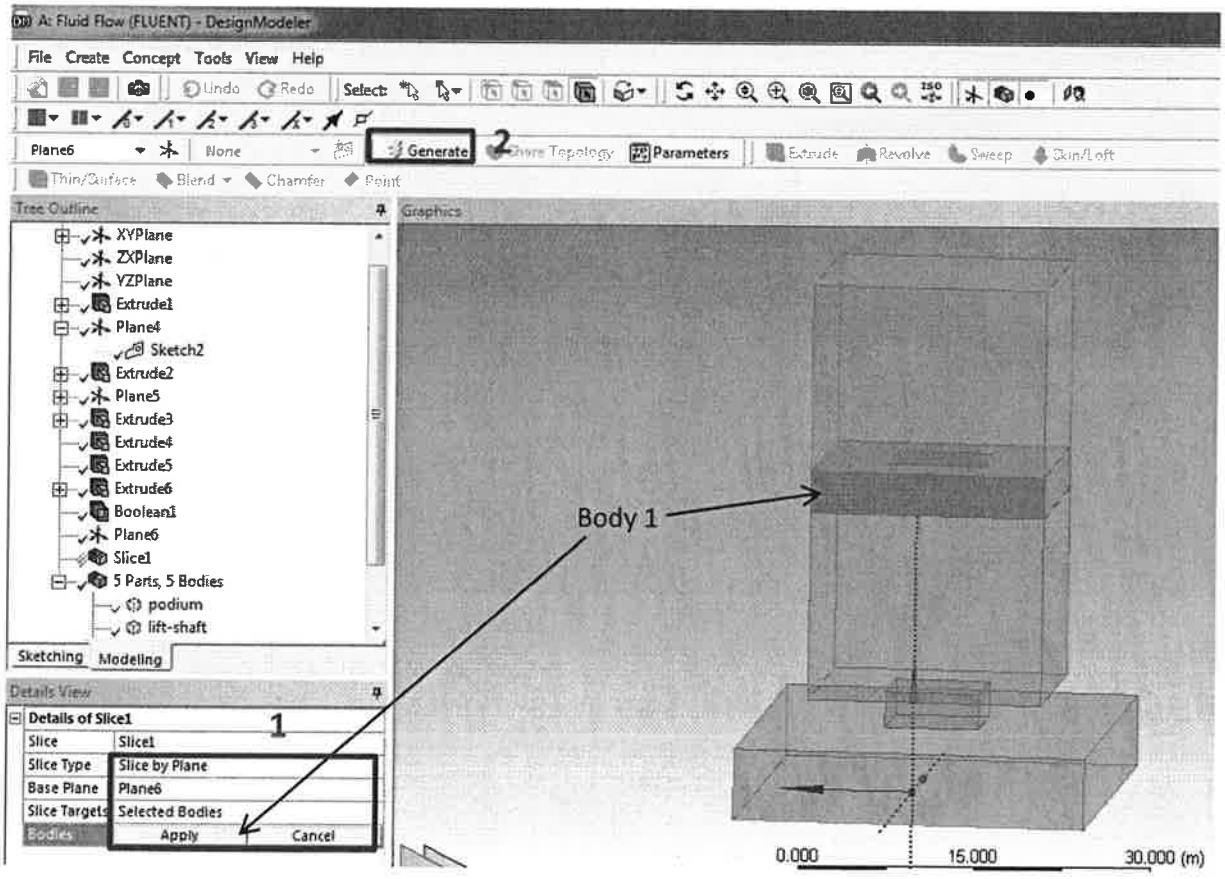


Create Plane6.

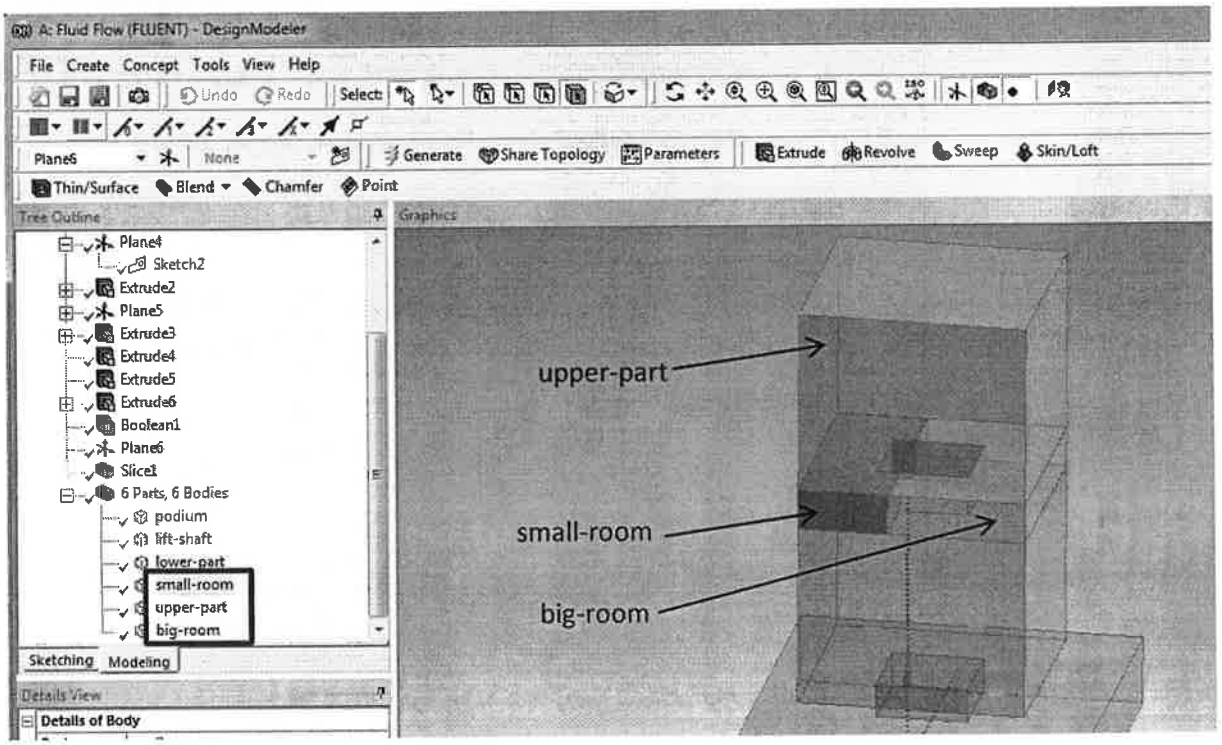


Use Plane6 to slice Body 1.



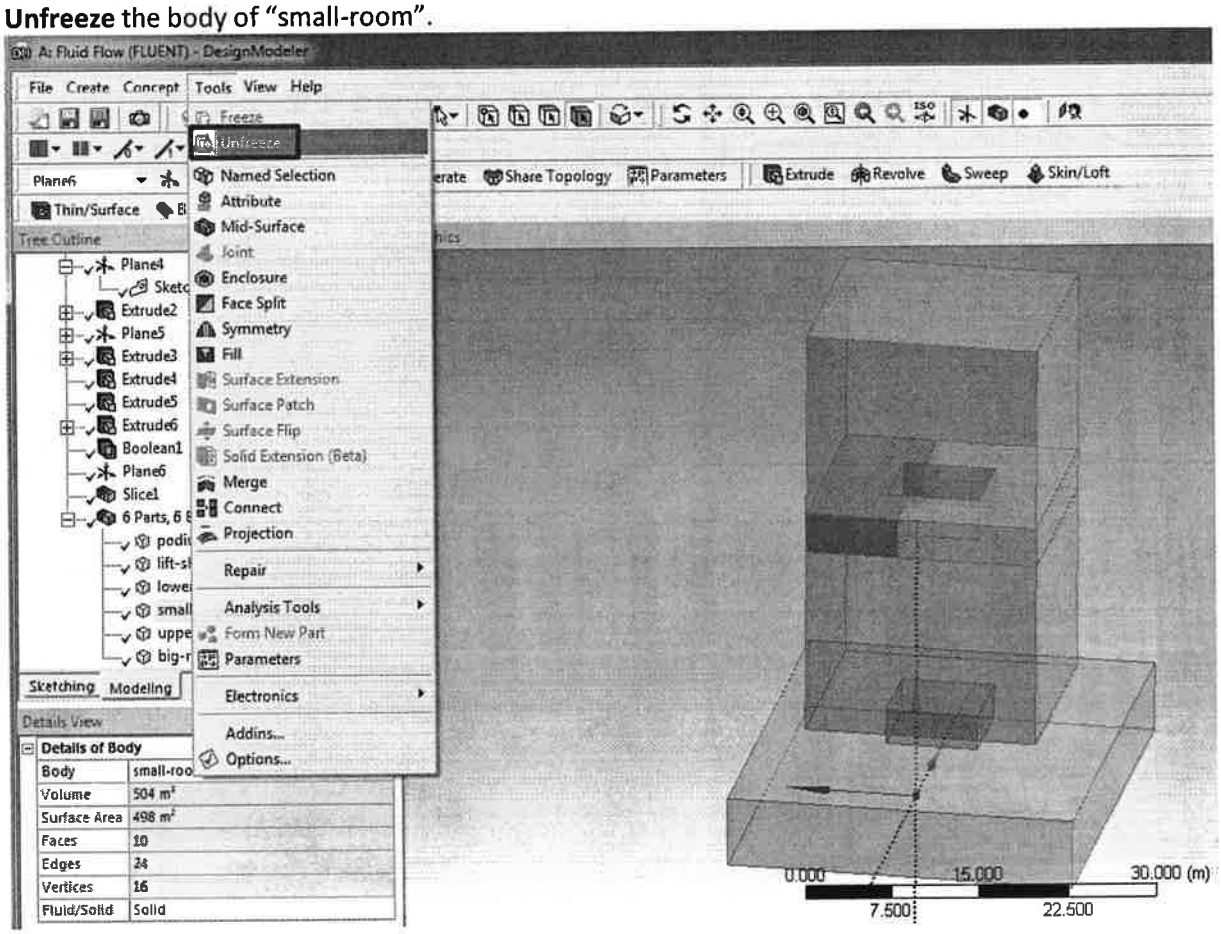


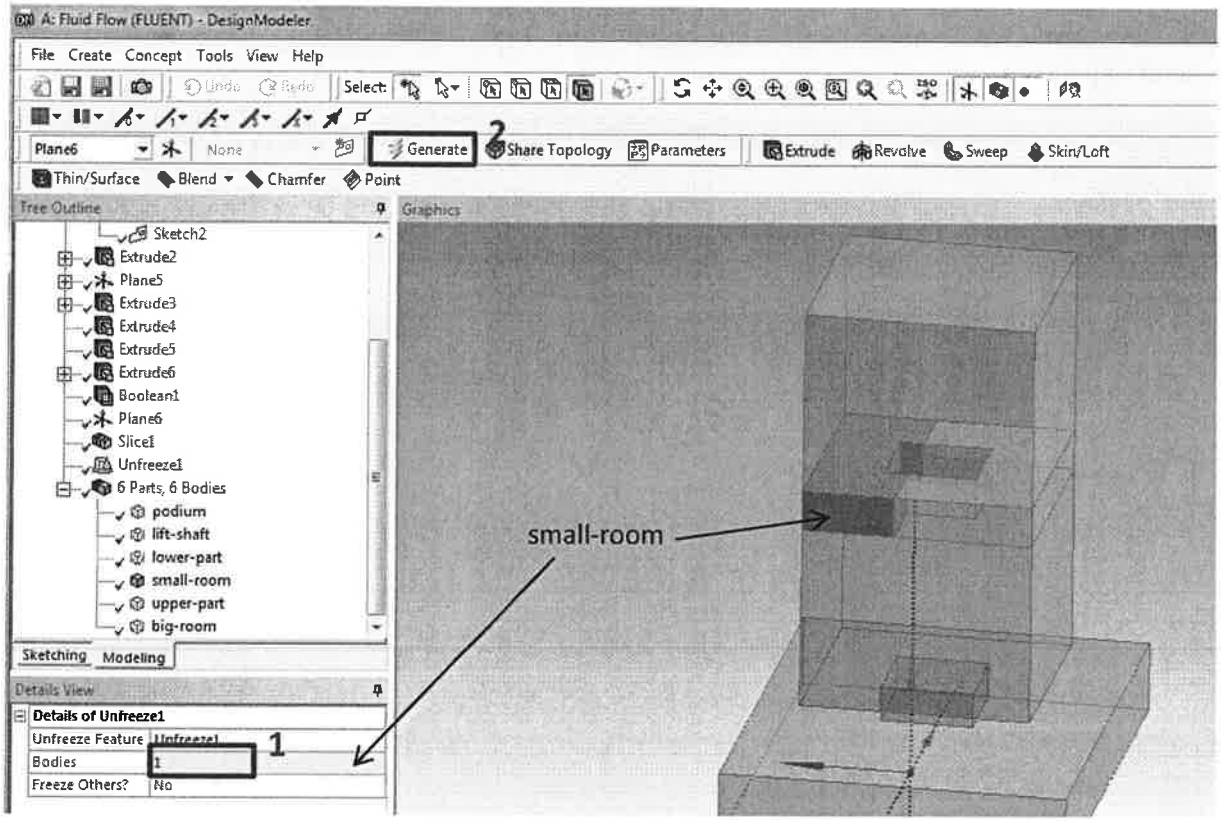
Rename the bodies.



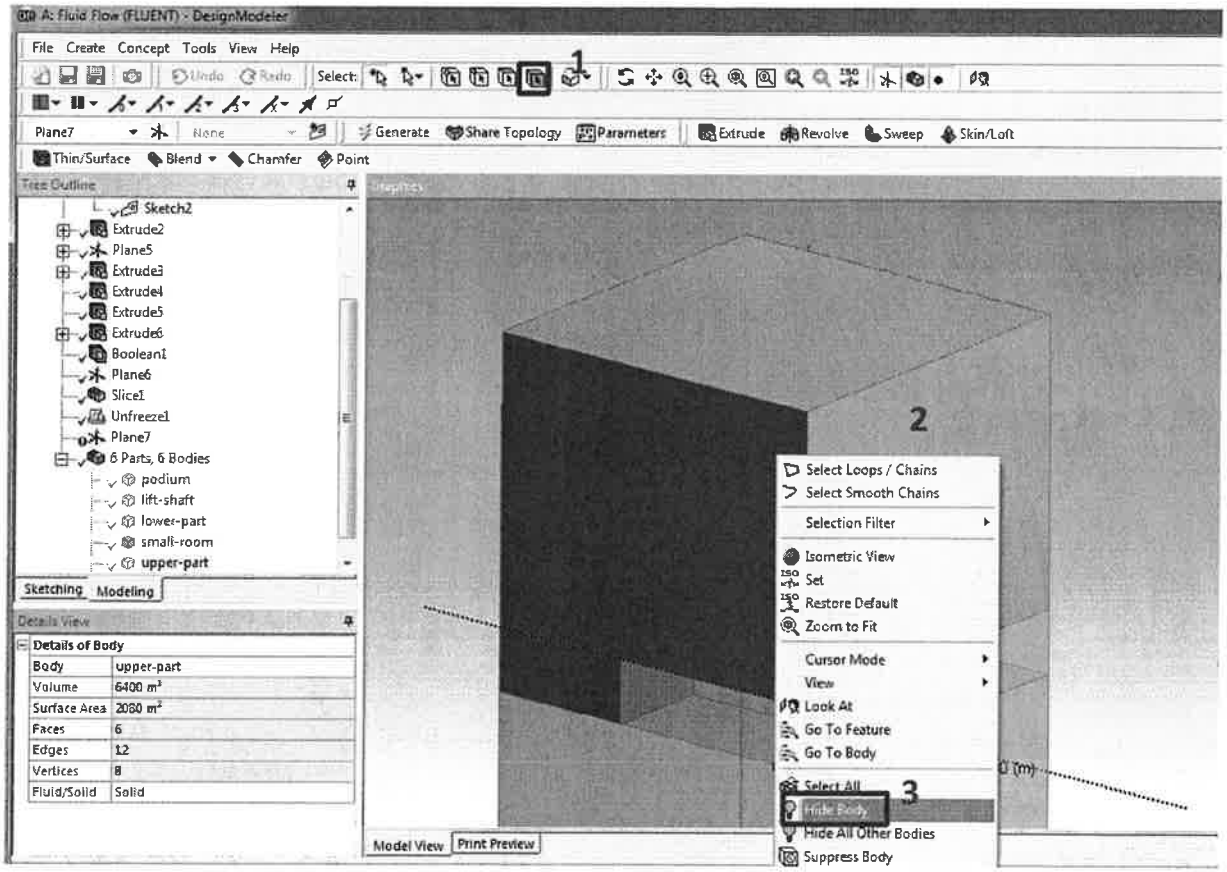
1.1.5 Create doors and windows on the rooms.

Unfreeze the body of "small-room".

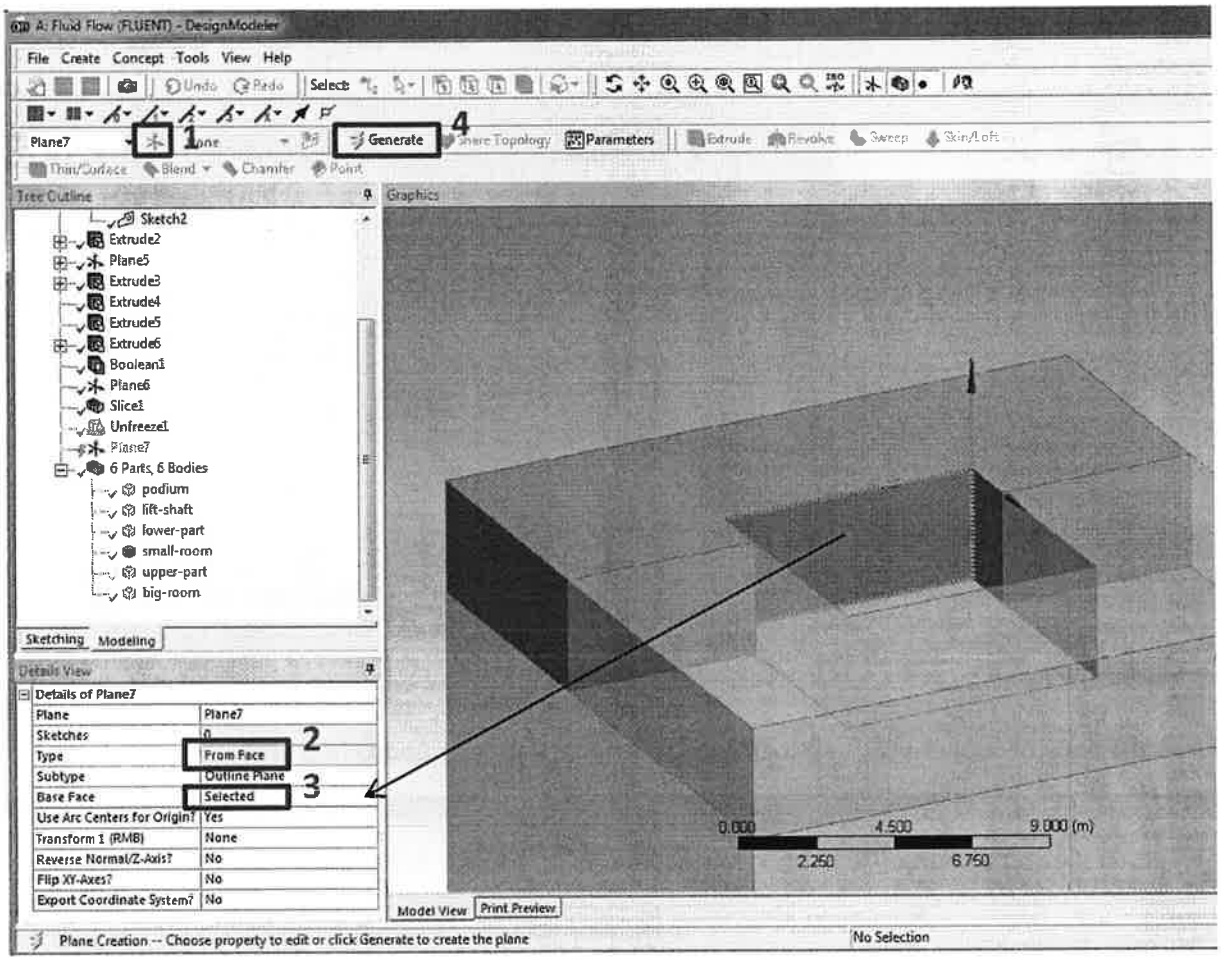




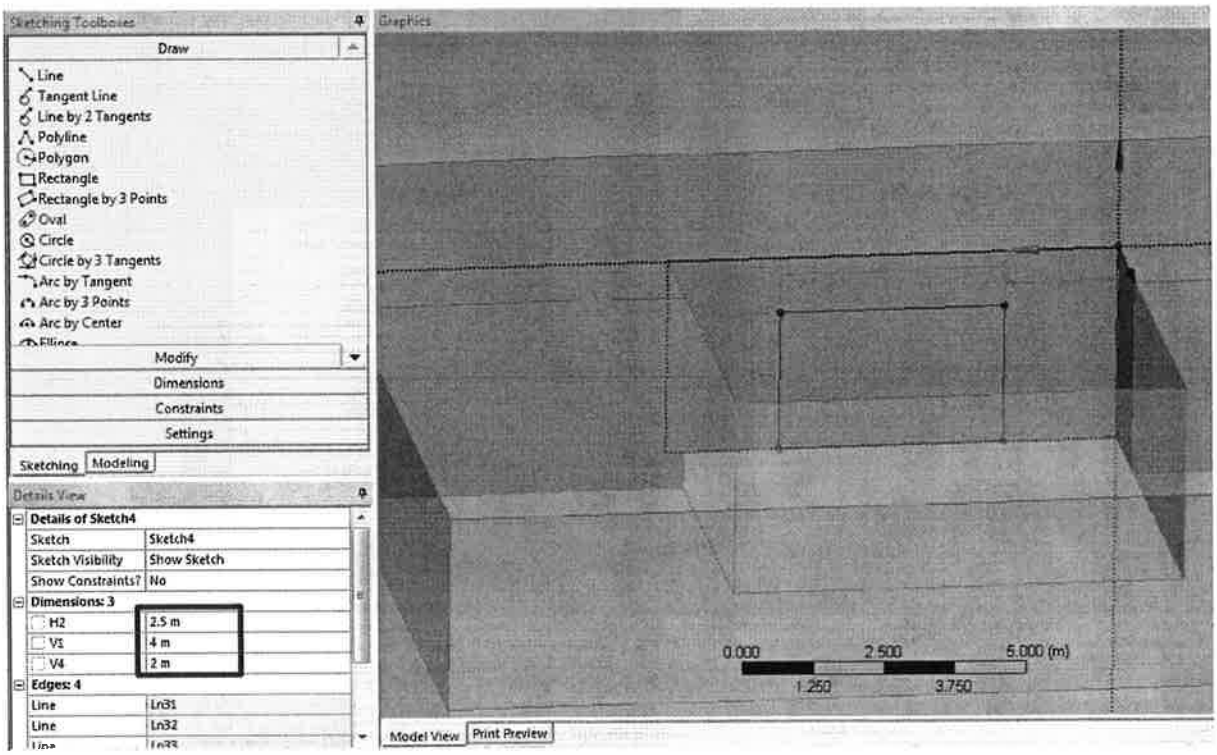
Select the body of "upper-part", right click on the model and **Hide Body**.



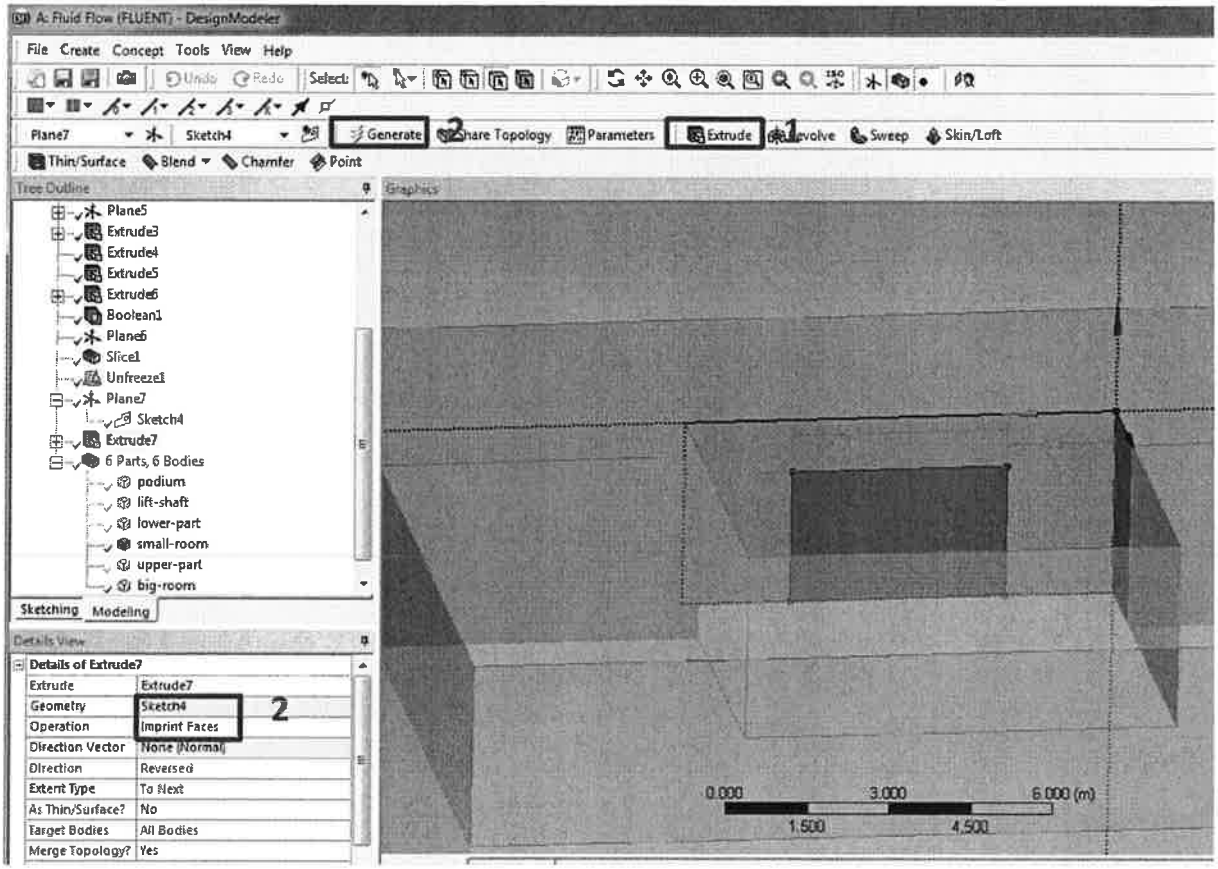
Click **New Plane** icon, choose **From Face** option from **Type**, select the face for **Base Face** and generate Plane7.



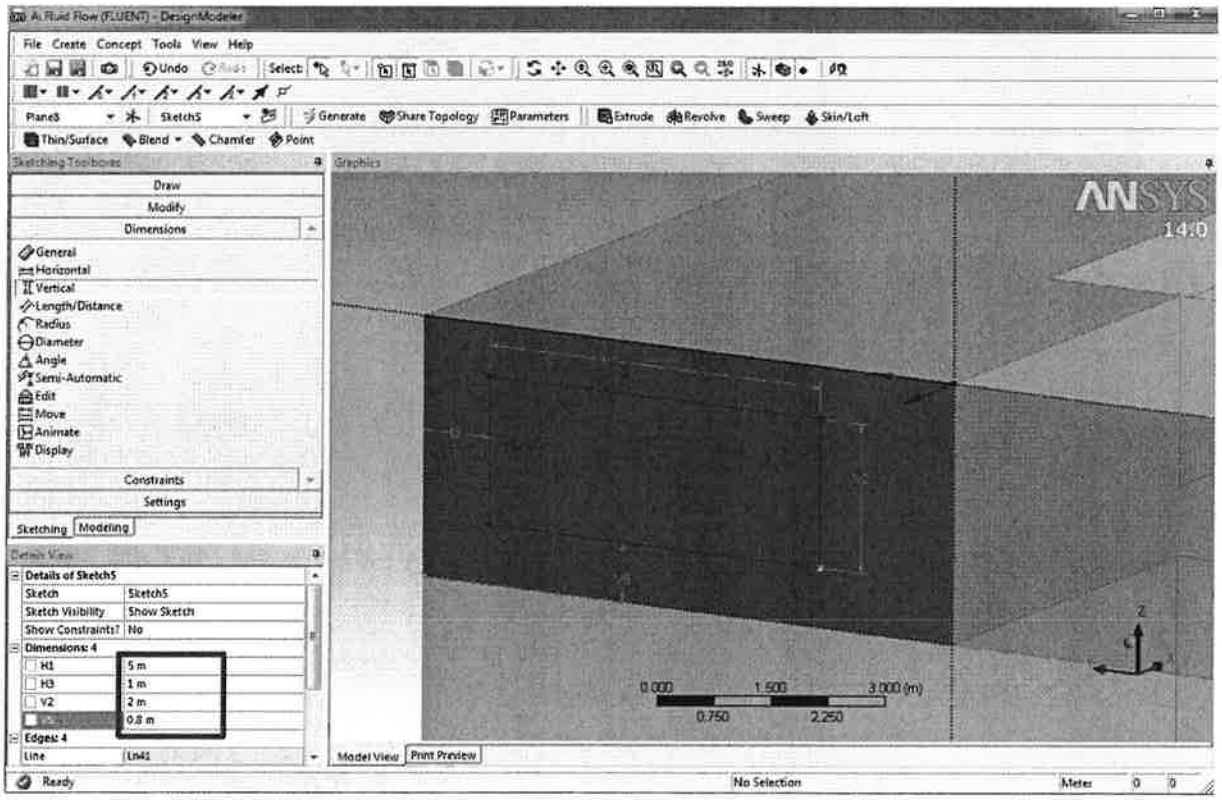
Draw Sketch4 on Plane7.



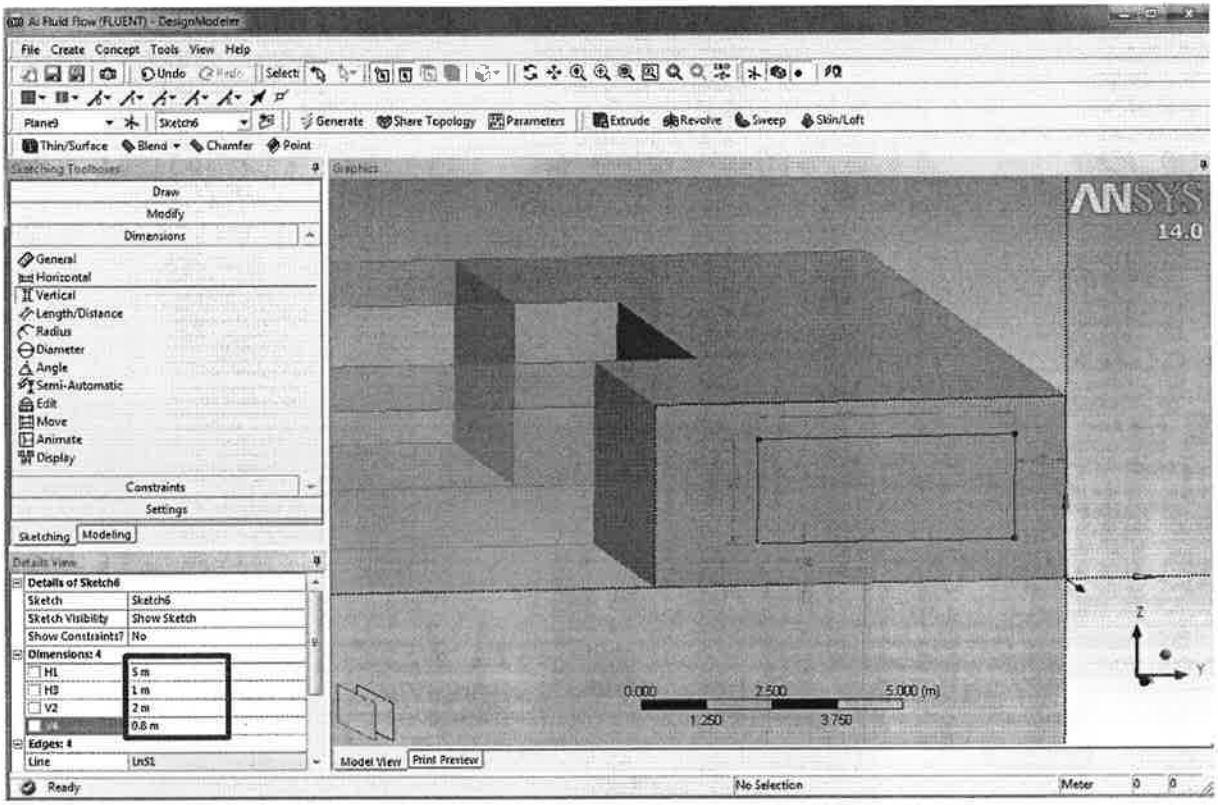
Click **Extrude**, select **Imprint Faces** option, click **generate** to create a door surface on the body of "small-room".



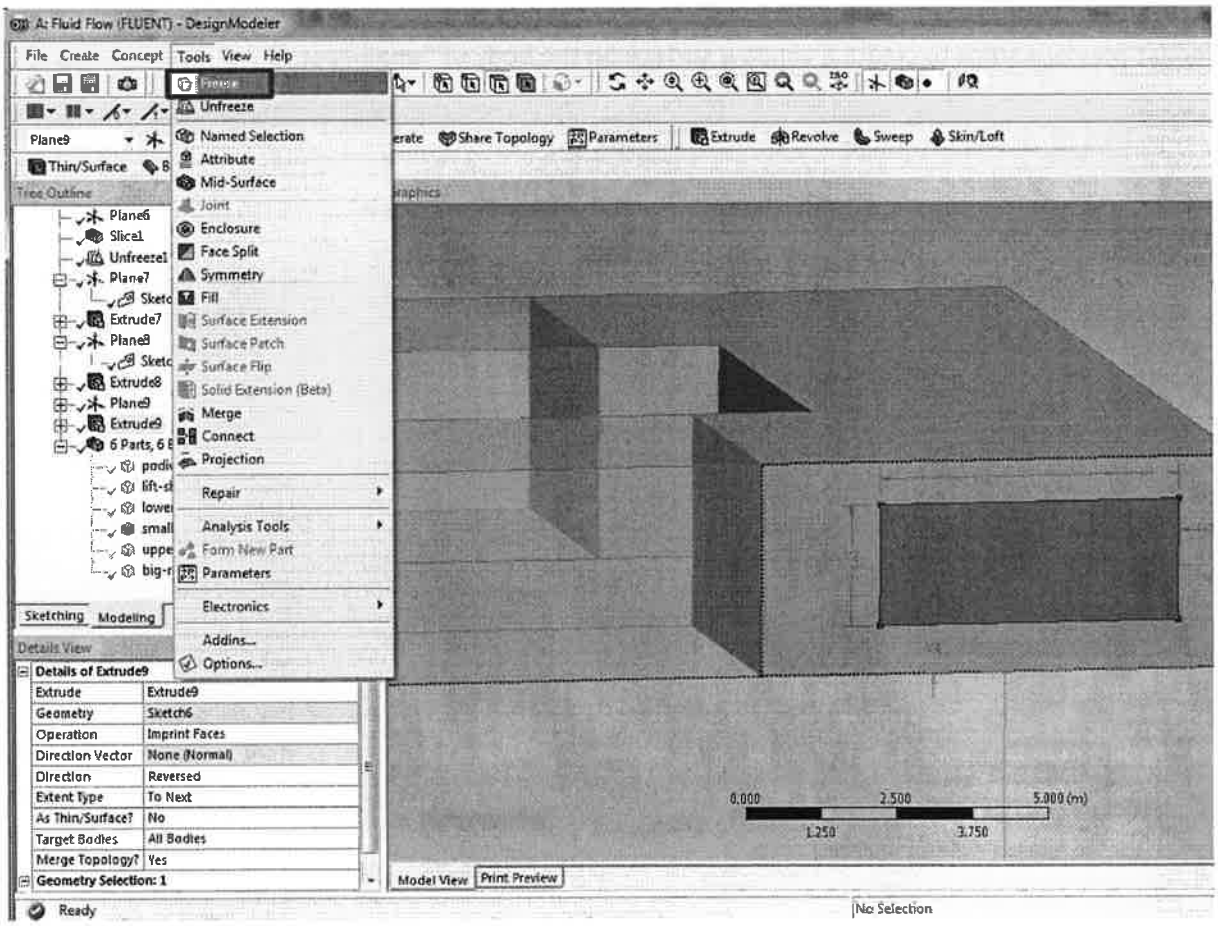
Repeat previous steps to create a window surface on the body of "small-room".



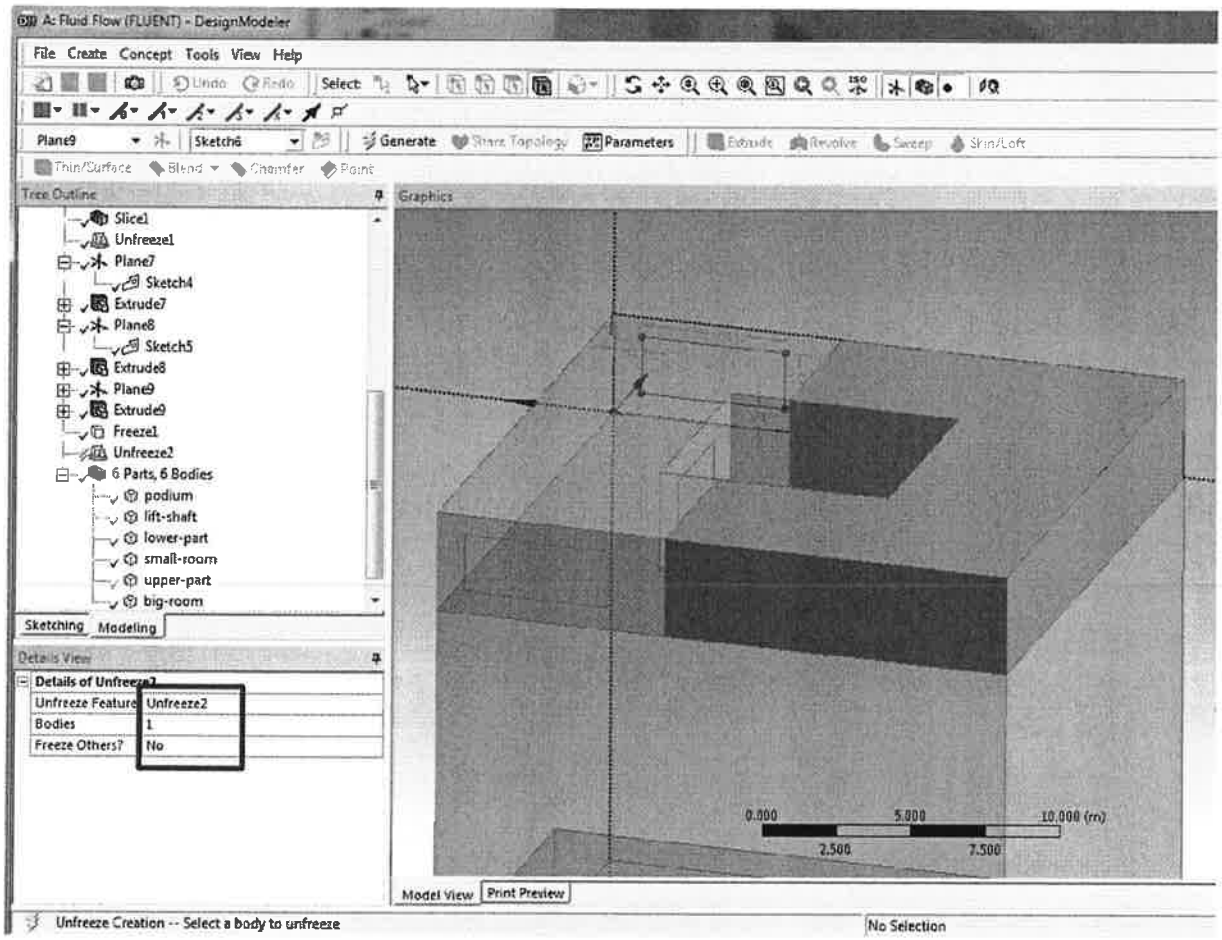
Repeat previous steps to create another window surface on the opposite side of the "small-room".



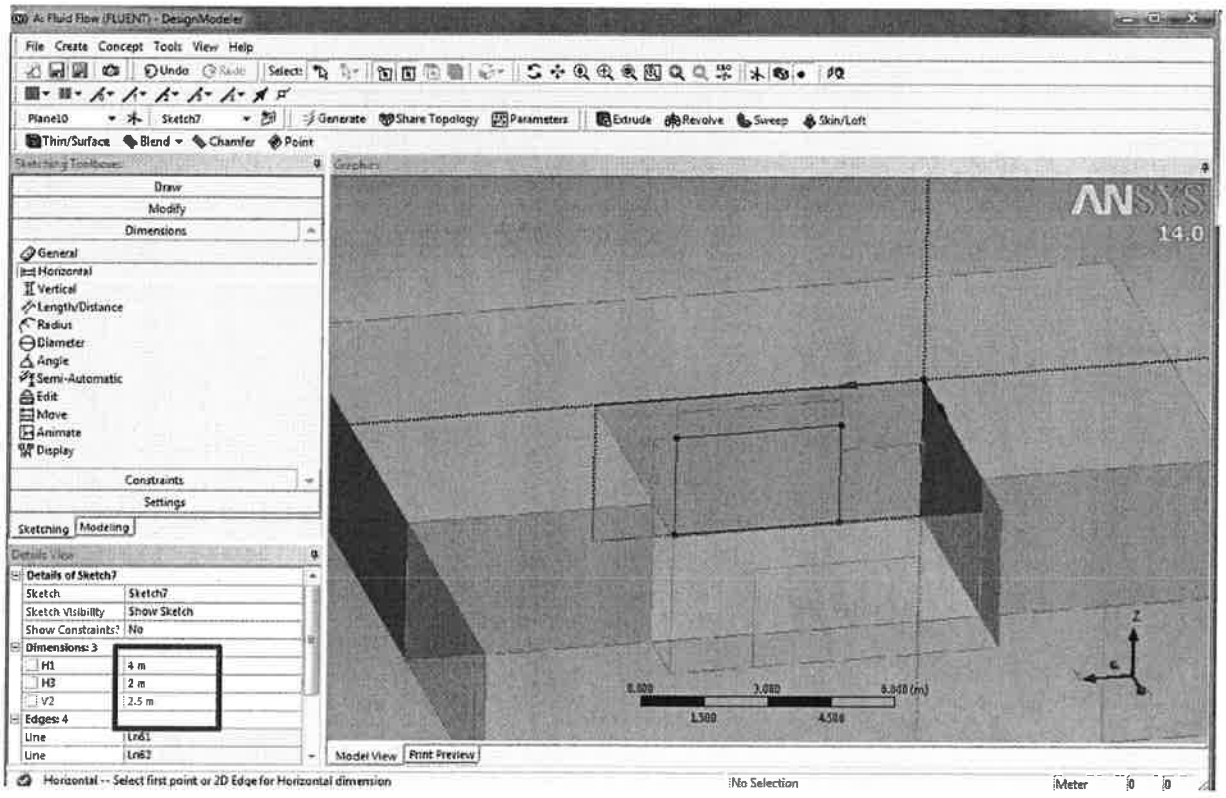
Click **Freeze** from **Tools** menu.



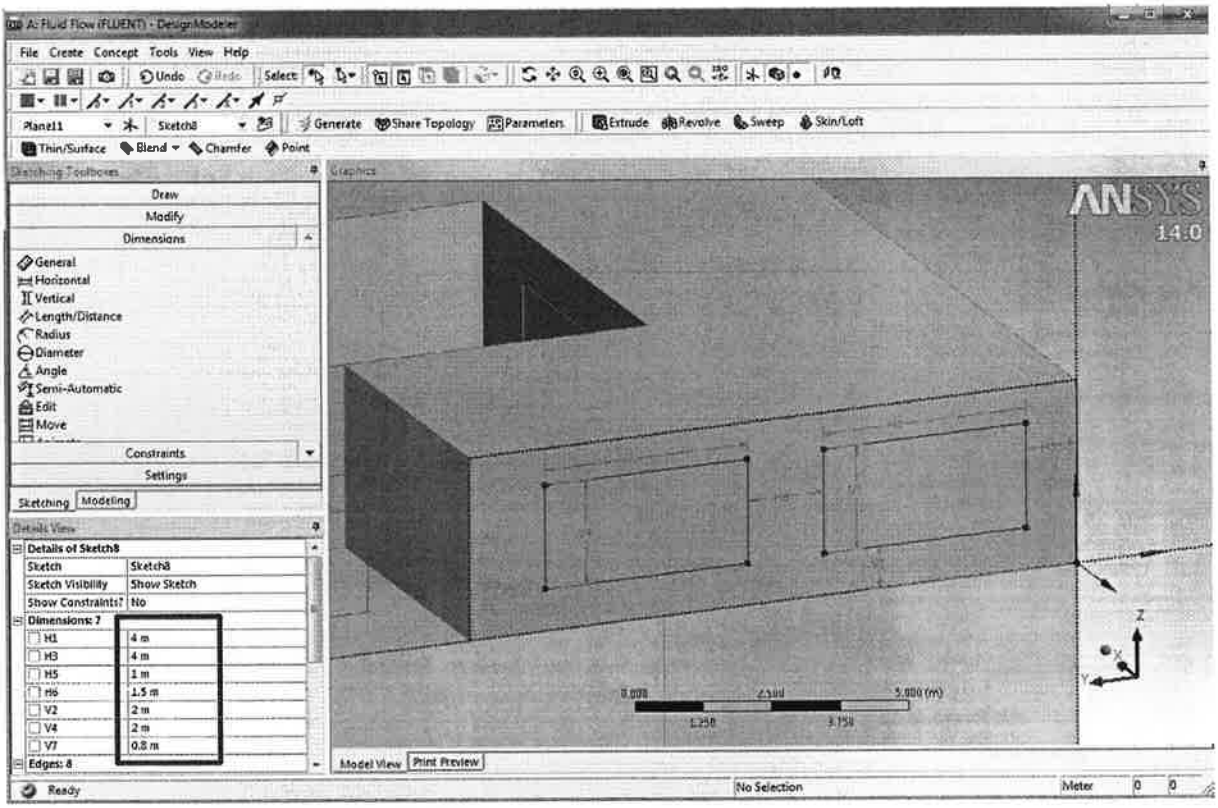
Similarly, **unfreeze** the body of "big-room".



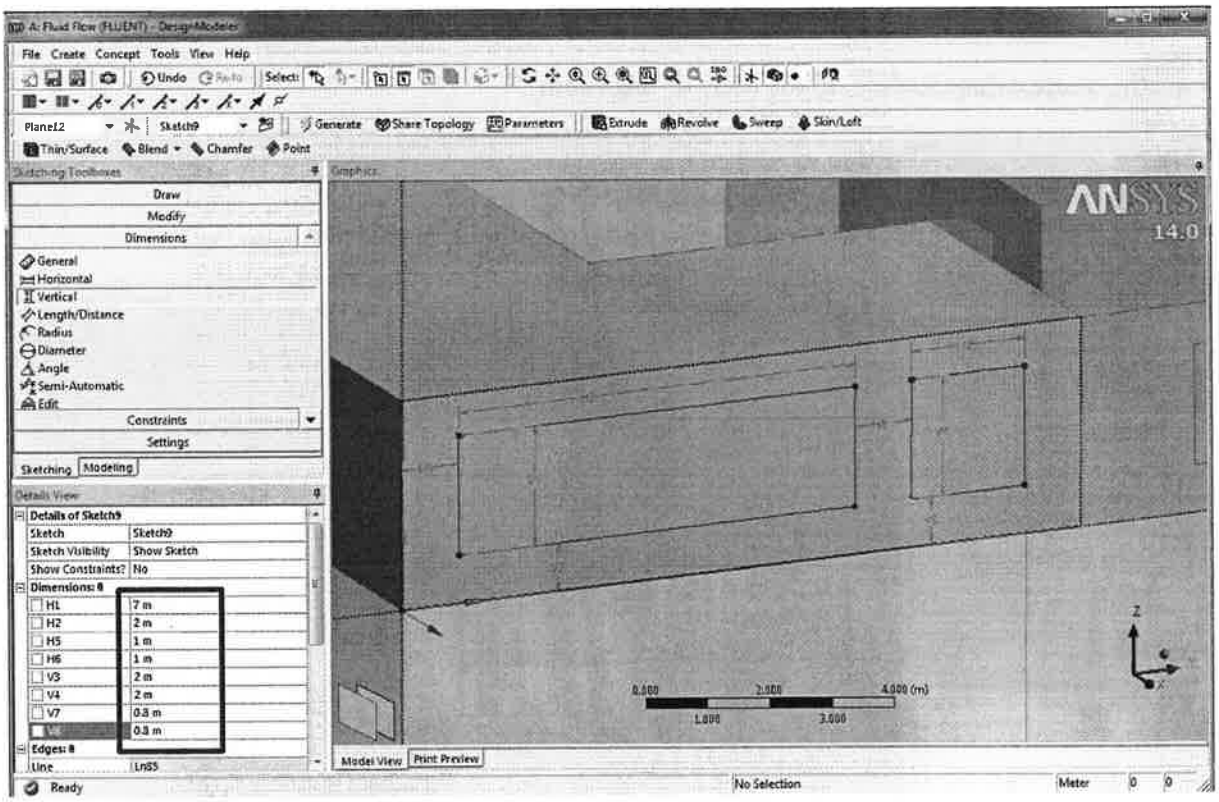
Similarly, create a door surface on the body of "big-room".



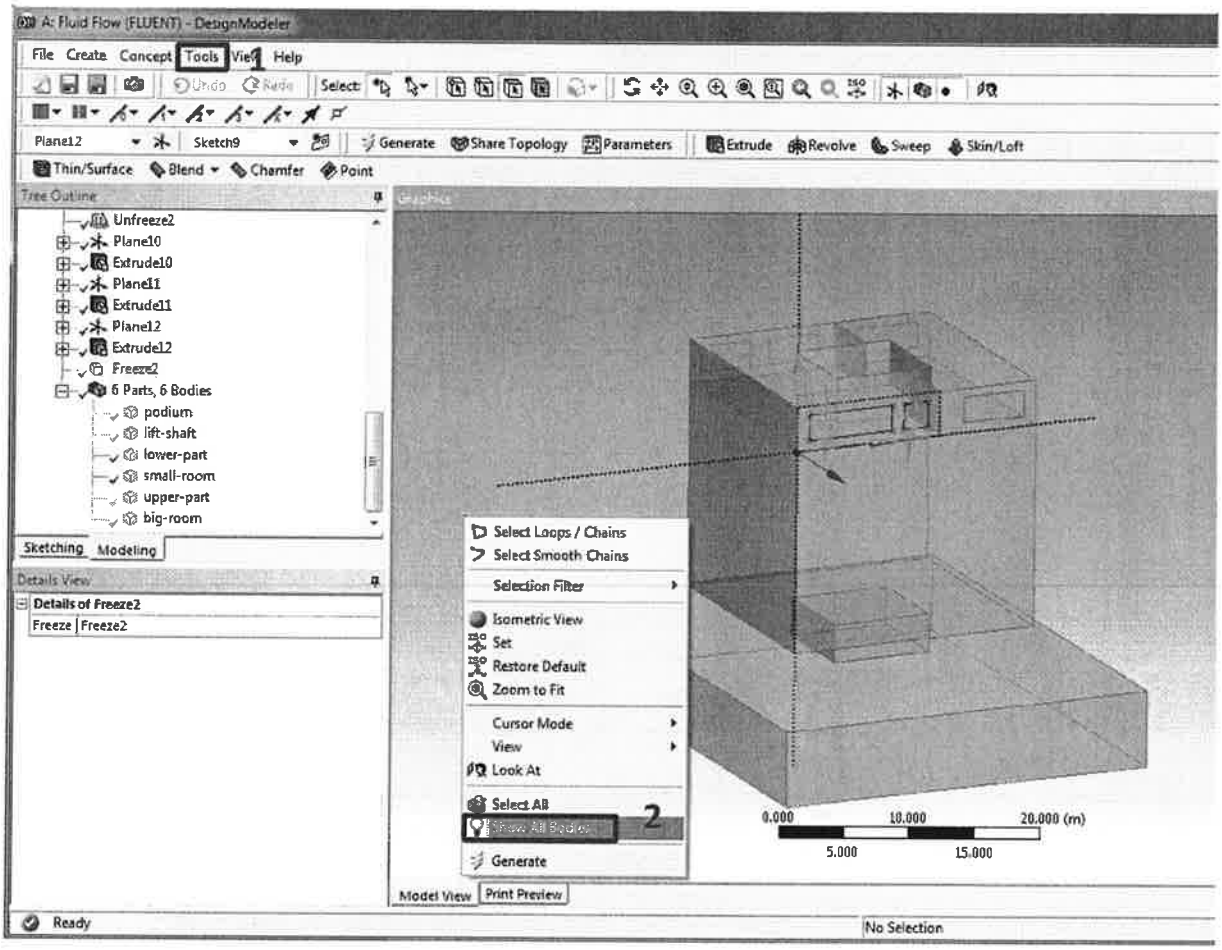
Create the window surfaces on the body of "big-room".



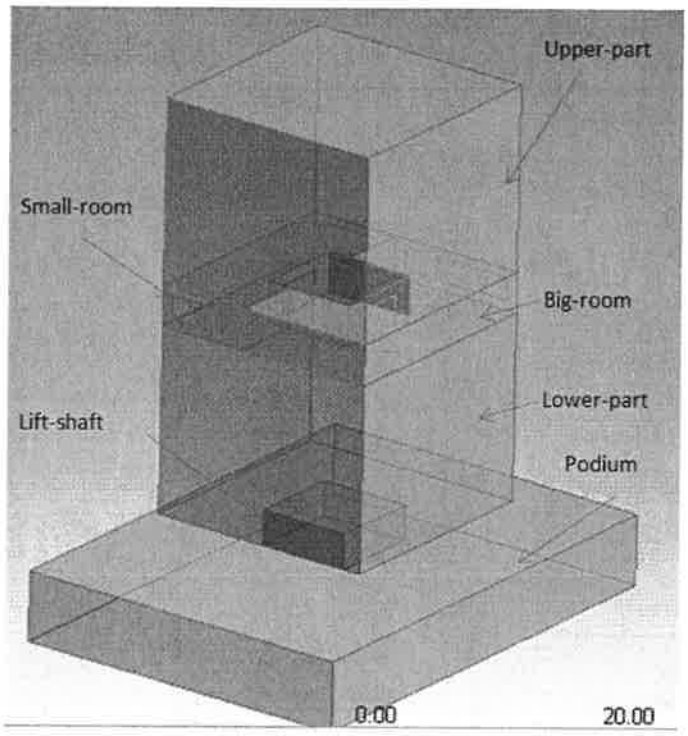
Create the window surfaces on the opposite side of the "big-room".



Click **Freeze** from **Tools** menu. Right click on the model and left click on **Show All Bodies**.

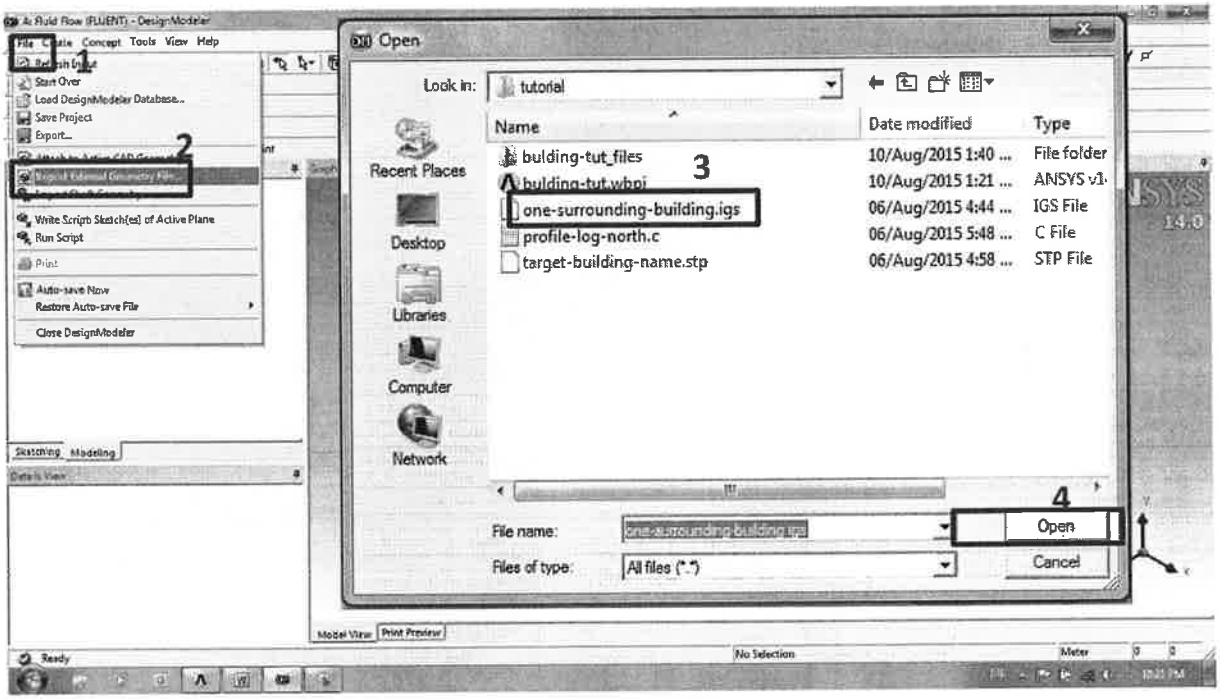


Note that the development consists of six bodies:

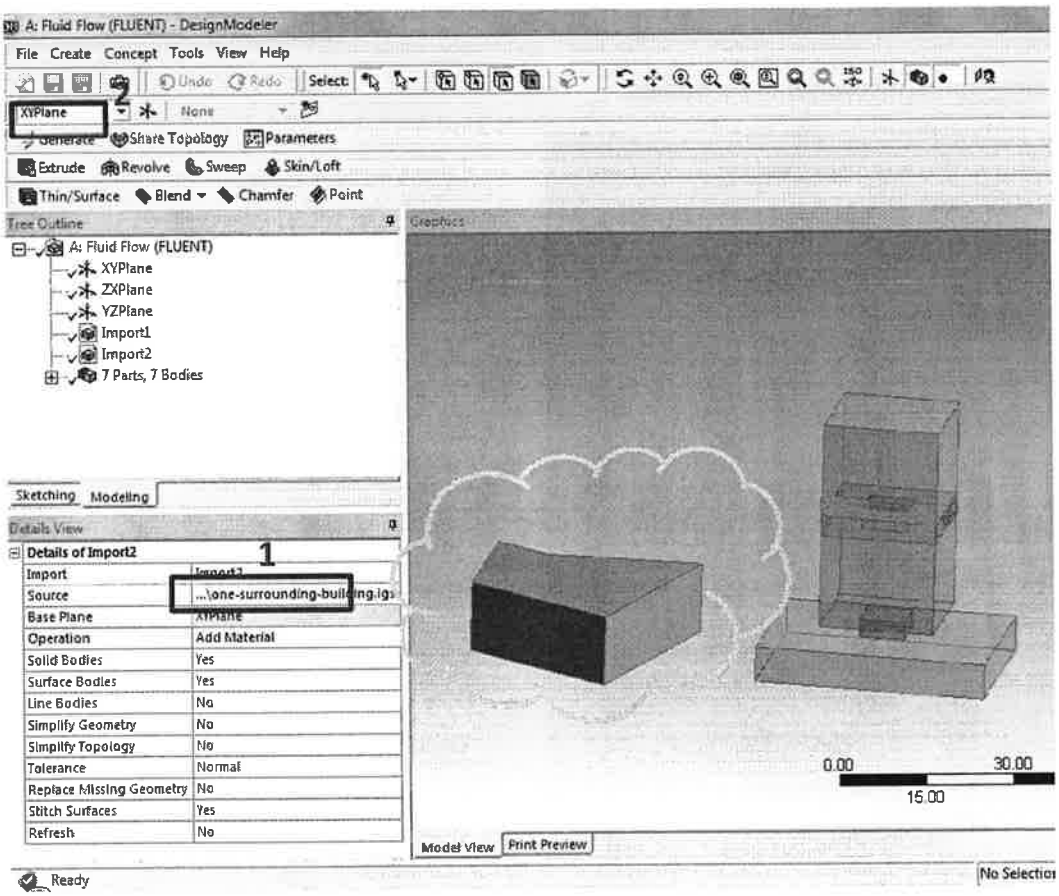


1.2 Creation of the surrounding building

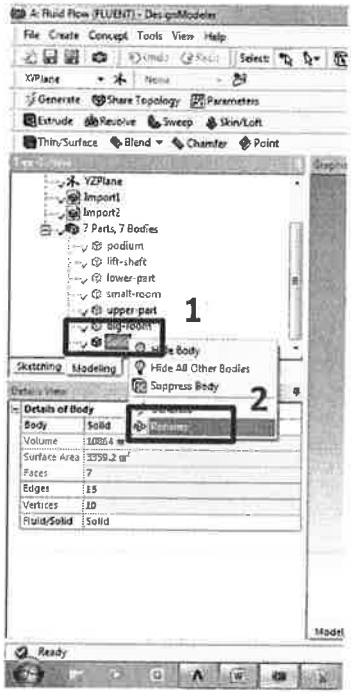
Similarly, we can create the surrounding building by following the procedure below: Import the surrounding building (one-surrounding-building.igs).

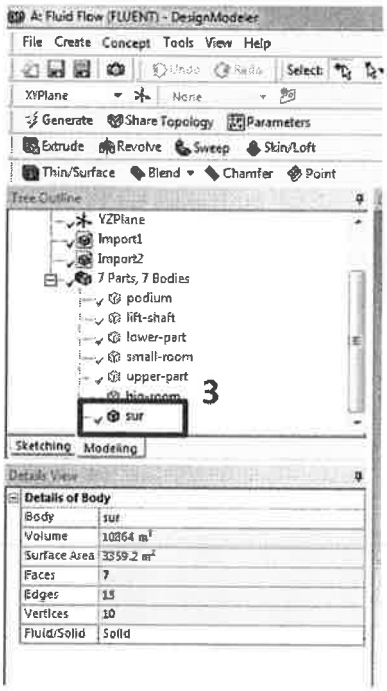


Generate the surrounding building in Design Modeler.



Note that the surrounding is assumed to be located ahead of the development right on the northern side. Then rename the surrounding building to be "sur".



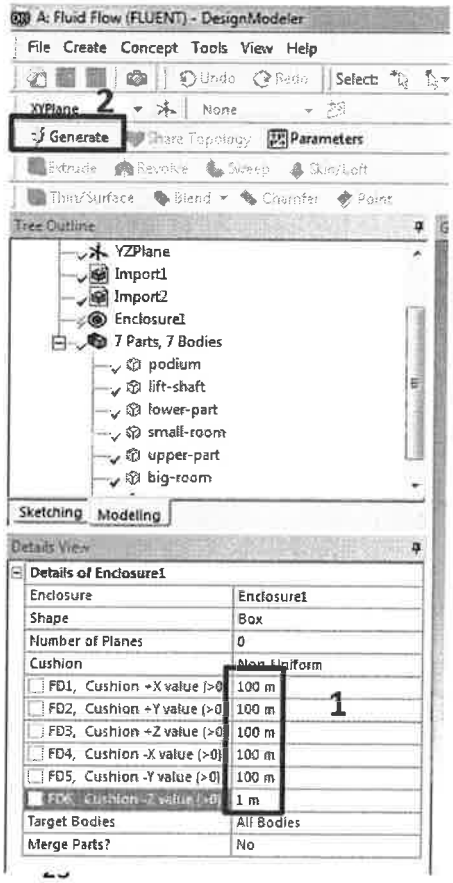


1.3 Creation of the enclosure as the computational domain

In this tutorial, the prevailing wind from the north intends to be simulated. A simple and relatively small computational domain composed of the development and its surrounding building will be created. It should be noted that the computational domain is not large enough as anticipated in GM code, due to the limited computational resources available in BCAA. Please refer to the BCA GM code about CFD methodology for details about the domain creation.

1.3.1 Create enclosure

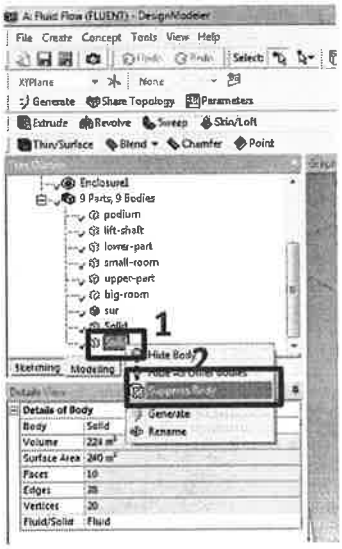
Choose *Tools > Enclosure*



Then two bodies will be created: one is the enclosure encompassing the development and the surrounding building, and the other is the solid space between the two rooms. The latter is assumed to be occupied by lift shaft and thus is not needed in simulation.

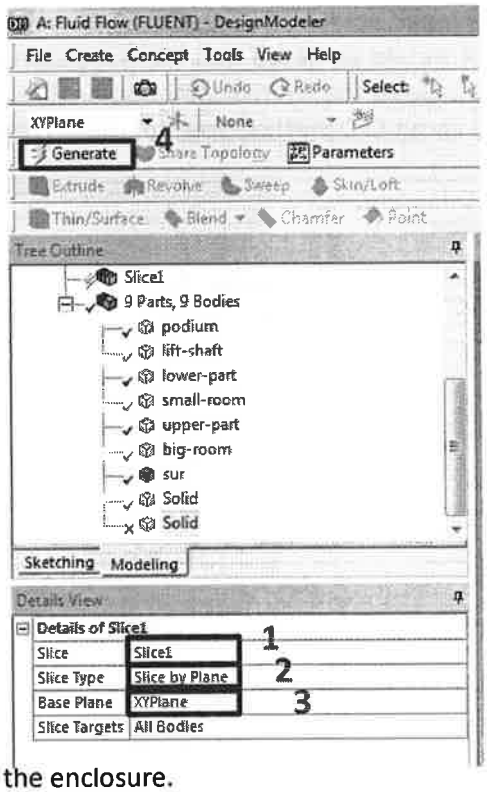
1.3.2 Suppress the extra body located between the rooms

Select the solid between the space of two rooms, and then right click the mouse button and choose ***suppress Body***.

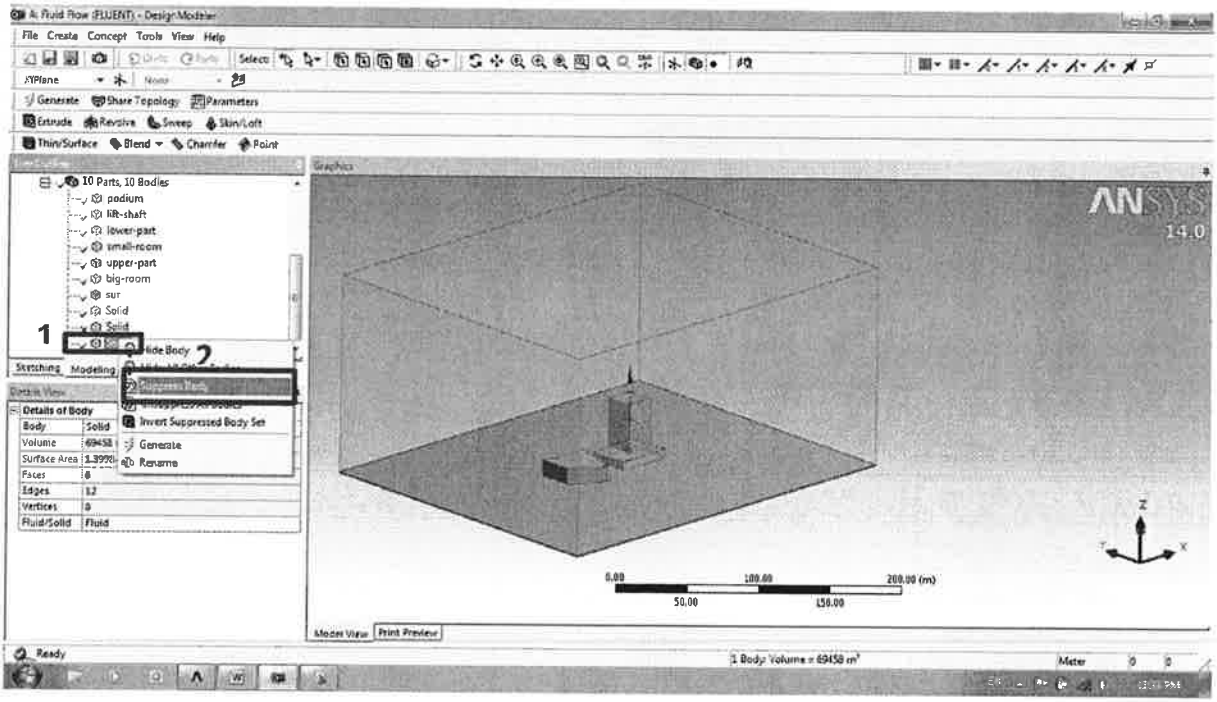


1.3.3 Trim the extra part (Z < 0) of the enclosure

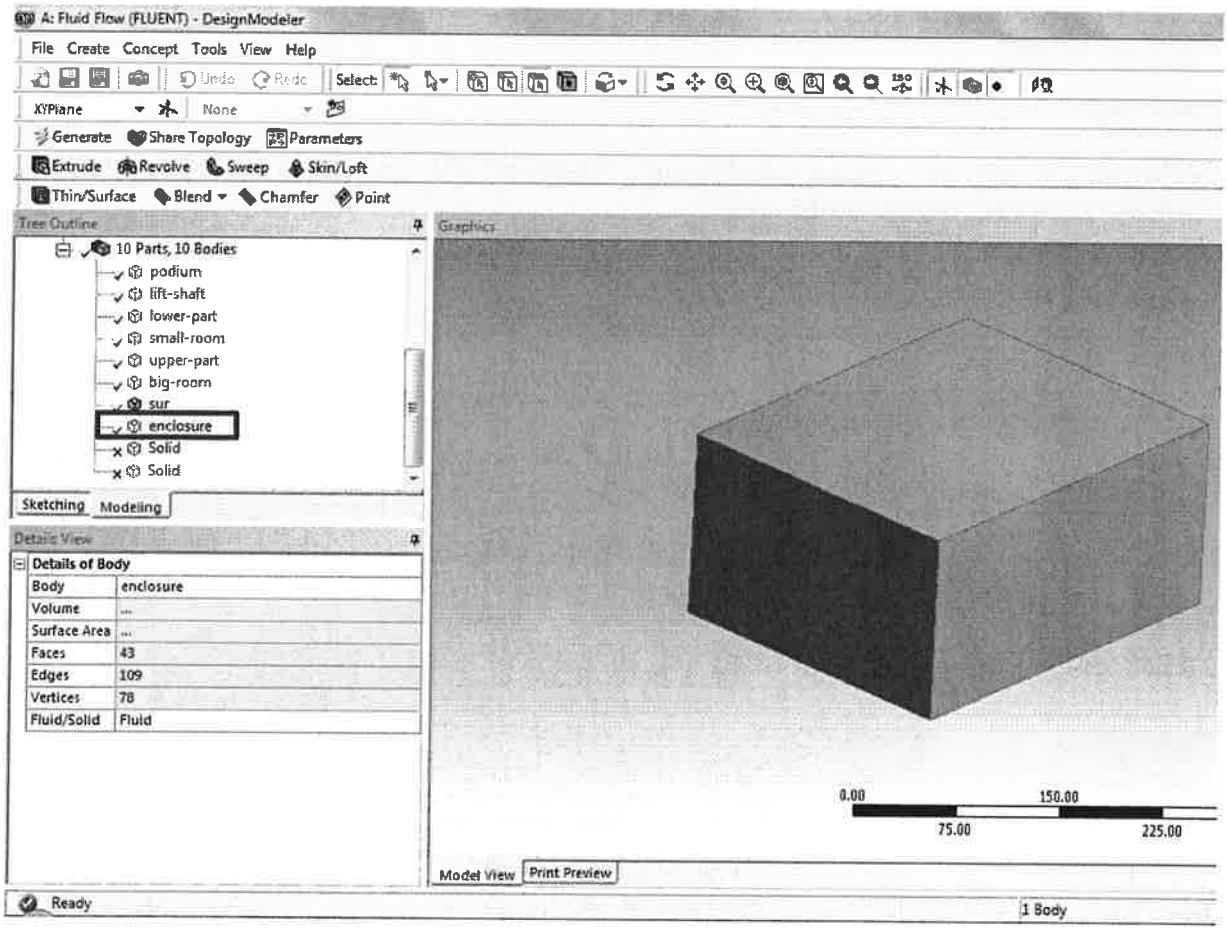
Choose ***Create > Slice***



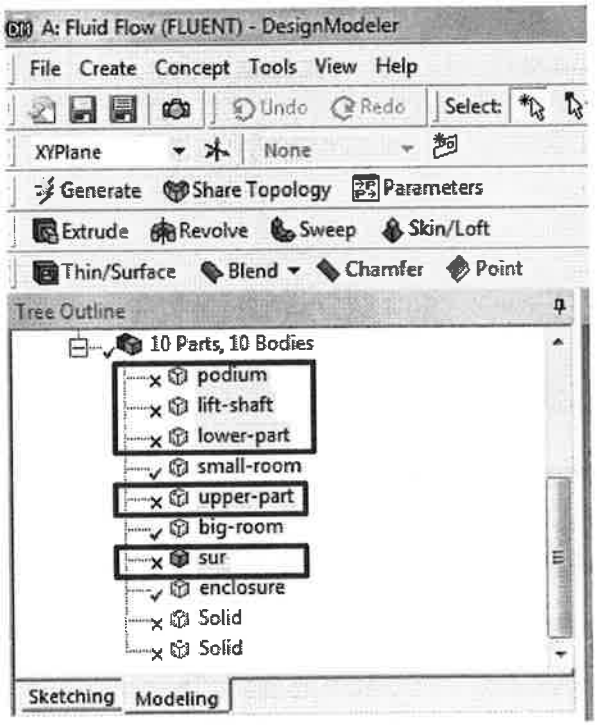
Then ***suppress*** the lower part of the enclosure.



Rename the remaining of the enclosure ("solid") to be "enclosure".

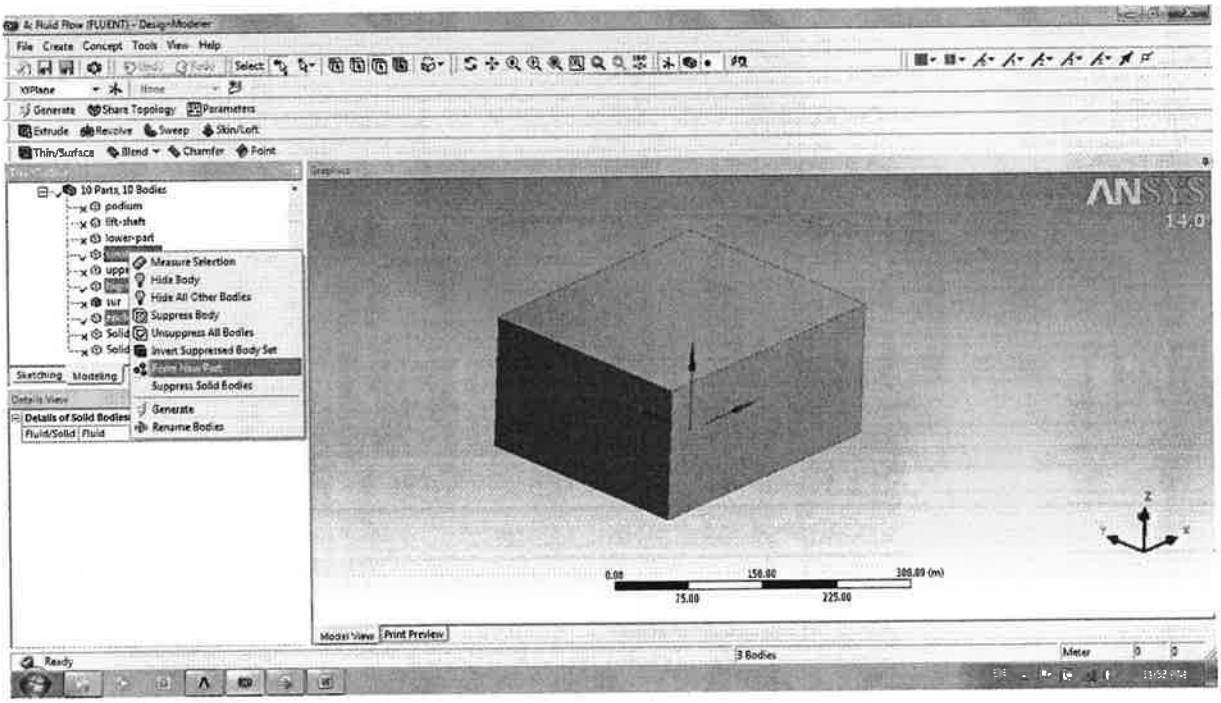


Suppress all bodies, except for "small-room", "big-room" and "enclosure". Holding CTRL key, left click respective bodies and then right click the mouse to choose ***Suppress Body*** option.

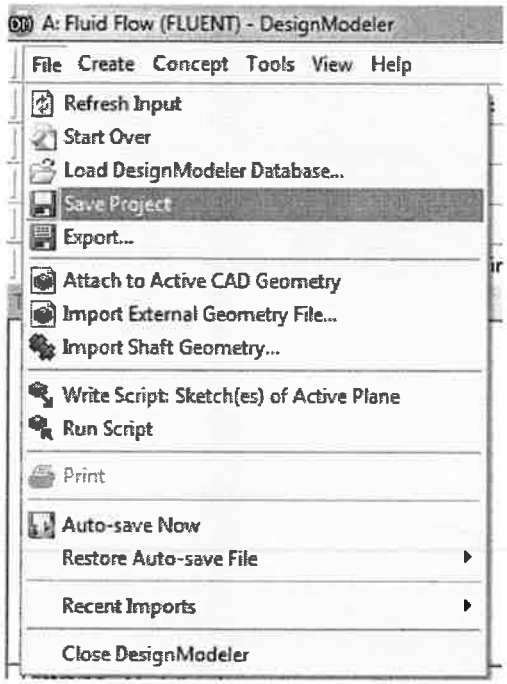


Form a single part with multiple bodies:

Holding the CTRL key and left click "small-room", "big-room" and "enclosure", and then right click the mouse to choose ***Form New Part*** option.



Save the project:

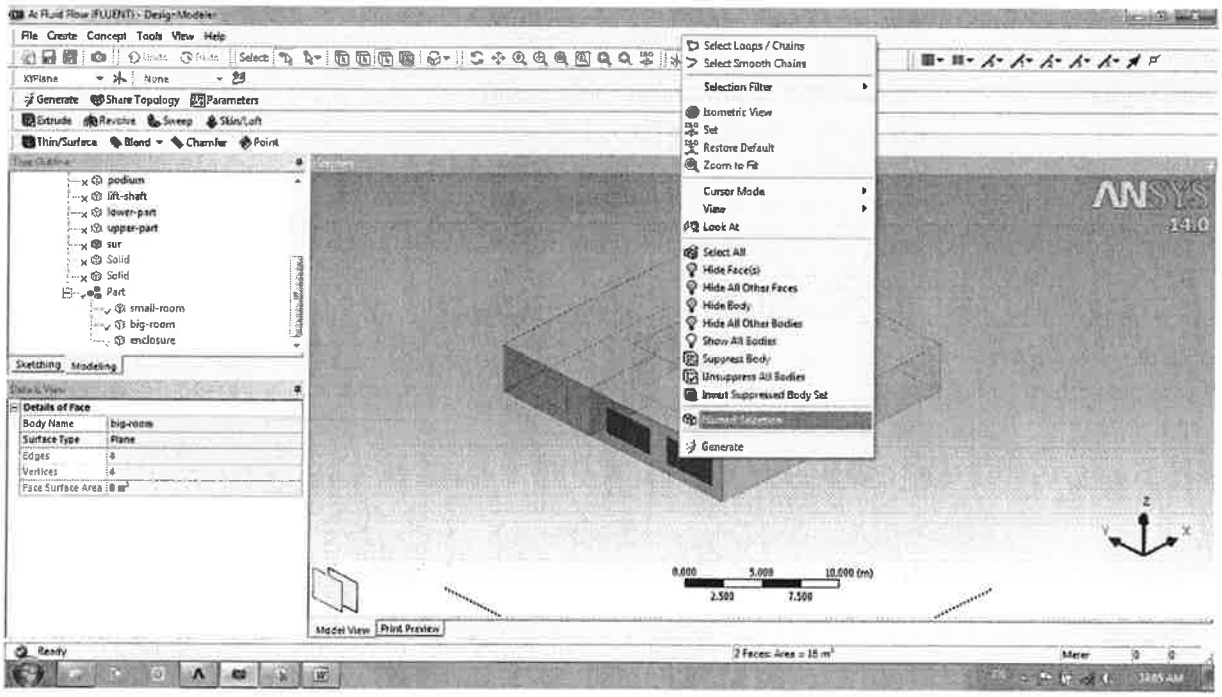


1.4 Creation of Name Selections

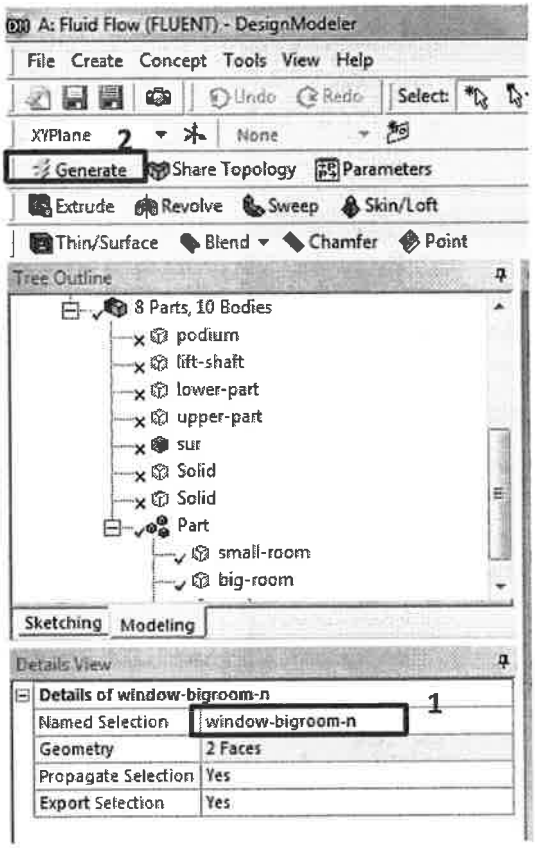
Hide body "enclosure" and fit the view for two rooms.

Define windows on the north of the big-room.

Holding the CTRL key, left click the two openings on –X side of big-room body and then right click the mouse button to choose ***Named Selection***.



Then define the selected faces as "window-bigroom-n" and generate the name selection.



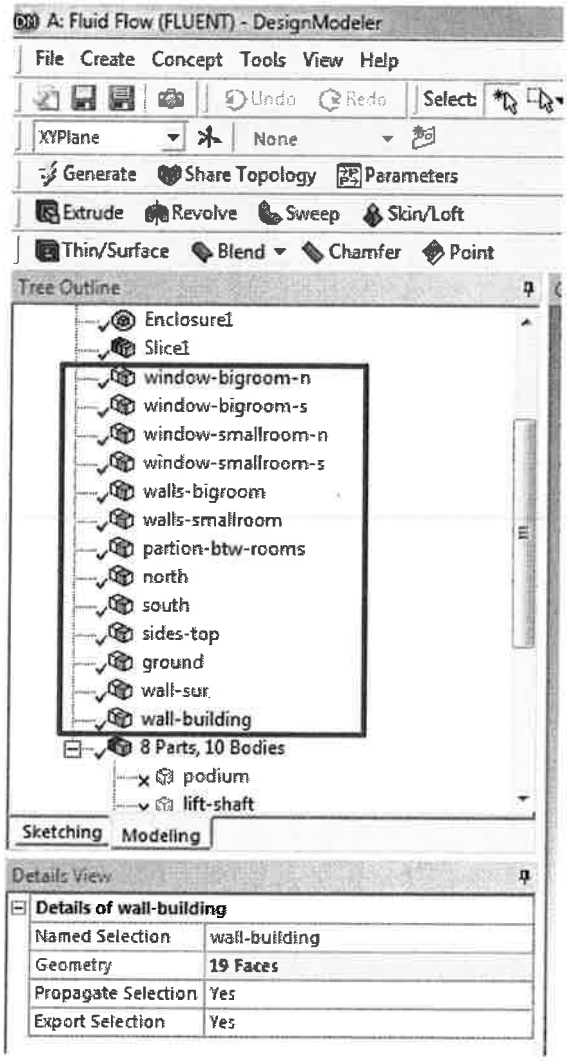
Similarly, define the other Name Selections as listed in the table below.

Name Selection	**Location**	Remark
window-bigroom-s	Two openings on +X side of big-room	
window-smallroom-n	Single opening on -X side of small-room	
window-smallroom-s	Single opening on+X side of small-room	
walls-bigroom	All vertical walls, floor and ceiling of big-room, except for the shared faces with small-room	Need to rotate the model
walls-smallroom	All vertical walls, floor and ceiling of small-room, except for the shared faces with big-room	
partition-btw-rooms	The two faces of small-room, which are shared with the body of big-room	Further hide the body of big-room

Furthermore, show the body of enclosure while hiding the other two bodies, then define the Name Selections as follows:

Name Selection	**Location**	Remark
north	The vertical face on -X side of enclosure	
south	The vertical face on +X side of enclosure	
ground	The horizontal face on the -Z side of enclosure	
sides-top	All other external bounding faces of enclosure	
wall-sur		Hide the external bounding faces and then use box selection to choose the faces for sur
wall-building		Hide the shared faces with rooms and then use box selection to choose all faces for the development

Then the feature trees with Name Selections will look like this

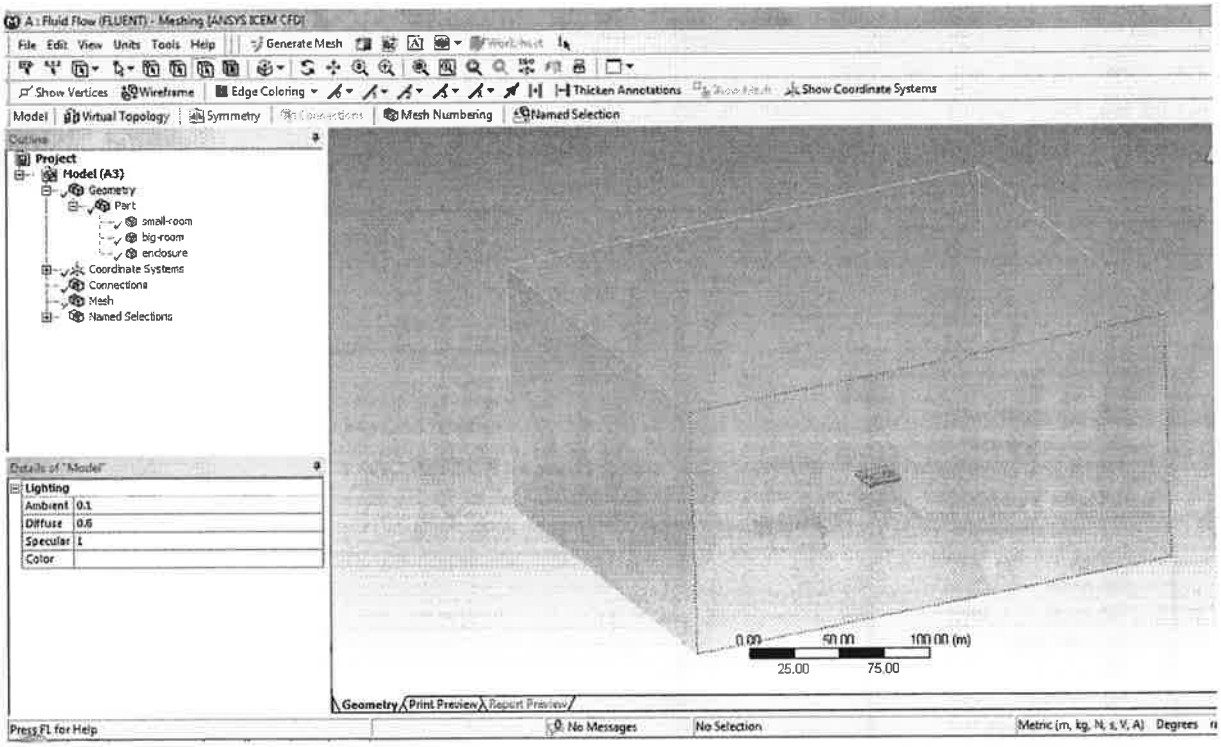


Save the project again.

Then exit Design Modeler : ***File > Close DesignModeler***

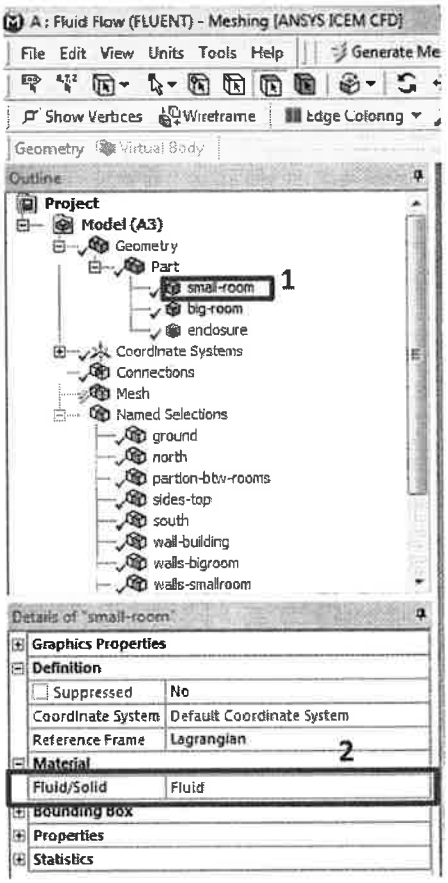
2. Mesh Generation

Double click Mesh icon to launch ANSYS Meshing.



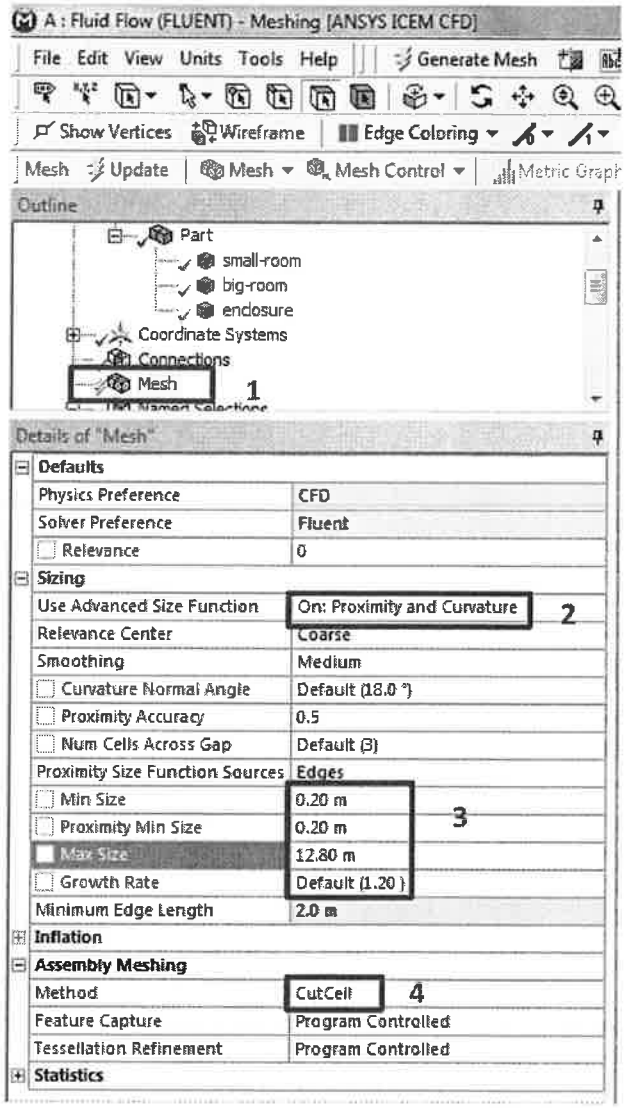
2.1 Update the material type of bodies

Change **material type** to be fluid for small-room. Repeat the process for the other two bodies.



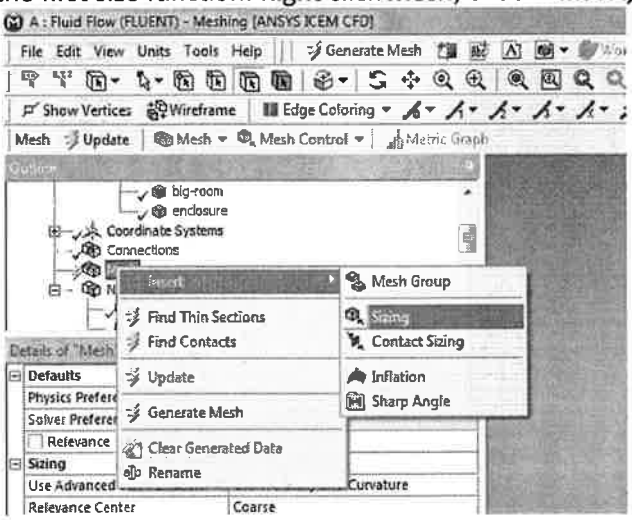
2.2 Assign global mesh settings

Choose **Cutcell** method for assembly meshing, for the sake of high-quality hexa-dominated elements and reduced number of elements.

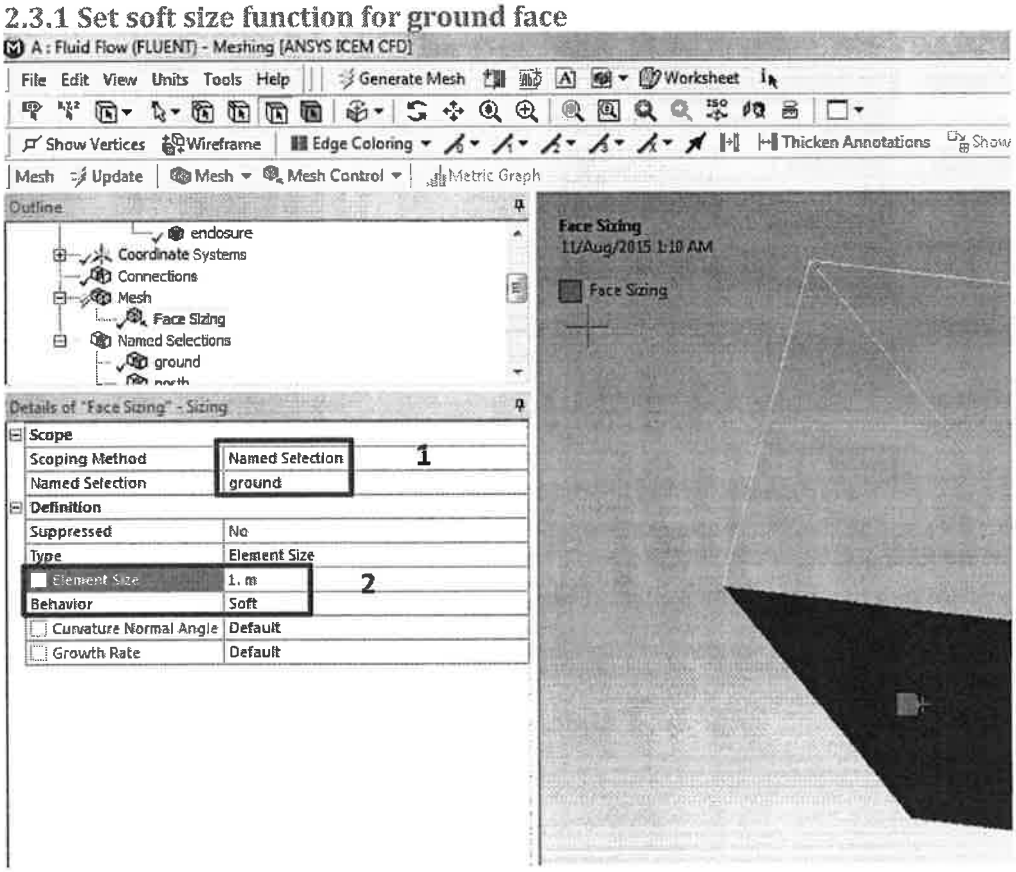


2.3 Define local size functions

Create the first size function: Right click ***Mesh***, choose ***Insert, Sizing***

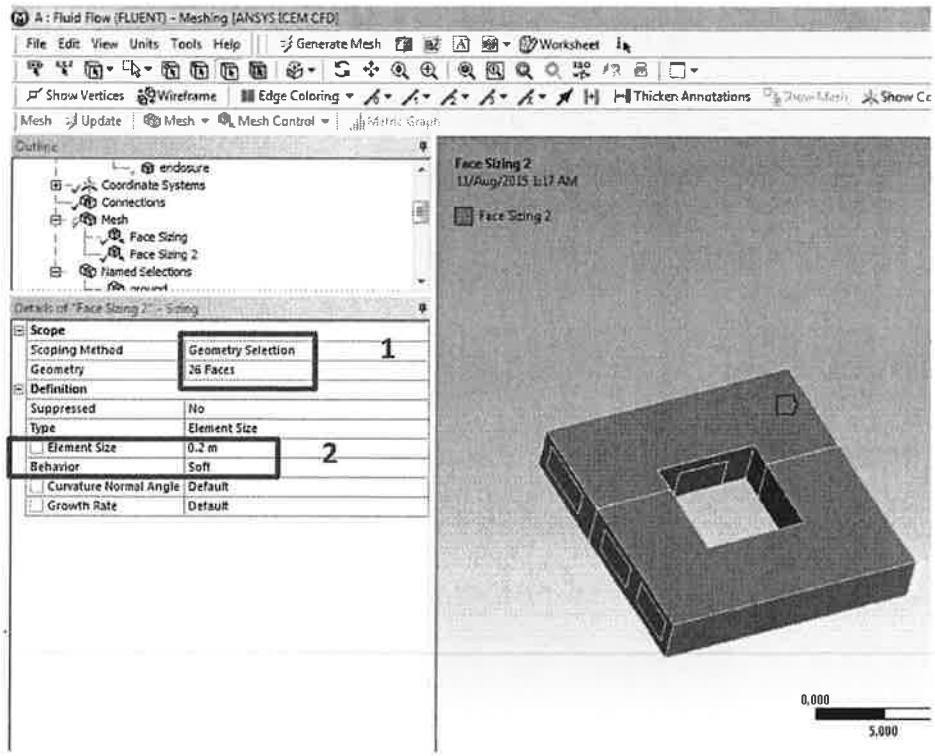


2.3.1 Set soft size function for ground face



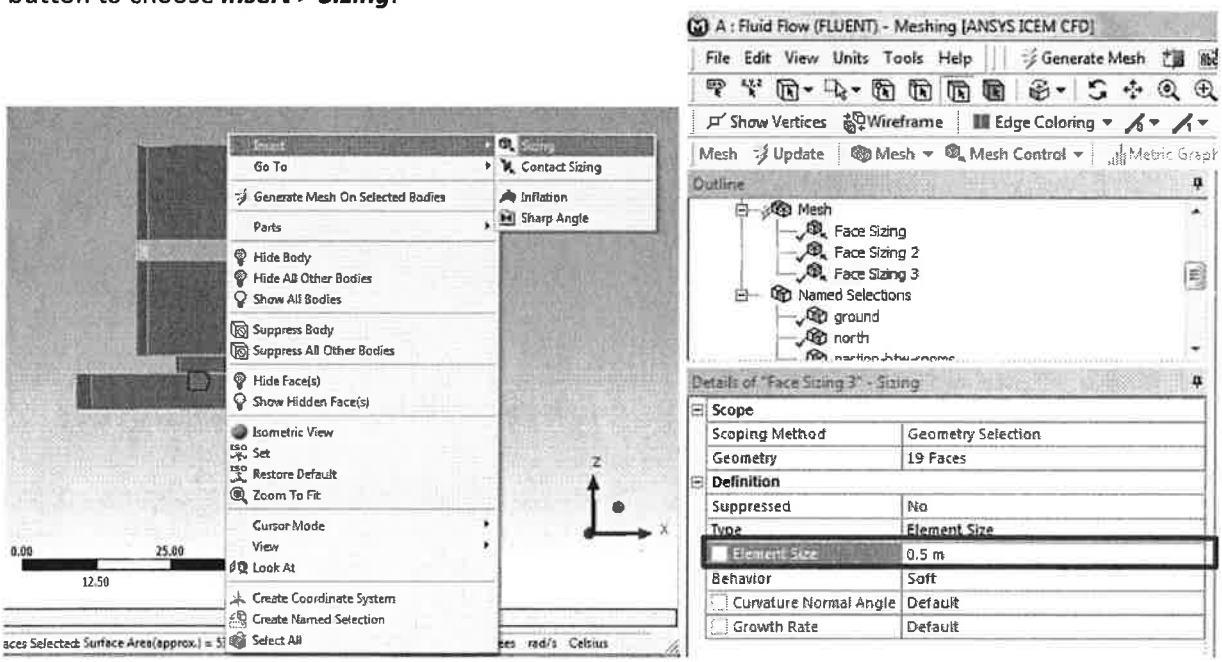
2.3.2 Set size function for faces of rooms

Hide the body of Enclosure. Then use **box selection** to choose all faces for two rooms and then set the size function accordingly.



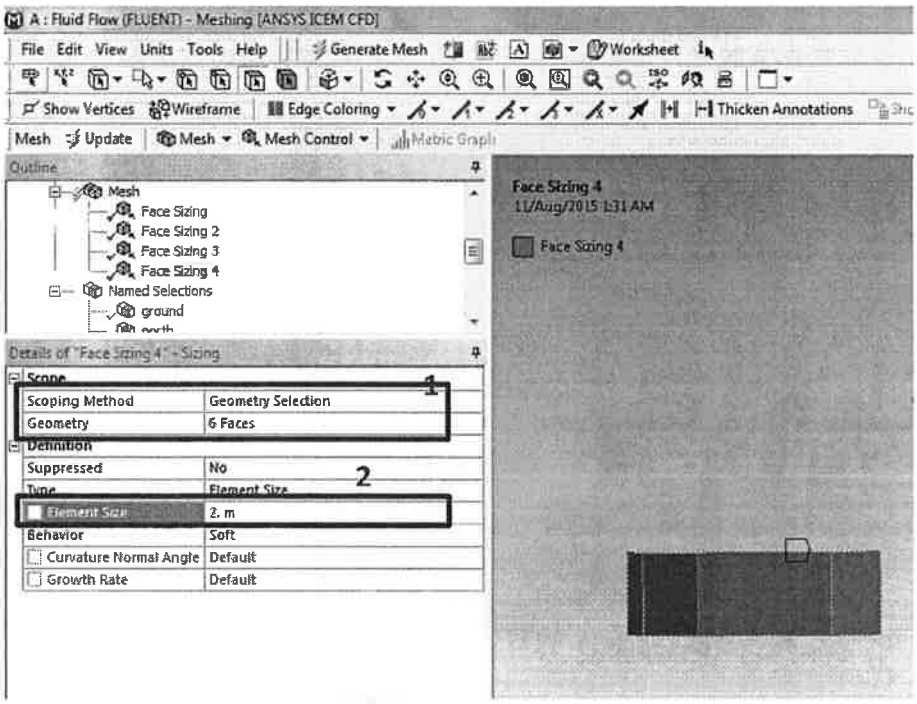
2.3.3 Size function for the development

Hide both bodies for rooms and ***show*** the body of Enclosure only.
Hide all external faces of enclosure body.
Hide shared faces with rooms using box selection.
Then use ***box selection*** to choose all remaining faces for the development. Right click the mouse button to choose ***Insert > Sizing***.



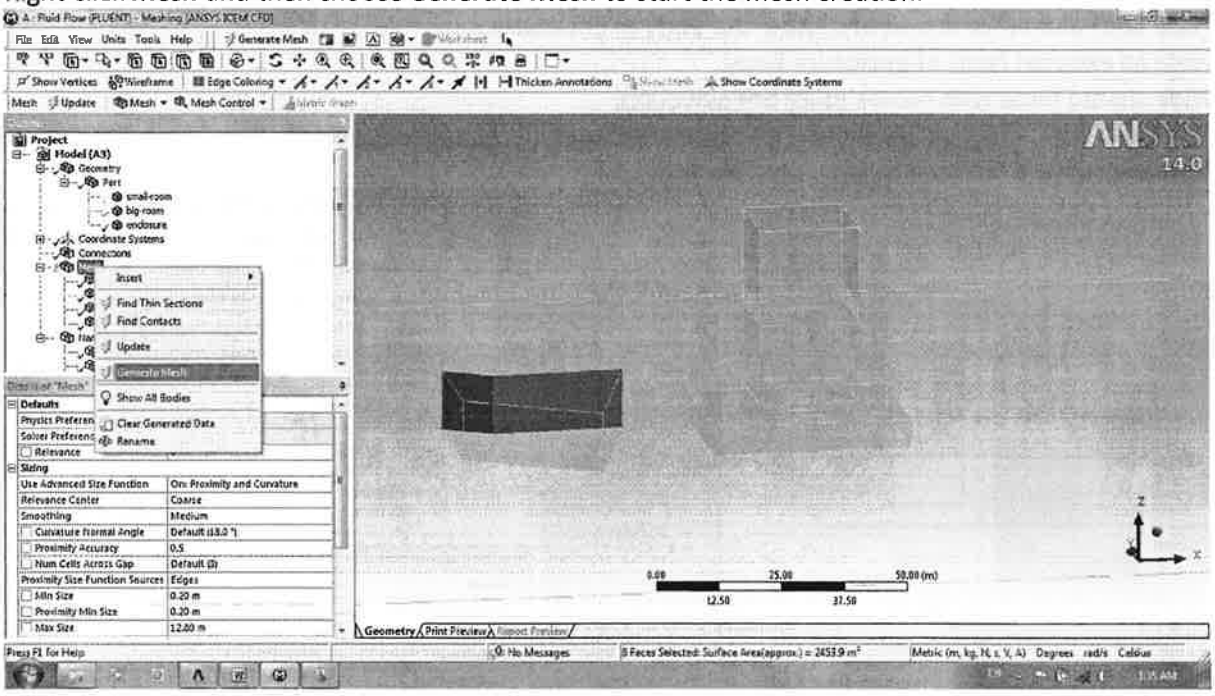
2.3.4 Size function for the surrounding building

Right click "Face Sizing 3" to choose ***Duplicate*** to create "Face Sizing 4". Update the faces by selecting the faces for sur body using box selection and then click ***Apply***. Set the element size to be no more than 2m in this tutorial.

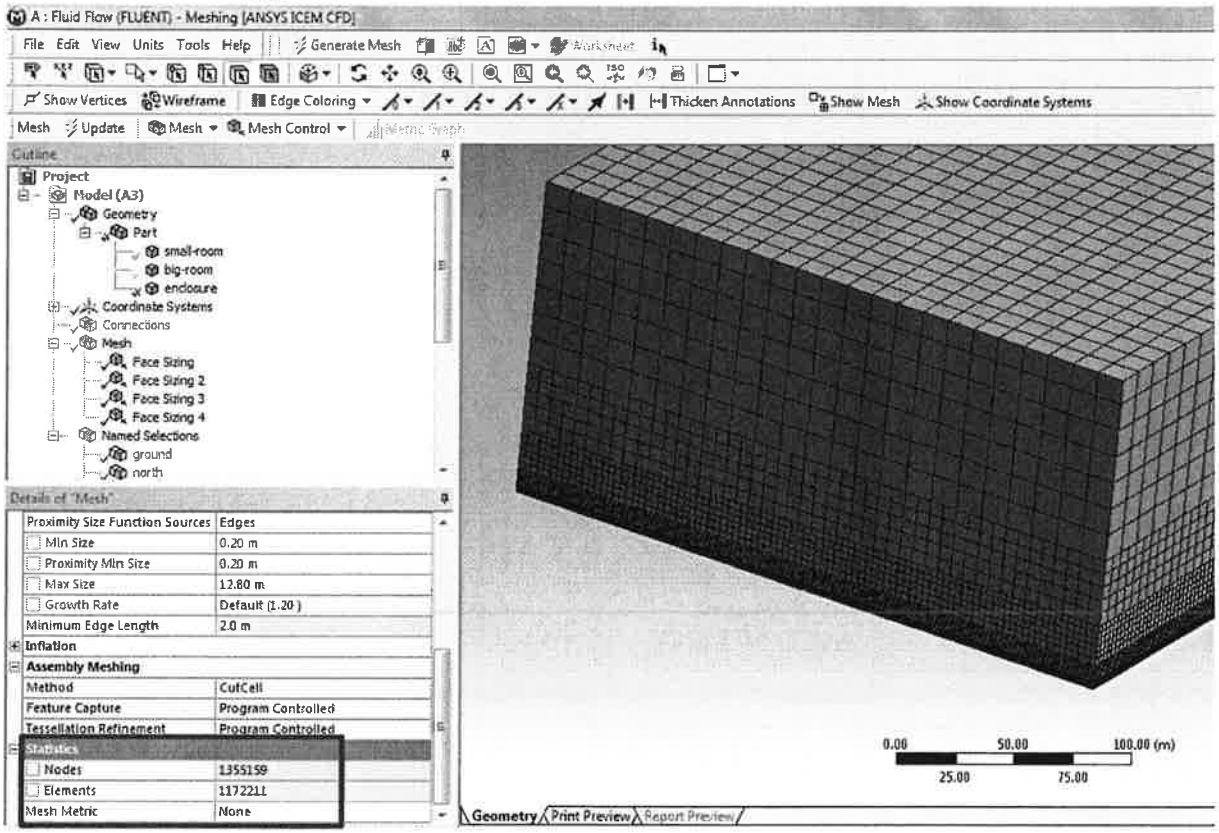


2.4 Meshing Generation

Right click ***Mesh*** and then choose ***Generate Mesh*** to start the mesh creation.



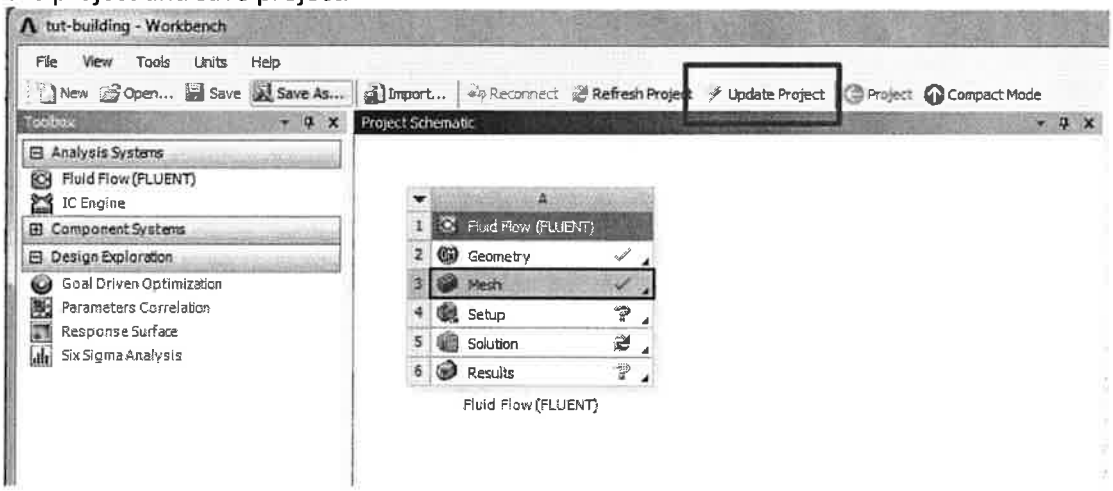
2.5 Check mesh



In total there will be around 1m hexa-dominated elements to be generated to spatially discretized the computational domain.

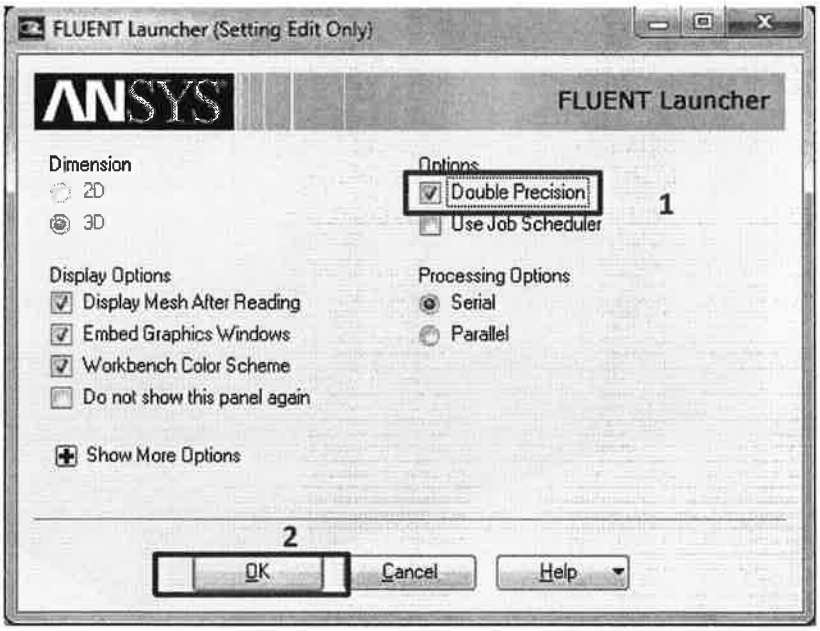
Exit ANSYS Meshing.

Update the project and save project.



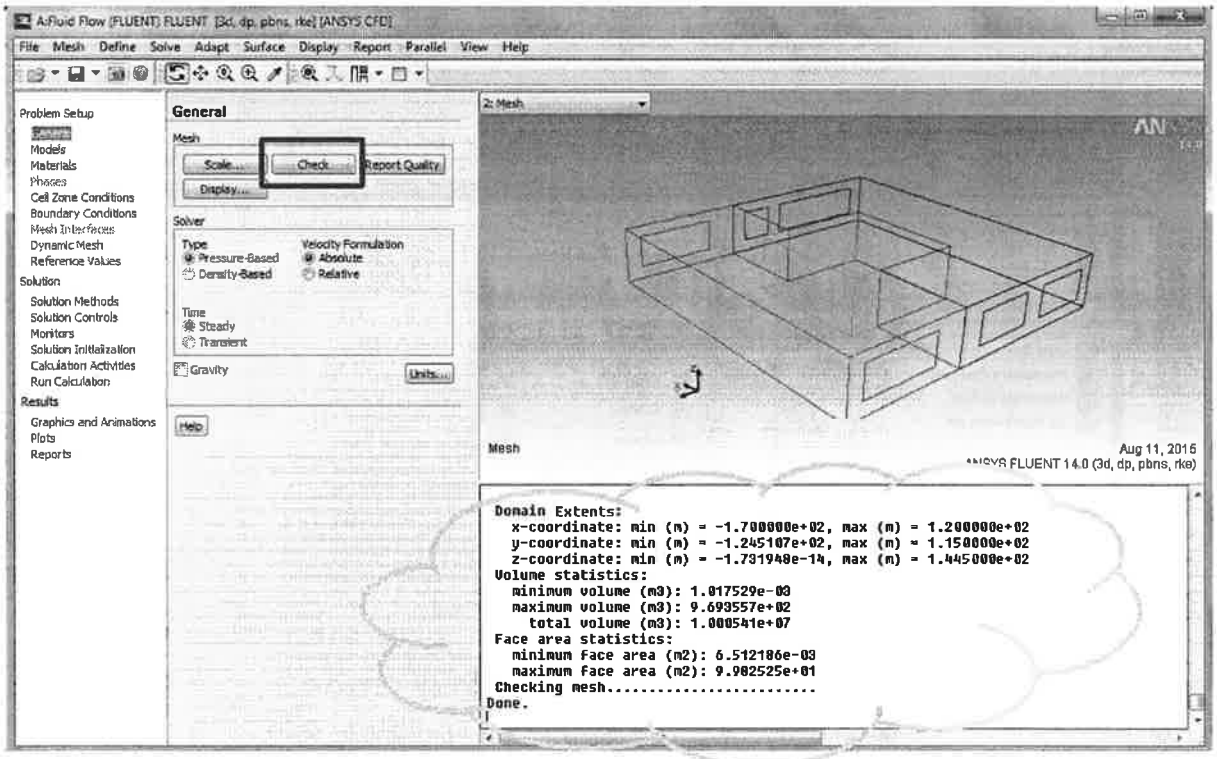
3 Numerical settings and computation

Double click Setup icon to launch ANSYS FLUENT. Turn on Double Precision option.



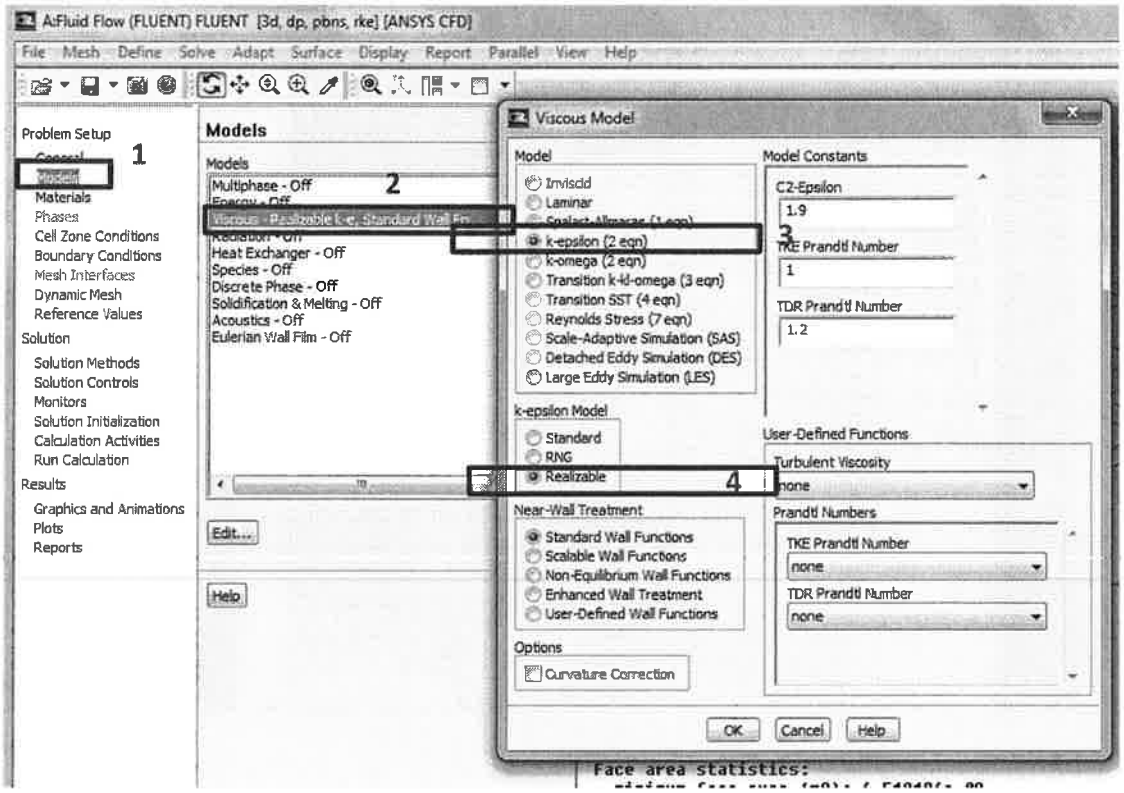
3.1 Check mesh

Make sure there is no error message about the meshing.



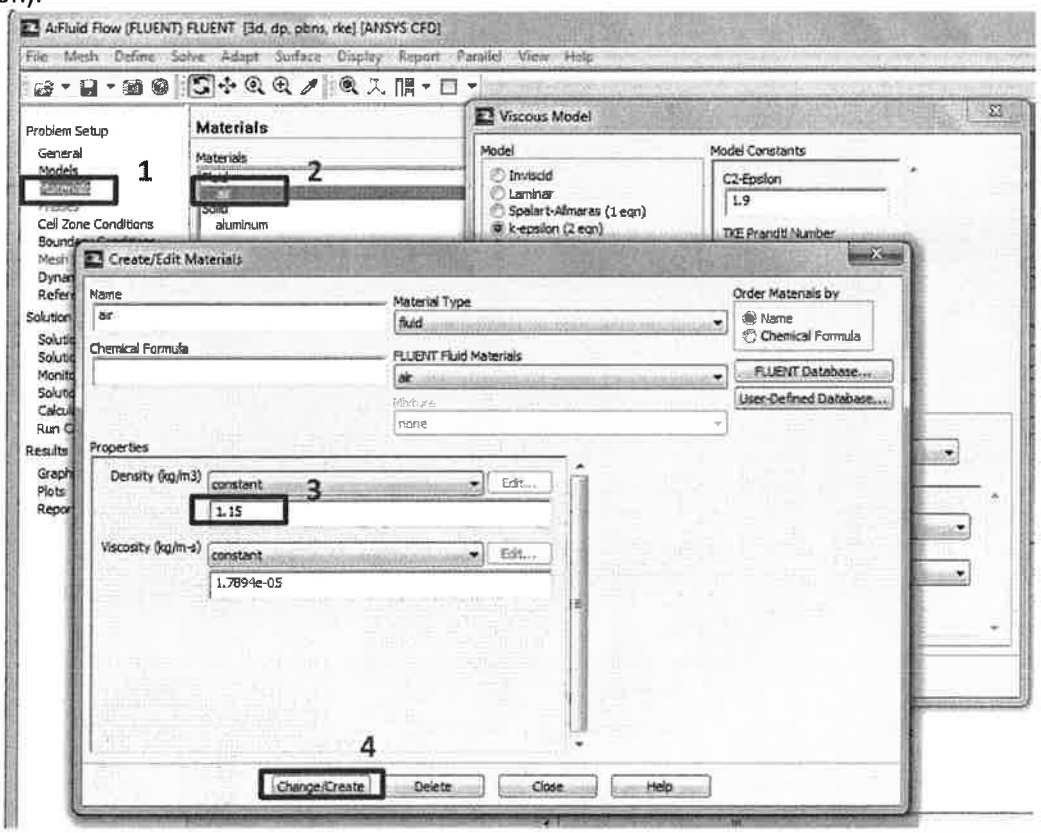
3.2 Model setting

Choose Realizable k-e turbulence model

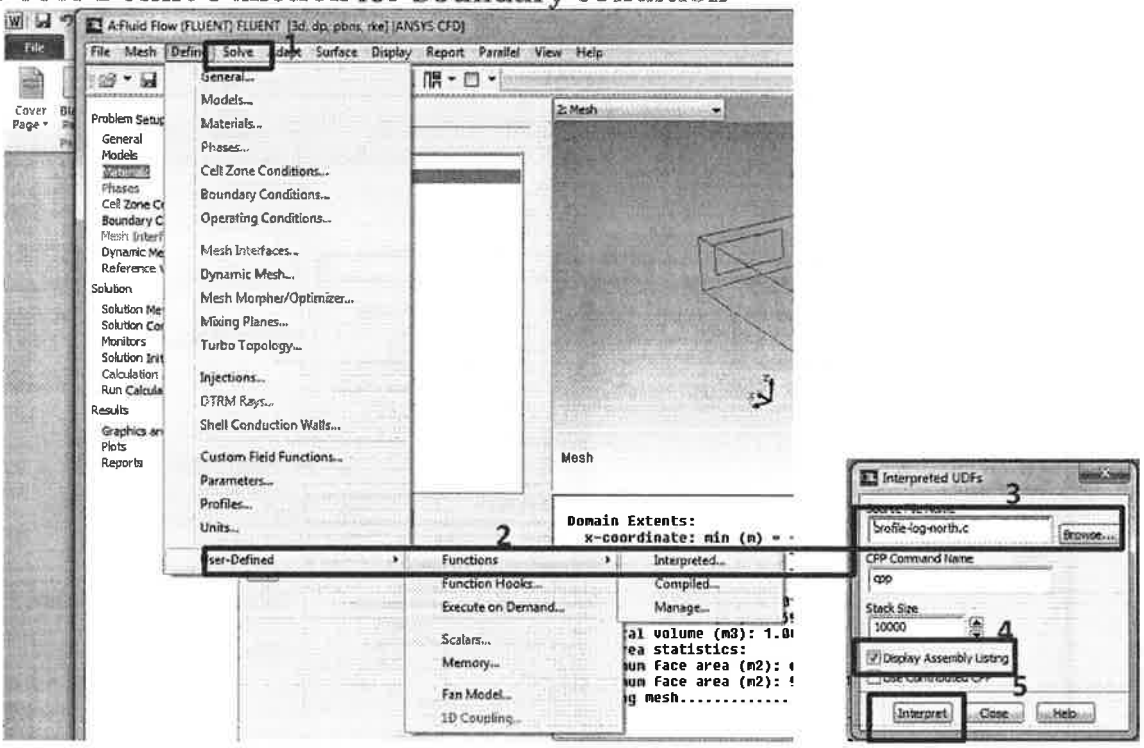


3.3 Material setting

Change air density to be 1.15 kg/m^3, in accordance with 33.1°C in temperature (Isothermal condition).

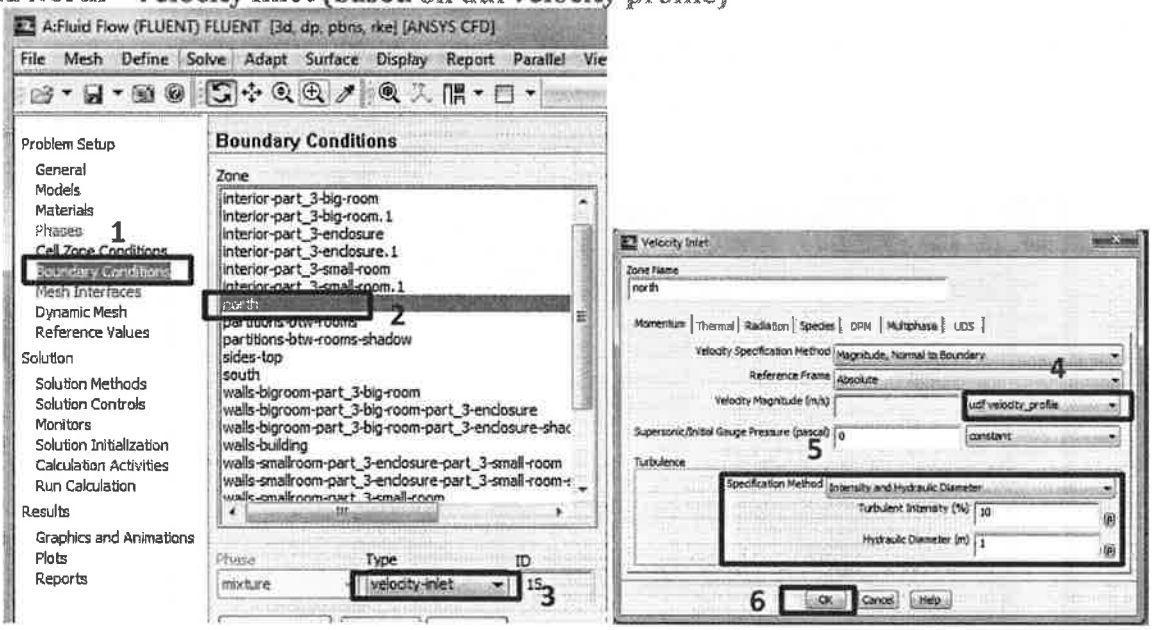


3.4 User Define Function for boundary condition

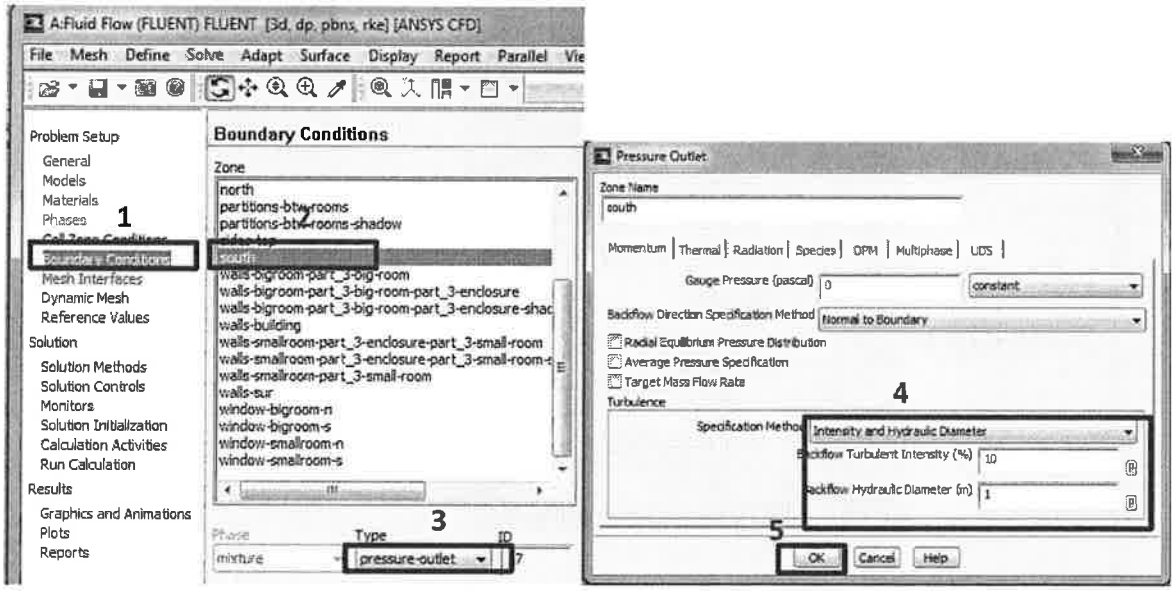


3.5 Definition of boundary conditions

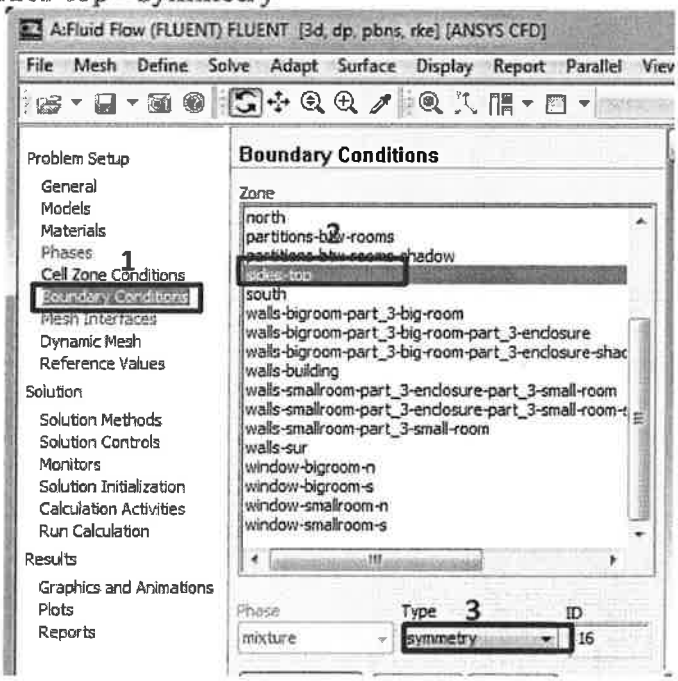
3.5.1 North – Velocity Inlet (based on udf velocity profile)



3.5.2 South – Pressure Outlet

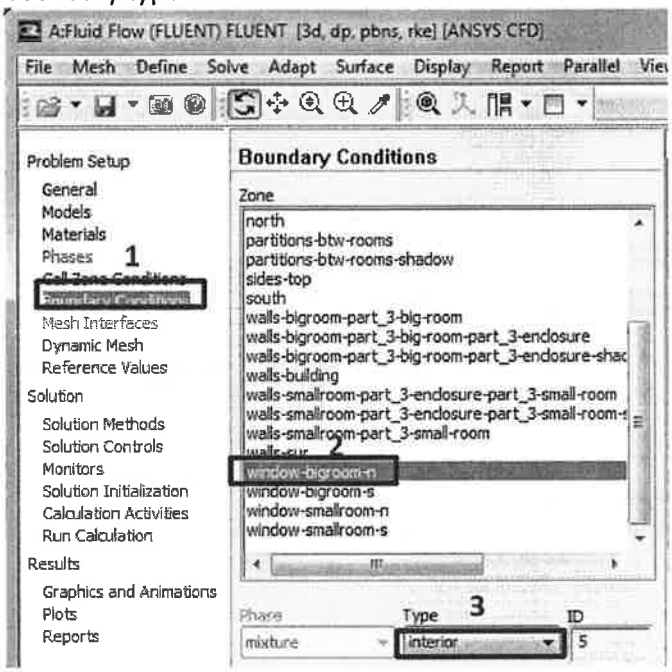


3.5.3 Sides-top – Symmetry



3.5.4 window-* - Interior

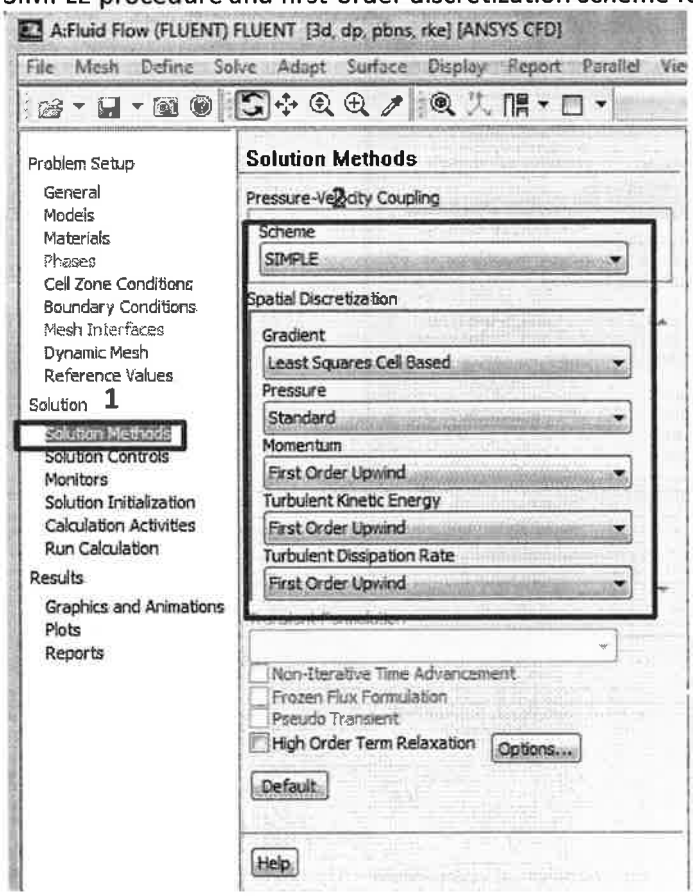
Change boundary type for all window-* Name Selections as Interior.



Repeat the changes for Window-bigroom-s, window-smallroom-n, and window-smallroom-s.

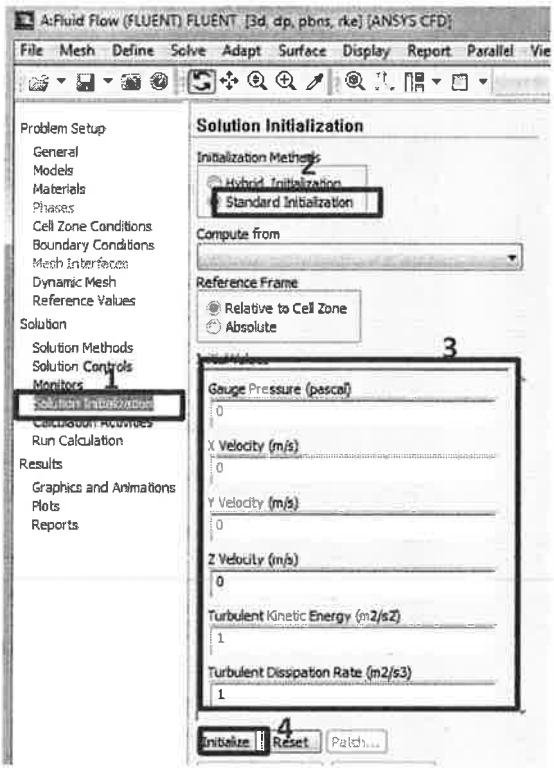
3.6 Solution method

Choose SIMPLE procedure and first-order discretization scheme for all quantities.



3.7 Initialization

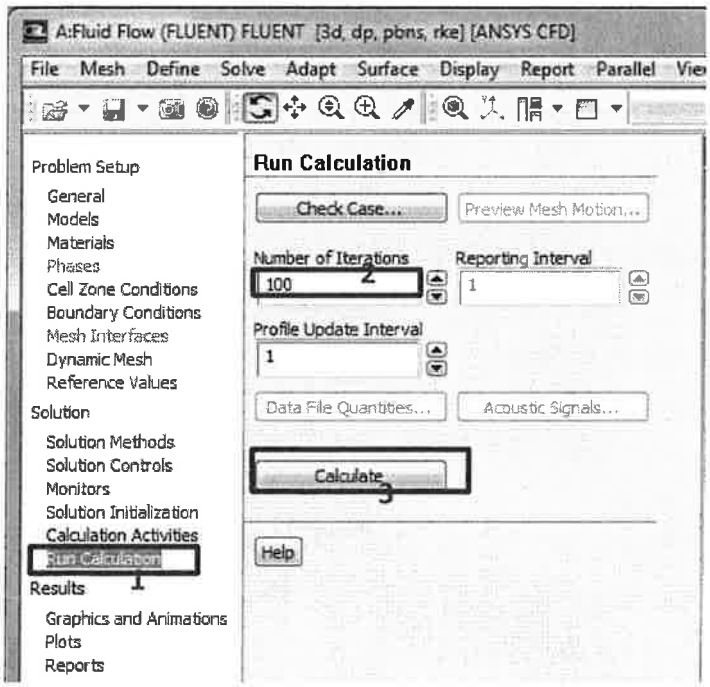
Standard initialization



3.8 Save Project

3.9 Run Calculation

Due to time constraint, only 100 iterations will be performed for the tutorial. Note that 100 iterations are not sufficient to reach convergence. Student can run more if time permits.

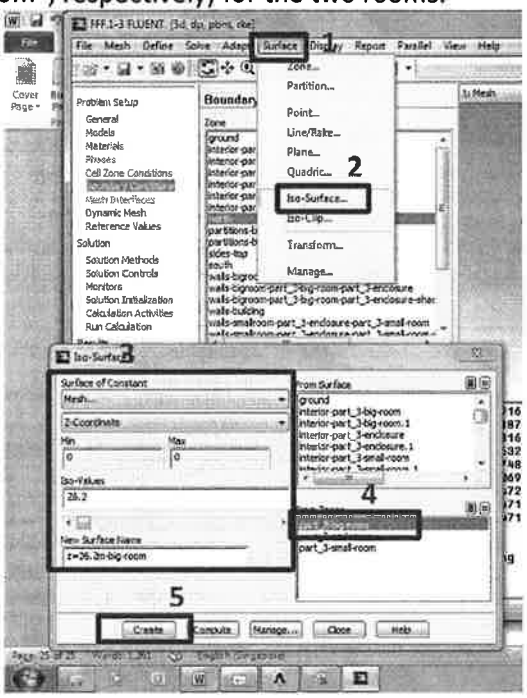


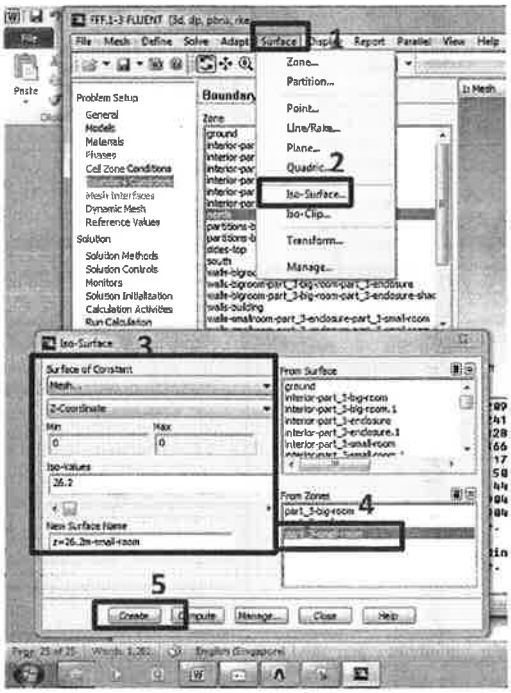
Once the simulation is completed, save project again.

4 Post processing simulation results using FLUENT

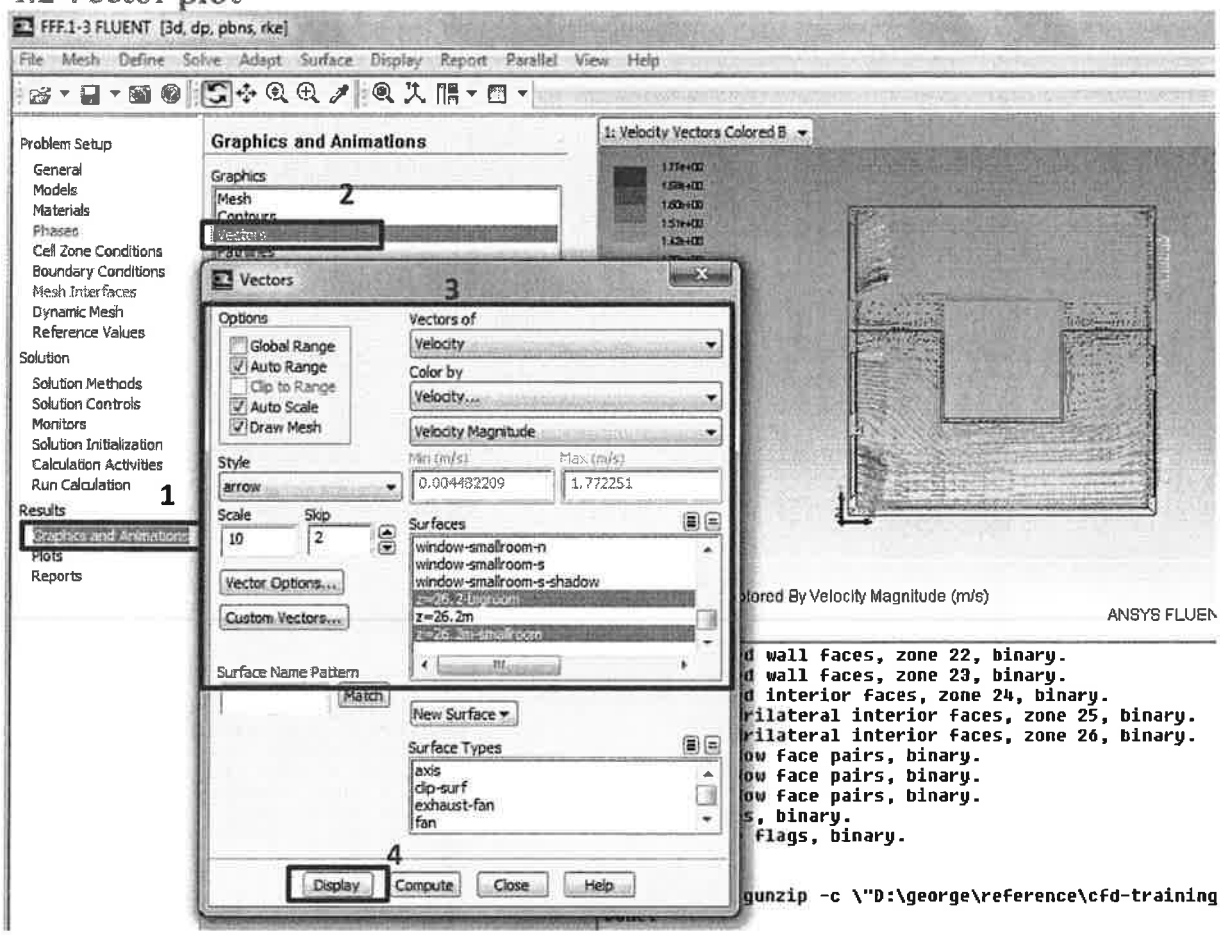
4.1 Definition of cutting planes

Z=26.2m @1.2m above the floor; two separate faces (named "z=26.2m-bigroom" and "z=26.2m-smallroom", respectively) for the two rooms.

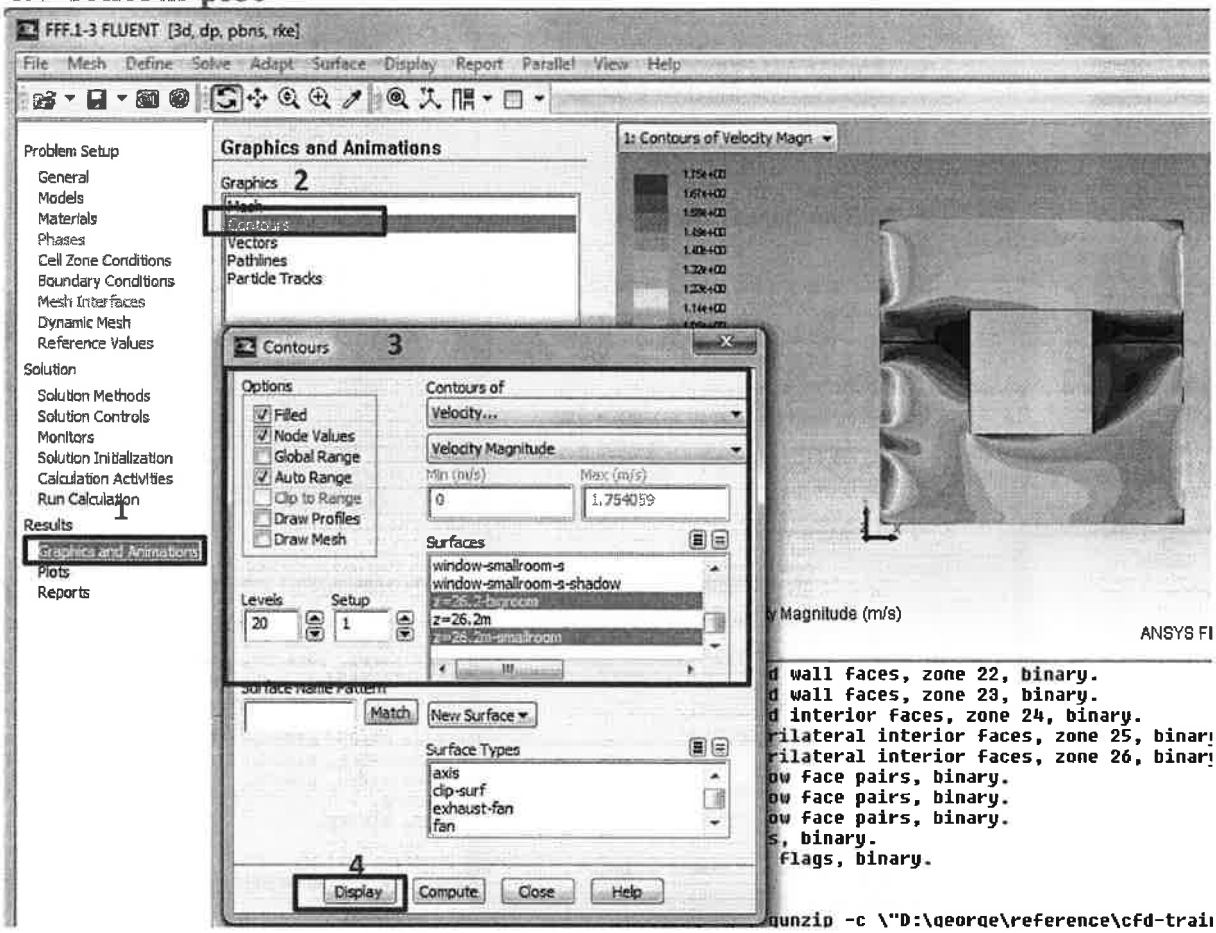




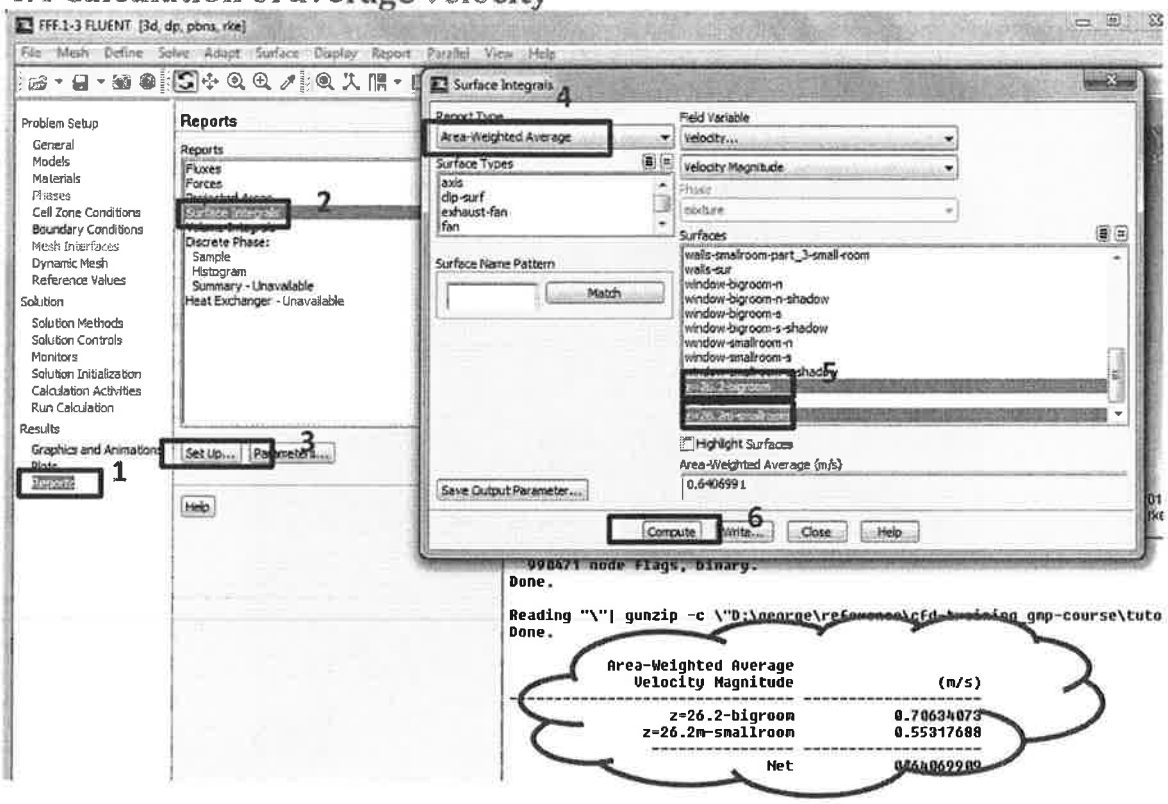
4.2 Vector plot



4.3 Contour plot

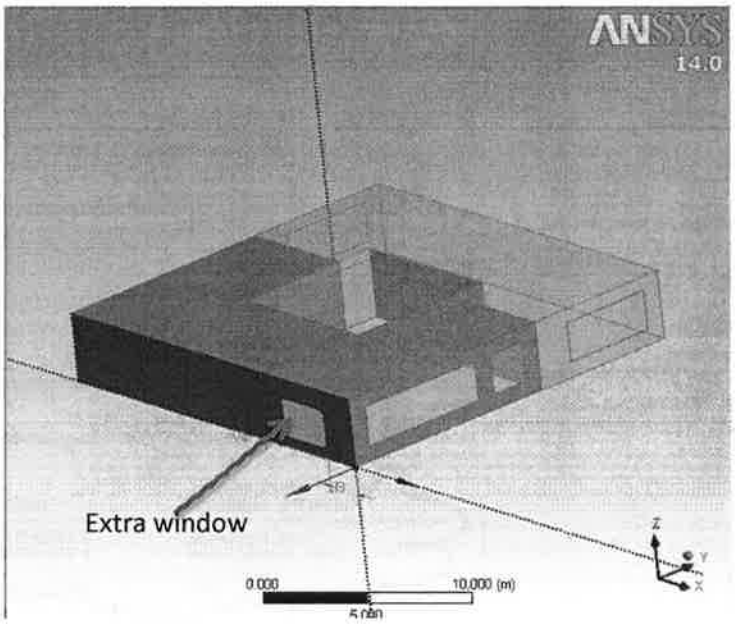


4.4 Calculation of average velocity



Appendix

Revised model: Add one more window for the "big-room" body

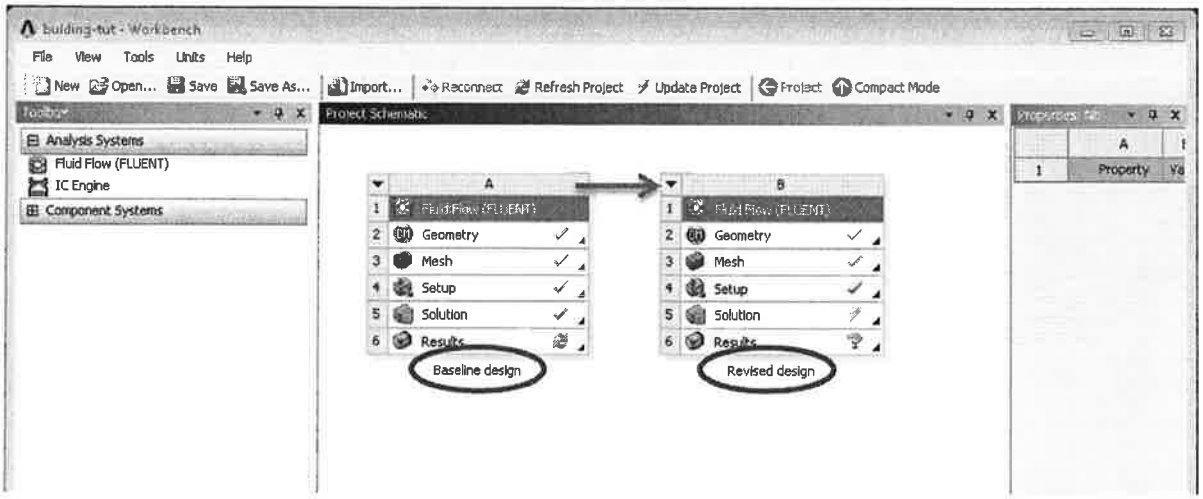


Please note that Design Modeler adopts parametric design approach in geometry creation, thus the "first-come, first serve" principle is applicable throughout the process in feature creation. Therefore, in order to add the window for Body "big-room", the operations should be inserted before the operation to import the surrounding buildings.

A.1 Duplicate the original scheme in Workbench

Duplicate the model in Workbench by ***right clicking*** the original scheme and then choosing ***duplicate*** option. ***Rename*** the original Schematic as "Baseline design" and the new one as "Revised design", simply double clicking the respective regions. Then the new case will be updated with the inclusion of a new window.

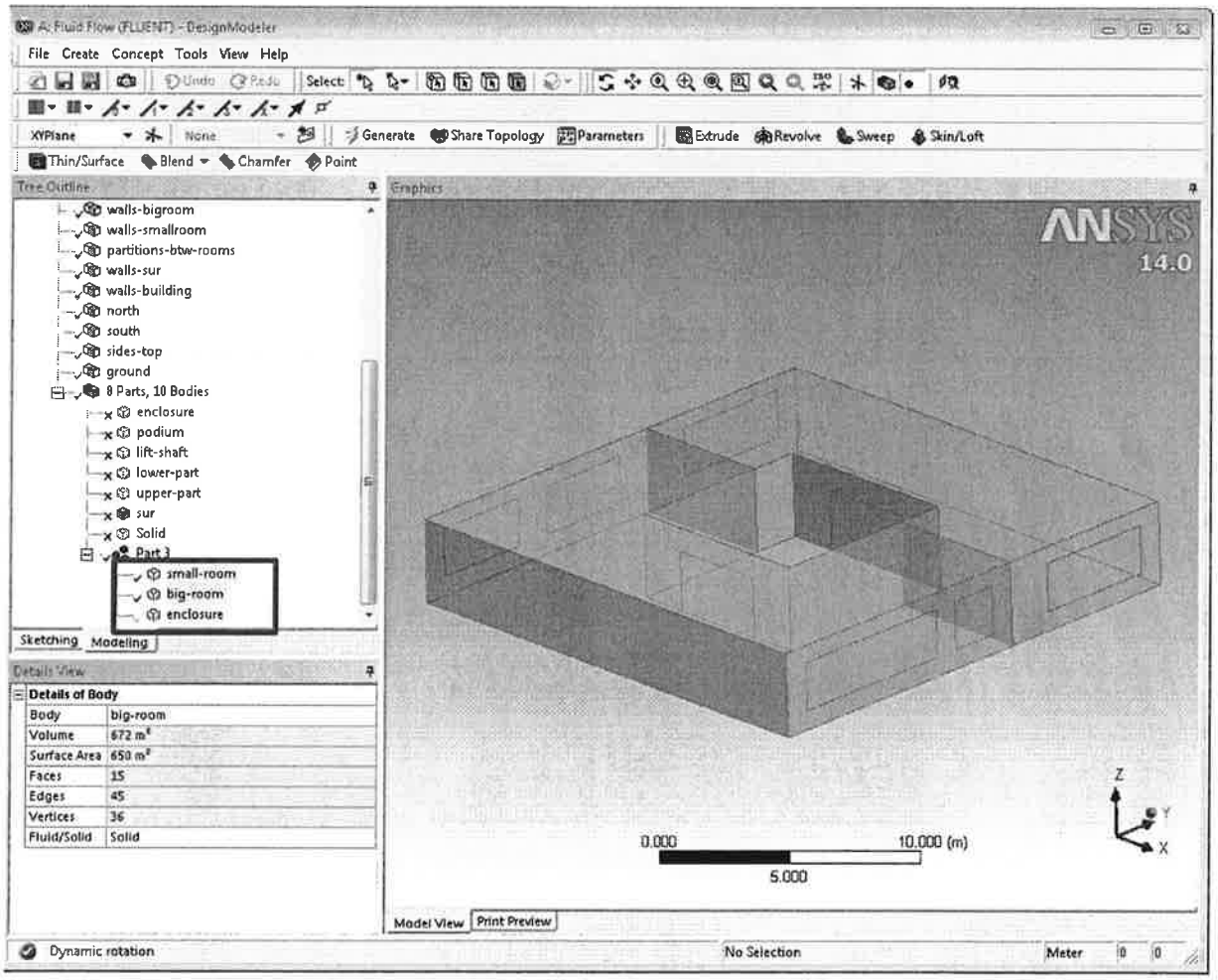
Double click the Geometry icon in the schematic for Revised design to launch Design Modeler.



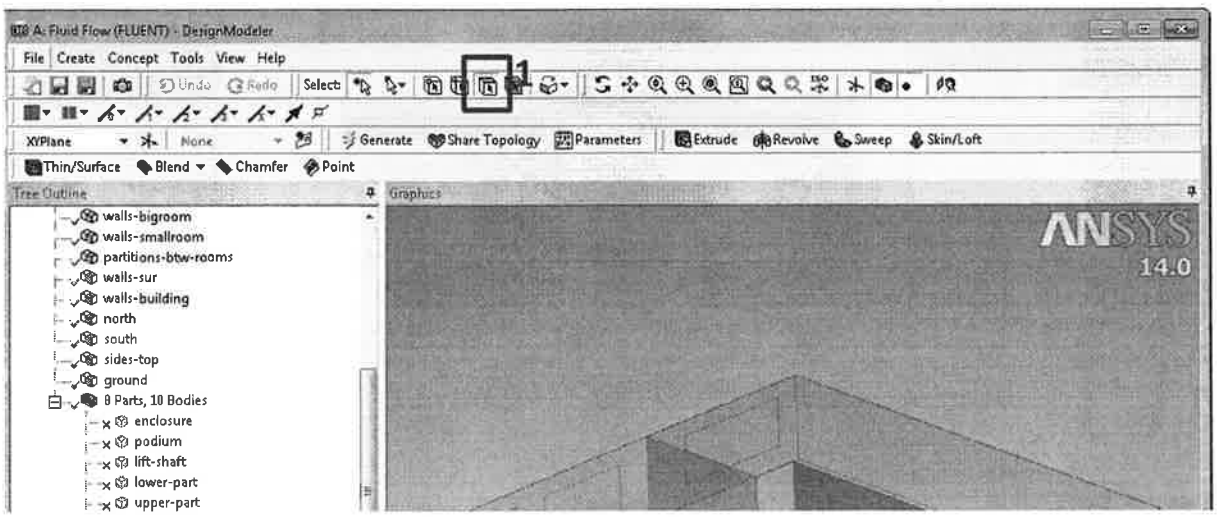
A.2 Creation of a new window for Body "big-room"

- Re-orient the model display

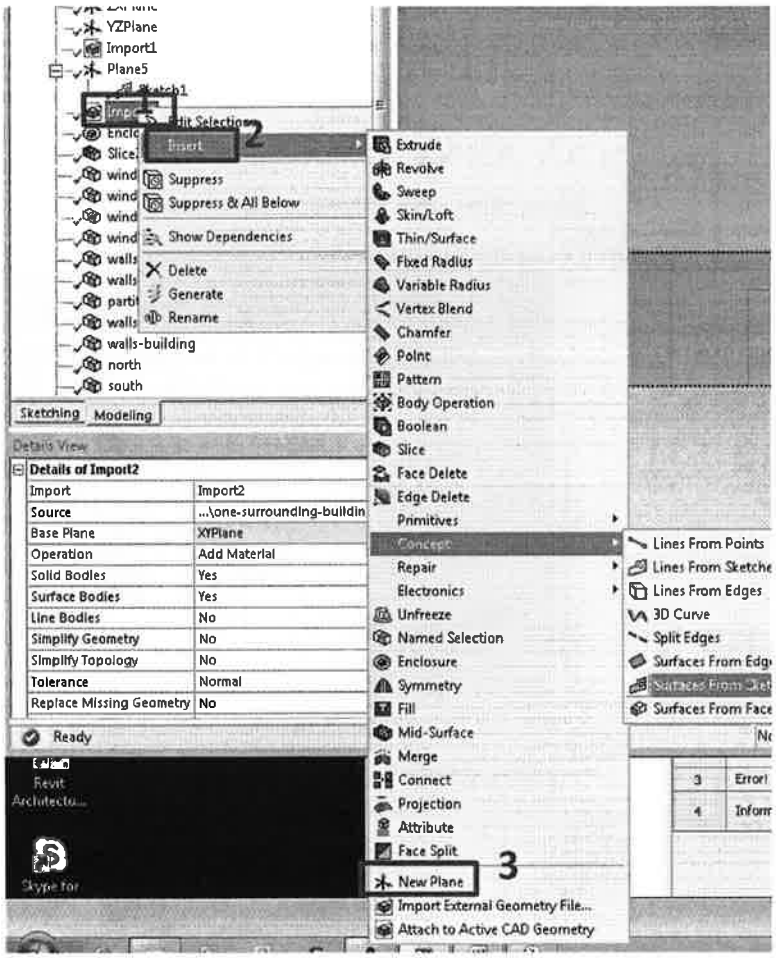
Hide body "enclosure" while keep bodies "small-room" and "big-room" **unhidden** and all solid bodies **suppressed**.



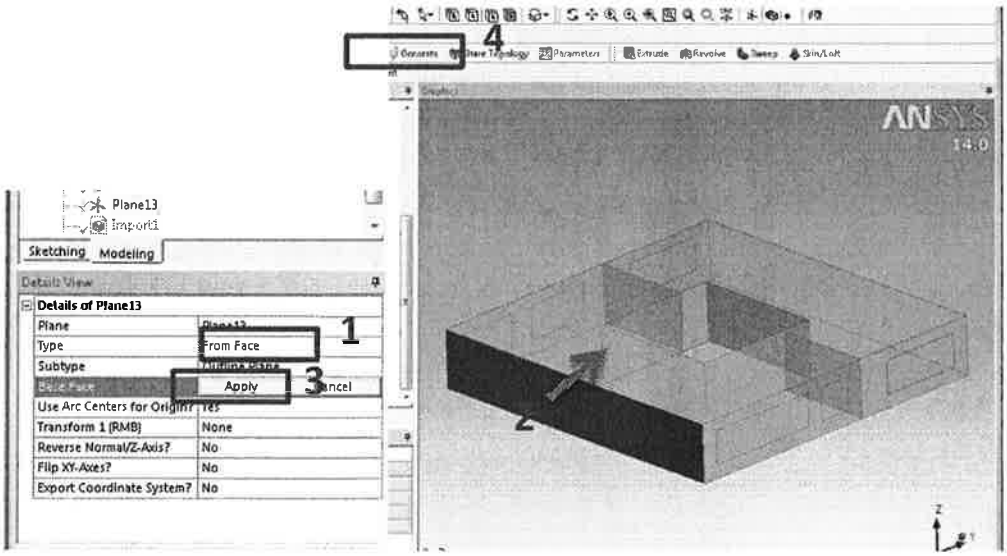
- Choose ***Face-selection*** filter.



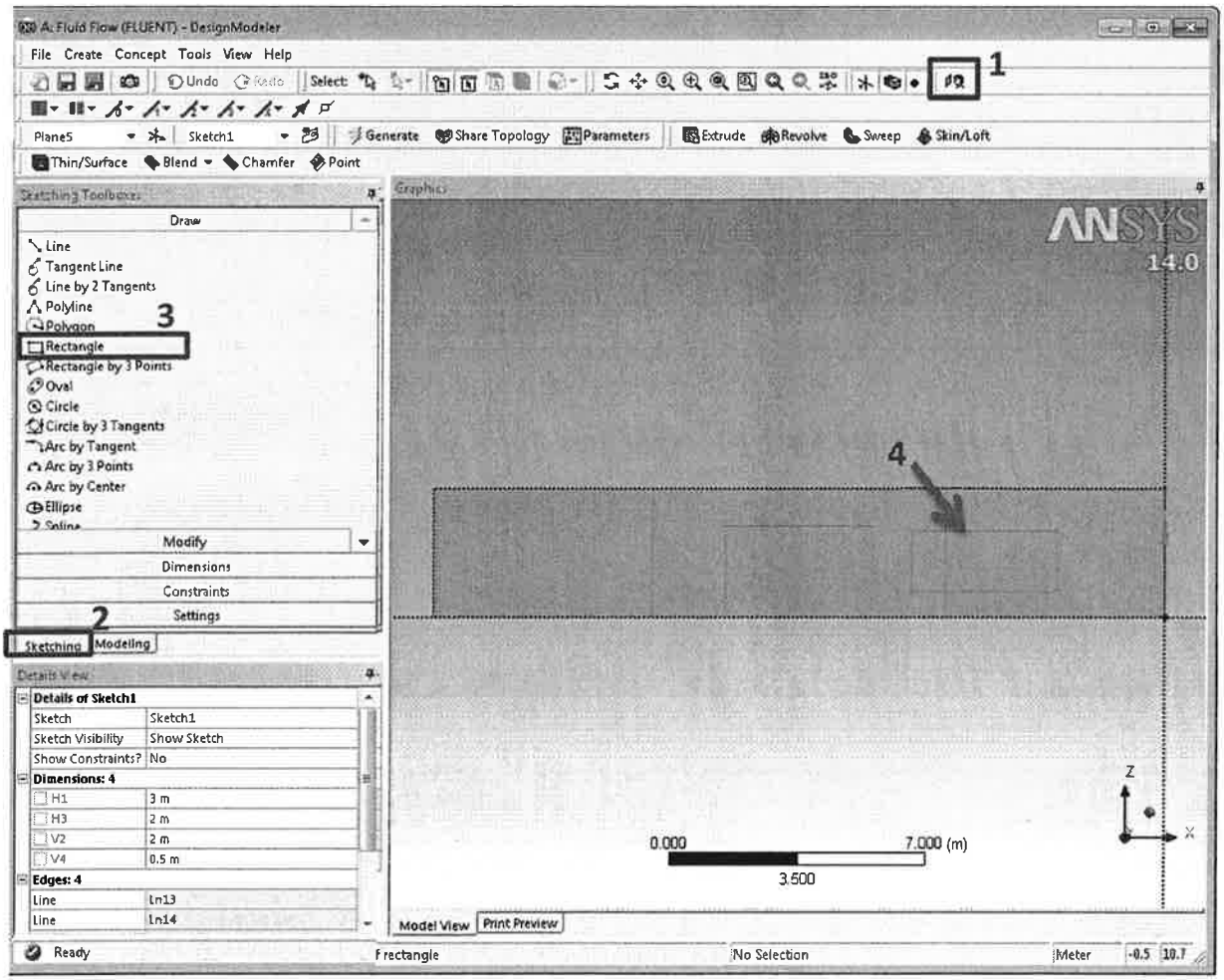
- Under **Tree Outline** side window, *right-click* **Import1** feature icon in the **Tree Outline** side window, then choose *insert -> New Plane*.



- Under **Detail View** side window, choose "**From Face**" Type, *left-click* the lateral face in the Model View screen, and then click "**Apply**" button next to Base Face option to finalize the selection. Then click "**Generate**" button to create the working plane.



- Select new plane feature (for example, "Plane13") in **Tree Outline** side window and then click **"Look at Face"** button to view the model from the angle normal to the face. Switch from Modeling mode to **Sketching** mode. Choose a **Rectangle** sketch in the **Sketching Toolboxes** side window and then draw a rectangle on the working plane in Model View window.



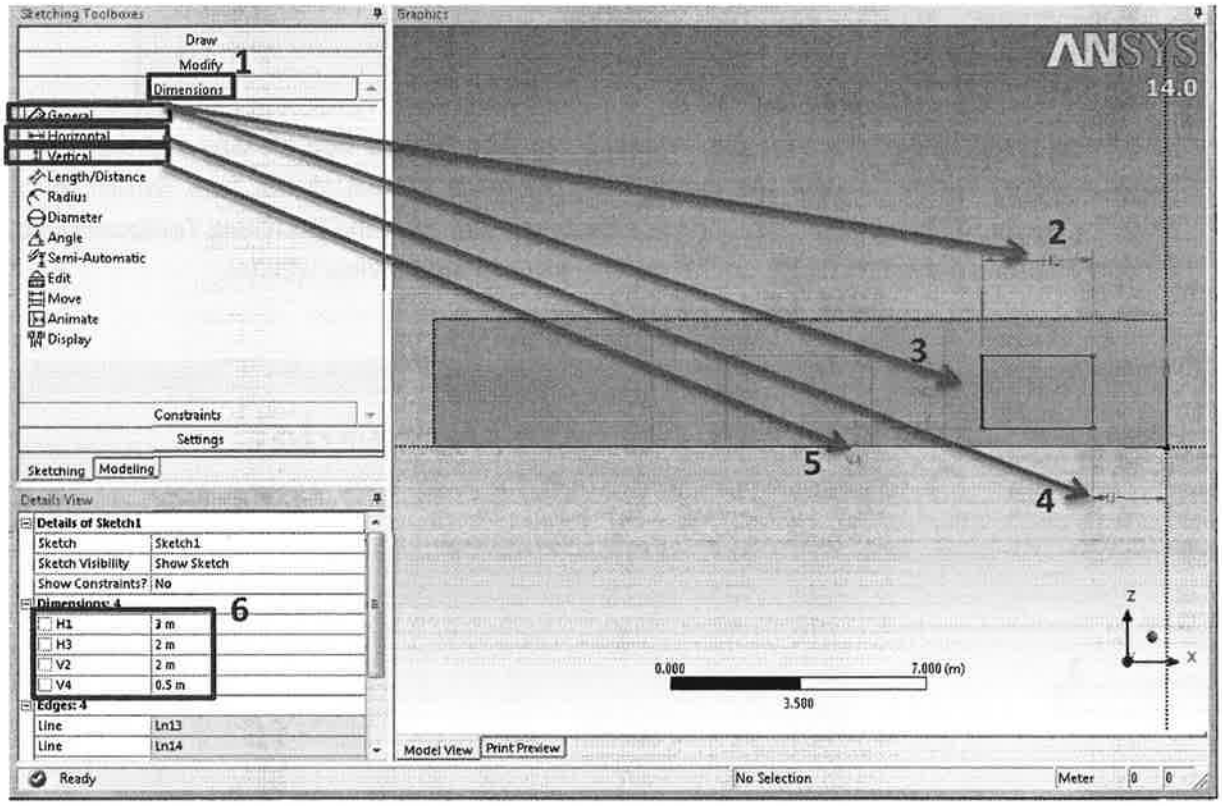
- Set dimensions for the new window.

Use "**general**" option for dimension H1 and V2, "**Horizontal**" for H3 and "**Vertical**" option for V4. For "**general**" dimension, just simply left-click the side of the rectangle, and drag the mouse upwards (for H1) or to the left (for V2) and then click the left mouse button.

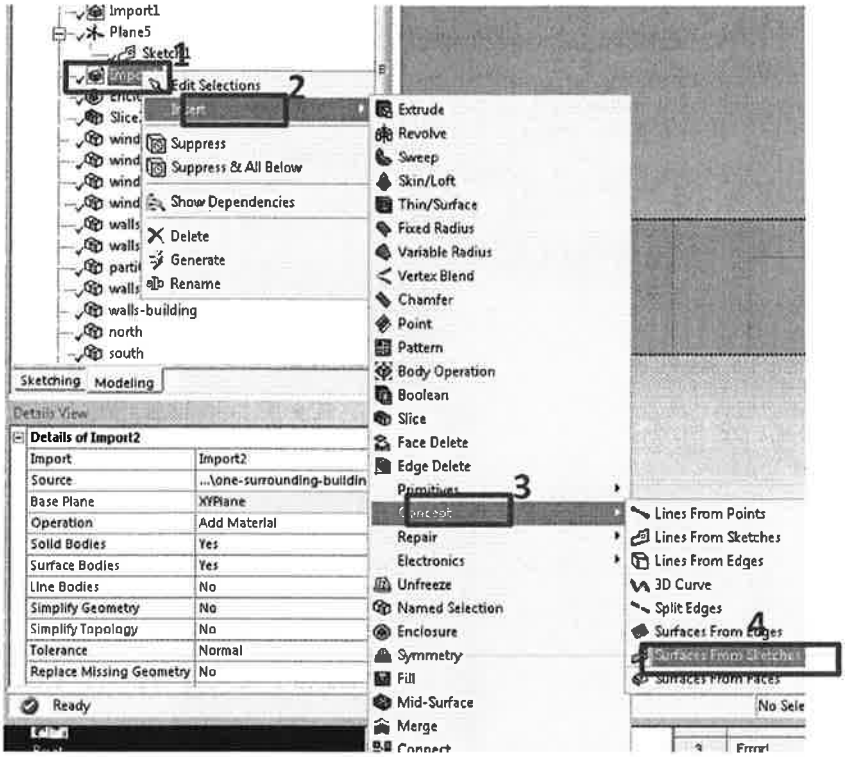
For "**Horizontal**" dimension, use the left mouse button to click the right edge of the rectangle first, followed by the local y-axis on the right, and then drag the mouse downwards a bit to click the left mouse button to position the dimension.

Similarly, for "**Vertical**" dimension, choose the bottom edge of the rectangle and local x-axis in sequence, and then drag the mouse to the left a bit to click the left mouse button to position the dimension.

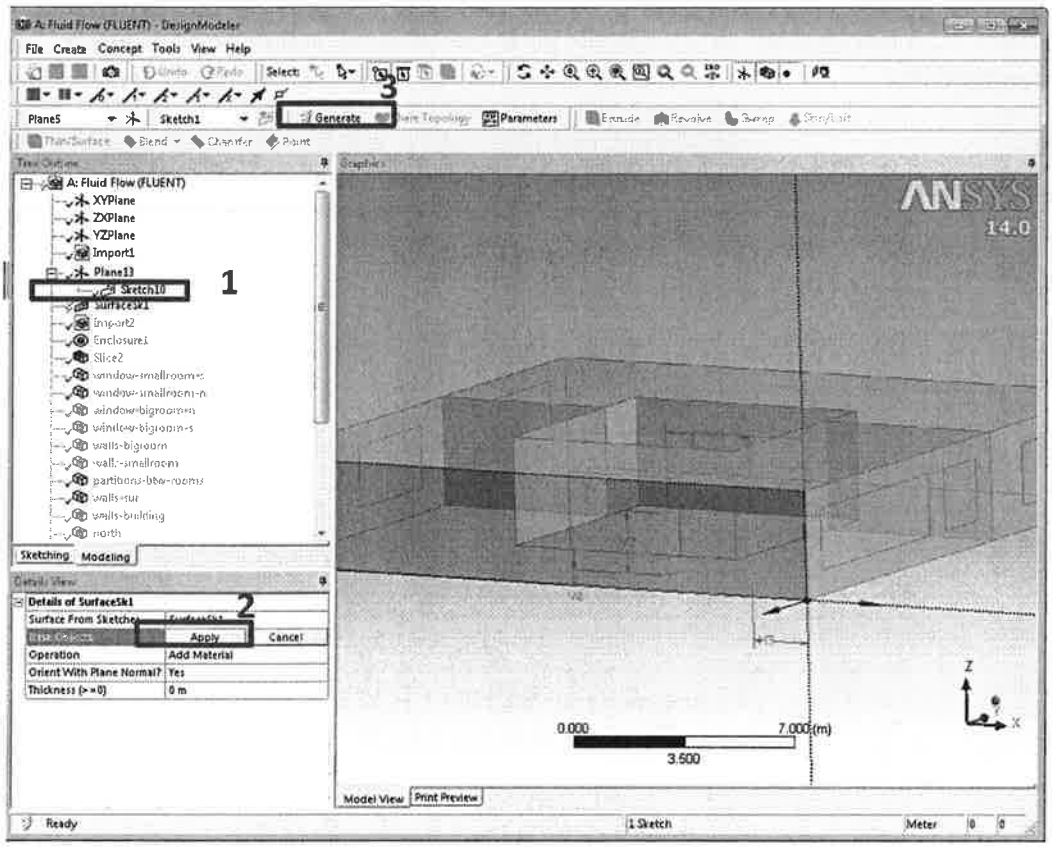
Set the values (H1 = 3, H3 = 2, V2 = 2, V4 = 0.5) in the **Details View** side window.



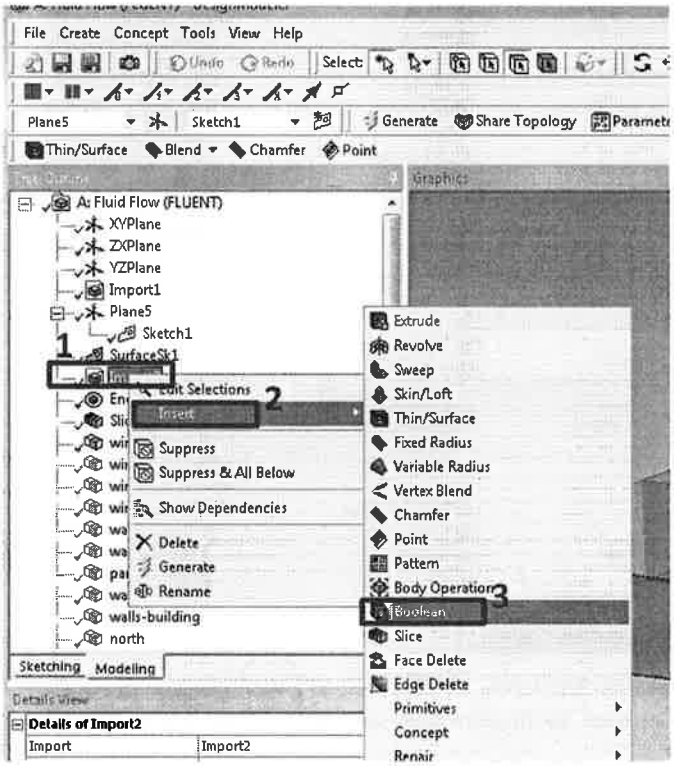
- Right click the feature "Import1" and choose *Insert ->Concept ->Surfaces from Sketches* to create a surface body.



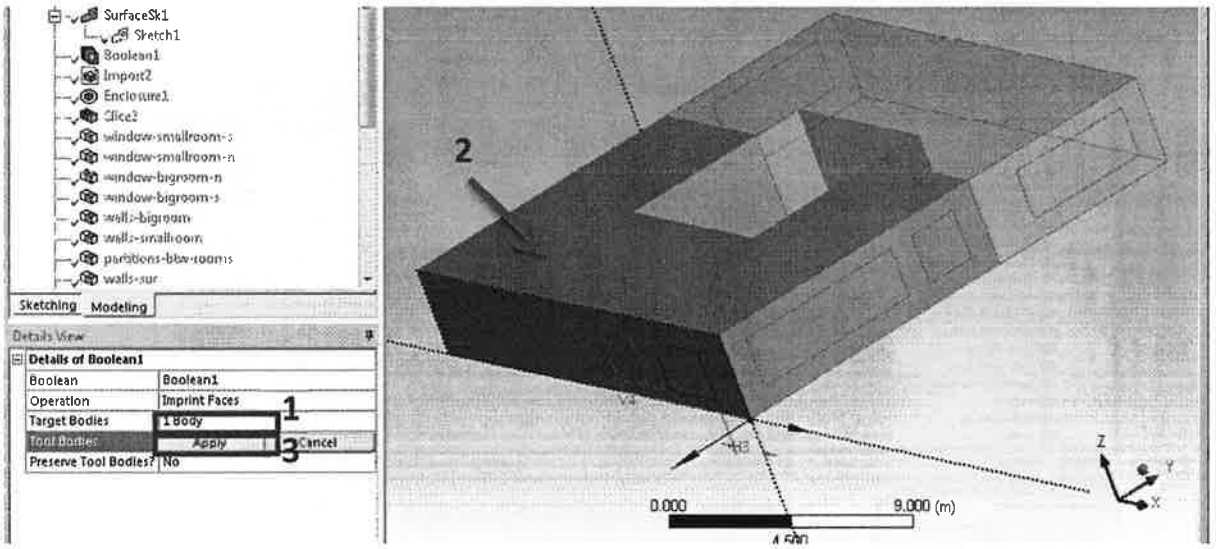
- **Left-click** the newly created sketch (for example, "**Sketch10**) within Plane13 feature in **Tree Outline** side window, then click "**Apply**" button to finalize the selection. Then click "**Generate**" button to create the surface body.



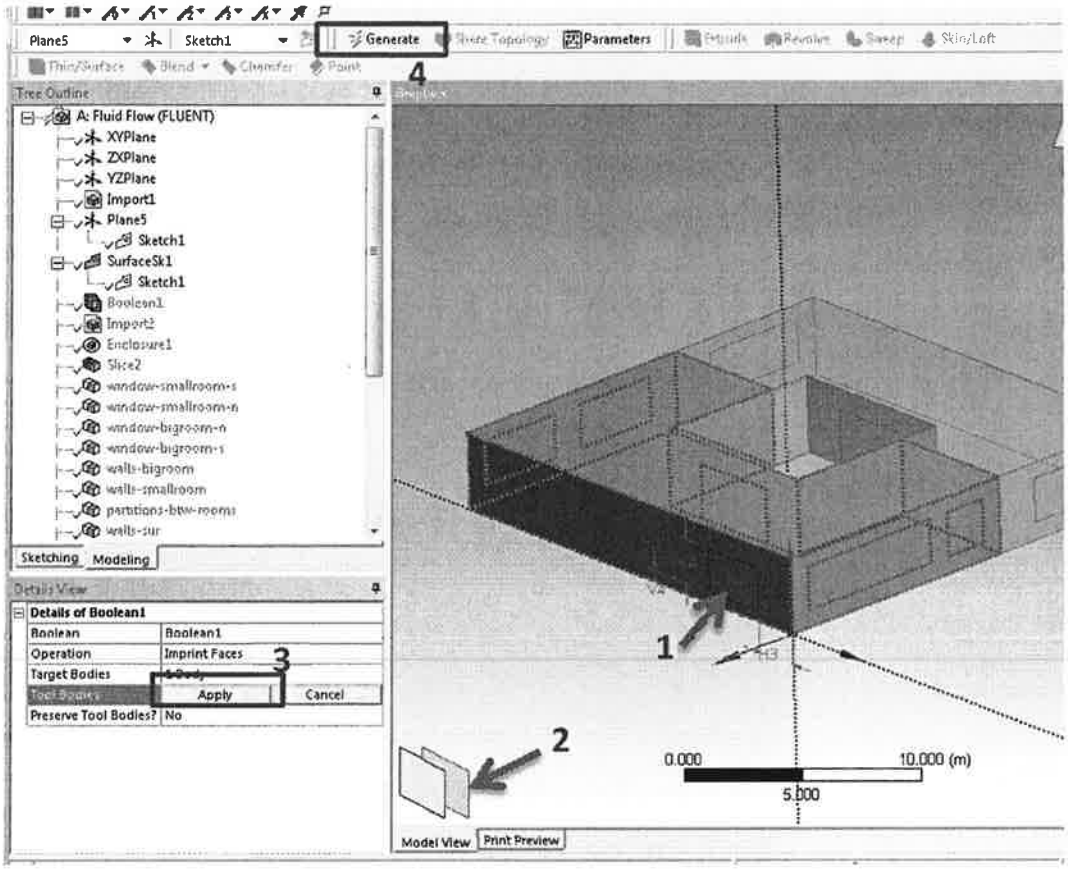
- Imprint the newly created surface body onto the "big-room" body.
  Right-click Feature "Import1" in **Tree Outline** side window to choose ***Insert -> Boolean***.



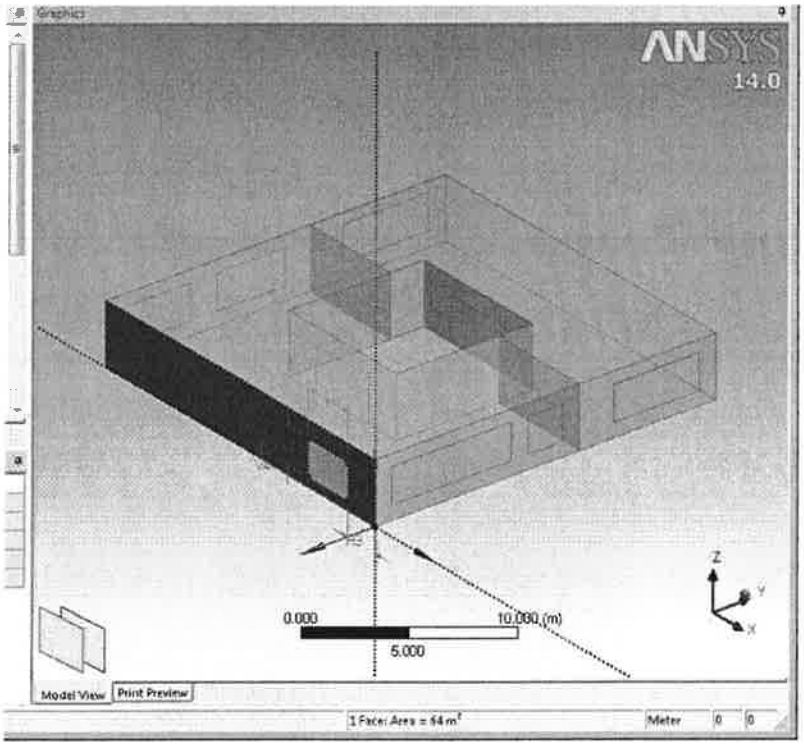
- Choose "**Imprint Faces**" Operation within the **Details View** side window at the left bottom first. left-click the "big-room" body in **Model View** screen and then click the "Not Selected" area on the right of "Target Bodies" in the **Details View** side window, and then click "**Apply**" button to select the body.



- Similarly, select the surface body as the Tool Bodies for the Boolean operation. Note that use the selection tool appearing at the left bottom of Model View Screen to choose the surface body by left-clicking the second tiny plane until it is in red. Click "**Generate**" button to complete the Boolean operation to split the newly created surface from its base face.

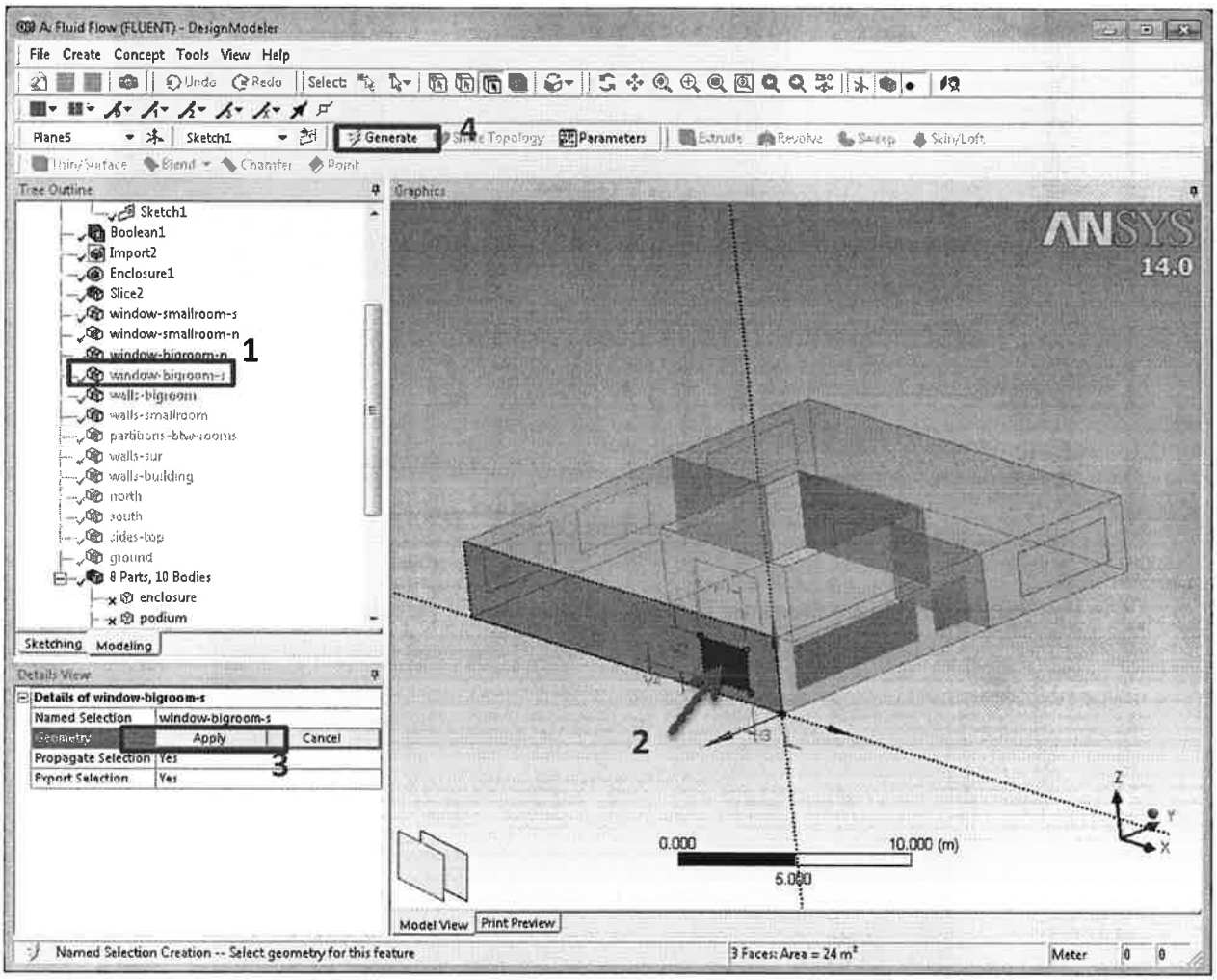


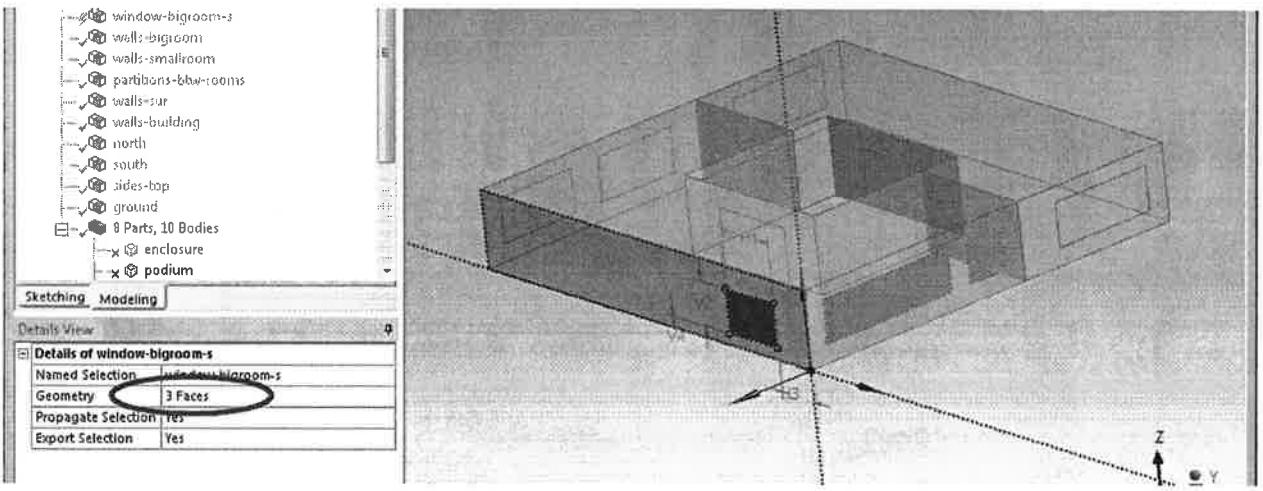
The final model should be the same as what is shown in the picture below.



A.3 Update the name selections

- Add the new window in the group of "window-bigroom-s" by right-clicking Feature "window-bigroom-s" in Tree Outline side Window. Hold *CTRL* key and then left click the newly created surface that reprsents the new window, and then *click* "**Apply**" Key to finalize the selection. ***Make sure that in total 3 faces are selected.*** Then, click "**Generate**" button to update the re-grouping.

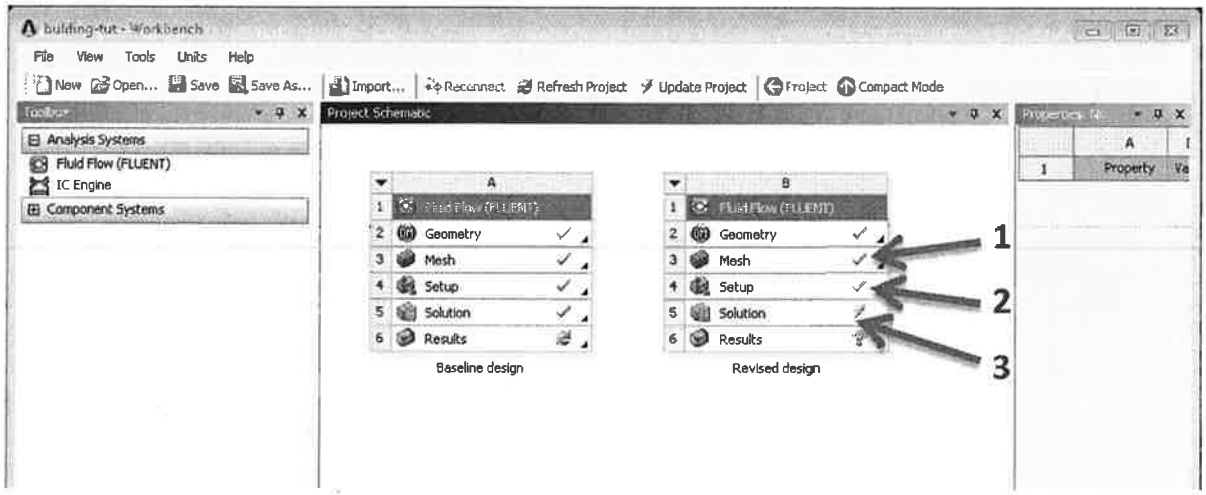




A.4 Save the project and then close Design Modeler.

A.5, update the Mesh, Setup and Solution, in sequence in the Workbench environment.

All the settings adopted in the model for baseline design will be automatically applied for the revised design. Please note that it will take some time to re-create the mesh, re-impose settings in FLUENT and re-simulate the case with the inclusion of one more window for Body "big-room".



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: 5pm, 1st March 2016

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.

Topic	Subtopic	Checks, advices
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
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
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

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)
Numerical	Discretization scheme	2nd order
Code		Latest version, updates, user forums
User		Training, experience, ask expert
Documentation		Full documentation of parameters, readable scale

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: Put all the files into the folder name and upload it to BCA IVLE 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)