Natural Ventilation, Thermal Comfort & Indoor Air Quality and Wind Driven Rain Analysis for Singapore Green Mark (GM2015) Buildings

Dr. Poh Hee Joo – Presenter
Chiu Pao-Hsiung, Venugopalan, Stephen Wan, Ooi Chin Chun, IHPC
Prof. Wong Nyuk Hien, Erna Tan, Osrithalita Gabriela, NUS
Hooff, T.A.J. van, Tu/E
Ken Po, Tan Phay Ping, BSD Pte Ltd
Leong-Kok Su Ming, Jeffrey Neng, Li Ruixin, BCA GM Department

20th Sep 2016

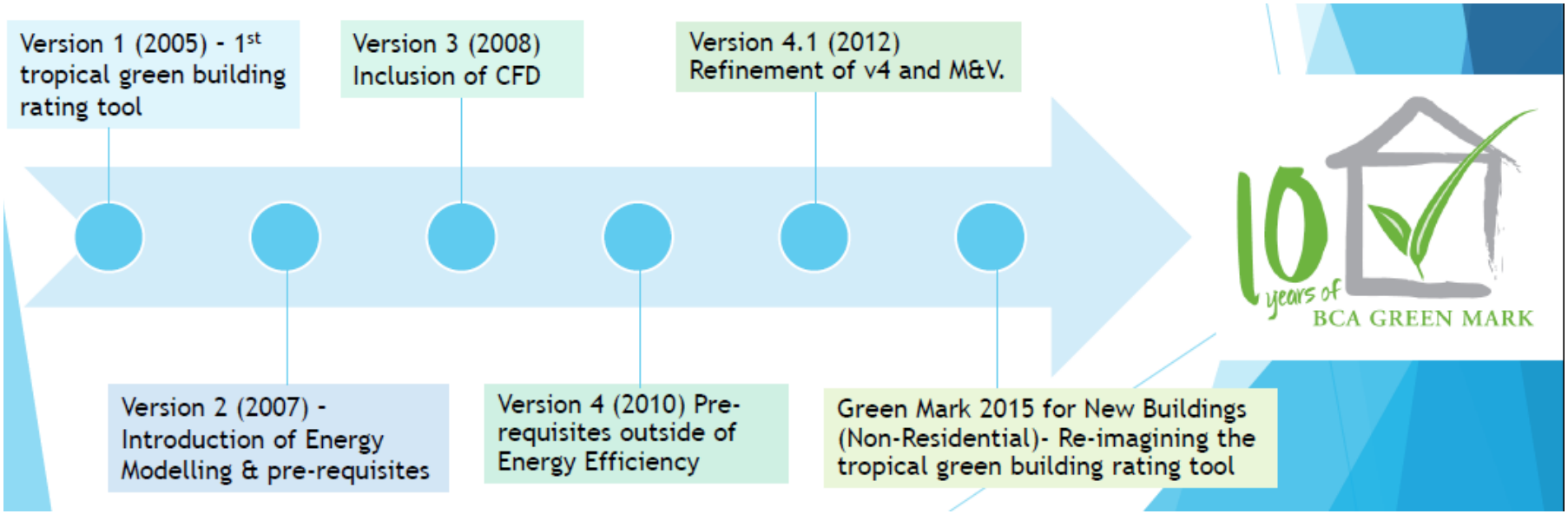
A*STAR IHPC

Content

- BCA Green Mark Certification Standard
- Natural Ventilation, Indoor Air Quality, & Thermal Comfort and Wind Driven Rain
- Test Case Study for GM2015 Non Residential Building Type
  - Industrial Facilities, Healthcare facilities, Commercial Atrium
- CFD Modelling for Tree Canopy, Louver & Fan
- Concluding Remarks

BCA Green Mark Certification Standard

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



Sharing with Permission from BCA

BCA Green Mark Certification Standard

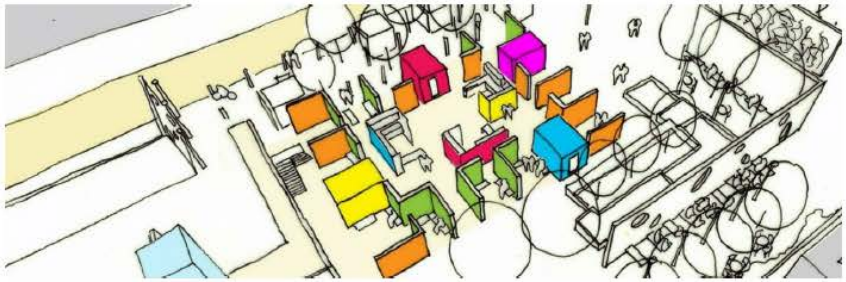
- Outcomes of Green Mark 2015

Climate Responsiveness



Conservation of Ecological Systems





Resource Stewardship

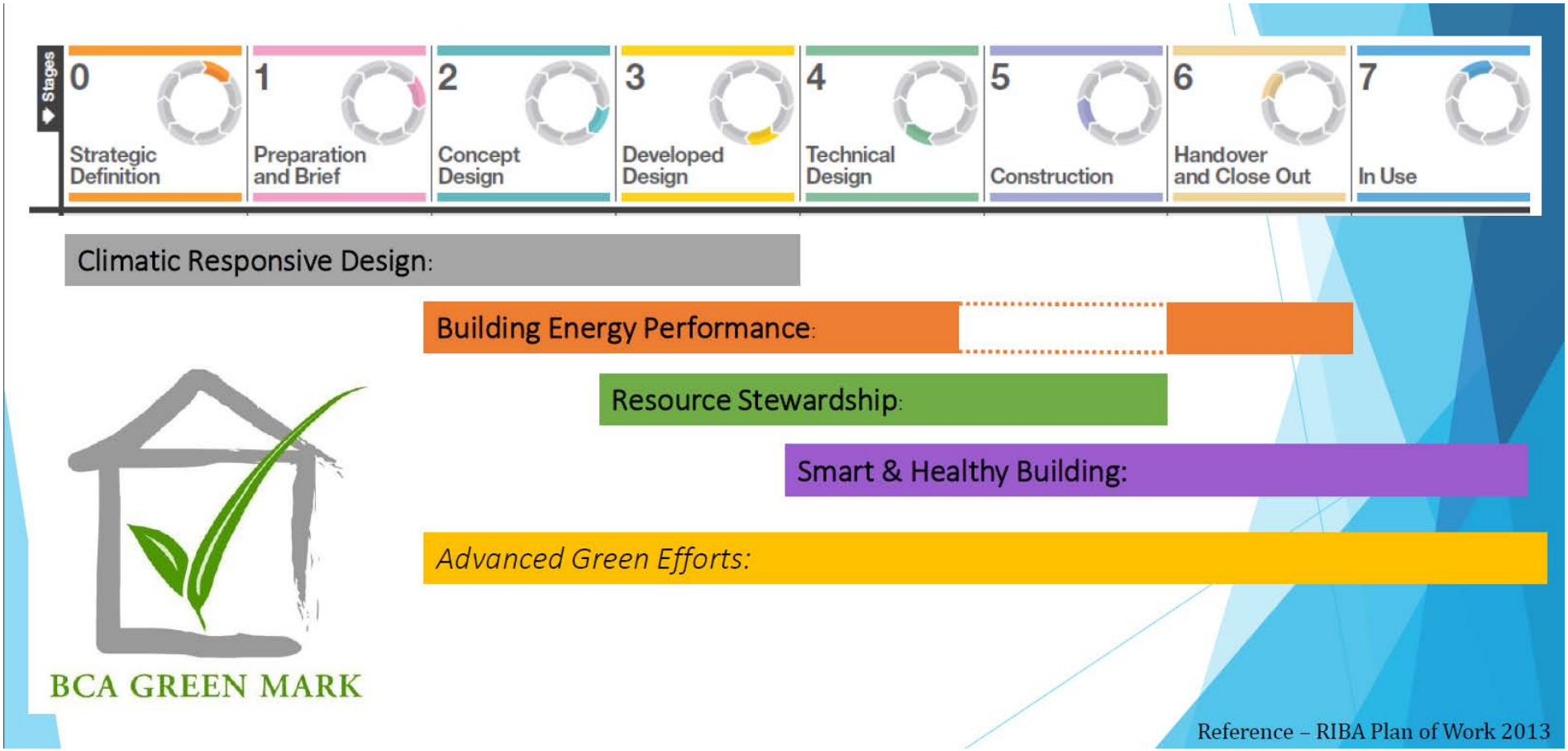


Health and Wellbeing

Sharing with Permission from BCA

BCA Green Mark Certification Standard

- Green Mark 2015 - Sections



Sharing with Permission from BCA

GM 2015 - 1.03c: Ventilation Simulation

NRB 1-4 Natural Ventilation / Mechanical Ventilation

(a) Natural Ventilation

Encourage building design that facilitates good natural ventilation.

(i) Proper design of building layout that utilises prevailing wind conditions to achieve adequate cross ventilation.

(ii) Use of ventilation simulation modeling and analysis or wind tunnel testing to identify the most effective building design and layout to ensure good natural ventilation.

Prerequisite Requirement :

Green Mark Platinum : Ventilation simulation modeling and analysis are to be carried out to ensure good natural ventilation with minimum weighted average wind velocity of 0.6 m/s within the functional spaces or units.

Enhanced: Simulation methodology more adaptive to NRB

Enhanced: Includes thermal comfort (PMV) or IAQ and the use of assisted ventilation in various circumstances

Enhanced: Bonus points for conducting Wind Driven Simulation

Sharing with Permission from BCA

Project Objectives for Natural Ventilation, Indoor Air Quality & Thermal Comfort, Wind Driven Rain

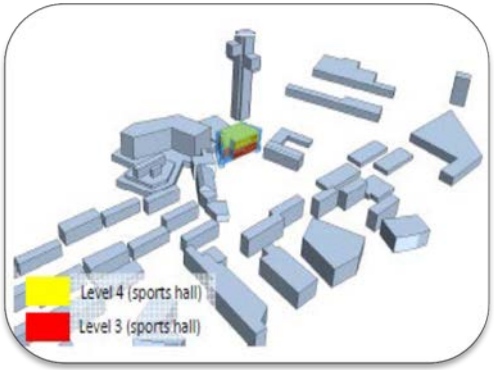
1. To develop new CFD simulation methodology and evaluation parameters to cover a holistic range of application in NRB including
   1. *hospitals, commercial atriums and industrial facilities – Phase 1*
   2. *schools, hawker centres, sports facilities – Phase 2*

   Improved parameters for urban airflows that are *suitable for project simulated in densely-built -up city areas* shall be established as well.
2. To develop passing criteria for an acceptable thermal environment in Non Residential Buildings (NRB). The alternative and more holistic evaluation methods shall cover both thermal comfort model and air quality index.
3. To develop Wind Driven Rain (WDR) simulation methodology based on Singapore's tropical rainforest climate and shall be established logically and economically feasible

Research Team

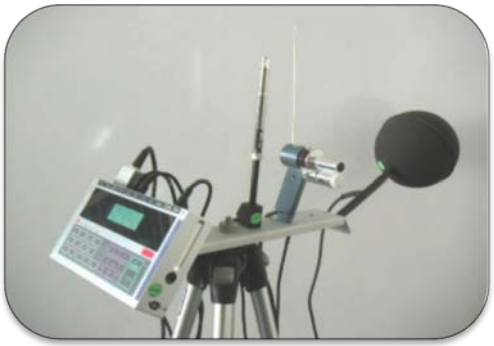
Project Manager

BCA – Dr. Li Ruixin



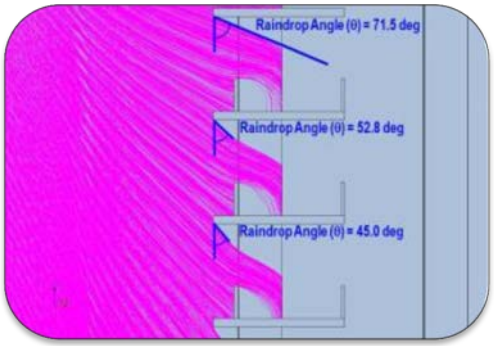
WP1- CFD Simulation

IHPC - Dr. Poh Hee Joo (PI)



WP2- Thermal Comfort & IAQ

NUS - Prof. Wong Nyuk Hien



WP3 – Wind Driven Rain

Tu/E + BSD -Prof. Bert Blocken

CFD Evaluation Criteria

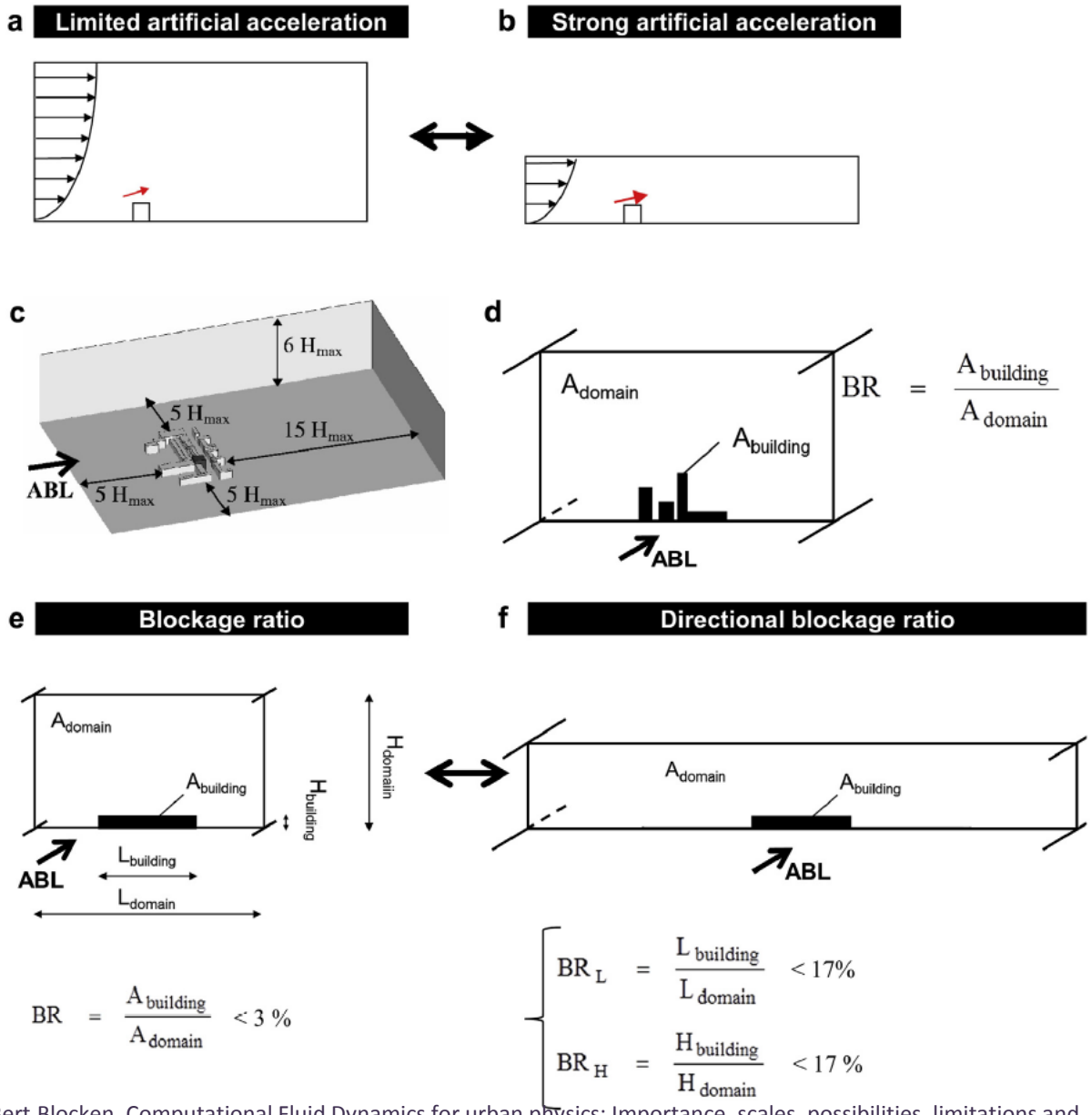
- CFD Parameter matrix evaluation
  1. Mesh independence studies
  2. Atmospheric Boundary Layer simulation
  3. Window modelling – any simplification?
  4. Mechanical fan modelling – simplified model?
  5. Heat source modeling (if any)
  6. Louver modeling (simplified model)
  7. Terrain modeling (resolution requirement)
  8. Surface roughness effect on estate airflow
  9. Baseline vs. Improved design comparison
  10. Framework for architect & CFD modeler interaction
  11. Surrounding buildings info to be obtained with reasonable accuracy
  12. Performance indicator – ACH instead of average velocity

Review on Existing GM Version 4.1

- Development of New methodology through test case study

Parameters	Current Methodology	Proposed Methodology for NRB
Computational grid size	• Units: 0.1-0.2m • Buildings and ground level: 0.5-1.0m • Far field boundary: 10m with min. 50m away from ground.	• Determine an appropriate grid size to optimize computing resources and turnaround time. • Boundary layer/inflation layer elements are advisable to be used in "units"
Domain size	• Min. three times the length of the longest distance across development boundary.	• Propose blockage area proposal
Roughness factor setting	Nil	• Consideration to model trees/ greenery/ terrain/ topographical features. • Determine/ quantify surface roughness factor.
Component/ device modeling	Nil	• More precise descriptions of boundary conditions for louvers, heat source etc.
Selection of unit	• Mid-height level of development; but not applicable for NRB	• Natural ventilated spaces in NRB may not start from the bottom; to consider all unit at particular level. • To determine an appropriate level(s).
Pressure difference selection point	• Midpoint of opening measured at 0.5m away from surface area; but not applicable for NRB	• Pressure difference taken only at midpoint is not representative. • To consider pressure differential over the entire surface.
Selection of units to determine area-weighted average velocity	• Five typical unit design layout for residential; but not applicable for NRB	• Determine appropriate area-weighted average velocity method for industrial facilities.

CFD Computational Domain for NV

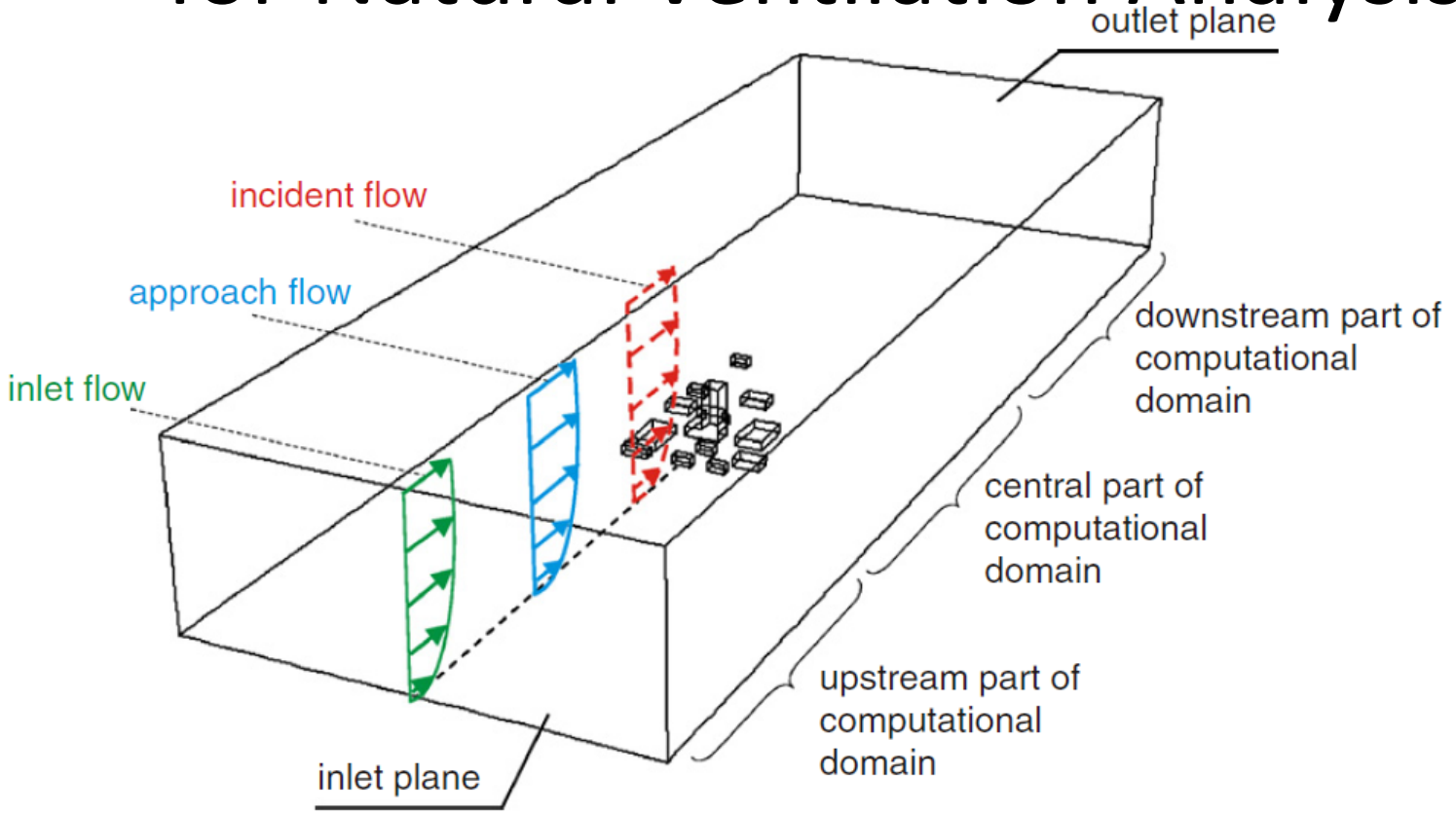


1. Minimum distance between urban/building models and boundary of domain
2. Maximum allowable blockage ratio
3. Combination of 1 & 2

- The surrounding buildings reside within 500 m distance stream wise and span wise from the edge of development of interest should be modelled explicitly

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

CFD Atmospheric Boundary Layer setting for Natural Ventilation Analysis



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

Checklist for Ventilation Simulation for NRB GM 2015

- Objective:

1. To maintain CFD quality control for GM submission
2. To keep track of NRB Nat. Vent. Design Progress
3. To assist BCA GM administrative process (e.g. submission, revision, iterations)

Checklist for Ventilation Simulation (Brief)

Checklist Items	Description/Selection	Response & Criteria
1. Submission Details	Client, architect, ESD consultants	Info on NV design
2. Building Type	industrial facilities etc	Describe building functionality
3. Problem Statement	Objective & Work Scope	Describe NV challenges, proposed solution, desired outcome and work scopes from the simulation model.
4. Site Information	3 times development length	Surrounding buildings, terrain, greenery
5. CFD Approach	Simulation Methodology	Choose well validated software with latest version,
5. CFD Approach	Assumption & Simplification	Describe simulation model assumption, limitations and geometrical simplification
6. CFD Domain	Computational Domain	Criteria on extent of CFD domain - blockage ratio and domain extension
7. CFD Meshing	Mesh independence	Mesh independence study at the functional unit NV space
7. CFD Meshing	Mesh size, distribution and quality	Proper domain decomposition. To use hexahedral cells if possible; 0.1 to 0.2 m within the functional space of interest

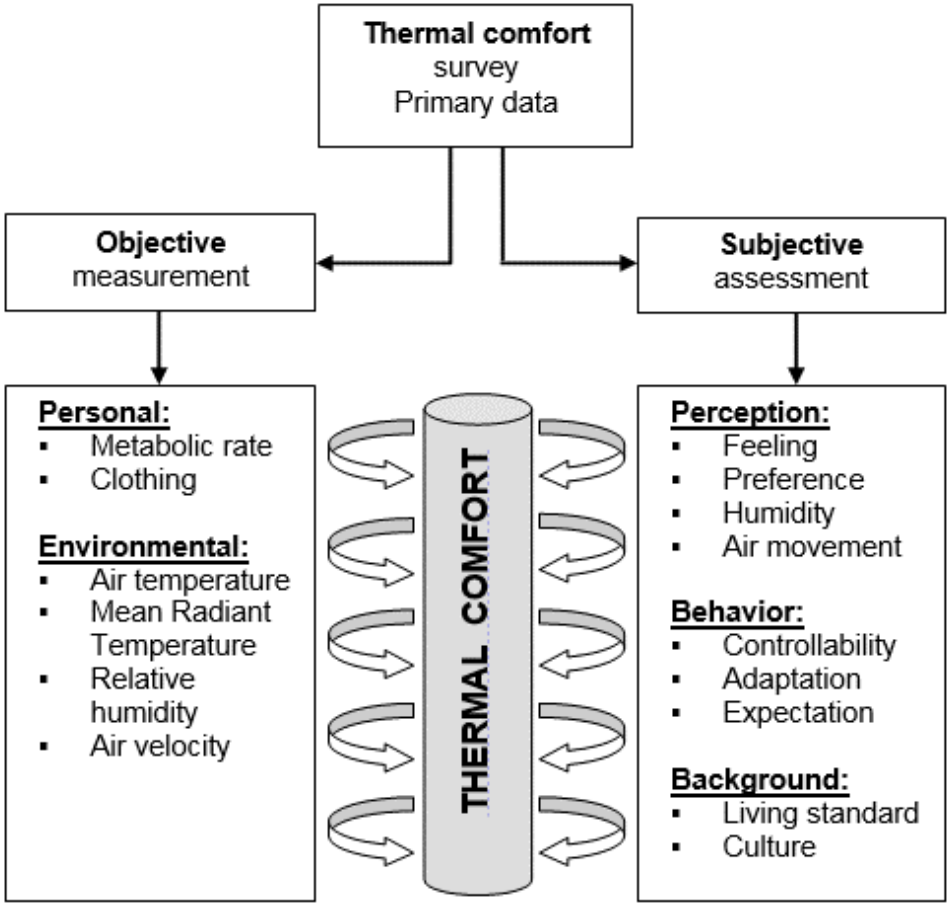
Checklist for Ventilation Simulation

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

Checklist for Ventilation Simulation

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

Thermal Comfort Methodology | Objective And Subjective Assessment



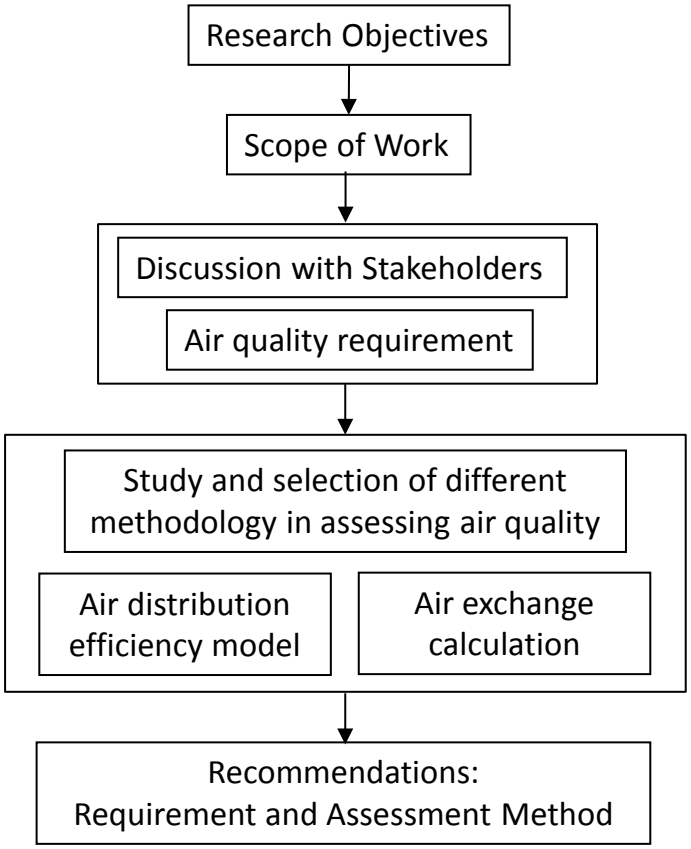
Courtesy from Prof. Wong Nyuk Hien, NUS

TC Methodology | Information from questionnaire form

Section	Topic of investigation	Description required
A	General Information	▪ Date and time the survey is taken ▪ Location
B	Personal	▪ Gender, race, age, height, weight ▪ Activities, clothing
C	Environment	▪ Temperature, Humidity, Wind speed, Mean RT ▪ Weather condition
D	Thermal perception	▪ Thermal sensation (ASHRAE scale) ▪ Preference to be warmer/cooler (McIntyre scale) ▪ Air freshness ▪ Perception on room humidity ▪ Perception on room air flow ▪ Thermal comfort (Bedford scale)
E	Control	▪ Window opening ▪ Usage of fan ▪ Usage of Air conditioner
F	Behavior	▪ Thermal comfort regulation ▪ Adaptive actions
G	Preference	▪ Preference on most comfortable room ▪ Utilize natural ventilation for various activities

Courtesy from Prof. Wong Nyuk Hien, NUS

Research Methodology | Air Quality



Courtesy from Prof. Wong Nyuk Hien, NUS

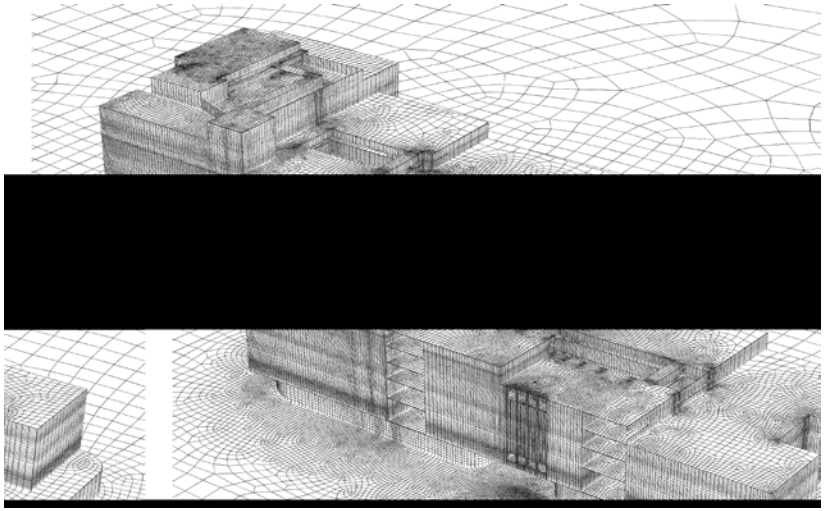
Air Exchange Effectiveness

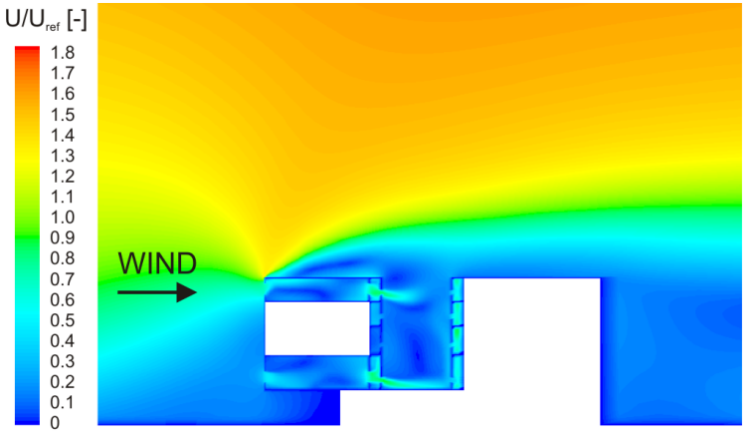
Comparison of Air Exchange Efficiency and Contaminant Removal Effectiveness as IAQ Indices(Novoselac & Srebric, 2003)

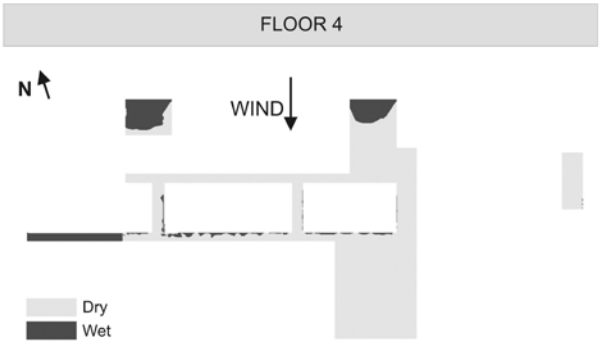
AEE for Characteristic Room Ventilation Flow Types		
Flow pattern	AEE	Comparison with the average time of exchange
Unidirectional flow	1.0 - 2.0	$\tau_n < \tau_{exe} < 2\tau_n$
Perfect mixing	1.0	$\tau_{exe} = 2\tau_n$
Short Circuiting	0 - 1.0	$\tau_{exe} > 2\tau_n$

Limits for AEE		
Upper limit	2.0	Ideal piston flow
Perfect mixing	1.0	Complete and instantaneous mixing
Lower limit	0	Bypass area and recirculation area are completely separated

Work Package 3 – Tu/E (T. van Hooff, B. Blocken)







- Wind flow pattern
  - Computational settings
    - Computational domain according to Franke et al. (2007) and Tominaga et al. (2008)
    - Realizable k-ε turbulence model (Shih et al. 1995)
    - Standard wall functions
    - Second-order discretization schemes
    - Inlet profiles based on Richards and Hoxey (1993)
    - Ground roughness based on Blocken et al. (2007)
    - Number of iterations before convergence was reached: 20,000

- WDR simulations
  - Terminal velocity of fall
  - Drag coefficients by Gunn and Kinzer (1949)
  - Four drop diameters: 0.5 mm, 1 mm, 2 mm, 5 mm
  - Determination of wetted area

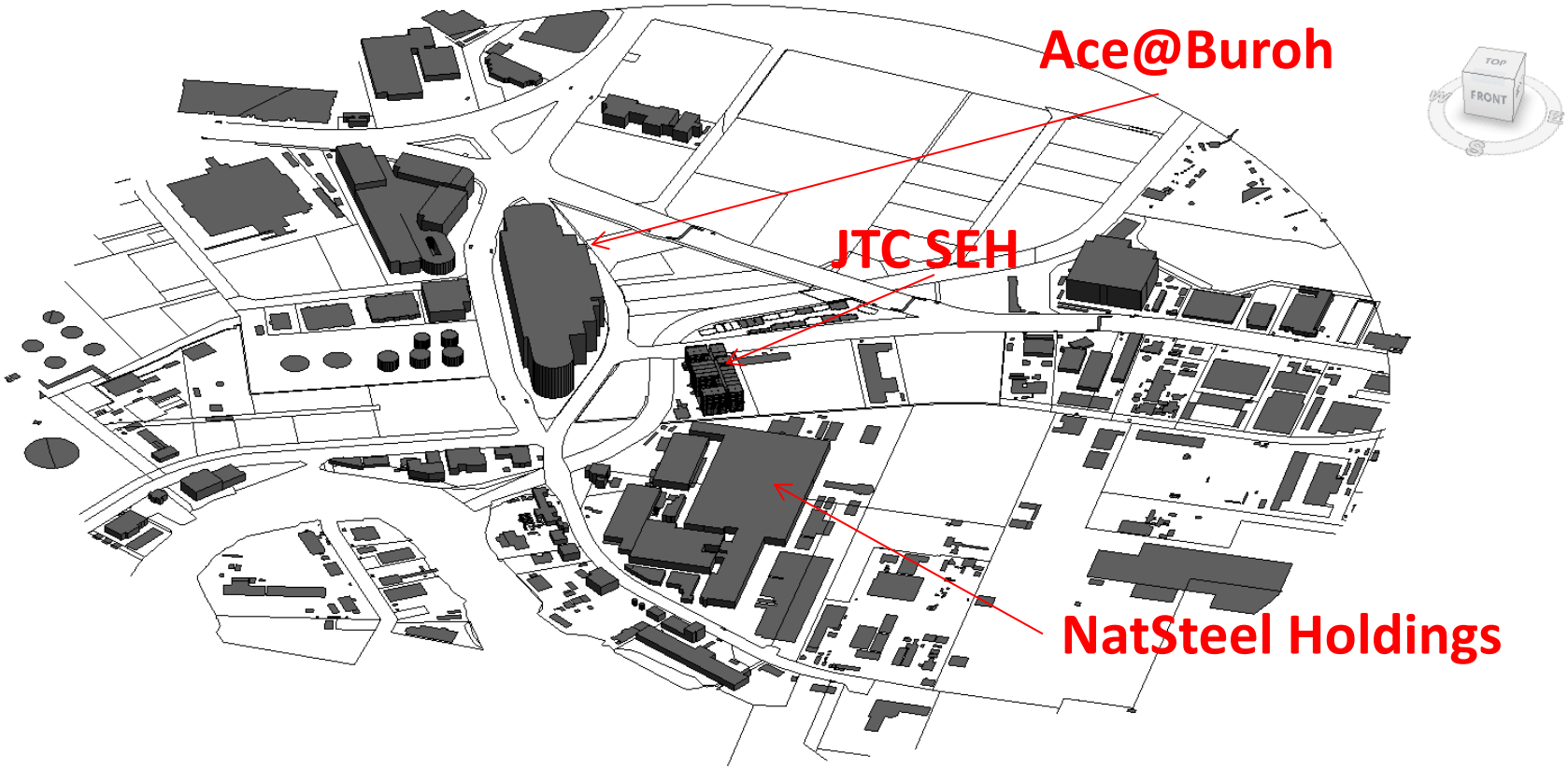
Courtesy from Prof. Bert Blocken, Tu/E

Test Case Study for GM2015 Non Residential Building Type

1. Industrial Facilities
    1. New Generation Factories - JTC Surface Engineering Hub @ Tanjong Kling – CFD NV
    2. New Generation Factories - JTC BioMed One @ Tuas – CFD WDR
    3. Standard Factories - 20 Woodlands Link
    4. Standard Factories - 22 Woodlands Link
    5. Flatted Factories - 26, Changi North Way
    6. Flatted Factories - JTC Space One @ Tanjong Kling
2. Healthcare facilities
    1. Khoo Teck Puat Hospital (KTPH)
    2. The Integrated Building, Changi General Hospital (CGH) – CFD NV & WDR
3. Commercial Atrium
    1. Plaza Atrium, National Library Building (NLB)
    2. Ci Yuan Community Club Atrium
    3. Atrium for Solaris Buildings – CFD NV & WDR

CFD methodology development for NV Design of GM 2015 NRB Buildings

- Test Case Building - JTC Surface Engineering Hub @ Tanjong Kling



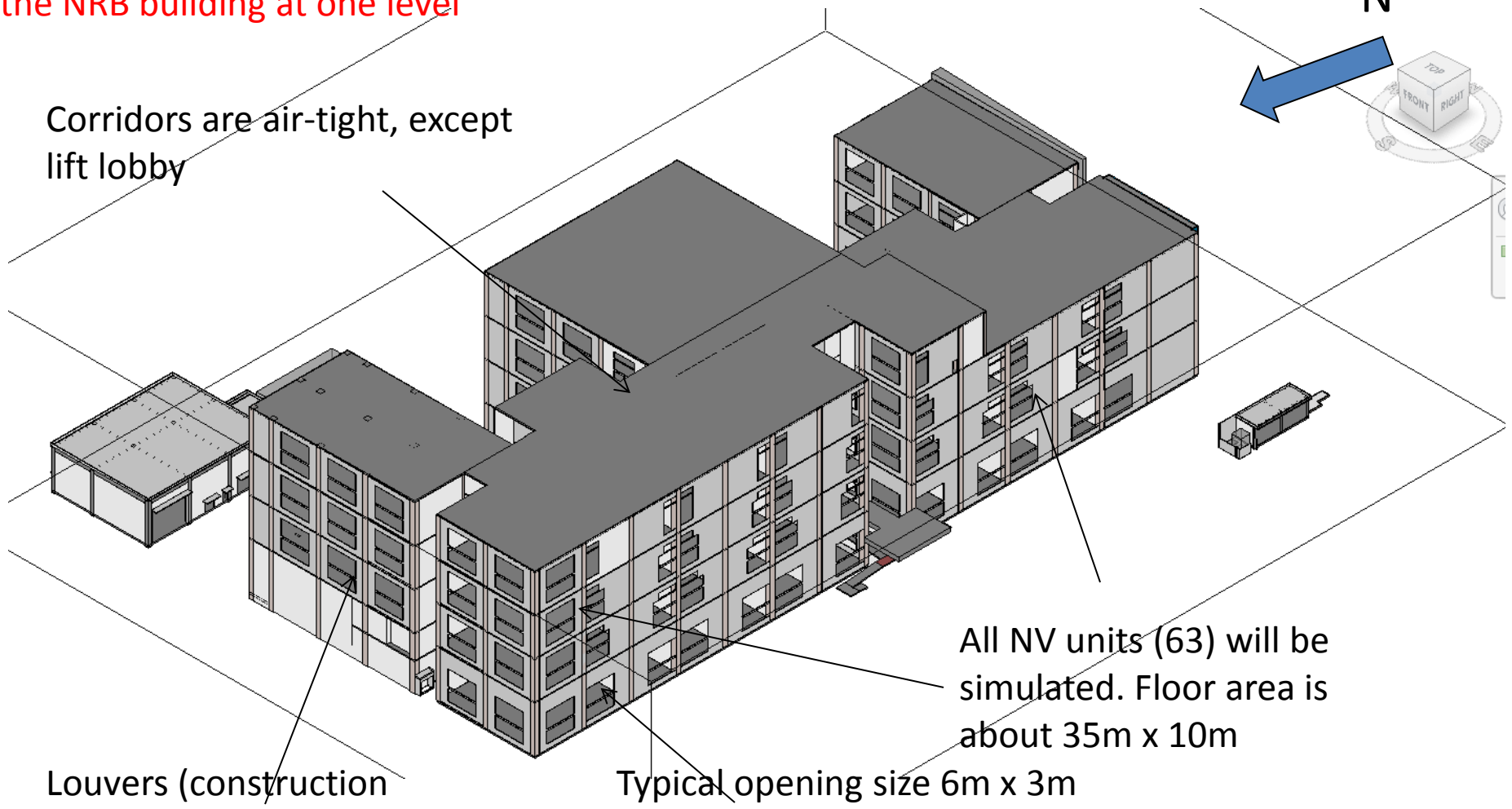
Sketch up drawing of surrounding buildings from JTC

Geometry Simplification on NRB

To model ALL functional units of the NRB building at one level

Level 2 – 4 is assumed closed for this CFD study

N



Corridors are air-tight, except lift lobby

All NV units (63) will be simulated. Floor area is about 35m x 10m

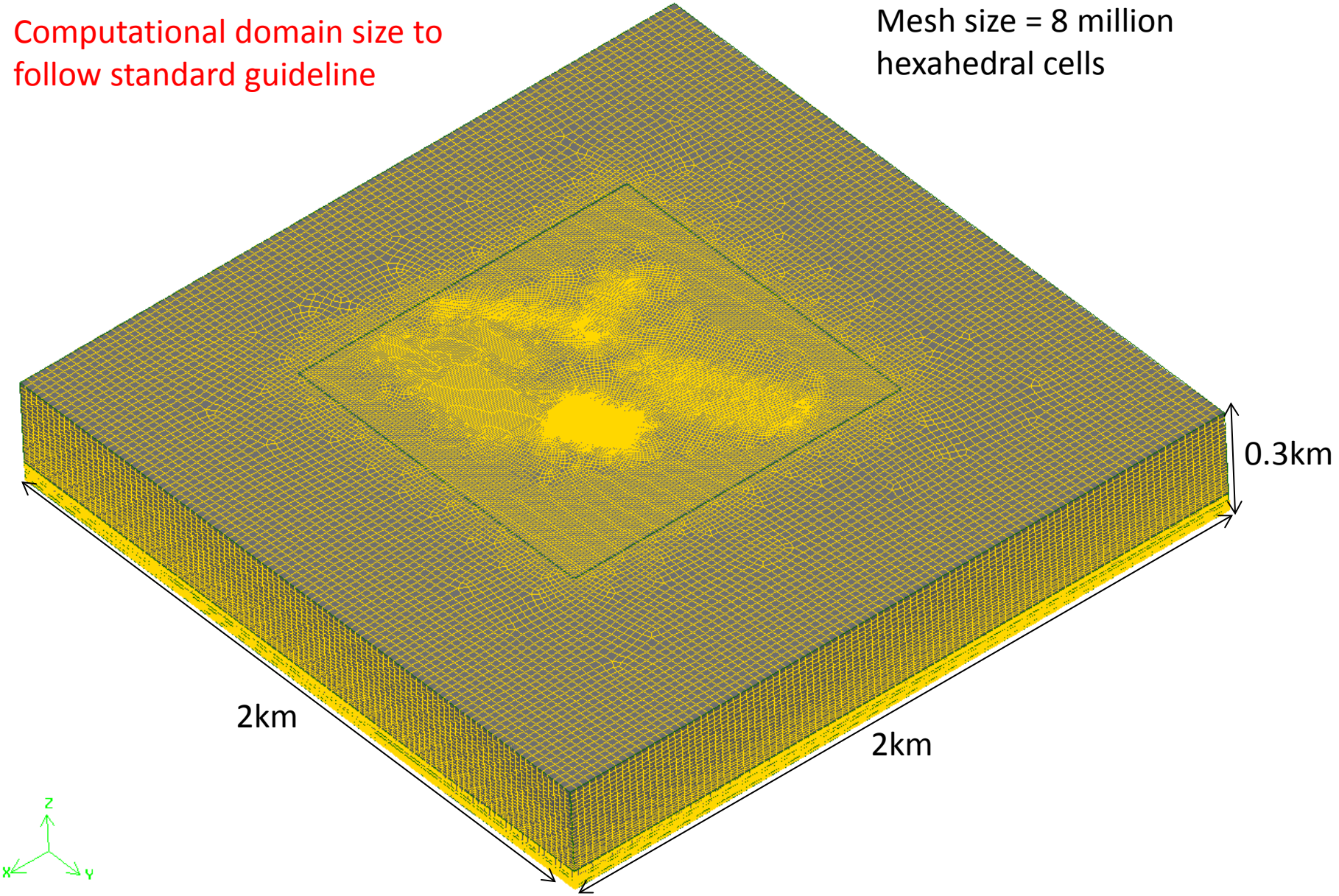
Louvers (construction drawings to be given)

Typical opening size 6m x 3m (approximate)

Computational Domain Decomposition & Meshing

Computational domain size to follow standard guideline

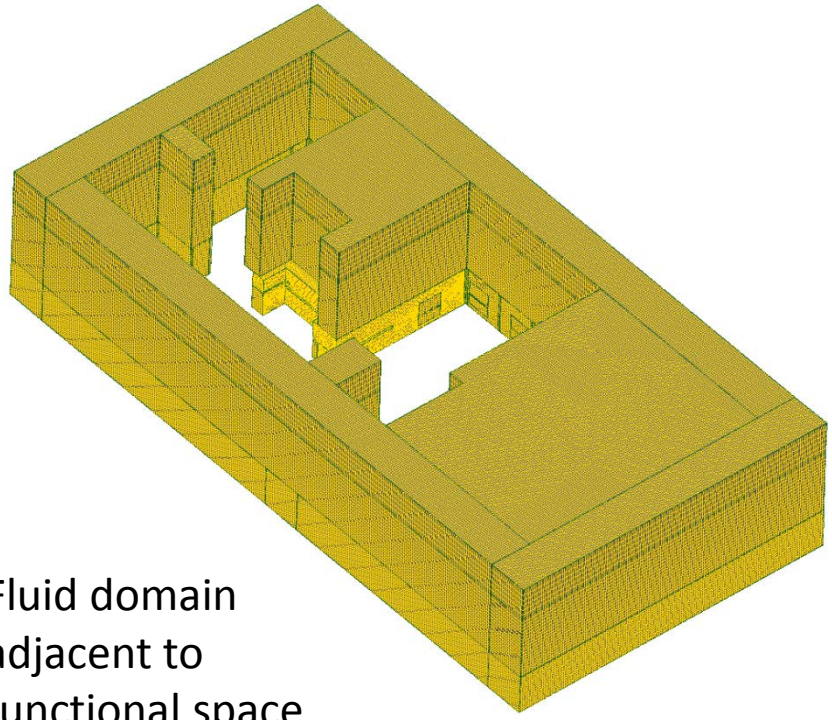
Mesh size = 8 million hexahedral cells



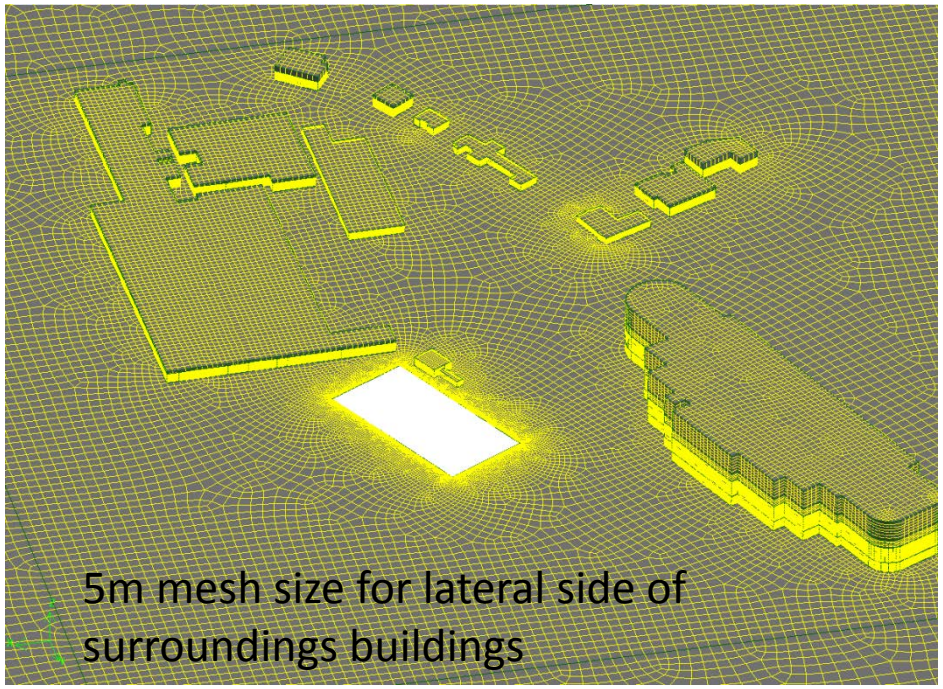
Computational Domain Decomposition & Meshing

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

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



Fluid domain adjacent to functional space



5m mesh size for lateral side of surroundings buildings

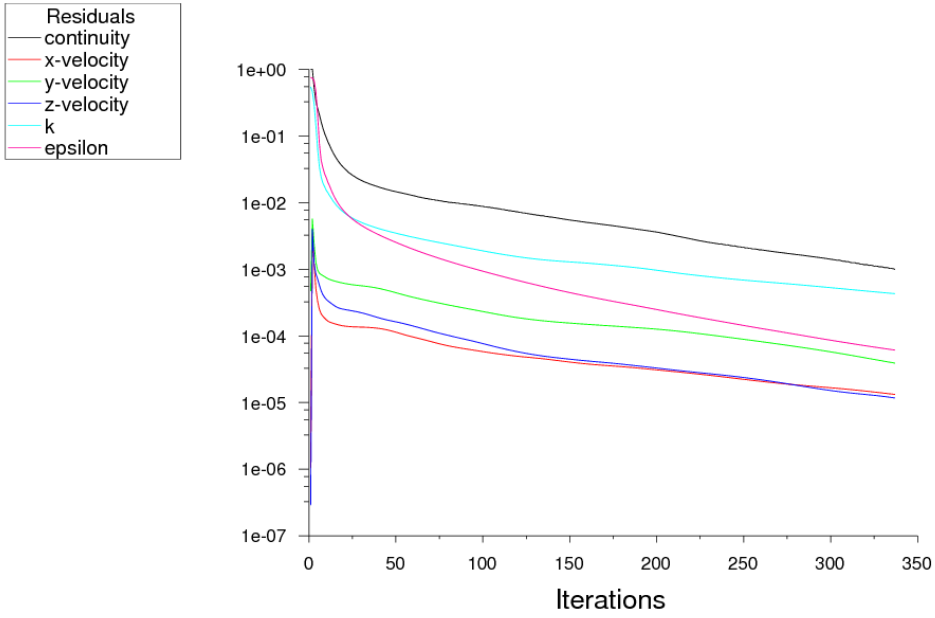
Simulation Results – Convergence Criteria

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

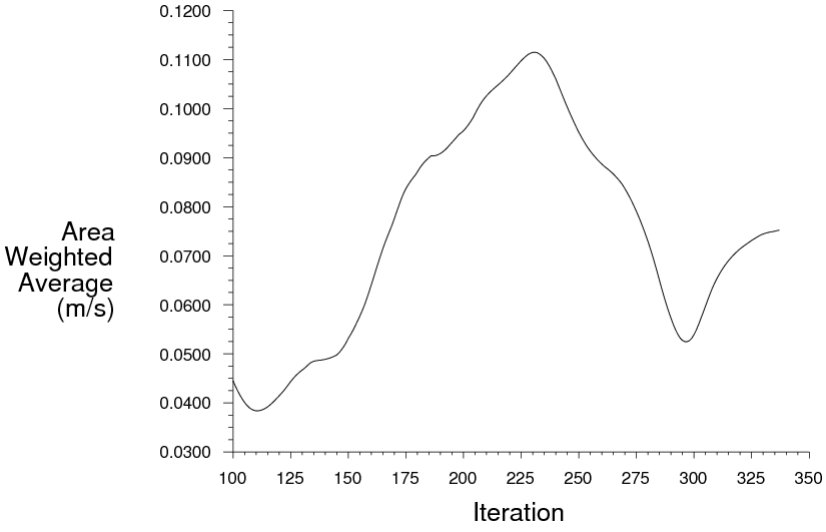
Convergence is reached when:

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

Residual, converged after 340 iterations



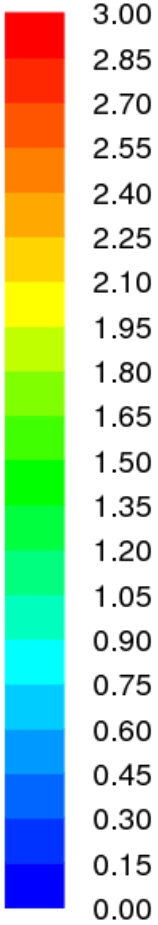
However, monitor point value is still fluctuating



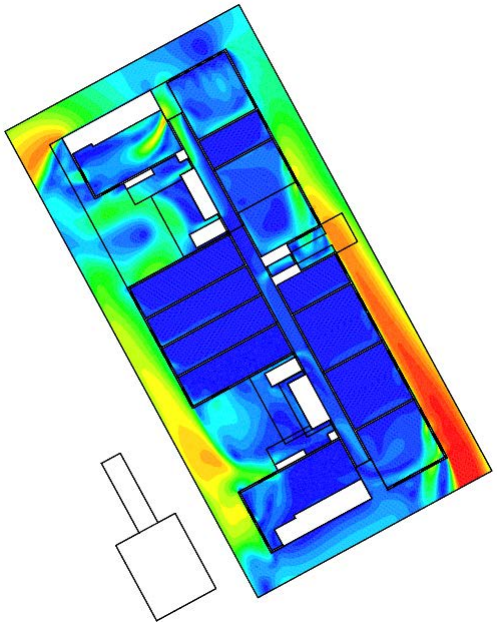
Simulation Results – Velocity @ 1.5m height

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

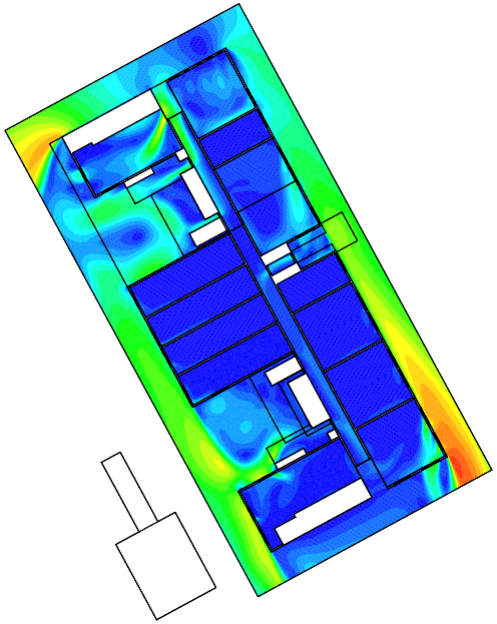
- Proper convergence criteria (e.g. residual setting & monitor quantity of interest) is crucial to ensure good accuracy results
- How about second order discretization scheme?



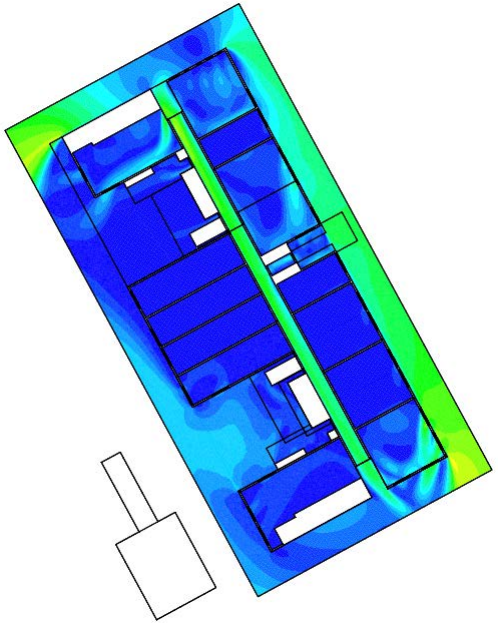
"Artificial" converged solution after 340 iterations, with convergence criteria for continuity set as 1e-3



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



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

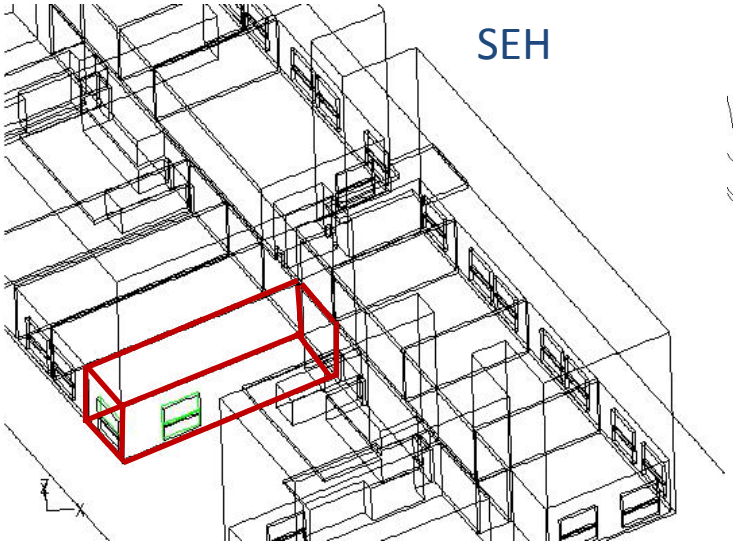


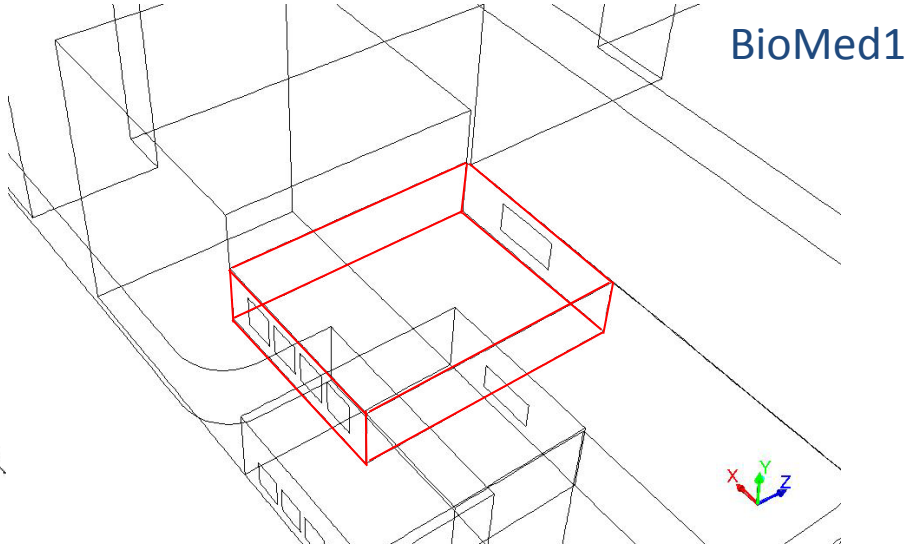
CFD simulation on Air Exchange Effectiveness

- For flow equations with turbulence model
  - incompressible Reynolds-averaged Navier–Stokes (RANS) equations
  - realizable $k - \varepsilon$ model
- For 2-species equations (air and CO_2).
  - incompressible ideal gas (IIS) Density model
  - Mass diffusivity is set as 1.6e-5 m^2/s (carbon dioxide in air)
- The ambient temperature is set as 33^oC

CFD simulation on Air Exchange Effectiveness

- Two building to be Investigated.
  - SEH
  - BioMed1
- CO_2 is uniformly distributed in the investigated unit as 1kg/s.





Air Exchange Effectiveness(AEE)

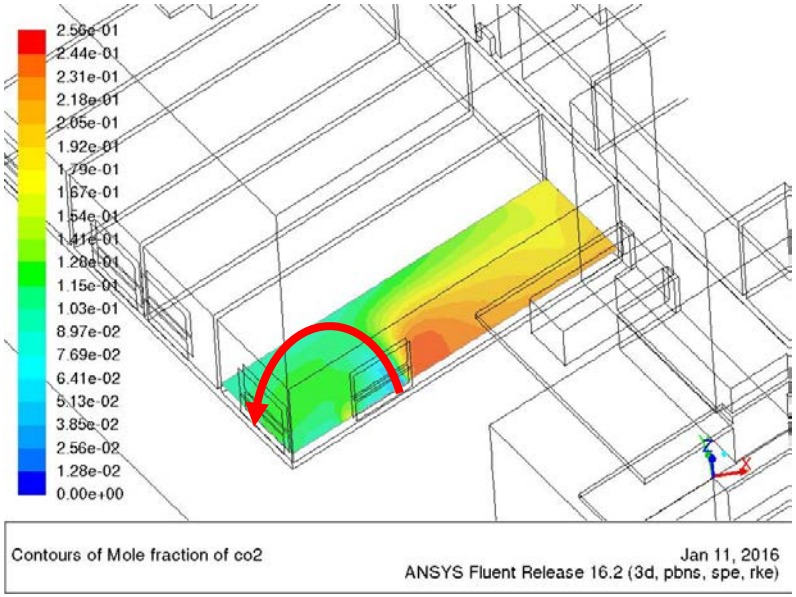
- AEE represents the ratio between shortest possible time needed for replacing the air in the room (τ_n) and the the average of local values of age of air $(\langle\tau\rangle)$

$$\text{AEE}=\frac{\tau_n}{\langle\tau\rangle}=\frac{C_{out}Vol/Cs}{C_{H=1.2}Vol/Cs}=\frac{C_{out}}{C_{H=1.2}}$$

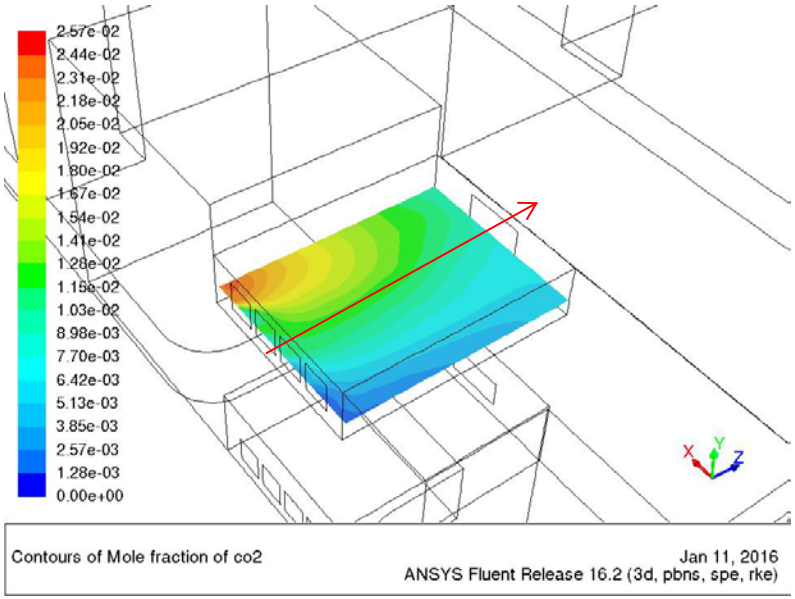
- C_{out} is area-averaged CO2 concentration at outlets, $C_{H=1.2}$ is area-averaged CO2 concentration at 1.2m height, Vol is volume of investigated unit, and Cs is the pollution source.

CFD - AEE results

AEE				
building	convergence tolerances	Density model	Mass diffusivity	AEE
SEH	1E-4	IIS	1.6e-5 m^2/s	1.08
BioMed1	1E-4	IIS	1.6e-5 m^2/s	1.8



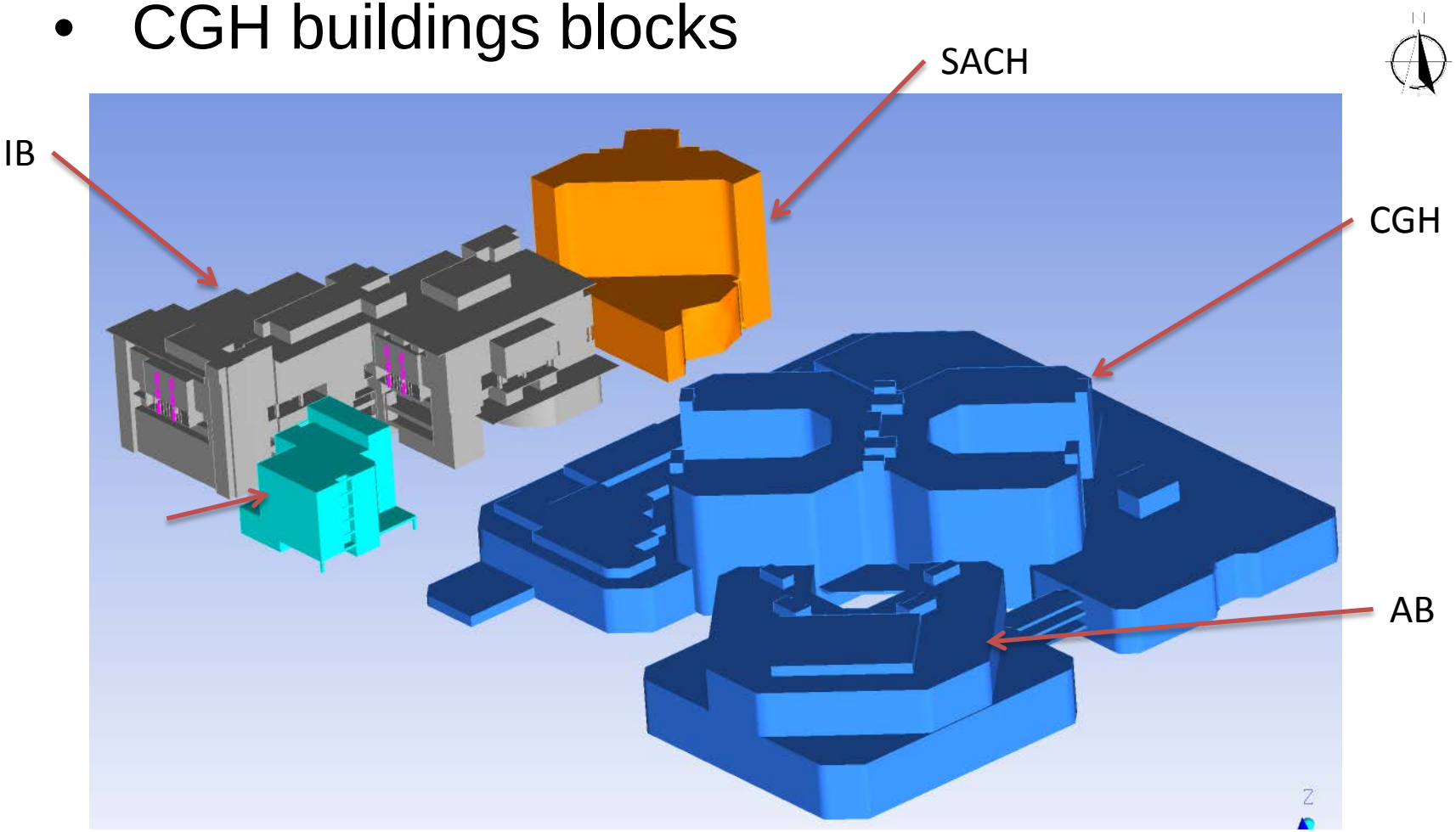
SEH, AEE = 1.08



BioMed1, AEE = 1.8

CFD NV simulation for Healthcare Facilities - CGH-IB, GM2015 methodology and passing criteria – meshing stage (work towards 2nd order discretization)

- CGH buildings blocks



Courtesy from Parson Brinkerhoff

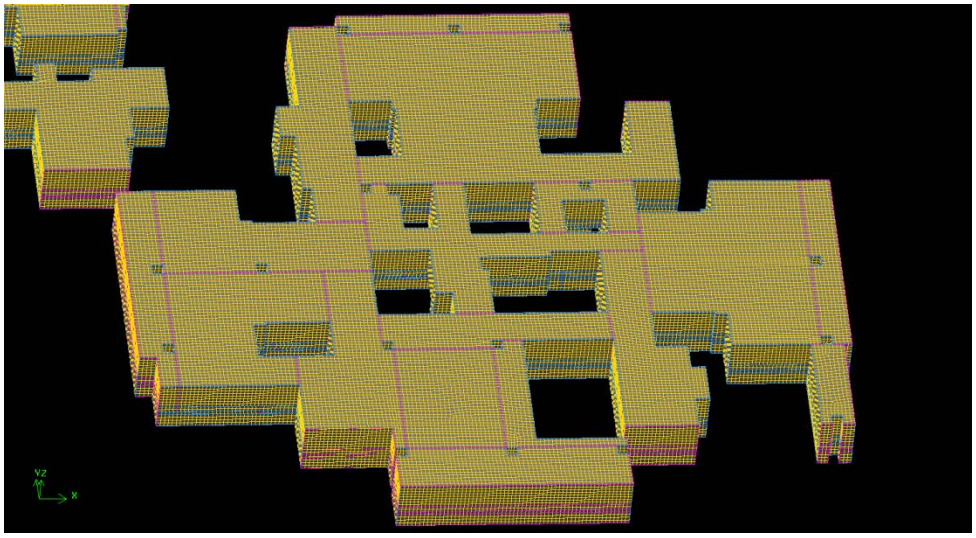
CFD NV simulation for Healthcare Facilities
CGH-IB, GM2015 methodology and passing criteria

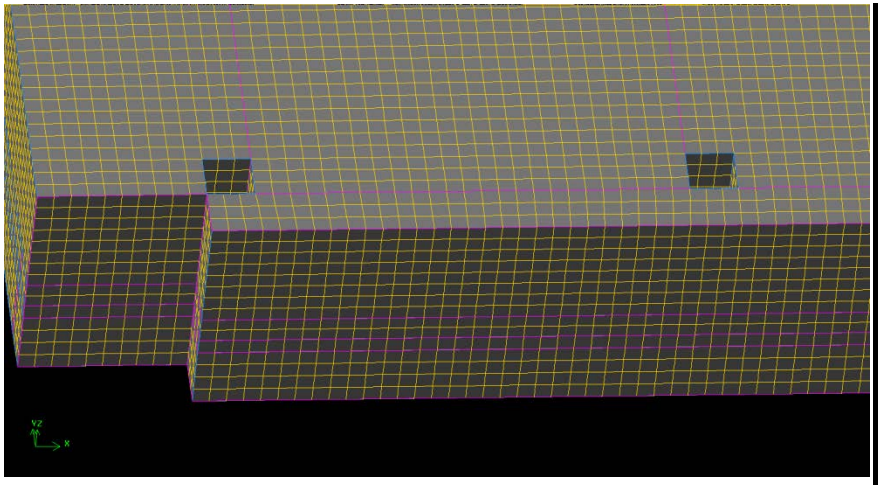
- To use hexahedral cells in the rectangular domain of NV space.
- Tetrahedral cells can be used to model the surrounding site features of the NV space.
- Hybrid pyramid or cut-cell mesh can be adopted at the interface.

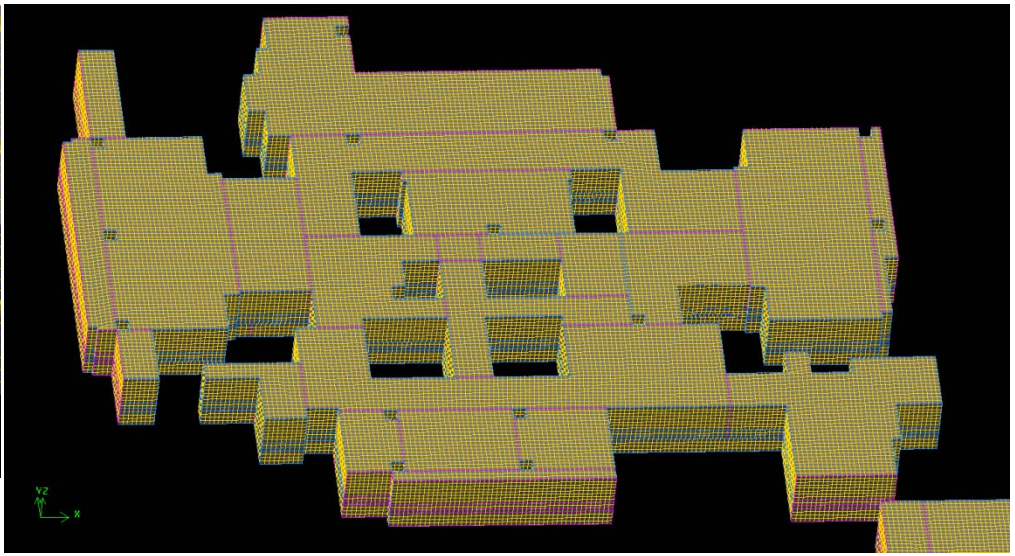
Location	Grid Size (m)
Within the functional spaces of interest	0.1-0.5
Building of interest	0.5-1.0
Surrounding building	1.0-5.0
From ground surface to 10m height in vertical direction	0.5 – 1.0
Ten to – H_{max} height vertical direction, (H_{max} is the height of the tallest building among the group of building modelled explicitly)	1.0 – 5.0

CFD NV simulation for Healthcare Facilities
CGH-IB, GM2015 methodology and passing criteria

Further creating high-quality mesh inside the functional unit







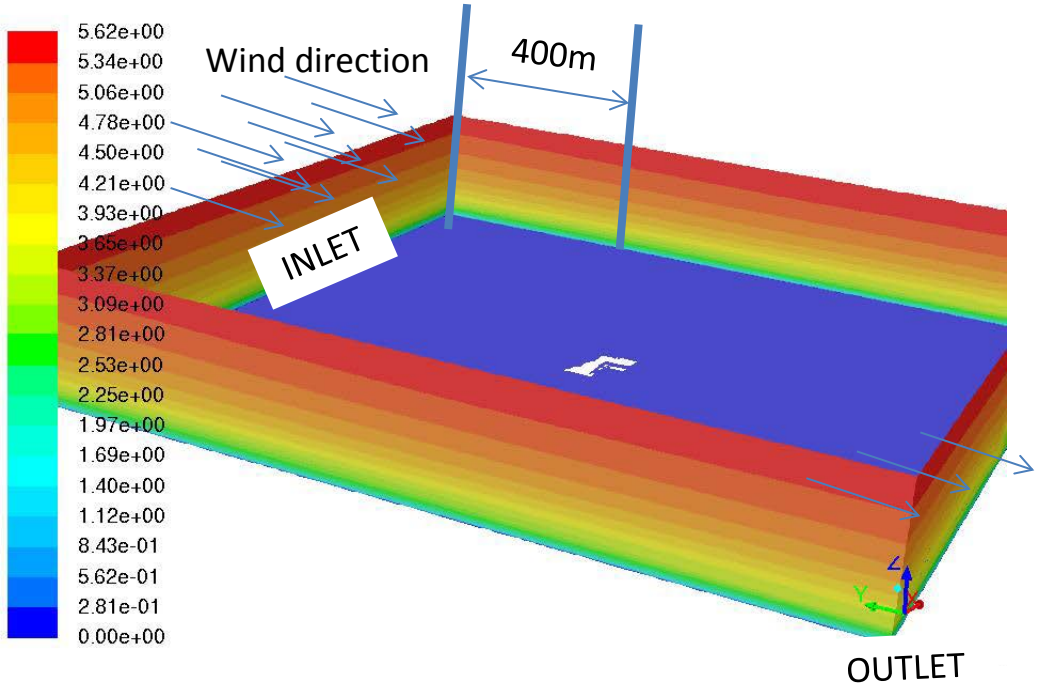
CFD NV simulation for Healthcare Facilities
CGH-IB, GM2015 methodology and passing criteria

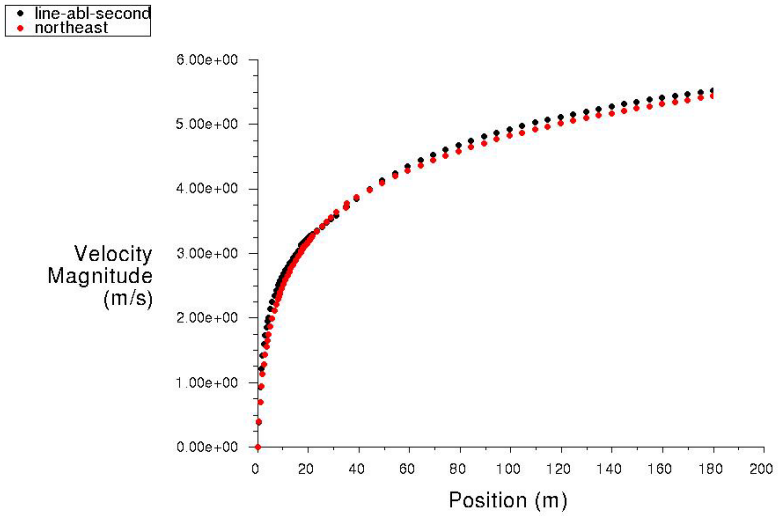
Simulation details

- Three dimensional RANS with realizable k-e model and standard wall function.
- Use customized function to implement the wind profile and corresponding turbulence quantities.
- Second-order scheme is used for momentum equations and turbulence equations.
- Convergence criteria is set as 1E-5 for all variables.
- SIMPLEC is used for velocity-pressure coupling
- Hybrid initialization is suggested to be used.

CFD NV simulation for Healthcare Facilities
CGH-IB, GM2015 methodology and passing criteria

Atmospheric Boundary Layer - Horizontal Homogeneity





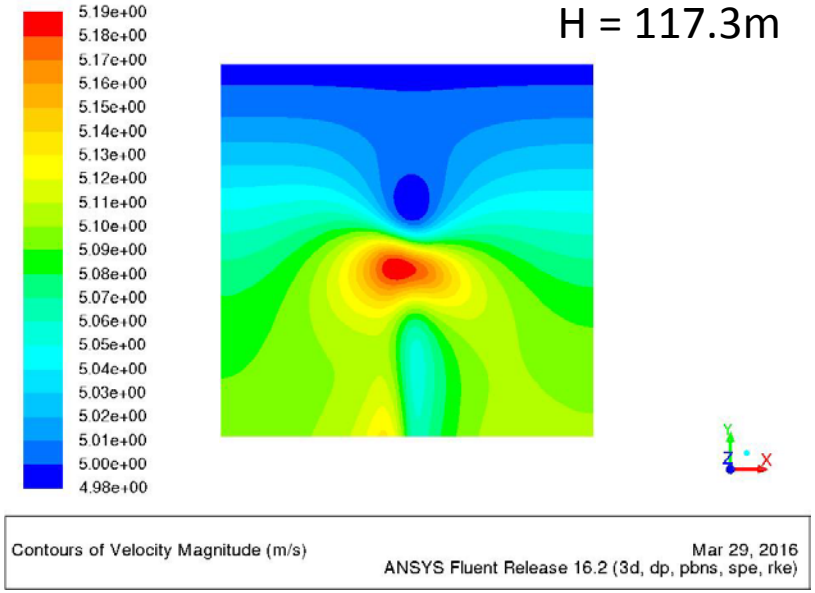
GM4.1 vs. GM2015 – Mesh Type & Blockage Ratio

	GM4.1	GM2015
Mesh size for functional unit	0.1~0.2m	0.1~0.5m
Vertical length	50m or 3 times	Blockage ratio < 17%
Horizontal length	3 times	Blockage ratio < 17%
Mesh type	No restriction	To use hexahedral cells in the rectangular domain of NV space

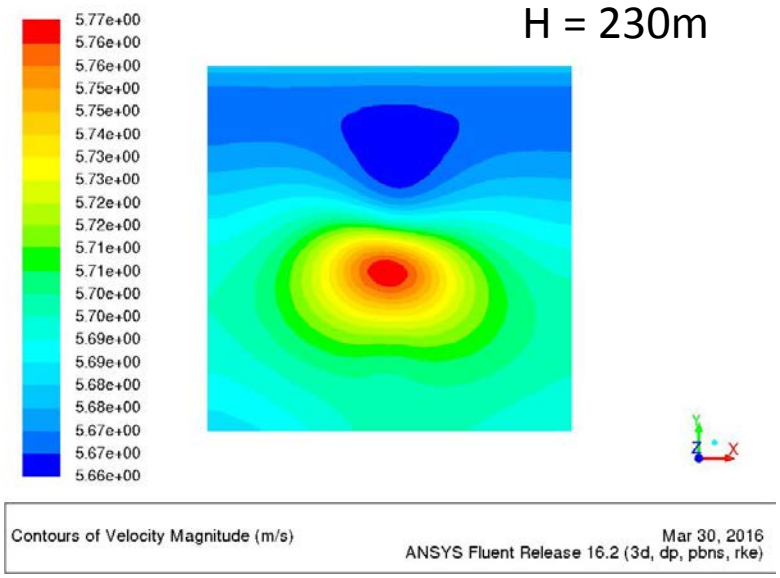
GM4.1 vs. GM2015 - Blockage ratio

- For GM2015, Blockage ratio < 17% (230m) is sufficient (less than 2%).
- For GM 4.1, it only requires 3 times height of the building (117.3m)

H = 117.3m



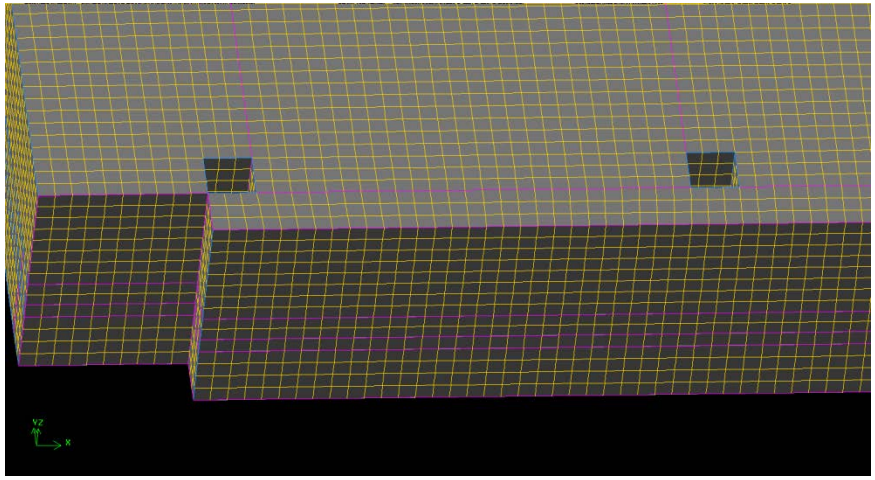
H = 230m



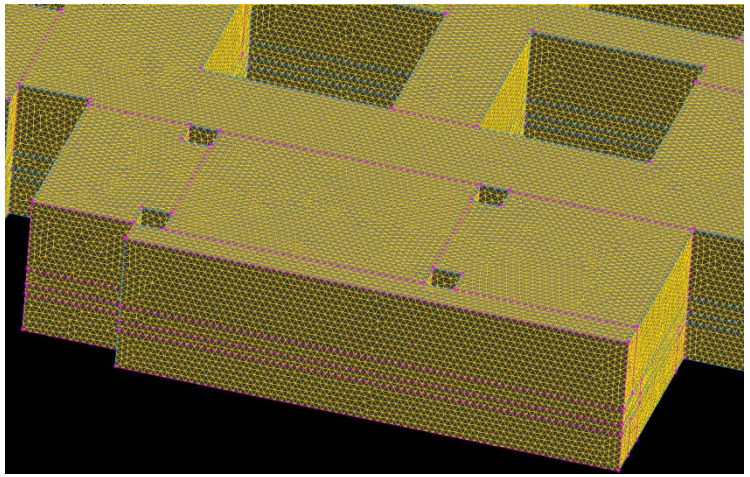
Height (m)	Blockage ratio (%)	Velocity difference (m/s)	Velocity difference (%)	maximum velocity (m/s)
117.3	33.3%	0.21	4.05%	5.19
230	17.0%	0.11	1.91%	5.77

GM4.1 vs. GM2015 - Hexa & Tetra

- Hexa to tetra leads to 4x increase in cell count & no converged solution guarantee



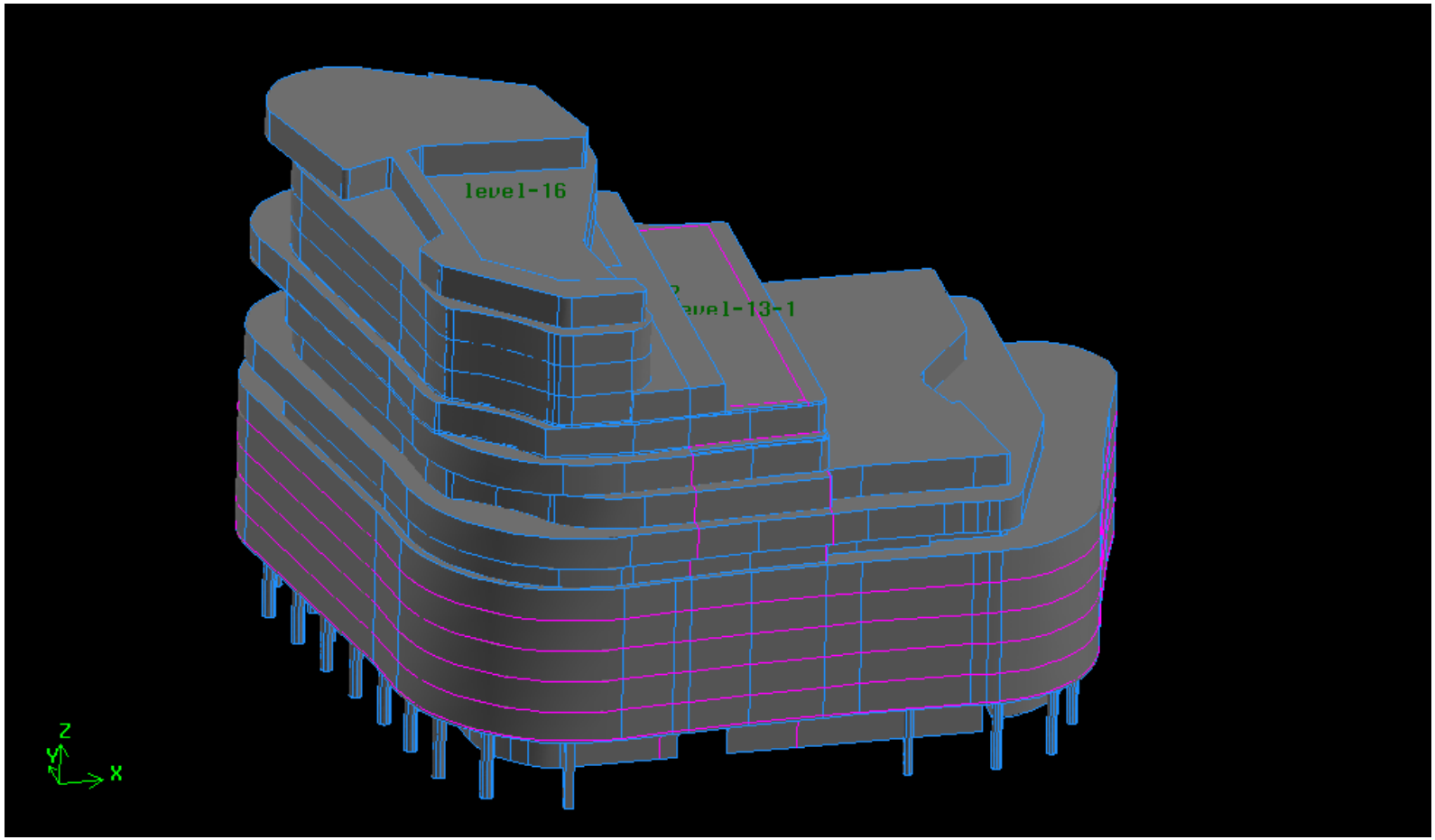
Hexa mesh



Tetra mesh

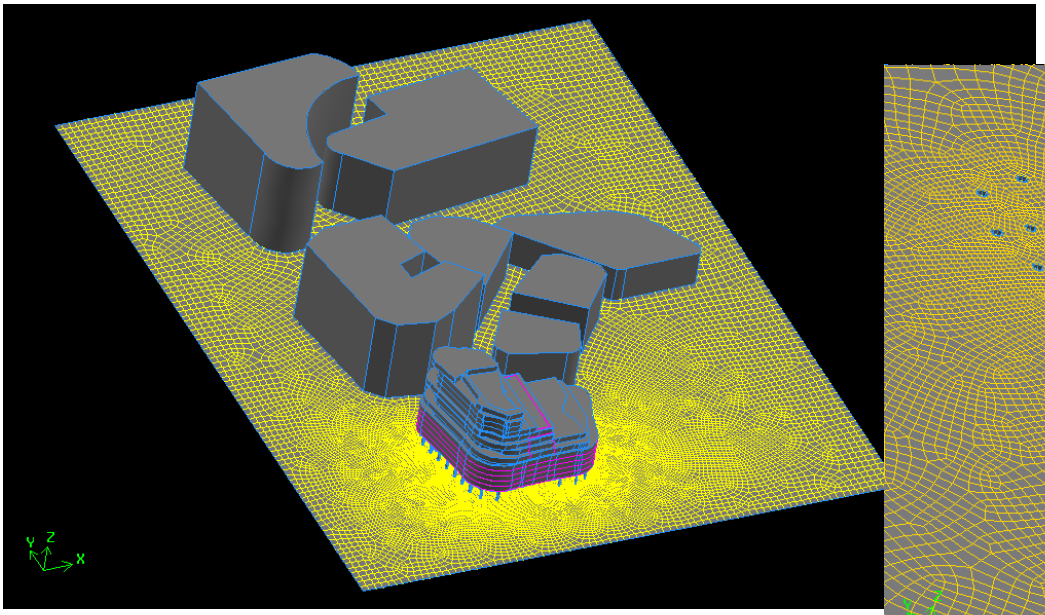
CFD NV simulation for Commercial Atriums - Solaris, GM2015 methodology and passing criteria – meshing stage (work towards 2nd order discretization)

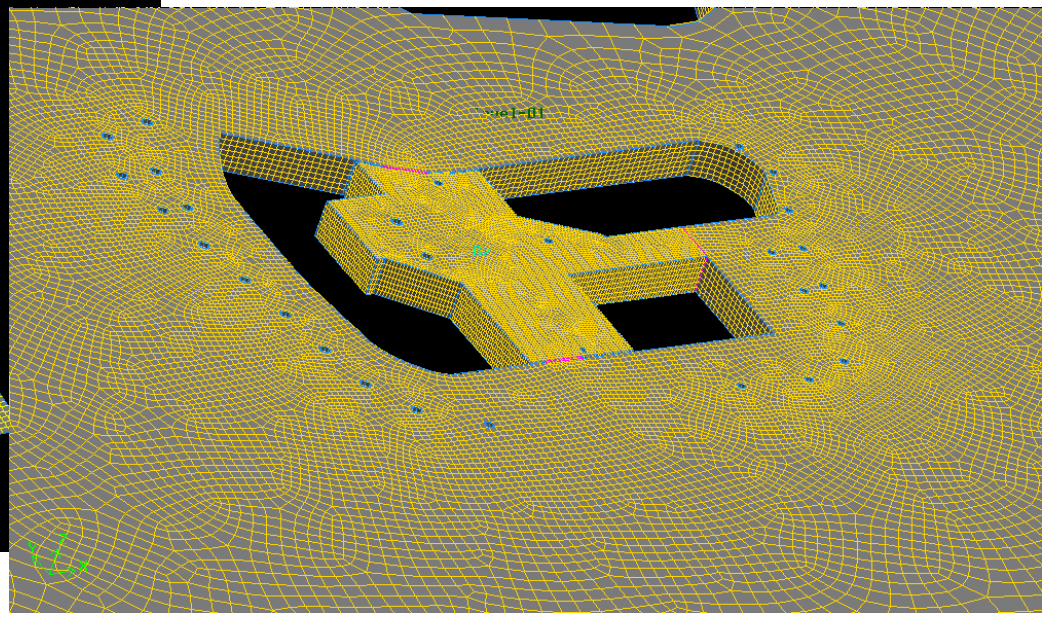
- Solaris Building



CFD NV simulation for Commercial Atriums
Solaris, GM2015 methodology and passing criteria

Including the surrounding buildings

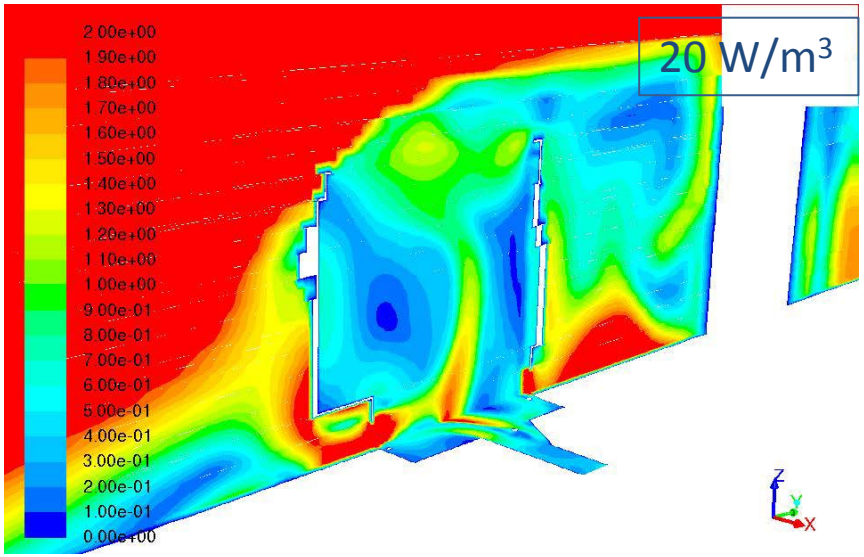


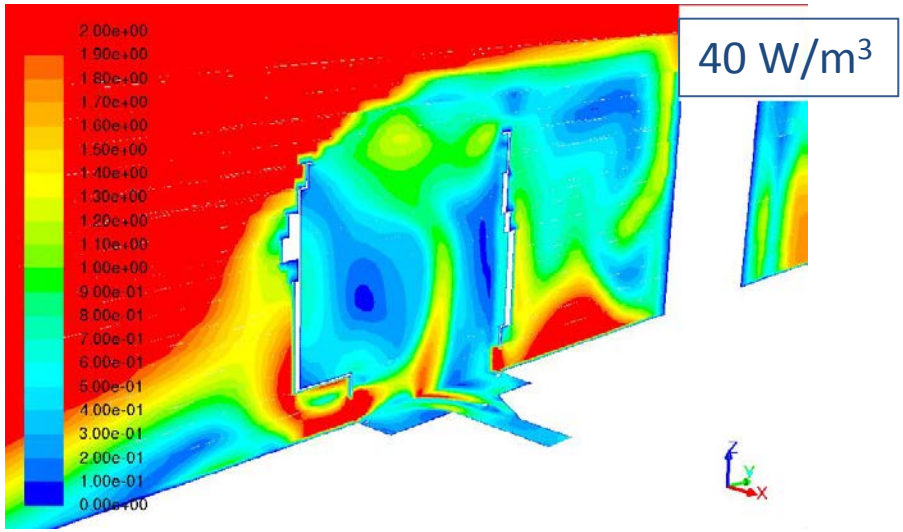


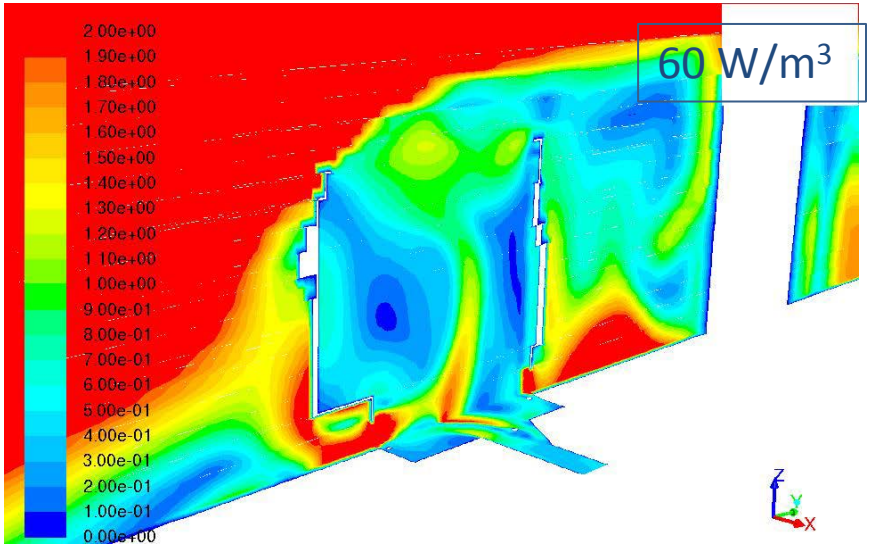
Location	Grid Size (m)
Within the functional spaces of interest	0.1-0.5
Building of interest	0.5-1.0
Surrounding building	1.0-5.0
From ground surface to 10m height in vertical direction	0.5 – 1.0
Ten to H_{max} height in vertical direction, (H_{max} is the height of the tallest building among the group of buildings modeled explicitly)	1.0 – 5.0

CFD NV simulation for Commercial Atriums
Solaris, GM2015 methodology and passing criteria

Velocity results @ x-cut plane from SE wind simulations







CFD NV simulation for Commercial Atriums
Solaris, GM2015 methodology and passing criteria

	Area-weighted velocity @ 1.2m	Temperature range	volumetric flow rate
isothermal	0.791 m/s	---	193.59 m^3/s
20 W/m^3 @level1	0.873 m/s	303-306 K	218.26 m^3/s
20 W/m^3 @0-1m	0.814 m/s	303-304 K	201.68 m^3/s
40 W/m^3 @0-1m	0.819 m/s	303-305 K	203.91 m^3/s
60 W/m^3 @0-1m	0.8288 m/s	303-306 K	207.18 m^3/s

CFD NV Methodology Development for GM2015 – component study

- Tree Canopy Modelling - Completed
- Fan Modelling – In Progress
- Louver Modelling – In progress
- Heat source modelling – To start
- Turbulence modelling – To Start

Tree canopy modelling for aerodynamic effects : RANS k-e model

Continuity equation

$$\frac{\partial \rho \langle u_i \rangle}{\partial x_i} = 0$$

Momentum equation

$$\frac{\partial \rho \langle u_i \rangle}{\partial t} + \frac{\partial \rho \langle u_i \rangle \langle u_j \rangle}{\partial x_j} = \frac{\partial}{\partial x_i}\left(\langle p \rangle + \frac{2}{3}k \right) + \frac{\partial}{\partial x_j}\left\{ \mu_t \left(\frac{\partial \langle u_i \rangle}{\partial x_j} + \frac{\partial \langle u_j \rangle}{\partial x_i} \right) \right\} + S_u$$

Transport equation for turbulent kinetic energy k

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho \langle u_i \rangle k}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma} \frac{\partial k}{\partial x_j} \right) + P_k - \rho\varepsilon + S_k$$

Transport equation for dissipation ϵ

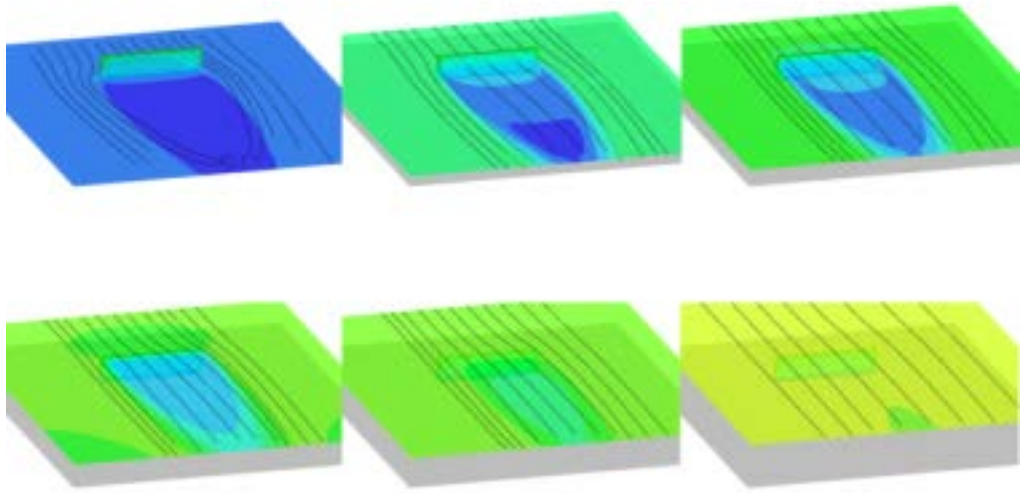
$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial \rho \langle u_j \rangle \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma} \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{\rho\varepsilon}{k}\left(C_{1\varepsilon} P_k - C_{2\varepsilon}\varepsilon \right) + S_\varepsilon$$

Where the production term P_k is

$$P_k = \mu_t \left(\frac{\partial \langle u_i \rangle}{\partial x_j} + \frac{\partial \langle u_j \rangle}{\partial x_i} \right) \frac{\partial \langle u_i \rangle}{\partial x_j}$$

The nature for the aerodynamic effect of tree

- When the wind passing through the tree canopy
  - decreases in velocity
  - increases in turbulence
  - increases in dissipation
- The corresponding source/sink terms should be added to model these effects.



Tree canopy modeling for k- ε model

- $S_u = -C_d\, LAD\, u_i\, U$ (sink term)
- $S_k = C_d\, LAD\, \beta_p\, U^3 - C_d\, LAD\, \beta_d\, U\, k$ (source term)
- $S_\varepsilon = C_d\, LAD\, \alpha_p\, \beta_p \frac{\varepsilon}{k}\, U^3 - C_d\, LAD \alpha_d\, \beta_d \frac{\varepsilon}{k} U\, k$

C_d :drag coefficient of tree canopy

LAD : leaf area density (m^2/m^3)

u_i : velocity component

U : $\sqrt{|u_i u_i|}$

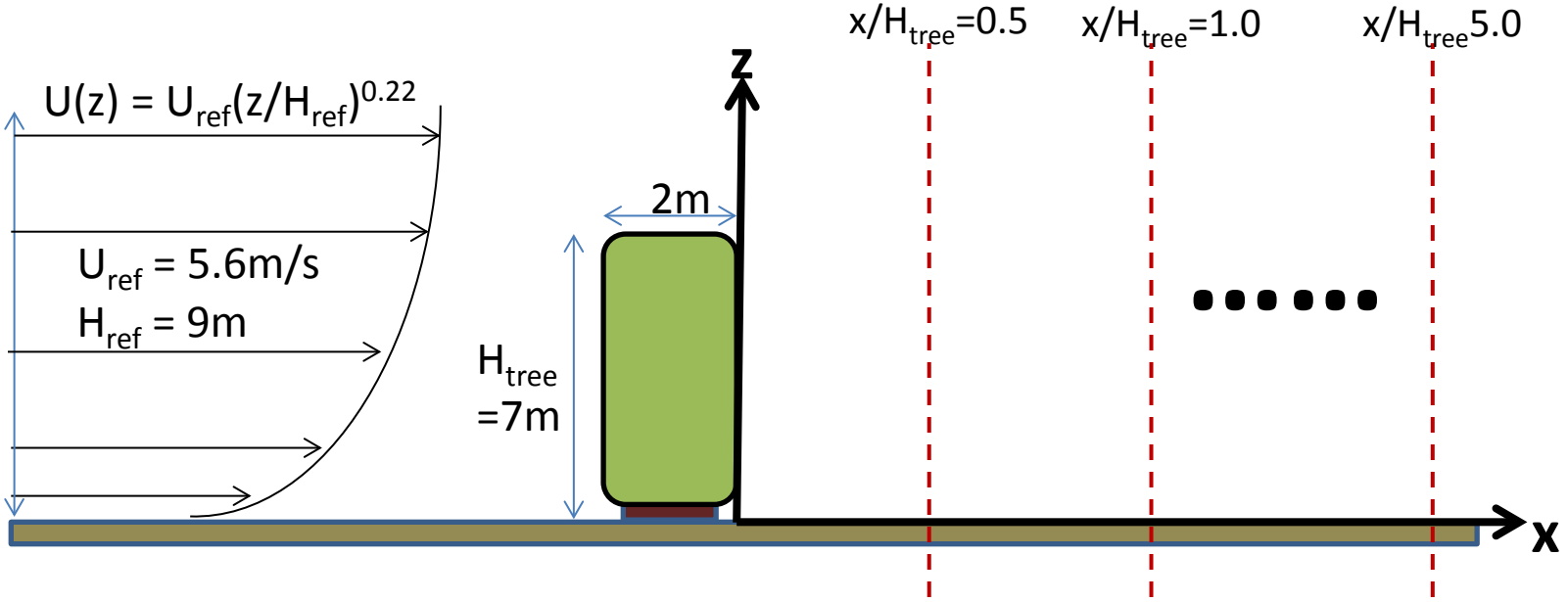
β_p : fraction of mean flow kinetic energy being converted to wake-generated energy by canopy drag, typical value = 1

β_d : magnitude of energy losses from interactions with obstacles, typical value = 4

α_p and α_d : model coefficient, typical value = 1.5

Y. Yang, Z., Xie, X. Jin, M. Gu, "*Verification of a tree canopy model and an example of its application in wind environment optimization*", Wind and Structures, Vol. 15, No. 5 (2012) 409-421

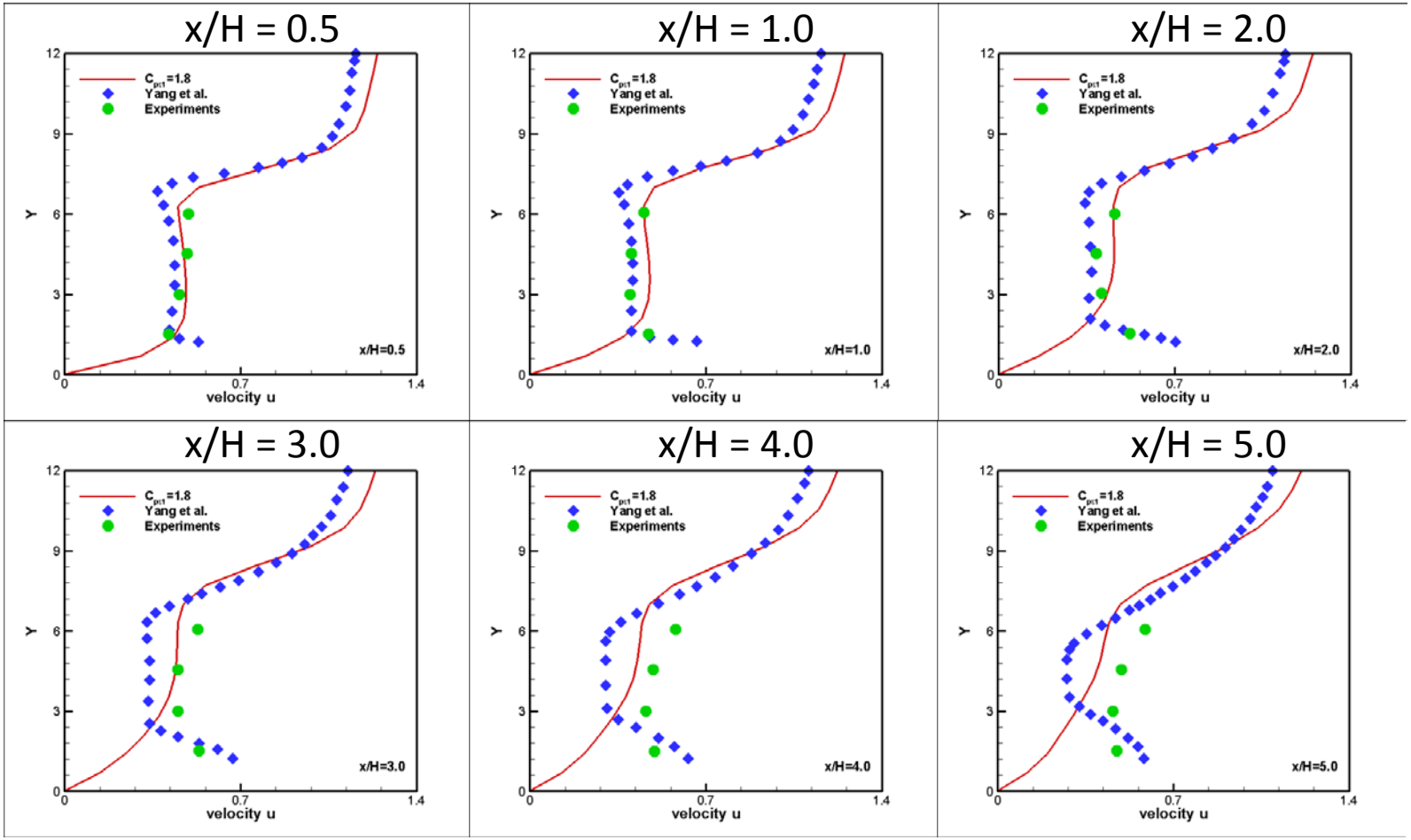
Benchmark test: flow behind pine trees



- Simulations are performed and compared with results of Yang et al. for x/H_{tree}= 0.5, 1.0, 2.0, 3.0, 4.0 and 5.0

Y. Yang, X. Jin, L. Yang and M. Gu, "*Verification and application of a tree canopy model for wind environment simulation in urban areas*", The Fifth International Symposium on Computational Wind Engineering (CWE2010)

Comparisons results



Note that in this study we only use standard wall function.

Y. Yang, X. Jin, L. Yang and M. Gu, "*Verification and application of a tree canopy model for wind environment simulation in urban areas*", The Fifth International Symposium on Computational Wind Engineering (CWE2010)

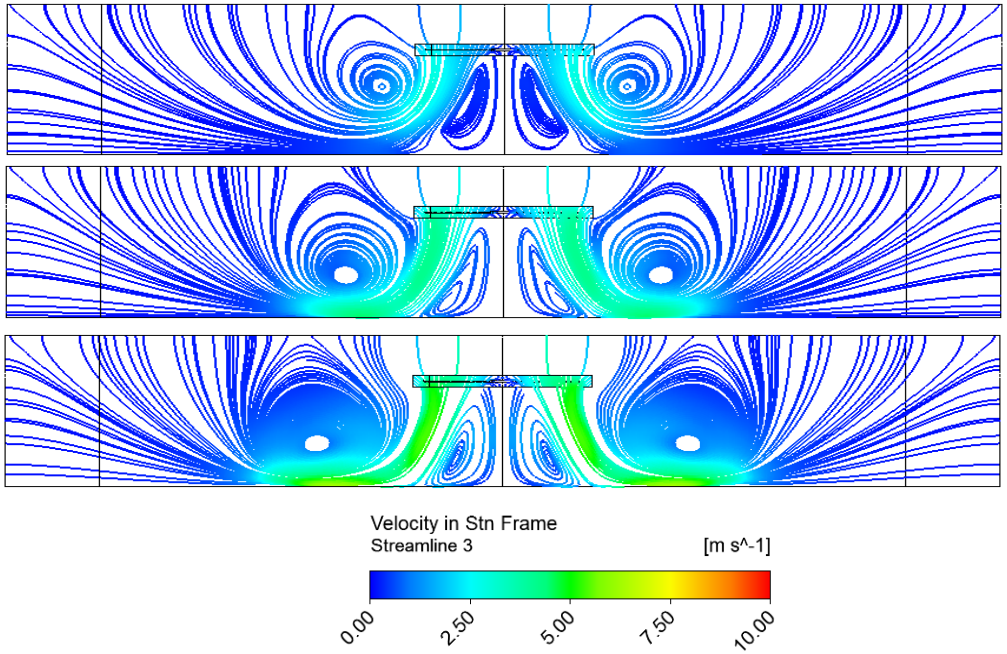
Low Speed High Volume Fan modelling

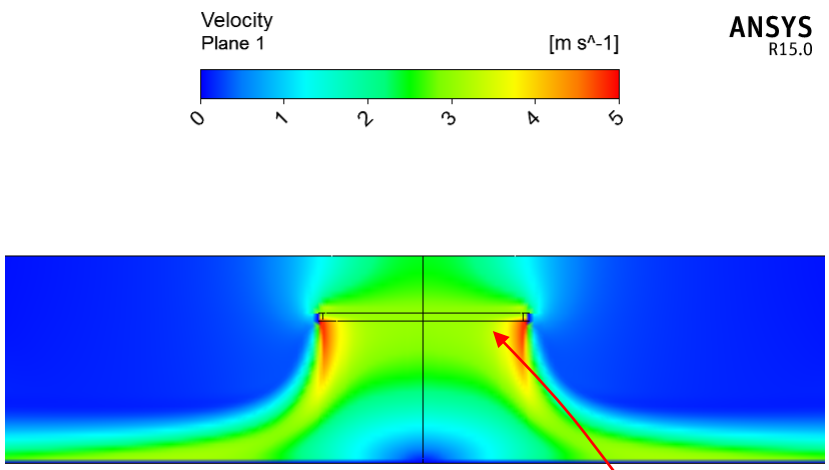
Methodology:

- Multiple reference frame (MRF) and immersed solid method, on ANSYS CFD for rapid characterization and exploration of new ideas
- At a later stage, to use in-house developed overset technique on OpenFOAM for detailed calculations (massively parallel simulations)
- Calibration with model scale fan (collaboration with Newcastle University in Singapore)
- Simplification by use of annular momentum source

First exploratory study:

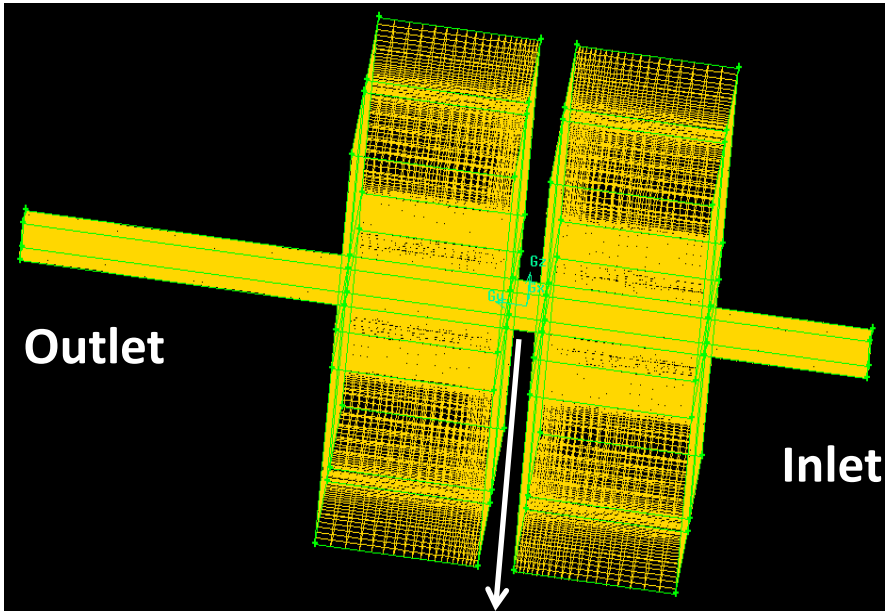
- Investigate effect of fan blade pitch angle on flow-field
- Eppler 420 blade profile (STOL blade)
- 24’ diameter fan
- Speed of 60 RPM



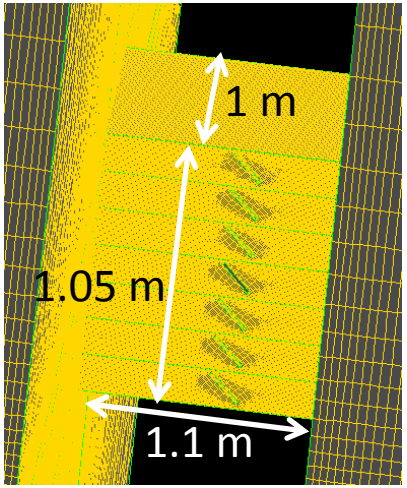


Simulation to to reproduce the flow using a circular /annular momentum source that varies in value in the radial and circumferential directions

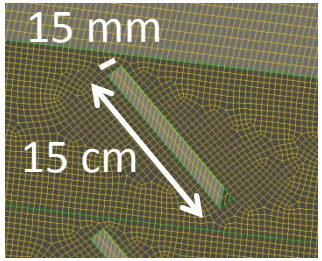
Louvers Modelling



Central Louver Block



6+ million cells



CFD Model

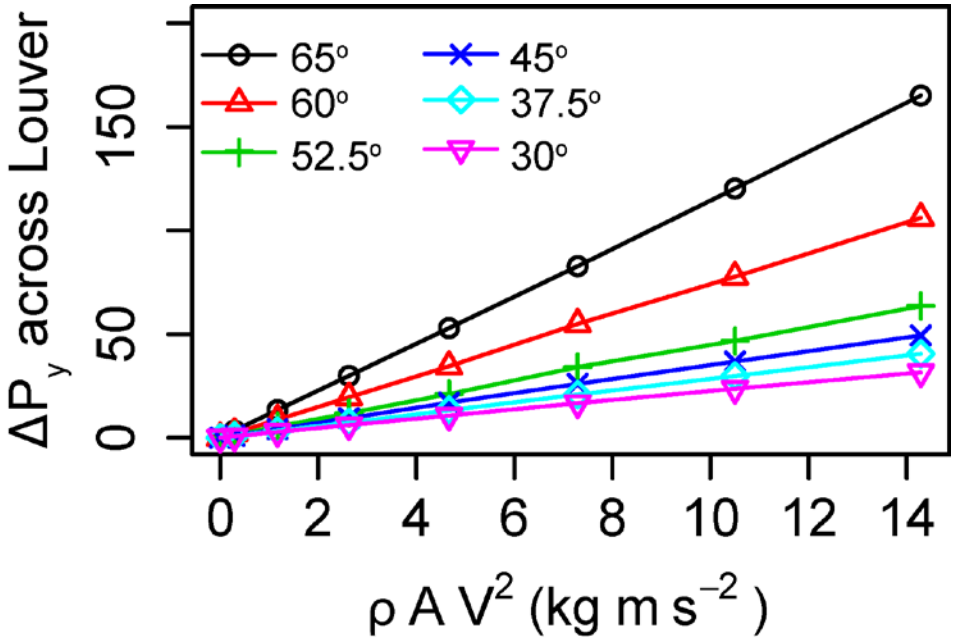
- 1 x 1.05 m^2 cross-sectional area
- 7 louvers
- Single-blade louver
  - 15 cm long
  - 15 mm thickness
- Aligned at θ to wind direction

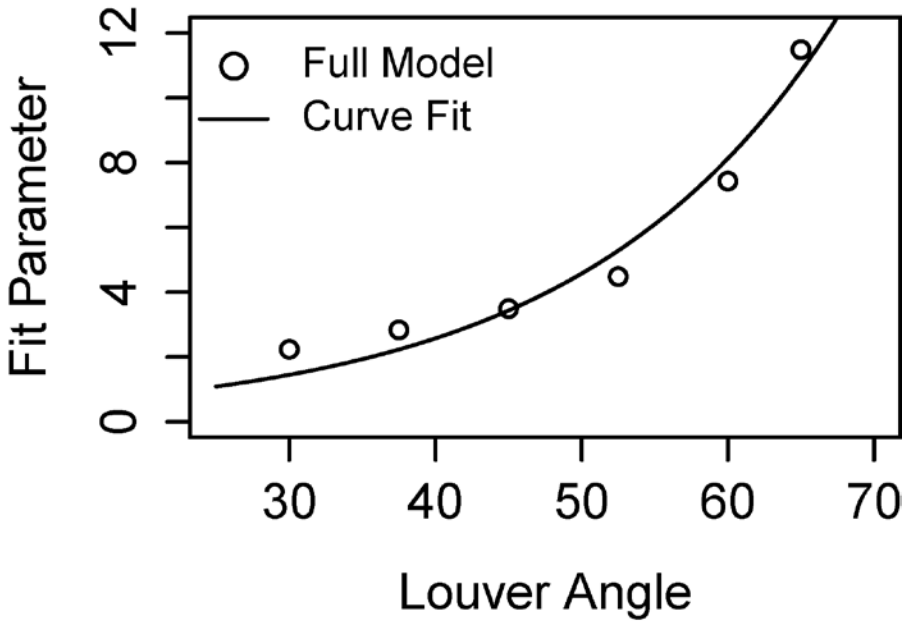
- Volumetric flow rates tested
  - 0.5 to 3.5 m^3/s
  - 0.5 m^3/s intervals

- Turbulence modeling
  - k-ε model
  - Second order scheme
  - Residual tolerance = 1E-4

Louvers Modelling - Model Derivation

- Simulated 30º, 37.5º, 45º, 52.5º, 60º and 65º degrees for 7 flow rates (0.5 – 3.5 m^3/s)

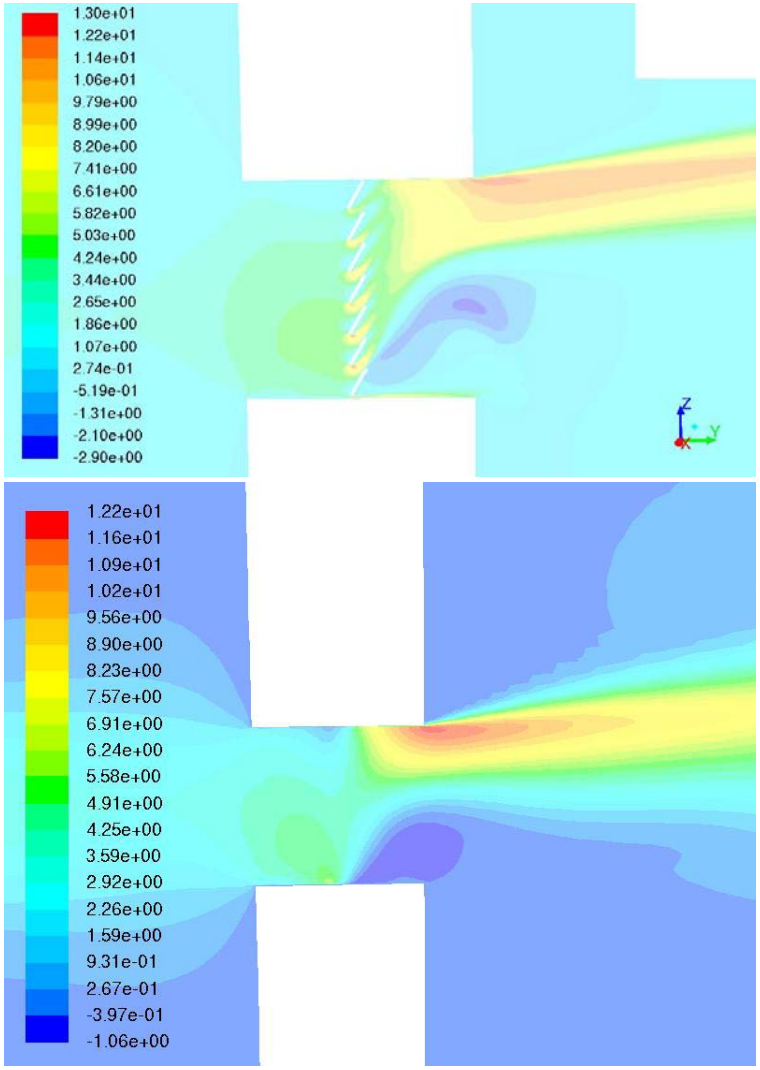




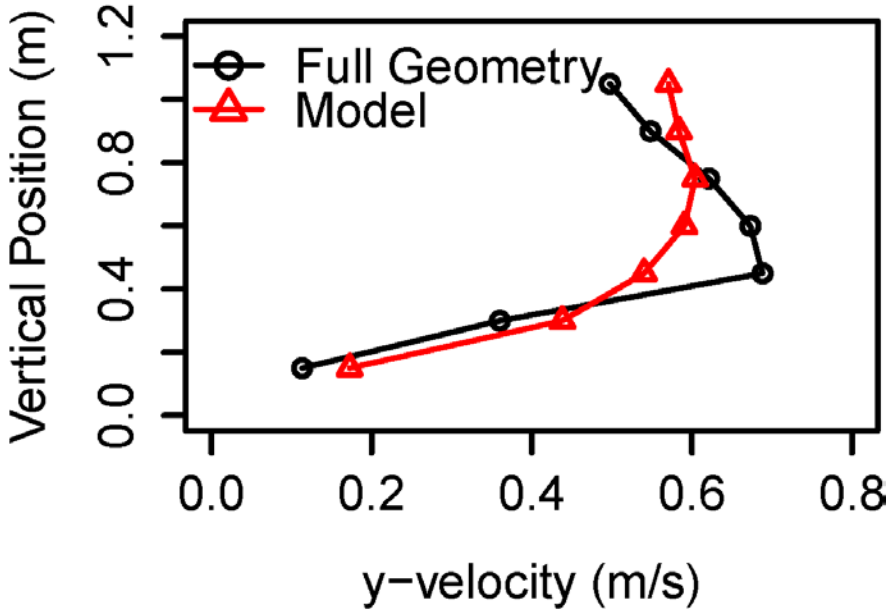
Obtained curve-fit parameters for dependence on louver angle and flow rate.

→ Fit to exponential form: $e^{a\theta+b}$

Louvers Modelling - Model Output



y-velocity contour plots for (Top) full geometry and (Bottom) reduced model for 60^{o} angle



y-velocity distribution along the vertical direction on exiting the louver region.

Directional effect is incorporated.

Concluding Remarks

- Improvements/modification have been made to current GM Ventilation Simulation Methodology
  - Domain size specification – follow the standard guideline, pay special attention to blockage ratio
  - Mesh quality – good mesh and cell size requirement, hexahedral mesh at functional unit is preferred
  - ABL simulation with the near ground surface fine mesh distribution – to ensure horizontal homogeneity
  - Tree canopy modelling – aerodynamic drag but not suitable to include heat sink at this stage
  - Fan modelling - Multiple reference Frame with Steady/Transient simulation, Momentum source method, customized 3D fan module with pressure drop/fan curve specification
  - Louver modelling – pressure drop with directional changes and gap effects to be included
  - Check list to ensure CFD quality & track NV design changes
  - Provision for AEE calculation in NV simulation
- Ongoing process to gather feedback from stakeholder and task force committees