Green Mark **2021**

CFD

Computational Fluid Dynamics Simulation

GREEN MARK 2021

%

EE

Energy Efficiency

GREEN MARK 2021

Hw

Health & Wellbeing

Revision Log

Revision	Description	Effective Date
R1	1st Version	1/11/2021

GM Guideline on Computational Fluid Dynamics (CFD) Simulation

0. General

This guideline on Computational Fluid Dynamics (CFD) Simulation should be used in conjunction with the thermal comfort requirement stipulated in

- Green Mark 2021 Health and Wellbeing (Hw) section for thermal comfort and/or
- Green Mark 2021 and Green Mark SLE Residential Energy Efficiency (EE) section for ventilation performance.

This guideline encompasses 6 sections: (1) general ventilation simulation requirement, (2) ventilation simulation guideline for non-residential projects, (3) ventilation simulation guideline for residential projects, (4) wind driven rain simulation guideline, (5) guidance note on simulation, and (6) documentation requirements.

1. General Ventilation simulation requirement

1.1 Simulation Software

The CFD modeling shall be carried out using the well validated software. The CFD solver shall have the minimum capability of solving the Navier-Stokes fluid flow equations for a three-dimensional incompressible flow at steady state. Turbulence modeling shall also be included with the minimum requirement of using the standard k-ε turbulence model, coupled with the standard wall function. [*Note: It is recommended to use the enhanced RANS eddy viscosity model (apart from the minimum realizable k-ε turbulence model) and RANS Reynolds Stress Model.*]

1.2 Conditions

All simulation models shall be carried out under isothermal conditions at steady state condition. If the impact of heat sources is significant, heat source modeling shall be included. (*Note: The aggregated heat load from heat dissipating devices shall be modelled. Boussinesq or variable density can be used.*)

1.2.1 Computational Domain and Surrounding Buildings

The computational domain shall include the development of interest and the far field boundary which should be located far enough from the building model to avoid artificial acceleration of the flow. As a general guideline, the direction blockage ratio (BR_L & BR_H) along lateral and vertical directions should be less than 17%.

$$BR_L = \frac{L_{Buildings}}{L_{Domain}} < 17\% \qquad\qquad BR_H = \frac{H_{Buildings,max}}{H_{Domain}} < 17\%$$

It is also important to ensure that the blockage ratio (BR) arising from the projection of building frontal to the domain enclosure is no larger than 3%.

The surrounding buildings residing within 500 m distance from the edge of development of interest should be modelled explicitly. In the event that the building and surrounding development are located within hilly terrain with elevation more than 10 m height, the topography information should also be included in the simulation models to capture the wind redistribution arising from terrain gradient and vicinity effect. The ground surface beyond surrounding buildings site can be modelled implicitly using the Davenport-Wieringa roughness classification.

1.2.2 Grid size

The computational grid generated for all simulations shall resolve the salient flow features in the naturally ventilated spaces and around the development. The recommended grid sizes are as follows:

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

As a guide, the dimension of the computational elements is advised to follow the principles such as:

- Proper domain decomposition should be carried out to ensure a good quality mesh can be obtained.
- Hexahedra or prism body-fitted grid are preferred.
- A grid independent test shall be performed at the functional space through grid refinements in areas with sharp gradients.
- In terms of the computational cell quality, the skewness of the cell is advised no greater than 0.9.
- The maximum stretching ratio for near building cell size should be kept to be less than 1.4.

1.2.3 Boundary Condition & Turbulence Modelling

(a)   Inlet Atmospheric Boundary Condition

Based on local climatic wind conditions, meteorological data on the precise wind direction and velocity of the proposed site location for the months of December, March, June and September shall be used for the CFD simulation. The prevailing wind conditions, such as the mean speed and direction for Singapore, shall be based on NEA's 18-year data at a reference height of 15.0 m as follows:

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

The inbound vertical wind profile shall be assumed to be given by the Logarithmic Law with reference height at 15.0 m. The wind profile shall be determined by using the following equations:

$$U(z) = \frac{u^*_{ABL}}{\kappa}\ln\left(\frac{z+z_0}{z_0}\right); \quad k(z) = \frac{u^{*2}_{ABL}}{\sqrt{C_\mu}}; \quad \varepsilon(z) = \frac{u^{*3}_{ABL}}{\kappa(z+z_0)}; \quad u^*_{ABL} = \frac{U_{ref}\,\kappa}{\ln\left(\dfrac{h+z_0}{z_0}\right)}$$

Where

   u^*_{ABL}: Atmospheric boundary layer (ABL) friction velocity
   κ: von Karman constant (0.42)
   C_μ: A constant, generally taken equal to 0.09
   z_0: Aerodynamic roughness length
   U_{ref}: The specified velocity at reference height h

The aerodynamic roughness length z_0 for wind profile should be selected from the updated Davenport-Wieringa roughness classification as follows, to match the terrain category of the development site of interest, including the tree/greenery effect.

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

(b) Ground Surface

Using appropriate roughness parameters is an essential component for accurate simulation of Atmospheric Boundary Layer (ABL) flow. The two types of roughness parameters, (i) aerodynamic roughness length z_0 and (ii) equivalent sand-grain roughness height k_S, should be applied on different surface areas as listed:

Ground Surface Area	Roughness Parameter
Area 1: From domain Inlet boundary to the boundary of explicitly modelled buildings	Aerodynamic roughness length z_0
Area 2: Within the region of explicitly modelled buildings	Aerodynamic roughness length z_0
Area 3: Within the site boundary of the development of interest	Equivalent sand-grain roughness height k_S

The region of inlet, approach and incident flow at the upstream of computational domain should be modelled with appropriate aerodynamics roughness length z_0 as well as the relationship between equivalent sand-grain roughness height k_S with the corresponding aerodynamics roughness length z_0.

(c) Top and Lateral Surface of Domain

Use zero velocity gradients and zero normal gradients, i.e. "symmetry" condition, for all variables at the top and lateral surface when the top and lateral boundaries of the domain are far away enough from the buildings (refer to the requirements on the domain size).

(d) Outlet Surface of Domain

Use zero static pressure as the boundary condition at the outlet surface of computational domain.

1.2.4 Discretization Schemes

In all circumstances, the users should attempt to apply 2nd order discretization schemes, which are preferred over 1st order discretization schemes to avoid numerical diffusion.

1.2.5 Convergence Criteria

To ensure the changes in solution variables from one iteration to the next are negligible, residuals with at least 4 orders of magnitudes shall be achieved. In addition, monitoring points should be defined in the region of interest and the velocities at those points should be recorded to ensure that the flow has reached steady values when simulation is converged properly.

1.3 Design Iteration

There shall be at least two iterations of simulation models to assess the wind flow conditions and air-flow pattern within the development to demonstrate the improvement in natural ventilation design. The simulation modelling can be conducted based on the two best prevailing wind directions for the building development that are North or North-East and South or South-East.

2. Ventilation simulation for Non-residential projects

Green Mark Health and Wellbeing Section HW 1.3a Thermal Comfort provides the performance levels required for thermal comfort. The methodology described below shall be used to demonstrate performance under thermal comfort simulation of effective cross ventilation.

2.1 Effective Cross Ventilation

Simulation follows the methodology outlined in section 1 for the naturally ventilated occupied spaces. Where there is repetition, space shall be selected at the lowest level for simulation. All naturally ventilated functional spaces are to be included in the simulation model except for the enclosed spaces such as storerooms or CD shelters.

From the simulation results, the area-weighted average wind velocity of each simulated space shall be determined by considering the air flow conditions of the applicable areas. The area-weighted average wind velocities of these areas are to be computed at a horizontal-plane of 1.2 m above the floor level.

2.2 Thermal Comfort Simulation Methodology and Requirements

The thermal comfort assessment, where required, shall be carried out using Predicted Mean Vote (PMV) equation to identify the most effective building design and layout for the development. The assessment and simulation results and recommendations derived are to be adopted to meet the intent of the criteria.

For occupied spaces with natural ventilation performance utilising mechanically assisted ventilation, thermal comfort modelling shall be performed, the wind speed shall be determined from the methodology described in section 1 with the additional modelling of the mechanical assistance. The PMV shall meet the thermal comfort criteria for naturally ventilated spaces in tropical climate. Thermal comfort assessment shall be based on the PMV equation as follows:

$$PMV = a + b \times DBT + c \times WIND$$

Building Type	Value of a	Value of b	Value of c	Baseline of DBT(°C)
Industrial buildings	-4.974	0.202	-0.181	30
Healthcare facilities	-8.405	0.322	-0.686	30
Commercial atrium	-9.252	0.343	-0.747	31*
Hawker centres	-13.075	0.443	-0.460	32
Sport faciliteis	-9.945	0.379	-1.658	30
Schools	-6.805	0.267	-0.87	31
Note: The PMV value is to be rounded up to one decimal point.				

Where

> *DBT: Indoor air temperature (°C).*
>
> **DBT value for commercial atrium can be derived from the result of simulation that considers solar radiation, thermal load from human and equipment for event.*
>
> *WIND: Indoor wind velocity (m/s). The value shall be derived from the result of indoor ventilation simulation via the Ventilation Simulation Methodology and Requirements in this annex. Natural ventilation simulation with fan modelling can be performed based on occupied spaces alone, without the inclusion of external domain with prevailing wind flow condition.*

2.3 IAQ Methodology and Requirements

The Indoor Air Quality simulation can be used to demonstrate performance where draughts are crucial to avoid, but comfort and air quality are critical. This option is only applicable for specific sports facilities and industrial buildings with occupancy densities less than 50m^2 per person.

2.2.1 Air Change Rate

The air change rate calculation shall be calculated based on the following equation:

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

Where

> *Q: Air flow through the occupied spaces (m^3/s), determined from the result of indoor ventilation simulation via the Ventilation Simulation Methodology and Requirements in this annex*
>
> *Vol: Volume of the room (m^3)*

2.2.2 Air Exchange Effectiveness (AEE)

In addition to ventilation simulation, species transport modelling shall be performed based on steady state. The tracer gas is recommended to be released through constant volume dosing within the occupied spaces of the natural ventilated building. In order to avoid the impact of gravity on the concentration distribution, the molecular weight of the injected tracer gas should be similar to air. The calculation of air exchange effectiveness shall be based on the concentration of tracer gas within the occupied spaces following the rule of mass balance.

The AEE shall be calculated based on the following equation:

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

Where

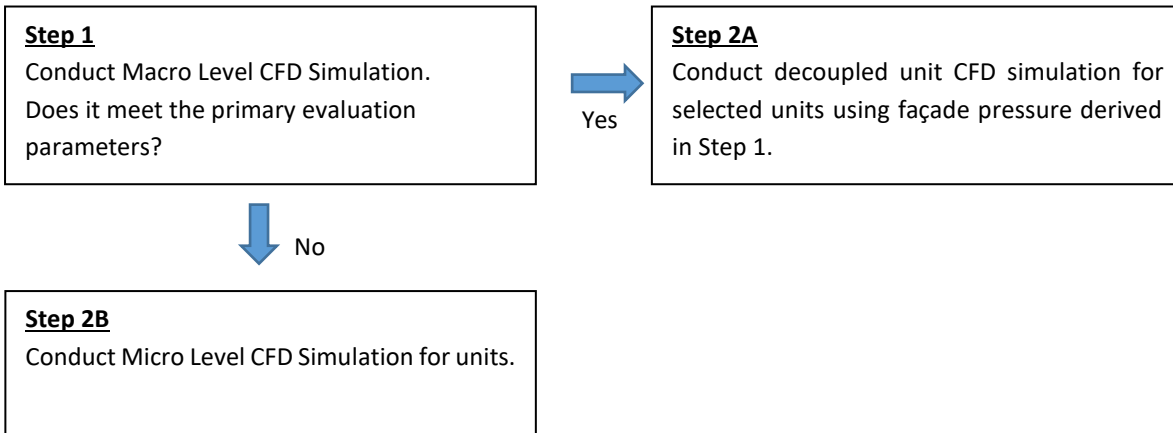> ε_a: Air exchange effectiveness
>
> A_i: Face element area at the out-flow openings
>
> $C_{i,out}$: Concentration of tracer gas in the outflow air
>
> C_{eq}: Average tracer gas concentration in the room when steady state is reached, which represents the equilibrium conditions

3. Ventilation simulation for Residential projects

This part is applicable for residential projects using CFD simulation and analysis. The simulation is recommended to be carried out with a steady temperature of 29.5°C. The simulation results and the recommendations derived are to be implemented to ensure good natural ventilation. Simulation shall follow the methodology outlined in section 1. Besides, the process shall be conducted as follows to meet the performance criteria under the EE section, or for thermal comfort under Hw section.

Step 1
Conduct Macro Level CFD Simulation. Does it meet the primary evaluation parameters?

Yes →

Step 2A
Conduct decoupled unit CFD simulation for selected units using façade pressure derived in Step 1.

No ↓

Step 2B
Conduct Micro Level CFD Simulation for units.

Determine up to five (5) typical unit design layouts that have the majority number of units. If the proposed building development comprises less than 5 typical unit types, all the typical unit design layouts are to be selected for the simulation.

There shall be at least two iterations of simulation models to assess the wind flow conditions and air-flow pattern within the development to demonstrate the improvement in natural ventilation design. The simulation modelling can be conducted based on the two best prevailing wind directions for the building development that is North or North-East and South or South-East.

3.1 Ventilation simulation modelling for pressure differential analysis

- Conduct a large-scale ventilation simulation modelling for development using the specified computational domain and grid stated to assess the wind flow conditions around the proposed building development and adjacent buildings. Natural ventilated corridor linked to the unit should be taken into consideration for the simulation models.
- From the simulation results, determine the wind pressure taken at 0.5 m from every assumed opening of all units at mid height level (capped at 20 storey height) and the pressure difference (i.e. the difference of the maximum and minimum wind pressure) of each unit. In instances, where all or some of the typical unit layouts are not designed at mid-height level, the average wind pressure and respective pressure differences should be determined for these typical units located at the level closest to the mid-height level.
- Calculate the global pressure differential by summing all the pressure difference of all units divided by the total number of units (at mid height level).
- If the development level simulation result meets the primary evaluation parameters, the project would have deemed to satisfy the performance requirements to conduct a decoupled unit CFD simulation using the façade pressures derived.

Primary evaluation parameters:

- A minimum 60% of Dwelling Units with window openings facing the prevailing north or north-east and south or south-east directions AND a minimum 2.7 Pa of Global Pressure Differential of Dwelling Units located at building mid height level

  OR

- If < 60% of Dwelling Units with window openings facing the prevailing north or north-east and south or south-east directions, to meet a minimum 4.3 Pa of Global Pressure Differential of Dwelling Units located at building mid height level.

3.2 Step 2A Decoupled Ventilation simulation modelling for units

- Unit simulation to assess the air-flow patterns within all the five selected typical dwelling unit types. The façade wind pressure result taken from 3.1 shall be prescribed as the boundary condition. All living spaces in the dwelling unit are to be included in the modelling except for enclosed space, such as storeroom or CD shelter. All windows & doors are assumed to be fully opened as designed except for the main door which is assumed to be closed at all time unless a door gate or grille is provided by the developer.
- For residential buildings, the applicable areas refer to living room, open kitchen (which is connected to the living room), study rooms and all bedrooms. The area weighted average wind velocities of these areas are to be computed at horizontal plane 1.2 m above the floor level.
- The selected units area-weighted average wind velocity of the unit shall meet the prescribed performance in GM 2021 EE and/or Hw sections. The overall percentage of units achieving good natural ventilation is given by:

$$\frac{\sum (\text{No. of Selected Units for Each Layout} \times \text{Area-Weighted Average Wind Velocity}) \times 100\%}{\text{Total Number of Selected Units} \times \text{prescribed windspeed (m/s)}}$$

3.3 Step 2B Micro Level ventilation simulation modelling for units

- If the primary evaluation parameters were not achieved, from results of step 1 ventilation simulation for development, select the unit with pressure difference that is closest to the average pressure difference from each typical unit design layout. The maximum allowable margin of ± 10% difference from the average pressure difference is deemed acceptable.
- Conduct a large scale CFD simulation to assess the air flow conditions of these five (5) selected units. All living or functional spaces in the unit are to be included in the simulation modelling except for enclosed spaces such as storeroom or CD shelter. All windows & doors are assumed to be fully opened as designed except for the main door which is assumed to be closed at all time unless a door gate or grille is provided by the developer.
- From the simulation results, determine the area-weighted average wind velocity of each selected unit by considering the air flow conditions of the applicable areas. For residential buildings, the applicable areas refer to living room, open kitchen (which is connected to the living room), study rooms and all bedrooms. The area weighted average wind velocities of these areas are to be computed at horizontal plane 1.2 m above the floor level. The same applies to naturally ventilated functional spaces for non-residential buildings.
- The selected units area-weighted average wind velocity of the unit shall meet the prescribed performance in GM 2021 EE and/or Hw sections. The overall percentage of units achieving good natural ventilation is given by:

$$\frac{\sum (\text{No. of Selected Units for Each Layout} \times \text{Area-Weighted Average Wind Velocity}) \times 100\%}{\text{Total Number of Selected Units} \times \text{prescribed windspeed (m/s)}}$$

3.4 Thermal comfort modelling for units

For developments using thermal comfort modelling the prescribed % of selected typical dwelling units shall demonstrate a weighted average wind velocity of 0.2m/s, without the mechanically assisted ventilation to be provided and included for the thermal comfort modelling.

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

PMV= -11.7853+0.4232DBT-0.57889WIND

where   DBT is indoor air temperature (°C). Baseline of T is 29.5°C

WIND is indoor wind speed (m/s). The value shall be derived from the result of indoor ventilation simulation via the Ventilation Simulation Methodology and Requirements in this annex. Natural ventilation simulation with fan modelling can be performed based on selected dwelling units alone, without the inclusion of external domain with prevailing wind flow condition.
PMV is Predicted Mean Vote

4. Wind driven rain simulations

Projects are encouraged to consider the use of CFD simulations to identify and to reduce the severity of rain penetration into naturally-ventilated areas. The following Wind Driven Rain (WDR) simulation guide outlines the process.

Four different raindrop sizes are to be analyzed. From the simulation results, the depth of rain penetration (measured from the fenestration opening) into the functional spaces of the development shall be determined, and the most effective mitigation method to reduce the severity of rain penetration without manual behavioural intervention shall be identified. The simulation results and recommendations derived are to be adopted to meet the intent of the criteria if points are to be scored.

The severity of rain penetration are as follows:

	Depth of rain penetration
1. Very good (no noticeable penetration of WDR)	Depth of rain penetration ≤ 0.20 m
2. Good (some but acceptable degree of penetration of WDR)	Depth of rain penetration ≤ 0.40 m

The frequency of WDR occurrence are as follows:

	Return Period
1. Type 1 Rain (no WDR risk)	Return Period ≤ 2 months
2. Type 2 Rain (low WDR risk)	Return Period ≤ 6 months

Simulation shall follow the methodology outlined in section 1 and adapt it to the following requirements as highlighted below:

4.1 Software

The software shall be also be capable of second-order discretization schemes with Lagrangian particle tracking.

4.2 Boundary Condition & Turbulence Modelling

(a) Inlet Atmospheric Boundary Condition

The inbound vertical wind profile shall be assumed to be given by the Logarithmic Law with reference height at 15.0 m. The prevailing wind condition during raining period such as the wind velocity magnitude of different return periods for Singapore shall be based on NEA 32-year data at a reference height of 15.0 m as follows:

Wind Direction [deg]	NORTH 000°	NORTH-EAST 045°	EAST 090°	SOUTH-EAST 135°	SOUTH 180°	SOUTH-WEST 225°	WEST 270°	NORTH-WEST 315°
Return Period (months)	Wind Velocity (m/s)							
1	2.9	3.8	4.4	4.5	4.4	3.5	3.4	3.0
2	3.9	4.9	5.0	5.2	5.2	4.4	4.1	3.6
3	4.8	5.8	5.5	5.9	5.8	5.2	4.8	4.1
4	5.4	6.3	5.9	6.3	6.3	5.7	5.2	4.4
6	6.2	7.1	6.3	6.8	6.8	6.3	5.7	4.8
9	7.0	7.8	6.7	7.3	7.4	7.0	6.3	5.2
12	7.5	8.4	7.0	7.7	7.8	7.4	6.6	5.5

(b) Rain Drop Size

Four different raindrop sizes shall be analysed, and the respective terminal velocity ($V_{terminal}$) of different raindrop sizes are as follows:

Raindrop Diameter [mm]	$V_{terminal}$ (m/s)
0.5	-2.0
1.0	-4.0
2.0	-6.5
5.0	-9.0

(c) Drag Coefficients

The drag coefficients for the raindrops (C_d) is a function of the relative Reynolds n umber (Re) and shall be taken from the table as follows:

Re (-)	C_d (-)	Re (-)	C_d (-)	Re (-)	C_d (-)	Re (-)	C_d (-)
1.80	15.0	269.0	0.671	1,461.0	0.498	2,772.0	0.615
9.61	4.20	372.0	0.607	1,613.0	0.503	2,905.0	0.635
23.4	2.40	483.0	0.570	1,764.0	0.511	3,033.0	0.660
43.2	1.66	603.0	0.545	1,915.0	0.520	3,164.0	0.681
68.7	1.28	731.0	0.528	2,066.0	0.529	3,293.0	0.700
98.9	1.07	866.0	0.517	2,211.0	0.544	3,423.0	0.727
134.0	0.926	1,013.0	0.504	2,357.0	0.559	3,549.0	0.751
175.0	0.815	1,164.0	0.495	2,500.0	0.575		
220.0	0.729	1,313.0	0.494	2,636.0	0.594		

(d) Other Settings

Piecewise integration of raindrop equation of motion performed under Lagrangian Particle Tracking shall not be larger than 0.05 m length step size.

The injection location of raindrops shall be located inside the computational domain and outside the zone that is influenced by the buildings. The raindrops shall be released high enough to enable them to reach their terminal velocity of fall. It is recommended to use the following general rules to select the raindrop injection planes inside the computational domain:

- The width and length of the plane should be larger than that of the building of interest

- The vertical location of the plane depends on the velocities and raindrop diameters:

  - Higher velocities require a lower vertical location

  - Larger raindrops require a higher vertical location

- The horizontal location depends on the chosen height of the injection plane, the raindrop diameter and the reference wind velocity chosen for the simulation (recommended to be located at least 15H from the target building)

5. Guidance Notes

The following are guidance notes to help project teams keep track of their natural ventilation design and simulation progress.

Guidelines	Description/Selection	Response & Criteria
Submission Details		Provide the project details (especially information on natural ventilated design, building massing/orientation, GFA of natural ventilated spaces, % of opening & windows, credible source of site information with surrounding buildings, vegetation and terrain, future development etc.)
Problem Statement	Objective & Work Scope	Describe natural ventilated challenges, proposed solution, desired outcome and work scopes from the simulation model. Whenever necessary, use the architectural drawing for explanation. Describe design stages and fix simulation details. Subsequent design change has to be supported by simulation results.
Site Information	Minimum 3 times the length of the longest distance measured across the boundary of the development, or within 500 m distance from the edge of development of interest, whichever that is smaller	Describe the site information (including surrounding buildings, terrain, greenery), and illustrate how the geometrical info is incorporated into the simulation model (conversion process). Surrounding buildings within 500m distance stream wise and span wise from the edge of development of interest should be modelled explicitly; while the greenery can be modelled implicitly with tree canopy approach. Terrain effect can be ignored if elevation is less than 10m.
CFD Approach	Simulation Methodology	The CFD solver shall have the minimum capability of solving the Navier-Stokes fluid flow equations for a three-dimensional incompressible flow at steady state. Turbulence modelling shall also be included with the minimum requirement of using the standard k-ε turbulence model, coupled with standard wall function
	Assumption & Simplification	Describe simulation model assumption, limitations and geometrical simplification. Whenever necessary, use the published literature data (including software manual) and comparison between architectural & CFD model for explanation
CFD Domain	Computational Domain	Describe the domain decomposition methodology; and relevant meshing type for each domain within the site. Describe the domain that be modelled implicitly with Davenport Roughness classification
CFD Meshing	Mesh size, distribution and quality	To carry out proper domain decomposition. To use hexahedral cells in the rectangular domain of NV space. Tetrahedral cells can be used to model the surrounding site features of the NV space. Hybrid pyramid or cut-cell mesh can be adopted at the interface. For implicit modelling of terrain roughness effect, prismatic or hexahedral cells is recommended to be used. As a guide, the dimension of the computational mesh should be set at 0.1 to 0.2 m within the functional space of interest, 0.5 to 1.0 m for building of interest and 1.0 - 5.0 m for surrounding buildings. The computational element size in vertical direction should be set at 0.5 - 1 m from ground surface to 10m height; followed by 1 - 5m to Hmax height. Reporting on skewness and

		aspect ratio of the mesh is required.
CFD Model	Atmospheric Boundary Layer	1. To ensure horizontal ABL homogeneity in upstream and downstream 2. To ensure sufficiently high mesh resolution in vertical direction near ground (e.g. height of first cell < 1m) 3. To know the relationship between equivalent sand-grain roughness height (k_s) and corresponding aerodynamic roughness height, z_o 4. To ensure first cell center point (y_p) to be larger than physical roughness height (k_s)
	Buoyancy	To use Boussinesq or variable density and check gravity direction, if thermal simulation is performed.
	Turbulence model	To use the steady Reynolds-Average Navier Stokes (RANS), with minimum requirement k-ε turbulence model equation for NV flow.
CFD Boundary Conditions	Inlet wind profile	To ensure the vertical profile for wind velocity and turbulence in the ABL should be modelled by assuming constant shear stress with height.
	Top & lateral sides of domain	To use zero velocity gradients and zero normal gradients of all variables.
	Outlet plane	To use zero static pressure.
	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 radial and swirl flow components.
	Louver modelling	To use simplified porous zone with appropriate pressure drop components and directional effect.
	Heat source modelling	To use aggregated heat load from heat dissipating devices, such as cooking stalls for hawker center facilities, heat generators for industrial facilities and etc.
CFD Numerical	Discretization scheme	To use 2nd order for momentum equations
	Convergence criteria	To ensure solution is converged and monitored points at functional space reach steady values
User		Training, experience, consult expert
Documentation		Full documentation of parameters, readable scale
Design Iteration	Baseline case Modified case Optimal case	Baseline case – fixed building massing and layout Modified case – to highlight improvement on NV design Optimal case – final design with incorporation of NV features or passive innovative ideas

6. Documentation Requirements

Design Stage

The Qualified Person (QP) and the other appropriate practitioners shall ensure that the following report and building 3D model are available as evidence to demonstrate compliance with the ventilation simulation framework. The report should comprise the following items:

2.0 Cover page with a proper title, design image of development, developer's information (including developer's name and address and person-in-charge), consultant's detail (including the principal's name and authorized signature, firm's address and person-in-charge)

3.0 Table of Contents

4.0 Executive Summary
- Background of the development
- Main findings
- Concluding remarks

5.0 Background/ Introduction
- Building and site information
- Design strategies
- Detail of natural ventilation spaces (location, area, window to wall ratio etc.)

6.0 Methodology
- Describe methodology used in the study

7.0 Geometrical Model
- Isometric view of the development from various angles
- Domain size used
- Plan and 3D isometric model of units from various angles

8.0 Simulation settings
- Boundary conditions
- CFD software/ models used/ numerical scheme
- Mesh / cell sizing
- Solution control-convergence criteria

9.0 Result and Discussions
- Simulation results for the development for all directions showing the main graphical plots of the plan pressure and velocity vector and salient findings
- Tabulation showing the listing and details of all simulated NV spaces and the area-weighted average wind velocity within each simulated space where applicable

10.0 Conclusion

11.0 Appendix: The following plots are to be placed in the appendices:
- Simulation results for the development for each direction
  - Static pressure (plan view-ground & mid elevation and at the level of simulated NV space, isometric views on building façade)
  - Velocity vector and contour showing the plan view at ground & mid elevation and at the level of simulated NV space, and a few isometric sectional cut plans to show air-flow patterns across the development
- Simulation results for the natural ventilated spaces for each direction
  - Static pressure (plan view at the level of simulated NV space)
  - Velocity vector and contour showing the plan view at the level of simulated NV space, and a few isometric sectional cut plans to show air-flow patterns across the NV space

If thermal comfort modelling or air quality assessment is attempted, a corresponding chapter in the report shall be added to show the relevant calculations.

If WDR simulation is carried out, a chapter in the report shall be added to show the results of Wind Driven Rain penetration under different wind directions and the calculation of green mark points for WDR performance. The report also shall contain the following information:

- Injection location of the raindrop into the computational domain (plan and sectional views)
- Raindrop trajectory into functional spaces (plan and sectional views)
- Tabulation showing the listing and details as well as the corresponding depth of rain penetration of all occupied spaces where applicable.

Verification Stage

- The project team shall declare if any changes had been made in actual built layout compared to the submitted 3D ventilation simulation model in the design stage. The re-assessment of ventilation simulation will depend on the extent of changes and their impacts on NV performance.

- If thermal comfort modelling assessment is attempted, the percentage of opt-out decisions of home buyers should be table.

Acknowledgements