CHAPTER 3

METHODOLOGY

3.1 Description of methodology

After passing through the study on problem statement of row house in Phnom Penh city, this thesis is deliberated about the improvement of indoor ventilation by using the effect of trombe wall technique in study case of Phnom Penh row houses. CFD program XFlow would be used as a simulation tool for the experiments. The experiments divided into two categories.

- First group is about the simulation on top-stair chamber. In this group, there contains 5 different cases based on the change of heat flux input from 200W/m², 400W/m², 600W/m², 800W/m², and 1000W/m².
- Second group is whole house simulation to see the real performance in practice case. There are also 5 cases study for this group: existing house (NTW); new house designed with Trombe wall by heat flux 200W/m² (TW200W/m²) and 1000W/m² (TW1000W/m²); and two other cases which were added temperature on top ceiling surface and walls of stair block, one without Trombe wall (HC-NTW), and another with Trombe wall by heat flux 1000W/m² (HC-TW1000W/m²). To attempt the consequences, the process of the experiment follows the steps below.

An accurate 3D model of the sample house was set up in Solid work program. In this study, the location of trombe wall was be on the top floor next to the stairblock. In accordance with the theory of trombe wall, the location of the wall must be in the hottest orientation from the sun in the afternoon period, usually in South and West. By doing so, this provides more heat to store in trombe wall which expected to gain better result by the system. Position and sizes of trombe wall was measured and installed according to the best ratio recommended by previous research documents. The process of studying this topic is briefly summarized by the diagram in Figure 3.1.

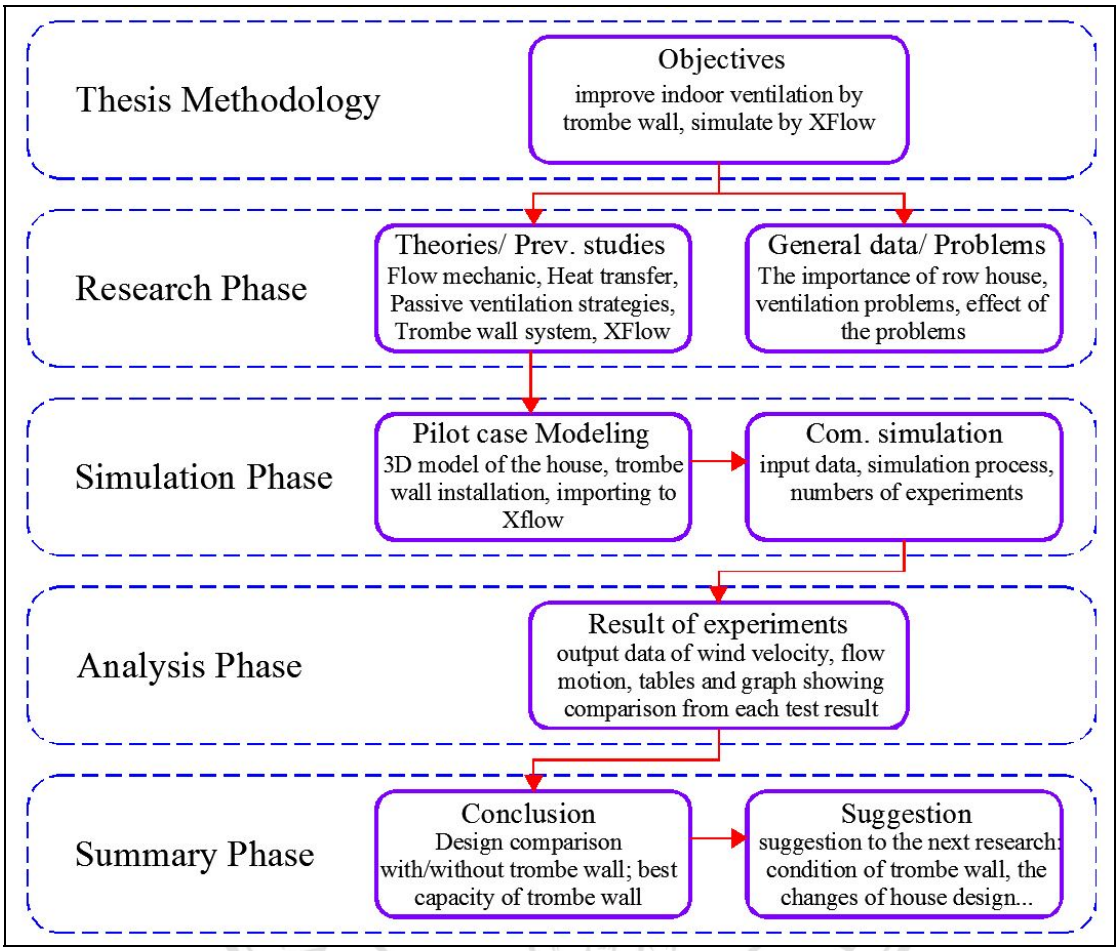


Figure 3.1 Diagram of research methodology study

Next step, all models would be imported into CFD XFlow program. Before running the simulation, some important input data must be inserted correctly. Those input data are: the position of inlet and outlet, the inlet temperature and wind speed, pressure, and material type (gas). Those data can be found by the research from Phnom Penh weather data and real testing on row house environment. Furthermore, for the first simulation group, the number of experiments are based on the change of heat flux varies from 200-1000W/m² inputting into the system. The test may notice on how much the minimum heat flux needed in Trombe wall to make the system started to work, until the maximum heat flux which Trombe wall is predicted to gain up to, to make the flow reaches its best performance. Likewise, the second group would refer to experiments on whole house ground floor area to examine about air flow circulation pattern from inlet to outlet, while roof of stair block was added some heat to familiarize with the real situation.

After all, in post processing step, the result of the test would be analyzed by the graphs, the image of the flow in term of the amount of air speed, temperature and mass flow rate provided by each case of simulations. These results would respond back to the standard of comfort zone declared in chapter of literature review. Thus, this study would bring discussion about the condition needed to make the system capable to work well, and how much “trombe wall” could help to activate indoor ventilation comfort.

3.2 Modeling set up and boundary conditions

3.2.1 Modeling set up

The house model is based on the real dimensions of existing house. Figures below show about building geometry dimension by 3D model before inputting into CFD program. There are a ground floor plan with living room and kitchen, and the linkage of the stair block in between, thus end up with the outlet at the top of the block. This house is simulated as an empty room without occupants and furniture. Normally housing size is 4.2m width x 16m length. At front house, there are two opening which are assumed as inlets. The big inlet is the main door of the house, and the small opening over the door is window or air-vent. The outlet is referred to the outlet of Trombe wall on top of the stair block with dimension of 3.01m width by 2.40m height.

By 3D model to be built, other no relevant floor is cutting out to help increasing simulation running time. For the existing house, there is an outlet with cement grills installed on the top of stair block chamber. But in drawing, those grills were taking out and maintained only the hole size with the purpose of reduce the geometry elements and to better see the flow. The inlets size are set equally to whole house simulation with Trombe wall, yet existing outlet size is 1.92m width by 0.15m height.

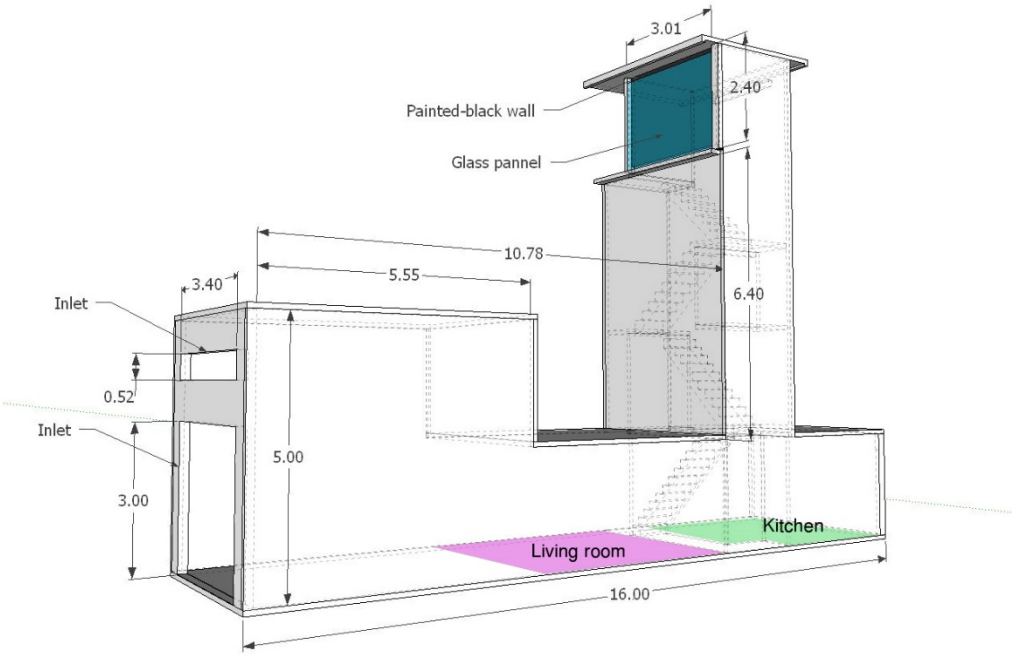


Figure 3.2 3D modeling of the study case showing the location of Trombe wall, in (m)

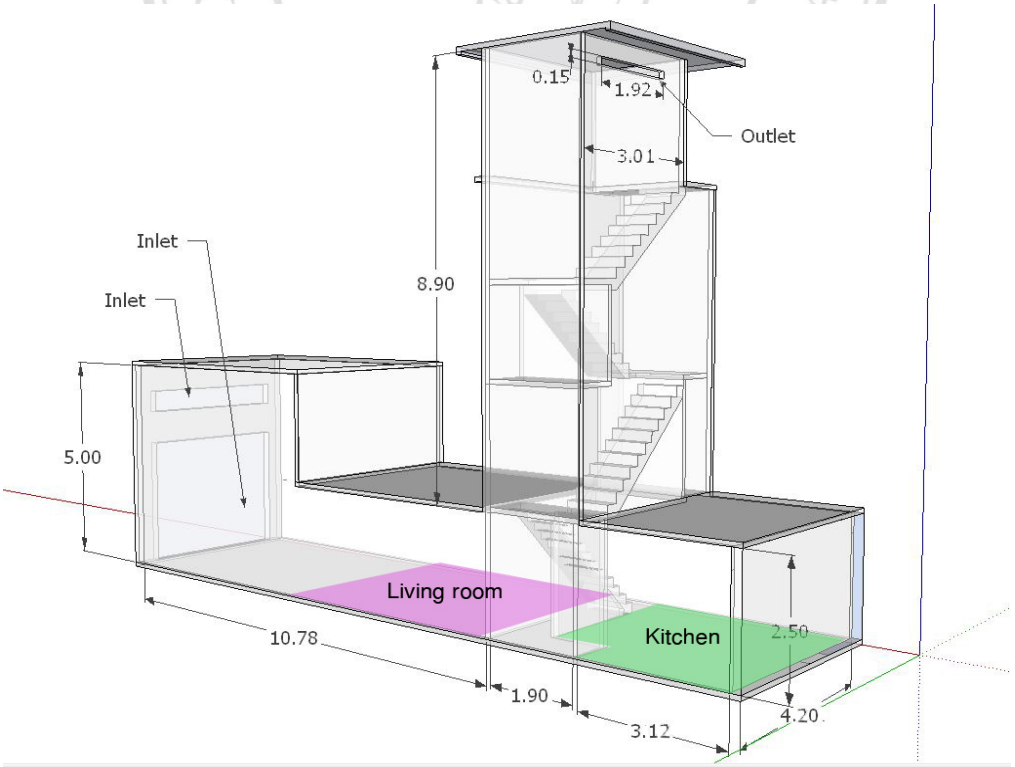
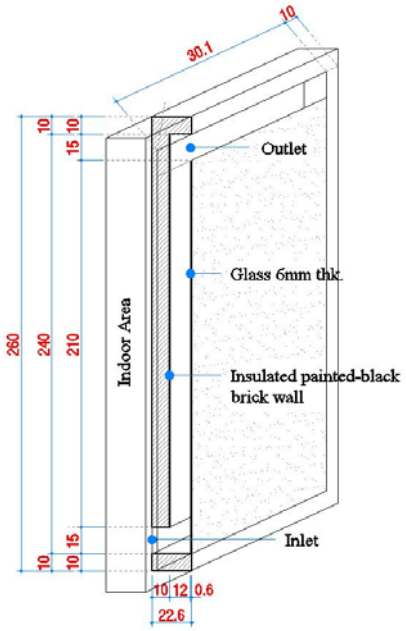


Figure 3.3 3D modeling of the study case showing the existing house with its outlet, in (m)

Trombe wall would be built up according to the appropriate available size of the top stair block area for the model of Phnom Penh's row houses. About Trombe wall dimension, the ratio of the gap between glazing to the wall is 0.05 which gives the best result of mass flow in the channel referred to the research of Mr. Phanuphong [23]. The inlet and outlet width of trombe wall was set equally with the channel's gap which followed the proportion of the study Gan [32]. So, the real hight of inlet and outlet, and depth which need to apply to the channel is equal 12cm. The wall inside channel was painted black with insulation inside to avoid the conduction of heat transmission into another surface of the wall.



Trombe wall model (cm)

Figure 3.4 Trombe wall dimension in (cm)

3.2.2 Model boundaries condition

After the modeling session, the model was exported into XFlow CFD program. Base on the program, to study about the performance of Trombe wall in different rate of solar radiation, the current investigation involved 5 cases testing by inserting uniform heat flux varies from 200-1000W/m² into Trombe wall. Primary boundaries condition were input such as earth gravity, indoor and outdoor temperature, also wall and glazing temperature. By those heat flux, we received a range of temperature on internal surface

of glass and wall due to Phanuphong [23]. Those temperature are converted from Celsius into Kelvin. To reduce the complexity of the simulation, all surfaces of the domain are considered as isothermal walls and no viscosity condition is set to the walls except for two cases studies while temperature on top of stair block are added to be more familiar to the real condition. Nonetheless, to better see the performance of the scheme, inlet pressure was set 2 Pa for first group simulation on top-stair chamber, and 5 Pa for whole house simulation. The initial wind speed is neglected because this will affect the performance of Trombe wall while the main objective is to investigate on the real capacity of Trombe wall without the help of prevailing wind speed. Anyway, for some real cases, blowing-in wind can't be regularly predicted due to house direction and wind direction.

The indoor temperature is set 27^{o}C according to ASHRAE psychometric chart of comfort zone [9], as that is the temperature which starts to give discomfort to occupant. Moreover, it is also made known that from real case experiment, the temperature in house is not lower than this [4]. The outdoor temperature is 33^{o}C, referenced from Phnom Penh airport weather data station as the average of hot temperature during summer from 2003-2012 [8]. For two cases of whole house simulation, the additional heat on surface of stair chamber ceiling was trial for the assumption amount which was expected that it is possible to achieve.

Table 3.1 Temperature on internal surface of glass and wall

Heat Flux	**Tem. glass**		**Tem. Wall**	
W/m^2	K	C	K	C
1000	356.35	83.20	334.15	61.00
800	349.62	76.47	331.82	58.67
600	338.95	65.80	326.58	53.43
400	332.82	59.67	321.32	48.17
200	325.62	52.47	317.35	44.20

In simulation phase, the flow was set to run by the program calculation until it reached its constant level. This phase needs a high performance computer to handle a faster time in simulation. The simulation were carried out on an Acer core i3 computer

with RAM of 3.0GB. The typical CPU times for running for group one, top chamber of stair block with Trombe wall, takes about 7 days to finish 200 seconds simulation time steps of each iteration case. Therefore, for second group of whole house simulation, it takes as approximately 5-7 days to finish 60 seconds time step for each case.

Table 3.2 Input parameter for simulation

Input	Unit	Data	
		Stair chamber group	**Whole house group**
Flow model		Internal flow	Internal flow
Gravity	m/s^2	9.81	9.81
Initial temperature	K	300.15	300.15
Operating temperature	K	306.15	306.15
Trombe wall, glass temperature	K	(Varies by heat flux)	(Varies by heat flux)
Trombe wall, wall temperature	K	(Varies by heat flux)	(Varies by heat flux)
Inlet pressure	Pa	2	5
Inlet temperature	K	306.15	306.15
Stair chamber roof ceiling	K	Non	Non, 318.15
Outlet		Gauge pressure outlet	Gauge pressure outlet
Simulation time	s	200	60
Resolve scale	m	0.1	0.1

3.3 Post processing analysis

Post progressing of simulation will be resulted in 3D field view, and vector of flow by time steps. The product will also show as range of velocity and temperature inside the channel. Later receiving velocity at specific place and outlet, those data will imported into Excel to find out the average velocity and characteristic of the flow at each specific position. Besides, mass flow rate reception will be noticed at outlet of each cases to see the quantities of mass which the system can produce.

On behalf of the analysis, the output data will be putting into groups in order to compare those 3 elements, velocity, temperature, or mass flow, from one to another heat flux support at each specific time of simulation. Thus, the result also discuss about flow characteristic inside the channel in essential positions such as near inlet, middle of channel, and near the outlet, at living room, and kitchen. Hence, 3D section views and flow vector would be attached.