CHAPTER 3

METHODOLOGY

3. METHODOLOGY

3.1 CFD simulation

In this work three-dimensional simulations have been carried out by means of commercial CFD software (ANSYS CFX v12.1). ANSYS CFX is one of the most widely used commercial codes for simulating engineering fluid flow due to its accuracy, robustness and convenience. ANSYS CFX solvers are based on the finite volume method. The fluid region is decomposed 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.

$$\frac{\partial}{\partial t}\int_V \rho \emptyset dV + \oint_A \rho \emptyset V . dA = \oint_A \Gamma \nabla \emptyset . dA + \int_V S_\emptyset \, dV$$

Continuous partial differential equations are discretized into a system of linear algebraic equations that can be solved on a computer. ANSYS CFX consists of four software's module that takes geometry and meshes and passes the information required to perform a CFD analysis (ANSYS CFX Introduction, 2009).

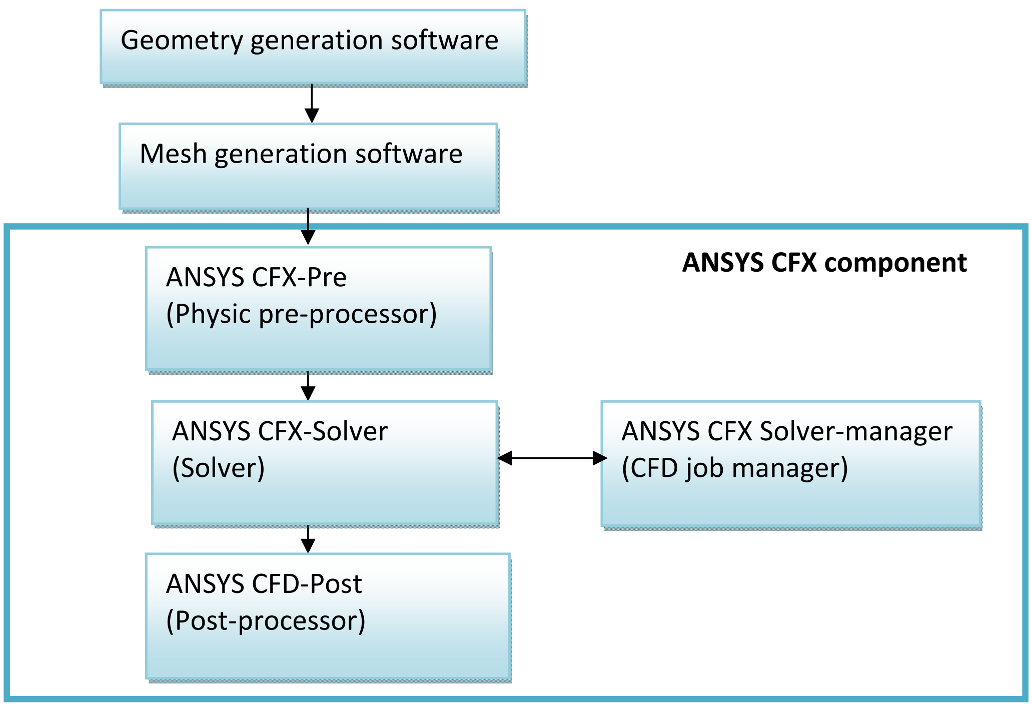

Figure 3.1: ANSYS CFX module (ANSYS CFX Introduction, 2009)

3.1.1 Pre-processing stage.

In the CFD simulation, pre-processing stage includes geometry creation, mesh development, physical properties set-up and the implementation of solving technique and parameters.

3.1.1.1 Geometry creation.

According to Norton et al (2007) there are three possible approaches to represent a model of a naturally ventilated building in computational domain of a CFD study. The possible model may consist of:

a) Both indoor and outdoor regions of naturally ventilated building.

b) Indoor and outdoor region divided into sub-domain and solved independently, with the solution being interpolated and match at the region interface.

c) Just the indoor environment.

The first methods offer the advantage of directly coupling both internal and external environment. However computation process may require large amount of time to complete because of the detail meshing required at the flow impingement and development area at the leeward and windward of the building respectively. However current development of more accurate meshing made this process more efficient. The second method was seldom being used. However some of large scale natural ventilation investigations have used it when direct coupling of indoor and outdoor environment cannot be done due to the insufficient computer resources. The third method has been shown to be flawed in some cases where the result of the simulation is different from the physical flow regime.

In this study, the first approach is taken where both indoor and outdoor region of the solar dryer greenhouse is created. The three dimensional computational domain (Figure 3.2) was 30m width x 30m length x 15m height including the modelled greenhouse with the drying chamber dimension of 4.5m width x 4.5 m length x 3.0 m height of drying chamber and 0.3m width x 0.3m length x 3.0m height of the chimney. This greenhouse was equipped with roof and side-wall opening. A roof opening is located on top of the chimney. Each wall was equipped with a window opening (1.5m × 0.5m) at 0.5m above the ground. The side windows opened by pivoting around their higher side. The greenhouse was covered with a 0.18mm thickness polyethylene films. For this simulation the greenhouse is empty, without any crop.

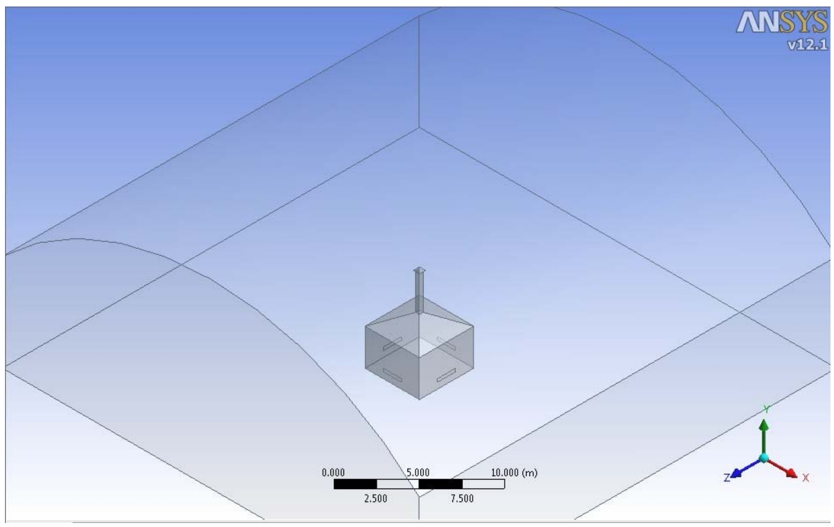

Figure 3.2: Three-dimensional computational domain.

3.1.1.2 Mesh development.

Adequate mesh resolution is important to ensure accurate result in CFD analysis. A tetrahedral type of mesh was used for this model (Figure 3.3).

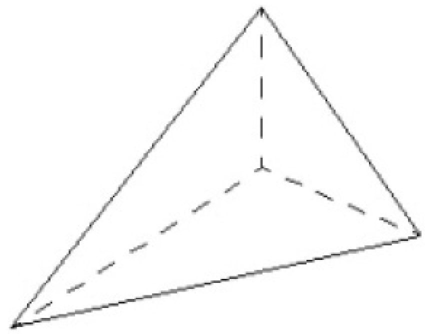

Figure 3.3: Tetrahedral mesh.

A maximum mesh surface size of 0.1 m is used for the greenhouse chimney and opening, 0.1 m for the greenhouse wall and floor and more coarse mesh size of 1.5 m was used for the environment. Figure 3.4 shows the model after meshing process.

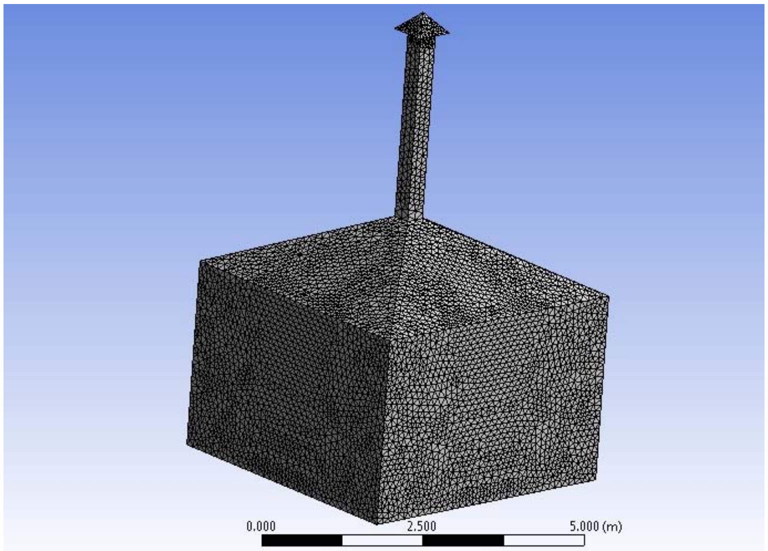

Figure 3.4: Model after the meshing process.

3.1.1.3 Physical properties set-up

The simulation is carried out for a steady state condition. Steady state condition is defined as those whose characteristics do not change with time and steady condition are assumed to have been reach after a relatively long time interval. For this simulation a physical timescale setting is used to make sure the steady state condition is achieved.

To predict the characteristic of the flow inside the solar dryer greenhouse, the Reynolds number is calculated based on the following equation:

$R_e = \rho u L / \mu$

Where R_e = Reynolds number

ρ = density (of air) (kg/m^3)

u = velocity (m/s)

L = characteristic length (m)

μ = dynamic viscosity (kg/ms)

From the calculation it shows that the Reynolds number is 26230 and it is predicted that the flow is going to be a turbulence flow. Therefore a turbulence model is added in the simulation.

For turbulence model the k-ε model has been used to represent the turbulence model for this simulation. k-ε model has proven to be stable and numerically robust and has a well established regime of predictive capability. The k-ε model is chosen because its offers a good compromise in term of accuracy and robustness.

For differencing scheme the high resolution differencing scheme is selected for this simulation. This scheme is chosen because it produces more accurate result compare to upwind and central difference scheme. The High Resolution scheme used a special nonlinear recipe for β at each node, computed to be as close to 1 as possible without introducing new extreme. The advection flux is then evaluated using the value of β and $\nabla\phi$ from the upwind node. This methodology involves first computing a ϕ_{min}

and ϕ_{max} at each node using a stencil involving adjacent nodes (including the node itself). Next for each integration point around the node, the following equation is solved for β to ensure that it does not undershoot ϕ_{min} or overshoot ϕ_{max}

$\phi_{ip} = \phi_{up} + \beta\phi.\Delta r$

Where ϕ_{up} is the value of upwind node and r is the vector from the upwind node to the ip. The nodal value for β is taken to be minimum value of all integration point value surrounding the node. The value of β is also not permitted to exceed 1.

ANSYS CFX uses the concept of domain to define the type, properties and region of the fluid, porous or solid. Domains are region of space in which the equations of fluid flow or heat transfer are solved. The computational domain of this study was divided into two. The first domain represents the air of the environment and the second domain represents the air inside the solar dryer greenhouse. The properties of both domains are assume to be the same and are as shown in Table 3.1. Both domains were connected using two types of domain interface. In this simulation domain interface is created for two purposes. First is to create a connection between two separate domains and second to be able to create a thin surface modelling. Thin surfaces modelling enable physics model such as heat transfer across a thin material or gap without needing to mesh the surface. The first domain interface (interface 1) represents the interface between the two domains at the opening of the greenhouse. Interface 1 is classified as general connection between fluid and fluid domain without any wall. The second domain interface (interface 2) represents the connection between the air inside the greenhouse and the environment with the wall in the middle. The second interface also classified as general connection but to model the heat transfer between the two domains with the greenhouse cover in between, thin material option is used where the material property for the thin material is set as polyethylene property with a thickness of 0.18 mm. Table 3.2 shows the property of polyethylene used in this simulation.

The solar dryer house ventilation is mostly developed based on buoyancy effect where the temperature difference between the outside and inside temperature will cause the air to move. For this solar dryer greenhouse the temperature different between inside and outside is expected not more than 30^{o}C even in the hottest time of the day, therefore the Boussinesq Approximation is selected for buoyancy model. As mentioned in chapter 2, the Boussinesq Approximation has been proven and used successfully in many CFD simulation studies previously.

Table 3.1: Domain properties

Properties	**Domain 1 and Domain 2**
Domain type	Fluid
Material	Air at 25^{o}C
Buoyancy model	Bussinesq approximation
Buoyancy reference temperature	25^{o}C
Heat transfer model	Thermal energy
Turbulence model	k-ε model
Thermal radiation model	Monte-Carlo
Spectral model	Gray

Table 3.2: Polyethylene properties

Properties	**Value**
Density	910 kg/m^3
Specific heat capacity	2700 J/kg.K
Thermal conductivity	0.52 W/m.K
Thickness	0.18 mm

The air is assumed to be the air at 25^{o}C, whose characteristic is as shown in Table 3.3.

Table 3.3: Air properties

Properties	**value**
Molar mass	28.96 kg/kmol
Specific heat capacity	1004 J/kgK
Dynamic viscosity	1.831e-05 kg/ms
Thermal conductivity	0.0261 W/mK
Refractive index	1
Absorption coefficient	0.01 /m
Scattering coefficient	0

3.1.1.4 Boundary conditions.

Boundary conditions are sets of properties or condition on a surface of a domain and are required to fully define the flow simulation. For domain 1, there are three boundary conditions which are the wall, inlet and outlet. In the simulation, solar radiation represents by the wall boundary condition. The solar radiation is set to be the directional radiation where the intensity value is the measured data at that particular time. The sun radiation direction for different times under consideration (10.30 am, 1.30 pm and 4.30 pm) is shown in Figure 3.5, 3.6 and 3.7. The wind speed and wind direction acting on the solar dryer greenhouse is represents as inlet and outlet boundary condition. The value for wind speed and direction are taken from the measured data of the actual condition.

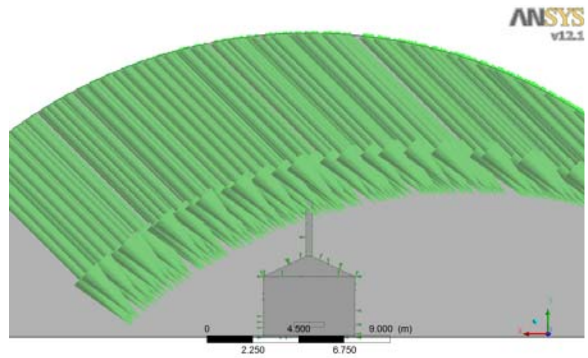

Figure 3.5: Sun radiation at 10.30 am

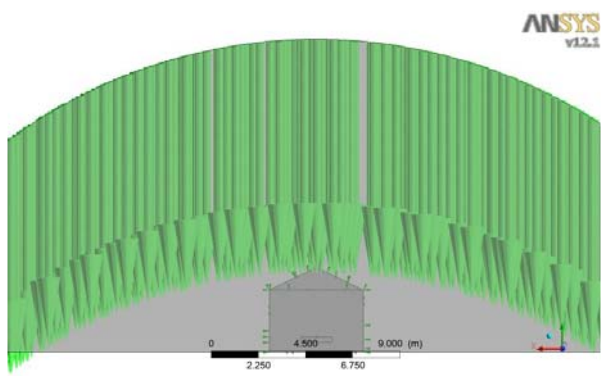

Figure 3.6: Sun radiation at 1.30 pm

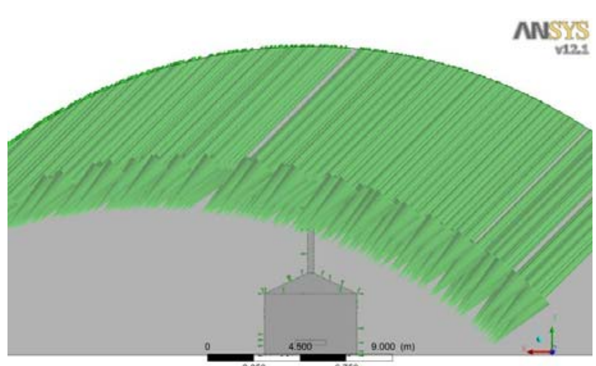

Figure 3.7: Sun radiation at 4.30 pm

3.1.2 Solving

Figure 3.8 shows the flowchart of solution process used in CFX solver (ANSYS CFX Solver theory guide, 2009).

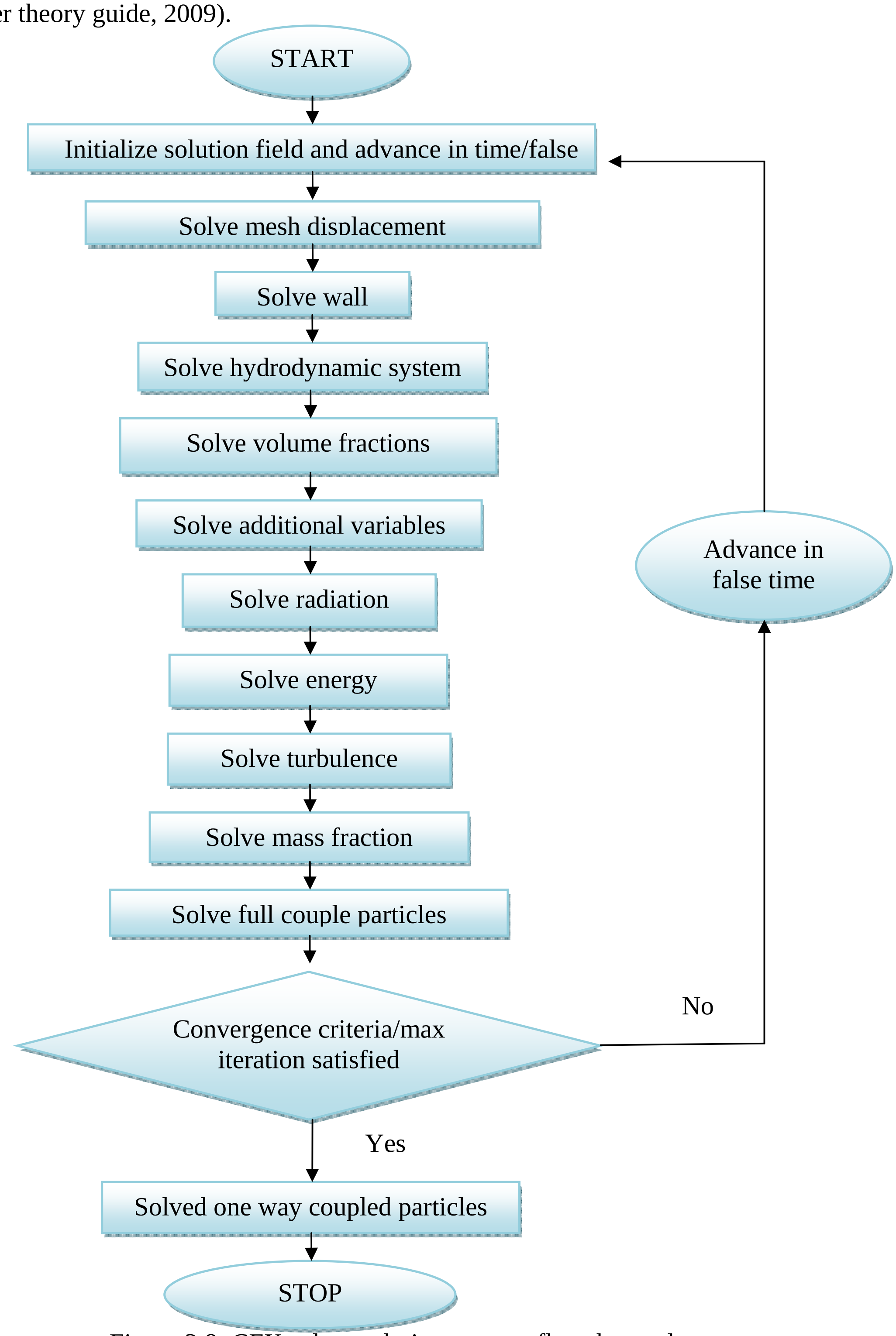

Figure 3.8: CFX solver solution process flowchart solver

(ANSYS CFX Solver theory guide, 2009)

3.1.3 Post processing

In the post-processing stage, the result from the ANSYS-CFX solver was visualized in the following forms;

i) Contour plot for temperature distribution.

ii) Vector plot for air flow velocity and direction.

3.2) Data collection

The data collection within the full scale solar dryer greenhouse was carried out with two main purposes. Firstly, is to measure the actual data as an input to the simulation. Secondly, is for validation of the CFD simulation. In this study, the physical measurement was carried out to measure the air temperature, solar radiation and wind speed inside and outside the solar dryer greenhouse. The data was collected for 24 hours on 13th March 2011 and 21st March 2011 for condition where roof and side opening is open and on 26th March 2011 and 04th April 2011 for condition where only roof opening is open. The equipment used to measure temperature is WatchDog data logger (Figure 3.9) with the specification as shown in Table 3.4. The equipment used to measure solar radiation is Silicon Pyranometer sensor (Figure 3.10) with the specification as shown in Table 3.5 and the equipment used to measure wind speed and direction is WatchDog 2000 series weather stations (Figure 3.11) with specification as shown in Table 3.6. Data for sun radiation, wind speed and wind direction were used as an input during the setup process.

Figure 3.9: Temperature sensor

Table 3.4: Temperature sensor specification

Brand name	WatchDog
Model	Model 150
Internal sensor	Temperature and relative humidity
Measurement range	-20^{o}C to 70^{o}C
Accuracy	±0.6^{o}C

Figure 3.10: Solar radiation sensor.

Table 3.5: Solar radiation sensor specification

Model name	WatchDog Silicon Pyranometer Sensor
Measure radiation range	300 to 1100 nanometre
Accuracy	±5%

Figure 3.11: Wind speed and wind direction sensor.

Table 3.6: Wind speed and wind direction sensor specification

Model name	WatchDog 2000 Series Weather Stations
Wind direction	2^o increment
Wind direction accuracy	$\pm 7^o$
Wind speed range	0 to 175 mph
Wind speed accuracy	±5%

The location of the temperature sensor inside the greenhouse is as shown in Figure 3.12:

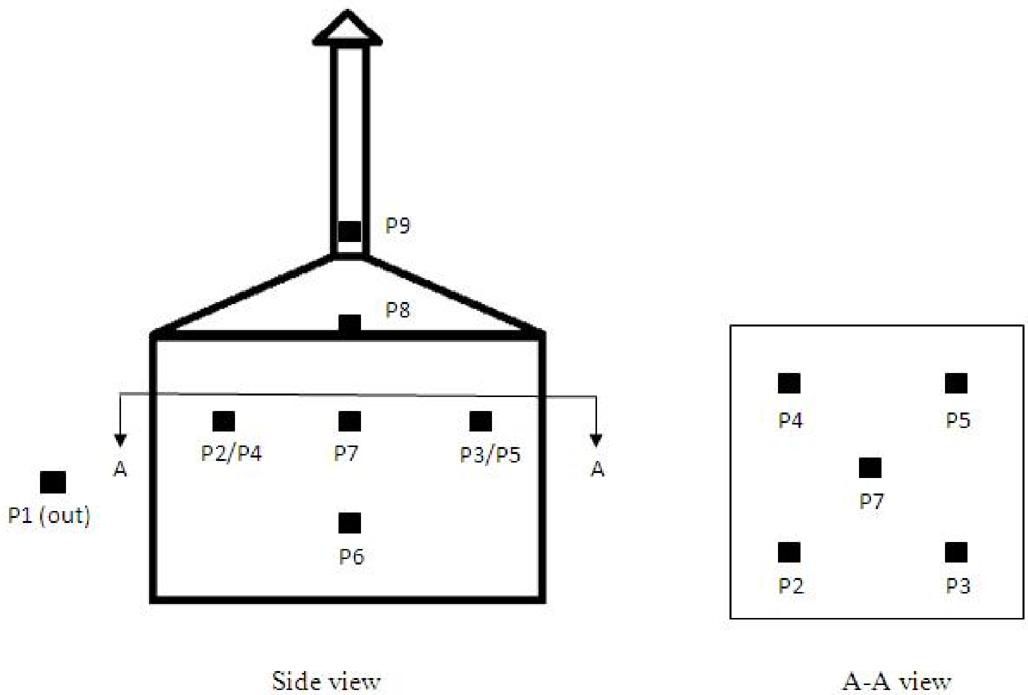

Figure 3.12: Location of temperature and humidity sensor inside the greenhouse

For wind speed sensor, due to limitation of the available equipment only one sensor was place inside the greenhouse, which is at the P7. One set of sensor to measure air temperature, solar radiation and wind speed was place 4m outside the greenhouse at 1.5m above the ground. Figures 3.13 and 3.14 show the actual sensors located inside and outside of the greenhouse.

Figure 3.13: Location of sensor inside greenhouse

Figure 3.14: Location of sensor outside greenhouse