CHAPTER 2

LITERATURE REVIEW

2. LITERATURE REVIEW

2.1 Solar dryer history

Drying is a simple process of removing excess water or moisture from a product in order to reach the requirement of standard specification moisture content. Drying is important especially to reduce the food product moisture content, as usually these have much higher water content than the one that is suitable for long preservation. Reducing moisture content of food product down to a certain level slows down the action of enzymes, bacteria, yeasts and molds. Thus food can be stored and preserved for long time without spoilage. Drying also is done with the aim of total removal of moisture until food has no moisture at all. Dehydrated food, when ready to use, is re-watered and almost regains its initial conditions. Drying is one of the most important post harvest process for agriculture product. It can extend shelf life of the product, improve the quality and reduce post-harvest losses due to waste. The transportation cost is also reduced as the weight is less since the water is taken out from the product during the drying process.

There are many methods available for drying process including hot air drying using heater, dielectric drying (radio frequency or microwaves being absorbed inside the material), freeze drying (drying method where the solvent is frozen prior to drying and is then sublimed) and solar drying. Increase in the fuel price and awareness to protect the environment had increase the potential usage of drying process using the solar energy. Murthy (2009) defines the solar drying process as a process where the solar radiation is used to evaporate the moisture present in the product. The solar energy is used to heat large volume of air and this air is allowed to flow over the product to remove and take away the moisture. Drying by solar energy is an economical procedure for agricultural products, especially for medium to small agriculture industry, to prevent excess of damage product after long storage. It is friendly to the environment. It used for domestic up to small size drying of crops, agricultural products and food product, such as fruits, vegetables and herbs where its contribute significantly to the economy of small agricultural communities and farms.

Traditionally, direct sun drying was performed by spreading the product on the platform directly under the sun without any cover. However, according to Belessiotis and Delyannis (2010), this method has many disadvantages including:

1) No scientific observations during long period of drying. The whole process depends on the experience of unskilled labour.

2) No standard control on the final quality of the final dried product, it is just based on observation and experience.

3) Very slow rate process, depending on the nature of the product and weather condition.

4) The product is exposed directly to all kinds of weather changes, such as rain and strong winds which can rot or destroy the material. Bad weather conditions on the other hand facilitate growing of bacteria and molds.

5) They have very large qualitative and quantitative losses due to natural attack conditions closely related to the open-air procedure such as dusting, rotting when weather conditions are not favourable, attacks by insects, rodents, birds and other unpredictable conditions.

In other words, the quality of finished product is inconsistent and difficult to control. To overcome these problems, solar dryer was developed. Solar dryer is an equipment which uses solar energy to heat up air and dry the food product. A solar dryer minimizes almost all the problems faced during conventional sun drying method, thus improving the quality of the dried product. According to Belessiotis and Delyannis (2010) in comparison to conventional sun drying, the use of appropriate solar dryers lead to a reduction of the drying time up to 50% and to a significant improvement of the product quality in terms of colour, texture and taste. Furthermore, contamination by insects and micro organisms can be prevented. The storage losses can be reduced to a minimum while the shelf life of the products can be increased significantly.

Several studies have been carried out to develope solar dryers for agriculture products. Figure 2.1 below shows the classification of available solar dryers for agriculture products based on the design of the system component and mode of utilization of solar energy (Fudholi et al., 2010)

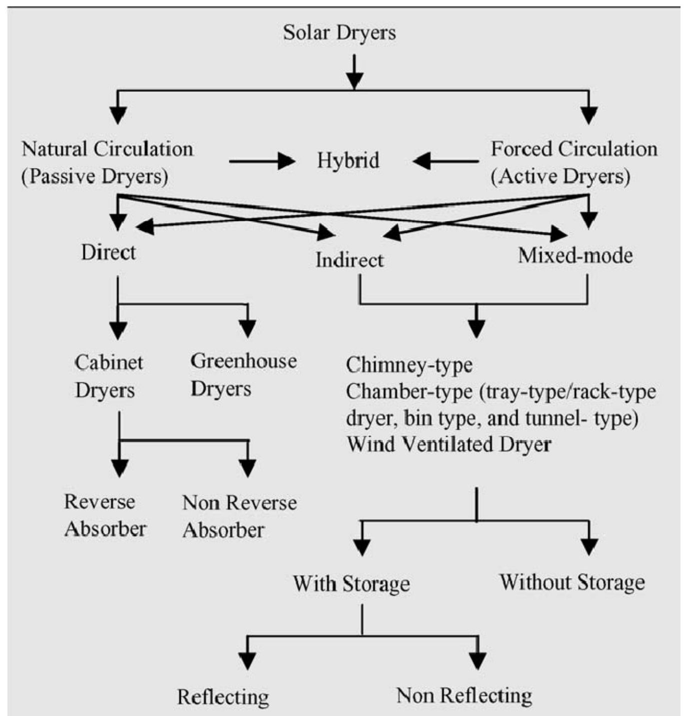
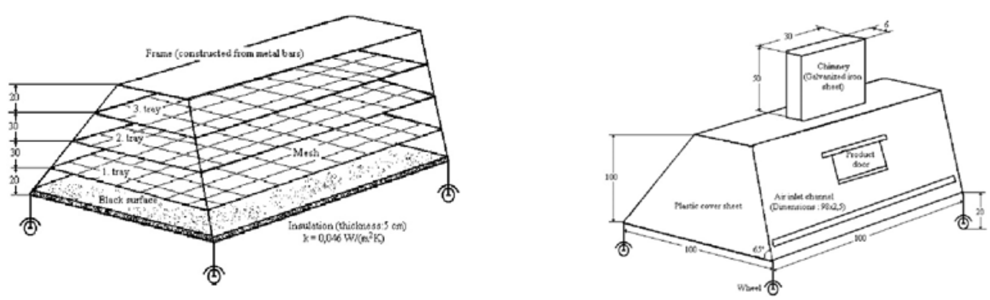


Figure 2.1: Classification of solar dryers and dryer modes.

(Fudholi et al., 2010)

2.2 Design and development of greenhouse solar dryer

As shown in Figure 2.1 there are two types of solar dryer greenhouse, which are natural circulation solar dryer greenhouse (passive dryer) and forced circulation solar dryer greenhouse (active dryer). Compared to passive greenhouse dryers, in the active solar dryer greenhouse the hot air was circulated by means of a ventilator. Naturally ventilated greenhouse for drying applications have been reported in the past study and it was reported that the dryer produced high quality dried food grades up to the desired moisture content level (Sethi and Arora, 2009). Several design of solar dryer greenhouse has been study from a simple small scale to a large scale solar dryer. Koyuncu (2006) have designed, constructed and tested the performance of two different types of natural circulation greenhouse type crop dryers (Figure 2.2a and Figure 2.2b). He have developed a small scale (1 m x 1 m) greenhouse type solar dryer consist of framework constructed from black coated metal bars, corrosion-resistant plastic mesh and a black coated solar radiation absorber surface. The frameworks of the dryers were clad with clear polyethylene sheet on the all sides. The cladding at rear side was arranged to allow putting the moist products into the drying chamber or getting dried product from there. The clear plastic cladding at the bottom edge of the front side and rear side was also arranged to allow air to flow into the chamber, while the rectangular stream at the top of the end served as the exit for the moist exhaust air. The results of the study show that the greenhouse solar dryers increase the ambient air temperature by 5 to 9^{o}C, and these dryers are 2 to 5 times more efficient than plastic mesh platform type open sun dryer. The dryers with a drying air outlet chimney give better value of air mass flow by increasing the air velocity. Ekechukwu and Norton (1996) have designed and developed natural convection solar dryers which are suitable for the drying of most crops. The design is a simplified design of the typical greenhouse type natural convection solar dryer (Figure 2.3). It consists of a cylindrical polyethylene-clad vertical chamber, supported structurally by a steel framework and draped internally with a selectively absorbing surface. They reported that performance of the dryer studied was dependent largely on the variations in ambient temperature and relative humidity. The results obtained from experimental solar chimneys in this study, if designed properly could maintain chimney air temperatures consistently above the ambient temperature which would enhance the desired buoyancy induced airflow through the chimney and drying rate. Linear correlations have been obtained between the drying rate measured experimentally and a group of ambient and crop parameters. Janjai et al. (2011) reported that the large scale tunnel type solar dryer using polycarbonate cover (Figure 2.4) have been tested and demonstrated potential of drying chilli, coffee and banana. The black painted solar absorber surface raises the efficiencies of the dryers. In general, the solar dryer offers much superior quality product compares to open sun drying



(a) (b)

Figure 2.2: Different types of natural circulation greenhouse type crop dryers.

(Koyuncu, 2006)

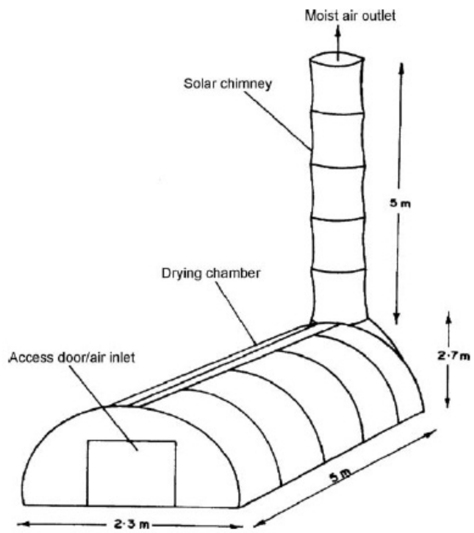


Figure 2.3: Natural convection solar dryer greenhouse.

(Ekechukwu and Norton, 1996)

Figure 2.4: Large scale tunnel type solar dryer using polycarbonate cover.

(Janjai et al., 2011)

2.3 CFD application in greenhouse development.

Computational fluid dynamics is a sophisticated design and analysis tool that uses computers to simulate fluid flow, heat and mass transfer, phase change, chemical reaction, mechanical movement, and solid and fluid interaction. The technique enables a computational model of a physical system to be studied under many different design constraints. CFD had the ability to efficiently develop spatial and temporal field solutions of fluid pressure, temperature and velocity, and has proven its effectiveness in system design and optimisation within many industries. The application of CFD in the agricultural industry is becoming more important due to the above mentioned factor. The versatility, accuracy and user-friendliness offered by CFD had led to its increased take-up by the agricultural engineering community to study and analyse the indoor climates of the greenhouse. This is reported by Norton et al. (2007) by the increase in peer reviewed papers of CFD applications in agriculture buildings (Figure 2.5).

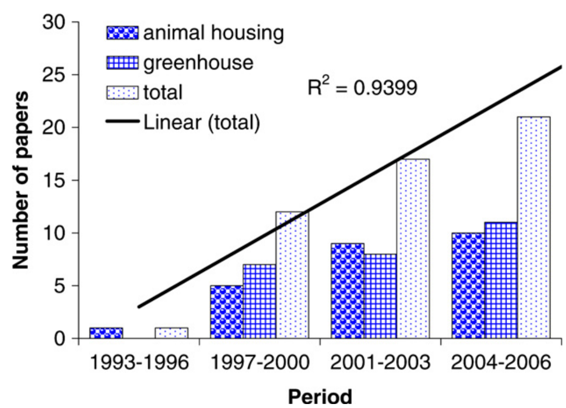


Figure 2.5: The number of published peer-reviewed publications of CFD applied to the ventilation of agricultural buildings.

(Norton et al., 2007).

2.3.1 Two dimensional and three dimensional CFD analyses.

Many studies around the world have been carried out to investigate the indoor climate pattern of greenhouse structure using the CFD simulation. When CFD first used to model airflow inside a room, it was assume that symmetric rooms with two dimensional boundary conditions had a two dimensional airflow pattern. Molina-Aiz et al. (2004) have done a study to analysed using computational fluid dynamics effect of wind speed on the natural ventilation of an Almer´ıa-type greenhouse. He is using the commercial program ANSYS/FLOTRAN v6.1 based on the finite elements method. The experiment was carried out in an Almer´ıa-type greenhouse equipped with top and side ventilation. The importance of roof ventilators for efficient ventilation in Almer´ıa-type greenhouses was observed. The air temperature distribution shows a gradient from the sidewalls towards the centre of the greenhouse due to the movement of the hot air rising towards the roof vent, and a vertical gradient due to the movement of the air above the surface of the ground absorbing solar energy at floor-level. Maximum air velocity inside the greenhouse was reached near the side vents, with the lowest values observed in the middle of the greenhouse. The velocity decrease produced in the windward opening between the outside and inside of the greenhouse was 75 to 85% in every case. The air velocity in the leeward area remained more or less constant around $0.3ms^{-1}$, as the result of the "chimney effect". The model was verified by comparing the numerical results with experimental data. The differences between values predicted by the CFD models and those measured were from 0.0 to $0.36ms^{-1}$ for air velocities, and from 0.1 to 2.1 $^{\circ}$C for air temperatures. Although this study shows a good agreement between predicted and measure data, later study found that many two dimensional CFD investigation should be view with caution unless there is proof that these pattern would occur in a physical situation because two-dimensional CFD predictions of such a large-scale structure cannot be generalised (Norton et. al 2007). Observation made by Molina-

Aiz et al. (2005) as quote by Norton et al. (2007) in a 2D study shows that ventilation rate is reduced of around 88% when the span of the building was increased from scale 1 to 5. Large temperature gradients were also observed in the middle of the building under all vent configurations, and insect screens were seen to greatly affect the ambient temperature and velocity difference between indoor and outdoor environments.

The study using three dimensional CFD using Airpack 2.10 Fluent Inc. Software was carried out by Pontikakos et al. (2006) to study the efficiency of natural ventilation in a commercial twin-span greenhouse. In his study, three-dimensional patterns for temperature and airspeed inside the greenhouse were generated, using specific boundary conditions. The CFD simulation was applied to an empty twin-span greenhouse with a floor area of 980 m^2 with low density polyethylene cover and two side continuous opening and one roof opening. The simulations assumed a sunny summer day with different airspeeds (0.0, 1.0, 2.0 and 5.0 m/s) in three directions for each non zero airspeed and three different temperature values for each airspeed (20.0, 25.0 and 30.0^{o}C). The results showed that the external boundary temperature, wind directions and airspeed are the crucial parameter on the pattern of the internal greenhouse temperature. Campen and Bot (2003) also have used the three dimensions CFD (using Fluent v.5.2 software) to study the ventilation of a Spanish 'parral' greenhouse. The simulation was verified by experimental result using tracer gas measurement. The result shows the simulation data and the experiment data was resembled within 15%. From his study he concludes that a three dimension CFD model was able to determine the greenhouse specific ventilation characteristic. The CFD calculation also indicates that the ventilation rate is largely depend on wind direction and the wind speed was correlated linearly with ventilation rate without the buoyancy effect.

2.3.2 CFD analysis for mono span greenhouse with pitched roof design.

The usage of CFD to investigate the mono-span type greenhouse with pitched roof design has been carried out by Shklyar and Arbel (2004). They study the effect of vent angles and wind direction on wind induced ventilation in an isothermal, pitched roof single span type of greenhouse. They found that by changing the vent angle from 20^o to 40^o can doubled the ventilation rate. They also found that the change in wind direction from 45^o to 90^o had more effect to ventilation rate compare to wind direction from 0^o to 45^o. Which mean ventilation rate induced by perpendicular wind was almost five times greater than when wind is parallel to the structure. Another study on pitched roof greenhouse was conduct by Campen (2005) to study the climate distribution inside the greenhouse under different wind speed and direction, various porous screen and different structure configuration. The result of the study shows that the resistant from a screen net gives more influence on the ventilation rate compare to wind direction. At low wind speed, 0.5 ms^{-1}, a greenhouse with roof and sidewall ventilation gave a large temperature increase inside the building. This was caused by both buoyancy and wind force counteracting one another. The temperature inside the building is increasing when the greenhouse design configuration was lengthened from 12m to 36m.

2.4 Governing equation

2.4.1 Navier-Stokes equation

CFD programs numerically solve Navier-Stokes and energy equation. The Navier-Stokes equations are the basic governing equations for a viscous, heat conducting fluid. It is a vector equation obtained by applying Newton's Law of Motion to a fluid element and is also called the momentum equation. It is supplemented by the mass conservation equation, also called continuity equation and the energy equation. The term a Navier-Stokes equation is used to refer to all of these equations. These equations constitute a micro model of fluid motion, and required time consuming iterative technique to solve them. In ANSYS CFX the instantaneous equation of mass, momentum and energy conservation can be written as follows (CFX-Solver theory guide, 2009):

a) The continuity equation

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho U) = \qquad (2.1)$$

b) The momentum equation

$$\frac{\partial(\rho U)}{\partial t} + \nabla.(\rho U \otimes U) = -\nabla p + \nabla.\tau + S_M \qquad (2.2)$$

Where the stress tensor, τ is related to the strain rate by

$$\tau = \mu(\nabla U + (\nabla U)^T - \frac{2}{3}\delta\nabla.U \qquad (2.3)$$

c) The total energy equation

$$\frac{\partial(\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla.(\rho U h_{tot}) = \nabla.(\lambda \nabla T) + \nabla.(U.\tau) + U.S_M + S_E \qquad (2.4)$$

Where h_{tot} is the total enthalpy, related to the static enthalpy h (Top) by:

$$h_{tot} = h + \frac{1}{2}U^2 \qquad (2.5)$$

The term $\nabla.(U.\tau)$ represents the work due to viscous stresses and is called the viscous work term

The term $U.S_M$ represent the work due to external momentum sources and it's currently neglected.

Besides Navier-Stokes equations, account must also be taken of the additional processes that may influence the dynamics of ventilation system. The governing equation may need to be modified with additional physical models or assumption to fully represent the physical situation. This may includes turbulence and porous media model, and models that describe occupant inside the building.

2.4.2 Turbulence model

Turbulence motions are usually associates with ventilation primarily due to high flow rates and heat transfer interaction involved in the flow field. Currently there are many turbulence models available and many studies have been conducted to validate these turbulence model. One of best performing turbulence model that being used in the modelling of agriculture buildings and application is the standard k-ε model (Norton, 2007). This model introduces two variables which are the turbulence energy, k and its dissipation rate, ε. The variable k corresponds to the turbulence velocity of the mixing length model and the variable ε corresponds to mixing scale. The model adds two transport equations corresponding to the two new variables to the usual transport equations describing the flow. In this way no mixing length needs to be defined and therefore complex flow governed by elliptic equation, such as recirculation flows can be solve. The standard k-ε model equation is written as follow:

For turbulent kinetic energy, k

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\varepsilon - Y_M + S_k \qquad (2.6)$$

For dissipation, ε

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(P_k + C_{1\varepsilon}P_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \quad (2.7)$$

Where,

μ_t is turbulent viscosity, P_k is production of k, S is the modulus of the mean rate of strain tensor, P_b is effect of buoyancy, $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, C_μ=0.09, $\sigma_k = 1.0$ and $\sigma_\varepsilon = 1.3$.

One important weakness of the standard *k-ε* model is that it does assume equilibrium. Meanings that one's turbulence energy is generated at the small wave number end of the spectrum (large eddies), it is equally distribute to the whole spectrum. Generally this is not the case because the transfer of energy from the large eddies, where turbulence is produce to the small eddies and turbulence dissipation occurs is not automatic. A considerable length of time intervenes between the production and the dissipation of turbulence. Moreover this process can be affected by the interaction with obstruction and walls (Mistiiotis et. al, 1997).

To overcome the above mention problem, two scale *k-ε* turbulence model have been introduced. Chen-Kim model and the renormalized group (RNG) *k-ε* model is the two scale *k-ε* turbulence model that introduce to model air movement within and around buildings. Chen-Kim model improves the dynamic response of the equation for *k* by introducing a second time scale, *k/p* where *p* is the volumetric production rate of *k*. Furthermore several of the standard model coefficients are adjust so that the models maintain good agreement with experiment data on classical turbulent shear layer flows. In RNG model, large scale is described by renormalized group method, where the effect of the small scale is represented by modified transport coefficient. In two dimension wind induced ventilation, Mistriotis et al. (1997) showed that better qualitative agreement with experiment observed flow patterns can be achieved with two scale k-ε model than with standard k-ε model. Later Roy and Boulard (2005) and Brugger et al. (2005) as quote by Norton (2007) also found a difference in ventilation rate predicted by two scale k-ε model and standard k-ε model. However when heat transfer is coupled with the field flow all k-ε model seem to perform similarly and shown similar agreement with experimental data.

Another new turbulence model is the Shear Stress Transport (SST) which combine the k-ε and k-ω models using a blending function was found to predict airflow in good agreement with experimental data not only in the near wall region of the flow but also in the free stream. Toma's et al (2007) suggests that this model should be considered when buoyancy force plays a major role in driving the flow within a ventilated space. Another turbulence model that has been used in previous study is the Reynolds stress closure models (RSM). Reynolds stress closure models (RSM) have exhibited far superior predictions for flows in confined rooms where adverse pressure gradients occur. These models have recently been shown to enhance the CFD predictions of primary and secondary flow patterns in empty-isothermal rooms and in loaded rooms with heat transfer (Toma's et al, 2007). However according to Toma's et al (2007) the weakness of RSM is extra computational time and memory required in solving the flow regime, alongside the difficulties in attaining good convergence behaviour.

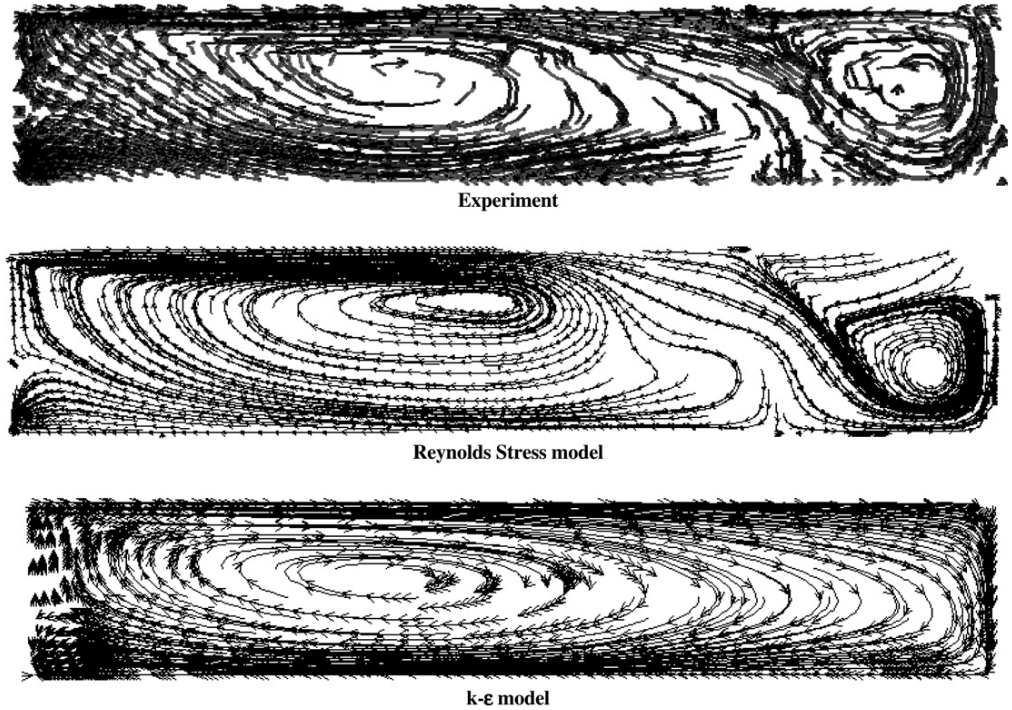


Figure 2.6: Influence of the turbulence model on the flow pattern at the symmetry plane of a confined flow.

(Moureh and Flick, 2005 as quote by Norton et al, 2007)

2.4.3 Heat transfer model.

In this study the main factor that influences the internal climate of the solar dryer greenhouse is the sun radiation. Heat transfer in a fluid domain is governed by the energy transport equation. In this study the radiation effect is significant compare to the convective and conductive heat transfer rates, therefore to account for radiation, Radiative Intensity Transport Equations (RTEs) are solved. Several radiation models are available in ANSYS CFX which provides approximate solutions to the RTE are (ANSYS CFX-Solver theory guide, 2009):

1) Rosseland Model (Diffusion Approximation Model)

2) P-1 Model (Gibb's Model/Spherical Harmonics Model)

3) Discrete Transfer Model (DTM) (Shah Model)

4) Monte Carlo Model

Each radiation model has its assumptions, limitations, and benefits. Rosseland model is a simplification of RTE for the case of optically thick media. Its introduce new diffusion term into the energy transport equation with a strongly temperature dependant diffusion coefficient. The P1 model is also a simplification of RTE. It assume that the radiation intensity is isotropic or direction independent at a given location in space. DTM assume that the scattering of the radiation is isotropic. The Monte Carlo Model assumes that the intensity is proportional to the differential angular flux of photon. The optical thickness should be determined before choosing a radiation model. For thin optical meaning that the fluid is transparent to the radiation at wavelengths where the heat transfer occurs and the radiation only interacts with the boundaries of the domain. While thick optical means that the fluid absorbs and re-emits the radiation. For optically thick media the P1 model is a good choice that gives reasonable accuracy without too much computational effort. Monte Carlo and DTM model also can be used for optically thick media, but the P1 model uses far less computational resource. For optically thin media, the Monte Carlo or DTM may be used but for models with long or thin geometries DTM can be less accurate. While Monte Carlo uses the most computational resources compared to DTM.

2.4.4 Modelling for buoyancy.

The ventilation in the solar dryer greenhouse in this study is natural ventilation. In natural ventilation, natural convection occurs when temperature differences in the air, resulting in density variations. This is called the buoyancy driven flow. Buoyancy driven flow is generated in the air when part of the air is heated or cooled by a surface. This causes motion in the air as the warm air rises and the cool air is then moved to the surface and will become heated. Climatic variables inside greenhouse are a function of varying flow properties caused by heating and cooling of the air. There are two main methods of modelling the density variation that occur due to buoyancy. First is the Boussinesq approximation. This has been used successfully in many housing application. The density variation according to Boussinesq approximation is as below (Toma's et al, 2007)

$$\rho = \rho_{ref}[1 - \beta(T - T_{ref}) \qquad (2.8)$$

Where β is the thermal expansivity:

$$\beta = -\frac{1}{\rho}\frac{\partial\rho}{\partial T}|_p \qquad (2.9)$$

And T_{ref} is the buoyancy reference temperature.

The assumption involved in this approximation involved:

a) Density differentials in the flow are only required in the buoyancy term of the momentum equation.

b) There is linear relationship between temperature and density, with all other extensive fluid properties being constant.

c) The temperature different in the flow field is less than 30^{o}C.

The main drawbacks of this approximation is that its only consider dry air as the fluid medium, while in actual condition most climatic flow will involved a mixture of dry air and moisture. The extended version of Boussinesq approximation was derived by Gan (1994) which describe the density of moist air as a function of temperature and moisture concentration. The extended version can be expressed as

$$\rho = \rho_{ref}[\beta(T - T_{ref}) + \beta_C(C - C_{ref})] \qquad (2.10)$$

Where C is water concentration and

$\beta = -\frac{1}{\rho}\left(\frac{\partial\rho}{\partial T}\right)_{P_{at}}$ and $\beta_C = -\frac{1}{\rho}\left(\frac{\partial\rho}{\partial C}\right)_{P_{at}}$ However despite the enhance relation, according to Toma's et al (2007) this model is not commonly used in the literature. For condition where there is a large temperature different, the Boussinesq approximation is not sufficiently accurate. Therefore another method is used where t is done by treating the air as an ideal gas and expressing the density different by means of the ideal gas equation:

$$\rho = \frac{P_{ref} W_a}{RT} \tag{2.11}$$

In this equation the density of the fluid is dependent on temperature and composition but not pressure. Although this equation may provide an accurate description of the density variation within the flow regime, it has been found to have impact on convergence behaviour of CFD solution.