International Food Research Journal 21(3): 1089-1096 (2014)
Journal homepage: http://www.ifrj.upm.edu.my

Simulation of spray drying on *Piper betle* Linn extracts using computational fluid dynamics

[1]Cher Pin, S., [1,*]Rashmi, W., [2]Khalid, M., [1]Chong, C. H., [3]Woo, M. W. and [4]Tee, L. H.

[1]*School of Engineering, Taylor's University, Subang Jaya, 47500 Selangor, Malaysia*
[2]*Division of Manufacturing and Industrial Processes, University of Nottingham, 43500 Semenyih, Malaysia*
[3]*School of Engineering, Monash University, VIC 3800, Australia*
[4]*School of Engineering, Monash University, 46150 Bandar Sunway, Malaysia*

Article history

Received: 23 September 2013
Received in revised form: 22 January 2014
Accepted: 23 January 2014

Keywords

Spray drying
CFD
Betel leaf extract
k-ε turbulent model

Abstract

The drying of *Piper betle* Linn (betel) leaf extract using a lab scale spray dryer was simulated using Computational Fluid Dynamics (CFD). Three different turbulent models (standard k-ε, RNG k-ε and realizable k-ε) were used in the present study to determine the most suitable model for predicting the flow profile. Parametric studies were also conducted to evaluate the effect of process variables on the final moisture content. Four different initial droplet sizes (36, 79, 123 and 166 μm) were tested with four sets of combination of hot air temperature (140 and 160°C) and feed rate (4, 9.5 and 15 ml/min). It was found that standard k-ε is the most suitable turbulent model to predict the flow behaviour Moreover, the lowest final moisture content present in samples was obtained at 140°C and a feed rate of 15.0 ml/min.

Introduction

Studies have proven that betel leaf possesses many beneficial effects such as antioxidant properties, antidiabetic activities and antimicrobial capabilities which results in improved human health. There is a great potential to extract the phytochemicals within the leaf and convert it into consumable food products. Spray drying technique can be used to convert the leaf extract into powder form to increase the life span of the extract (Arambewela *et al.*, 2005). Spray dryers are widely used in processes related to food, pharmaceutical, minerals and other product processing (Fletcher *et al.*, 2006). It is used to transform a fluid state feed into a dry particulate form by spraying the feed into a hot drying medium. The drying operation is a continuous process. Usually, the feed is concentrated before introducing into the spray dryer. The optimum condition can be manipulated at the atomization stage in order to obtain the desired characteristics of the dried product. Then, atomized feed is brought into contact with hot air. Approximately, 95% of the water contained in the feed evaporates in few seconds. Finally, the powder formed will be separated out using a cyclone separator. Wet scrubbers are used to purify and cool the air before it is released into atmosphere (Patel *et al.*, 2009).

Spray drying process is governed by a few critical parameters, for instance, inlet and outlet temperature of air, feed flow rate, aspirator rate, solid content, surface tension and even the material used to construct the nozzle in order to produce high quality product. The higher the air inlet temperature, faster will be the moisture evaporation rate. However, the chemical properties of the powder formed may be distorted due to high temperature. Moreover, the solid content must be taken into account in order to maintain proper atomization so that the formation of droplet is appropriate (Patel *et al.*, 2009). The quality of the end product using spray drying process depends on its processing conditions. Typically, the product from spray drying process, for instance, *Piper betle* L. extract considered to have good quality which has maximum hydroxychavicol content, minimum moisture content, minimum particle size and minimum hygroscopicity, respectively (Tee *et al.*, 2012). Different parameter settings will result different air flow pattern inside the chamber during the drying process. Therefore, the airflow pattern is said to be the main factor affecting the quality of the dried powder product (Kieviet *et al.*, 1997). However, the modelling of the flow profile in the spray dryer at different operating parameters is inadequately developed (Fletcher *et al.*, 2006). CFD can be used to predict the complex flow behaviour during the spray drying process by solving the governing equations numerically (Langrish and Fletcher, 2001). Simulation of the spray drying process using CFD had been started since late 1980s by Reay (1988). A few years later, Masters (1994) outlined the application of CFD by dryer manufacturers. This simulation software is commonly used to improve the spray dryer design and overcome some operational difficulties such as

*Corresponding author.
Email: *RashmiGangasa.Walvekar@taylors.edu.my*

wall deposition of dried product. It will illustrate the changes in both physical and chemical properties of the fluid during the spray drying process. This information can be used to improve the design of the spray dryer to obtain a desired product quality. With the aid of CFD, the fluid behaviour can be predicted without using the tedious, costly and time consuming experimental methods.

Different spray dryer models require a different simulation setup to model the drying process. Although there are few research studies reported on the modelling of the spray drying process using CFD, the selections of the physical phenomena and boundary conditions for the simulation setup for Mini Spray Dryer B-290, Buchi are not well explained and discussed. This will greatly affect the precision and accuracy of the simulation result predicted using CFD to model the spray drying process. The software uses this information to conduct numerical analysis by solving complex governing equations. The more detailed information provided with lesser assumptions, the simulation result will be more reliable with higher accuracy. Thus, the objective of this study is to determine the most suitable CFD setup model that can be used for modelling the spray drying process accurately. Then, the flow profiles for different initial droplet size at different hot air temperatures and feed rates will be studied. The results are compared with the drying kinetics studies (Roustapour *et al*., 2009; Tee, 2011).

Numerical Methodology

Geometry

The simulations were performed based on the experiments conducted by Tee (2011) in a lab scale cylinder-on-cone spray dryer (Mini Spray Dryer B-290, Buchi, Switzerland) as shown in Figure 1 and its schematic diagram is shown in Figure 2. The heating medium and the feed are flowing concurrently. This study is focused to perform simulation analysis of the fluid flow behaviour inside the spray dryer. Therefore, the computational domain (geometry) was only constructed for the spray dryer where the separation equipment (cyclone separator) was not included. Moreover, it was assumed that the air flow inside the dryer during the drying process was not swirling (swirl angle is assumed to be zero). To reduce the complexity of the simulation and also decrease the computational time, the scheme of the spray dryer's chamber is assumed to be axisymmetric and therefore 2D model was used. The geometry of computational domain was created using Design Modeller as shown in Fig. 3 with the actual spray dryer dimension. Its

Table 1. Characteristics of spray dryer

Part	Length (m)
Cone Height	0.4900
Cylinder Height	0.1100
Cone Diameter	0.0800
Air Inlet Diameter	0.0400
Nozzle Diameter	0.0070
Outlet Diameter	0.0175

Figure 1. Lab Scale Spray Dryer Buchi B-290 [15]

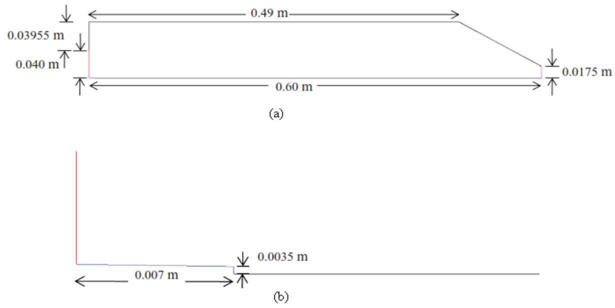

Figure 2. Dimensions of mini spray dryer (b) Dimensions of feed inlet

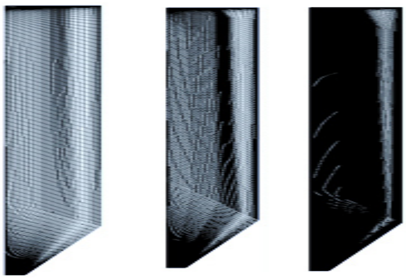

Figure 3. Geometry of spray dryer (a) 0.015 mesh size (b) 0.01 mesh size (c) 0.005 mesh size

characteristics are shown in Table 1.

Meshing

Once the computational domain was generated, geometry was sub-divided into smaller cells to have a better understanding of the fluid behaviour in different areas inside the spray dryer. The number of sub-domains, in fact, is governing the accuracy of the CFD simulation result. Typically, the higher number of sub-domains will have better accuracy for CFD solution. The fineness of the sub-domains will affect the accuracy of a solution, the cost of the computer

Table 2. Meshing for grid independence test

Type	Mesh Size (m)	No. of Mesh
1	0.015	10300
2	0.010	35442
3	0.005	128879

hardware and also the time required for the calculation. Meshes are considered to be optimum when they are non-uniform. Therefore, regions such as walls, inlet and outlet were having relatively finer meshes than the middle region. This is because the water flow inside the reactor will have greater variation in those specific areas and the changes of physical properties of the water were interested to know.

Grid independence test

A grid independent test is required in most of the CFD studies to check whether the mesh size has any significant effect on the accuracy of the simulation results. In order to conduct grid independence tests, three different sizes of mesh were created as shown in Figure 3. A similar flow problem would be run on these computational domains with different mesh size. Results showed that if there was no significant difference in the fluid flow behaviour, the mesh size would be said to be independent of the simulation accuracy. Else, the mesh size with simulation result which was closest to the experimental data would be chosen. Table 2 shows the number of elements and the maximum size of cells for the each mesh generated above.

Turbulence models

For turbulence models, since there is no swirling flow in the drying chamber, it is appropriate to apply the standard k-ε model for simulating this type of flow (Launder and Spalding, 1973). Since the fluid is in turbulent flow, it is very difficult to model every single particle behaviour in the flow. Thus, the properties of the fluid were fluctuating significantly due to the eddy motions which created strong mixing. Therefore, in order to simplify the modelling, the instantaneous transport properties could be discussed in terms of the mean and fluctuating components. When the mean and fluctuating value equation was substituted into the Navier-Stokes equations, it would form the Reynolds-averaged Navier-Stokes (RANS) equation (Jongebloed, 1973). The k-ε model is derived semi-empirically using two equation turbulence model which is dependent on an exact solution for turbulent kinetic energy (k) and the turbulent dissipation rate (ε). In order to model Reynolds stress, the k-ε model applies the Bousinesq approximation so that the Reynolds stresses can be related to the mean velocity gradients (Jongebloed, 1973). The transport equations for the standard k-ε model are for the transport of

Table 3. *Piper betle* L. extract properties

Properties	Value
Concentration	10 brix
Density	1.038 g/cm^3
Viscosity	2.29 cP
Thermal Conductivity	0.605 W/m.K
Specific Heat	2375 J/kg.K
Boiling Point	351

turbulent kinetic energy, k, and its dissipation rate, (Oakley, 1994).

Realizable k-ε model could be determined by obtaining the point that the average normal stress becomes negative. On the other hand, RNG k-ε model was developed to renormalise the Navier-Stokes equations. This approach had the ability to derive a turbulence model which was similar to the k-ε model. However, the equations were slightly modified as the effect of the different scales of motion through changes to the production term were taken into account (Yakhot *et al.*, 1992).

The flow in the dryer is two-phase. Therefore, a coupling between continuous and discrete phase was required to model this type of flow. The continuous phase calculation was performed to model the air and discrete phase was used to model the droplets (Huang *et al.*, 2004). The governing equations for continuous and discrete phases are shown as below:

Continuous Phase – Momentum Equation

$$\frac{\partial(\rho u_i u_j)}{\partial x_i} = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_i}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \rho\overline{u_i' u_j'}\right] + M_F \quad (1)$$

Discrete Phase

$$\frac{du_{pi}}{dt} = C_D \frac{18\mu}{\rho_p d_p} \frac{Re}{24}\left(u_i - u_{pi}\right) + g_i \frac{\rho_g - \rho}{\rho_g} + F_{xi} \quad (2)$$

Feed for spray drying

In this research, *Piper betle* L. extract was chosen to be the feed for the spray drying process. Similar research had conducted by Tee (2011) to experimentally determine the optimum operating conditions for the spray drying process. Therefore, the experimental values obtained are used for validation of the simulation results. The properties of the betel leaf required for the simulation were determined experimentally. These properties are tabulated in Table 3.

Boundary conditions

In this research, an integrated numerical scheme using ANSYS FLUENT™ was adopted. This scheme used different sub-models permitting mass, energy and momentum conservation equations to be solved in a time-dependent 3-Dimensional control volume.

There are two CFD sub-models (dispersed phase and turbulent models) associated with the general underlying physics of spray dryer. The hot air and the feed were entering the spray dryer from the atomizer with relatively high velocities. Therefore, the fluid flow inside the drying equipment will be highly turbulent. The hot air is entering the dryer from the air inlet as a normal flow while the feed is flowing into the dryer from the atomizer (feed inlet) as injections. Therefore, the discrete phase model was chosen for the feed. The injection point was created at the feed inlet of the geometry. The velocities of the hot air and the feed were 0.003257 kgs^{-1} and 0.41 ms^{-1}, respectively. The outlet pressure was set to be 4415 Pa. The residuals set for continuity and energy were 10^{-6} while x-velocity, y-velocity, k, ε and H_2O were set to 10^{-3}. The number of time step was set to be 730 for approximately 5 cycles in order to ensure the flow inside the spray dryer was fully developed.

In this research work, the k-ε model was chosen to be the most suitable turbulent model. This was because k-ε model generates good results for free-shear layer flows with relatively small pressure gradients (Bardina *et al*., 1997). Besides that, since the swirling effect in the drying chamber was not significant, k-ε models generate relatively accurate simulation results. All three k-ε models (standard k-ε models, realizable k-ε models and RNG k-ε models) were tested with the similar flow problem to determine the most suitable turbulent model for modemodellings process. After determining the turbulence model, the next step was to find out the most suitable pressure-velocity coupling scheme for this model. The function of the pressure-velocity coupling is to derive an extra condition for pressure by reformatting the continuity equation. The flow problem can be in two manners which are segregated or coupled. In FLUENT™, there are five different options for the pressure-velocity coupling algorithms. These algorithms were SIMPLE, SIMPLEC, PISO, Coupled and Fractional Step (FSM). When the flow problem was set to be transient, PISO was chosen for the pressure-velocity coupling. SIMPLEC was only chosen when the flow was run as steady. The turbulence intensity is the ratio of the root-mean-square of the velocity fluctuations to the mean flow velocity. Typically, turbulence intensity of 1% or less is said to be lower and greater than 10 % will be considered as high. In this work, the turbulence intensity is dependent of the upstream history of the flow. The intensity may be high if the flow is fully developed before entering the drying chamber. In this research, the turbulence intensity was set to be 9% as the flow was already fully developed.

Parametric studies

It was interested to study the effect of the process variables on the product quality during the spray drying process. The performance of the spray drying process was in fact highly depending on the parameters of the spray dryer such as the velocity of the drying air, humidity of the drying air and the viscosity of the feed. However, for this research only the inlet air temperature and the feed flow rate was studied. As for the quality of the dried product, the property studied in this work was the moisture content. Besides that, four different initial particle sizes were introduced to the simulation respectively to understand their effect on the product quality.

After determining the most suitable CFD model, parametric studies were conducted in the spray drying process. Four different sets of combination between the inlet air temperature and the feed flow rate as shown in Table 4 were investigated using the CFD setup model. Four initial particle sizes (36, 79, 123 and 166 μm) were tested separately under the four different sets of operating parameters. The simulated results were analysed based on the moisture content of the particle. The optimum operating parameter setup would produce the least amount of moisture content and would not form wall deposition.

Model validation

The moisture contents of the product obtained from the simulation results for all combinations of operating parameters were validated with the experimental results of Tee (Tee, 2011). If the difference between the simulation results and experimental values was significant, the CFD setup model would have to be modified by inserting more detailed information of the physical and chemical phenomena and also the boundary conditions. Similarly, the contour of the velocity profiles generated from the simulation would be compared with the previous researched done (Roustapour *et al*., 2009) to increase the reliability of the simulation results.

Results and Discussion

Grid independency test

The flow problem chosen to conduct the grid independent test was Set 1 operating condition shown in Table 4. The initial size of the droplet was 36 m. The moisture content obtained from simulation results for all three different fineness of meshes shown in Table 2 are tabulated in Table 5 and a graph of the results is plotted in Figure 4. According to Tee *et al.* (2012), the moisture content of the dried powder

Table 4. Operating parameter combination

Set	Hot Air Temperature (°C)	Feed Rate (ml/min)
1	160	9.5
2	140	4.0
3	140	15.0
4	160	15.0

Table 5. Grid independency test result

Mesh Size (m)	Moisture (%)
0.015	12.515
0.010	11.341
0.005	11.047

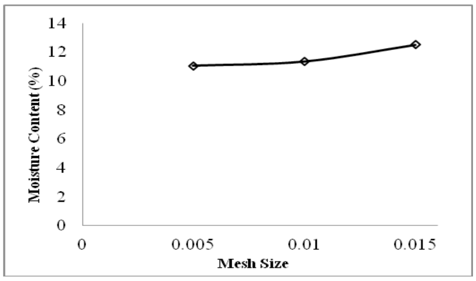

Figure 4. Simulation results at different mesh size

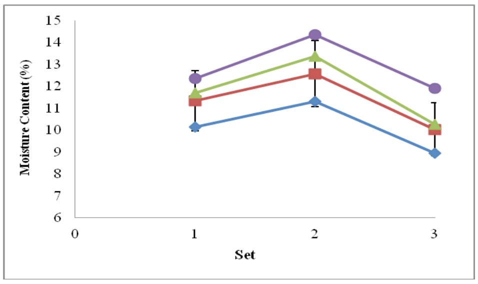

Figure 5. Comparison between experimental values and simulation results of different turbulence models

betel leaf extract at 160°C of inlet air temperature and feed flow rate of 9.5 ml/min was 10.157%. Based on the results obtained from the simulation at different mesh size, it shows that the simulation is dependent on the fineness of the mesh. The percentage errors for each different mesh size were 23.2%, 11.7% and 8.8%, respectively. Therefore, the finer the mesh is, the result obtained from the simulation is closer to the experimental value. This is because when the sub-division of the geometry is smaller, the governing equations solved for each mesh created will be more precise. Therefore, the flow inside the dryer will be described with detailed results. However, the computational effort is directly related to the number of equations to be solved in a simulation. In other words, increasing the number of governing equations by having a higher number of meshes will increase the computational effort in producing the simulation result. The time required to solve 0.015 m mesh size is the least among the rest followed by 0.010 m and lastly 0.005 m. Although 0.005 m mesh size gives the closest simulation result to the experimental value, the time required to generate the result is nearly three times more than 0.010 m mesh size. Besides that, the difference of the percentage errors between the 0.010 m and 0.005 m is not significant. Therefore, the 0.010 m mesh size is said to be the optimum mesh size for this simulation as it gives relatively accurate simulation result and it requires a reasonable amount of computational effort.

Table 6. Simulation results of moisture content for different turbulence models

Set	Turbulence model	Moisture content (%)
	Standard k-ε	11.341
Set 1	RNG k-ε	11.711
	Realizable k-ε	12.351
	Standard k-ε	12.580
Set 2	RNG k-ε	13.371
	Realizable k-ε	14.366
	Standard k-ε	10.035
Set 3	RNG k-ε	10.267
	Realizable k-ε	11.898

Table 7. Experimeεntal values of moisture content of different operating parameters

Set	Moisture Content (%)
Set 1	10.157
Set 2	11.303
Set 3	8.952

Table 8. Percentage errors for all turbulence models

Set	Turbulence Model	Percentage Error (%)
	Standard k-ε	11.7
Set 1	RNG k-ε	15.3
	Realizable k-ε	21.6
	Standard k-ε	11.3
Set 2	RNG k-ε	18.3
	Realizable k-ε	27.1
	Standard k-ε	12.1
Set 3	RNG k-ε	14.7
	Realizable k-ε	32.9

Selection of turbulence model

In order to determine the performance of three different k-ε models for this spray drying process. The tests were conducted on a similar flow problem. The operating conditions for different sets are shown in Table 4. The initial size of the droplet for all different operating conditions was set to be 36 µm. The moisture contents obtained from the simulation results are tabulated in Table 6 and the experimental values of the end product moisture content under these operating conditions is presented in Table 7. Figure 5 shows a comparison between both experimental values and simulation.

Based on the comparison shown in Fig. 5, it shows that the simulation results generated using Standard k-ε are relatively closer to the experimental values as compared to RNG k-ε and Realizable k-ε. Table 8 shows the percentage errors for all the turbulence models applied at different operating conditions.

According to the Table 8 above, the percentage

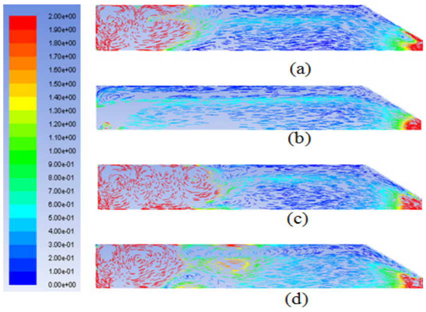

Figure 6. Initial size of 36 μm under (a) Set 1 (b) Set 2 (c) Set 3 (d) Set 4 operating conditions

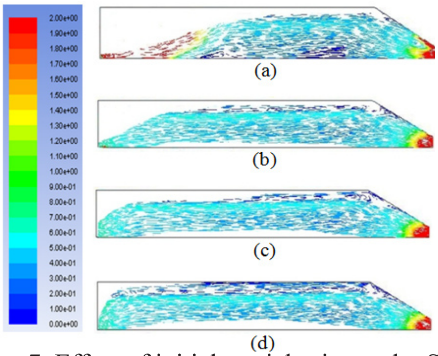

Figure 7. Effect of initial particle size under Set 1 operating condition at (a) 36 μm (b) 79 μm (c) 123 μm (d) 123 μm

error of standard k-ε is the llowest, followedby RNG k-ε and Realizable k-ε has the highest percentage error. From this set of results, it shows that standard k-ε is the most suitable k-ε turbulence model for simulating the flow pattern inside the spray dryer during the drying process of betel leaf extract. This is because the swirling flow is not significant in this particular spray drying process (Huang *et al.*, 2004). Therefore, standard k-ε turbulence model is able to predict the total velocity profiles accurately. Also, it can be used to determine the moisture content of the final product precisely. Moreover, the computational effort required to run the simulation using standard k- is relatively lesser than other models. However, if the flow is highly swirling, RNG k-ε turbulence model will predict a better simulation result than standard k-ε (Woo *et al.*, 2010).

Hot air temperature and feed rate

Hot air temperature and feed rate were the two parameters studied to understand their effects on the airflow pattern inside the dryer. The initial feed droplet size was set to be 36 μm throughout the first set of parametric studies. The flow problem was first run with steady behaviour. The velocity profiles of each different operating parameters generated by simulations is shown in Figure 6, respectively. Based on the results obtained, the general velocity behaviour in Set 1 is that the hot air entered the system with relatively high velocity and it started to decrease as it moved down to the bottom of the dryer. Further, near the exit its velocity increased tremendously due to the narrow exit diameter.

However, with the Set 2 operating conditions, there was no high velocity at the top of the dryer. Unlike the rest, the velocities of the air are very high with the presence of vortices near to the air inlet. This phenomenon will help to enhance the contact area between the hot air and the feed. The evaporation rate will increase as the contact area increases. This is because more energy from the hot air medium is transferred to the feed which results the water content of the feed to be vaporized faster.

The increased number of vortices formed are observed in Set 3. The recirculation of the hot air occurred under similar operating conditions is the longest as compared to other conditions. Therefore, the mixing of hot air and the feed is the most thorough in Set 3. As a result, the moisture content of the product is the least under this operating condition. On the other hand, the mixing occurred in Set 2 is the least. Hence, the moisture content of the product is the highest. Set 4 showed a decrease in velocity at the inlet and recirculation was observed compared to Set 3.

Initial particle size

To determine the flow behaviour with different initial particle size, the operating condition was fixed and the initial size was manipulated. The simulation results of the particle tracking for particle size of 36, 79, 123, 166 μm at 160 hot air temperature with a feed rate of 9.5 ml/min is shown in Figure 7. Based on the results obtained, it shows that larger feed particle size does not follow the core flow inside the dryer. They tend to move towards the inner wall instead of following the core flow path once they are fed into the drying chamber. The larger the initial particle size, the further they move away from the core flow. Small droplets follow the air flow pattern as they have insufficient momentum to escape from the core flow. However, as the droplet size increases, the momentum of the droplet will also increase. Therefore, larger droplet will have higher tendency to escape from the core flow. This will result the formation of wall deposition.

Steady and unsteady flow

Initially, all the flow problems were conducted in steady flow. Similar flow problems were then changed to transient flow to determine the difference. According the results obtained, it shows that velocity profiles for both steady and transient flow are having similar trends. Very minor difference was observed such as the positions of the recirculation of the flow were slightly different and more vortices could be observed for transient flow. Steady flow indicates that the flow does not change over time scale even though the turbulence fluctuations could be still present. In most of the studies conducted by previous researchers, the assumption that the flow behaviour inside the spray dryer is steady allows them to determine engineering data of interest. Moreover, to generate simulation results for steady flow requires lesser computational effort (Woo *et al.*, 2010).

However, experimental work conducted recently suggests that transient behaviour is getting more significant for the airflow pattern in the chamber. In the actual spray drying process, transient eddies will form in the near wall region. The central core flow will also tend to fluctuate sidewards. Transient behaviour is self-sustained fluctuations over long time scales. The pressure imbalances around the central jet help to sustain the fluctuation. The expansion ratio and operating conditions are key factors, whether the flow is steady or transient (Woo *et al*., 2010).

Conclusion

The fineness of the mesh will affect the accuracy of the simulation results. However, up to a point, an increase of the fineness of the mesh will not have a significant difference in terms of result accuracy. At the same, the computational effort required to solve fine meshing is significantly higher. Therefore, these two factors have to be compromised when selecting the size of the mesh. In this research, 0.010 m mesh size was chosen as it provided relatively accurate simulation results and required a reasonable amount of computational effort. The most suitable k-ε turbulence models to be applied in this research was the standard k-ε model. It managed to predict the flow profile reasonably close to the actual flow behaviour inside the spray dryer as compared to the RNG k-ε model and Realizable k-ε model. This is because the swirling effect of the flow inside the drying chamber is not significant. At the feed rate of 15 ml/min and 140^{o}C of hot air, the moisture content of the feed was the least as compared to other operating conditions. However, this is not optimum operating parameter of this spray drying process. This is because the moisture content is not the only product quality that is concerned about. Properties such as hygroscopicity and hydroxychavicol content have to be taken into consideration in determining the optimum operating parameter. The airflow profiles inside the spray dryer were generally similar. Very minor differences between the transient flow and the steady flow was observed. However, when the initial particle size was increased, the particle would tend to leave the core stream and travel to the inner wall. This is one of the reasons that cause wall deposition. Despite that, the simulation results are not completely validated by experimental data. Therefore, more experiments and numerical simulations are needed in order to have more precise findings in future.

References

Arambewela, L.S., Arawwawala, L.D. and Ratnasooriya, W.D. 2005. Antidiabetic Activities of Aqueous and Ethanolic Extracts of Piper Betle Leaves in Rats. Journal of Ethnopharmacology102(2): 239-245.

Bardina, J.E.,. Huang, P.G. and Coakley, P.G. 1997. Turbulence modeling validation, testing and development. NASA Technical Memorandum 110446. Moffett Field, California.

Fletcher, D.F., Guo, B., Harvie, D., Langrish, T., Nijdam, J. and Williams, J. 2006. What is Important in the Simulation of Spray Dryer Performance and How Do Current CFD Models Perform? Applied Mathematical Modelling 30(11): 1281–1292

Huang, L., Kumar, K. and Mujumdar. A.S. 2004. Simulation of a spray dryer fitted with a rotary disk atomizer using a three-dimension computational fluid dynamic model. Drying Technology 22(6): 1489-1515.

Jongebloed, L. 1973. Numerical study using FLUENT of the separation and reattachment points for backwards-facing step flow, Rensselaer Polytechnic Institute, New York.

Kieviet, F.G., Raaij, J.V., De Moor, P.P.E.A. and Kerkhof, P.J.A.M. 1997. Measurement and Modeling of the Air Flow Pattern in a Pilot-plant Spray Dryer. Institution of Chemical Engineer 75: 321-328.

Langrish, T.A.G. and Fletcher, D.F. 2001. Spray drying of food ingredients and applications of CFD in spray drying. Chemical Engineering and Processing: Process Intensification 40(4): 345-354.

Launder, B.E. and Spalding, D.B. 1973. The numerical computational of turbulent flows. Computational Methods in Applied Mechanical Engineering 3: 269-289.

Masters, K. 1994. Scale-up of spray dryers. Drying Technology 12: 235-257.

Oakley, D.E. 1994. Scale-up of spray dryers with the aid of computational fluid dynamics. Drying Technology 12: 217-233.

Patel, R.P., Patel, M.P. and Suthar, A.M. 2009. Spray Drying Technology: An Overview. Indian Journal of Science and Technology 2I(10): 44-47.

Reay, D. 1988. Fluid flow, residence time simulation and energy efficiency in industrial dryers. Proceedings of the Sixth International Drying Symposium, Versailles, France.

Roustapour, O.R., Hosseinalipour, M., Ghobadian, B., Mohaghegh,F. and Azad, N.M. 2009. A proposed numerical–experimental method for drying kinetics in a spray dryer. Journal of Food Engineering 90(1): 20-26.

Tee, L.H. 2011. Rheological properties and optimal spray drying process parameters of *Piper betle* L. (Betel) leaf extracts coated with different excipients. Master Thesis, Universiti Putra Malaysia.

Tee, L.H., Chuah, L.A., Pin, K.Y., Abdull, R.A. and Yusof, Y.A. 2012. Optimization of Spray Drying Process Parameters of *Piper betle* L. Journal of Chemical and Pharmaceutical Research 4(3): 1833-1841.

Woo, M.W., Mujumdar, A.S. and Daud,W.R,W. 2010. Spray Drying Technology Singapore.

Yakhot, V., Thangam, S., Gastski, T.B., Orszag,S.A. and Speziale, C.G. 1992. Development of turbulence models for shear flows by a double expansion technique. Physics of Fluids A 4(7): 1510-1520.