Science
Technology Progress
ISSN 2454–5147

Research Report

www.apjor.com

Received 30th November 2014
Accepted 31st May 2015

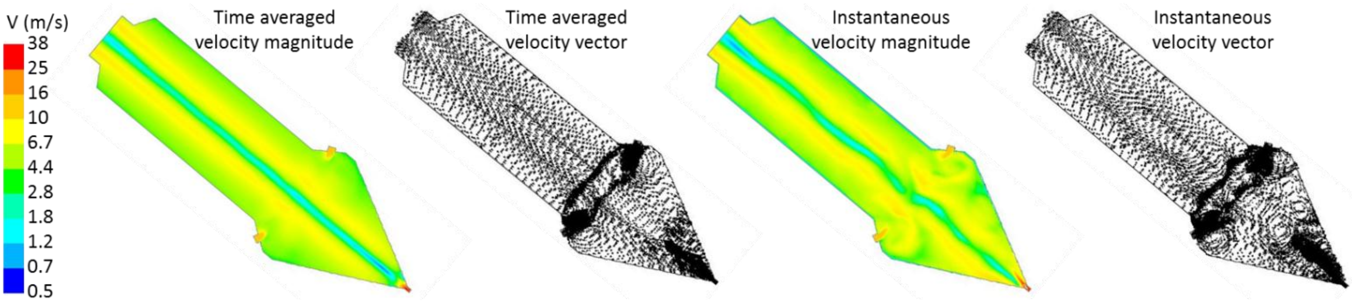
Keywords
Detached eddy simulation
Hydrodynamics
Counter-current spray dryer
Turbulence

CFD simulation of hydrodynamics in counter-current spray dryer tower

Siti Ferdaus Aspari[1], Woon Phui Law[1], Jolius Gimbun[†1,2]

[1]Faculty of Chemical & Natural Resources Engineering, [2]CoE for Advanced Research in Fluid Flow (CARIFF), Universiti Malaysia Pahang, 26300 Gambang, Pahang, Malaysia.
[†]E-mail: jolius@ump.edu.my

This paper presents a computational fluid dynamics (CFD) modelling of hydrodynamics in a counter-current spray drying tower. The simulations were performed using three different turbulent models, i.e. standard *k-ε* (SKE), Reynolds Stress Models (RSM) and the Detached Eddy Simulation (DES). The predicted airflow patterns inside the spray drying chamber were found to be in good agreement with the experimental data adapted from literature for all turbulence models tested in this work. A great potential of the DES for predicting the flow pattern in a counter-current spray dryer was uncovered as it provides more accurate predictions (around 10% deviation) compared to other models tested in this work.



Introduction

Spray dryer is a well-established method for converting liquid feed materials into dry powder products. Spray dryer is widely used for food processing such as whey, instant drinks, milk, tea and soups, as well as healthcare and pharmaceutical products, such as vitamins, enzymes and bacteria[1] also in production of fertilizers, detergent soap, and dyestuffs.

Many studies on computational fluid dynamics (CFD) of spray drying chamber reported in the literature such as Kieviet[2], Anandharamakrishnan et al.[3], Southwell and Langrish[4], Harvie et al.[5], and Huang et al.[6]. Most of the previous work deals with a common co-current flow spray drying. Although simulation of the tall counter-current spray dryer was reported by Wawrzyniak et al.[7], and Harvie et al.[5], but there are limited comparison made on the flow pattern inside the drying chamber. Bayly et al.[8] reported an extensive comparison between the experimental measurement and CFD simulation of a counter-current spray drying. The turbulence modelling was realised using a Reynolds Stress Models (RSM) model in their work, and it seems to give a good prediction of the swirling flow inside the drying chamber. However, there is a still discrepancy, especially in the prediction of gas axial velocity. Therefore, this work attempts to evaluate the performance of various turbulence models, namely standard *k-ε* (SKE), RSM and Detached Eddy Simulation (DES) for predicting the flow pattern in a counter-current spray dryer. The DES belongs to a hybrid turbulence model is a relatively new development in turbulence modelling, which blends Large Eddy Simulation (LES) away from the boundary layer and Reynolds-averaged Navier-Stokes (RANS) near the wall. This model was introduced by Spalart et al.[9] in an effort to reduce the overall computational effort of LES modelling by allowing a coarser grid within the boundary layers. The DES employed for turbulence modelling in this work is based on Spalart-Allmaras (SA) model and has never been previously used for modelling of a counter-current spray dryer tower, although has been employed to simulate a co-current spray dryer recently by Gimbun et al.[10]. Unlike the RANS based model, the DES does not suffer from the assumption of isotropic eddy viscosity. Since turbulence flow is anisotropic in nature, thus DES should provide a better prediction of turbulence flow in drying chamber.

Computational approach

A three-dimensional configuration of a counter-current tower spray dryer fitted with eight main inlets set around the tower hip was modelled using FLUENT 6.3. The main inlet cylinder shape was set 25° below the horizontal and 25° to the tower radius in the horizontal plane, which imparting a significant swirl to the flow in the tower. GAMBIT was used to draw the spray dryer tower diagram illustrated in Fig. 1, which has the same dimension to the one studied by Bayly et al.[8] The simulation was performed using counter-current spray drying tower composed mainly consisting of about (503k) hexahedral and tetrahedral cells. Earlier, Bayly et al.[8] employed 500k grid to yield a satisfactory prediction using the RSM turbulence model. Nevertheless, the grid dependent study was performed to confirm the suitability of the prepared grid. CFD simulation in this work was performed using a HP Z220 workstation with a quad core processor (Xeon 3.2 GHz E3-1225) and eight Gigabytes of RAM. The iteration time for 503k grid is almost twice as faster than that of 934k grid. The whole simulation takes about a week to complete. As it is shown in Fig. 2, there are minimal differences between the predictions obtained using both the 503k and 934k grid. Thus, the 503k grid was used for the remaining of this work in interest to minimise the computational time.

The total air flow through the eight main inlets to the tower is 3814 m^3/h and for the based inlet airflow is 239 m^3/h. The SIMPLE method was used for the pressure-velocity coupling and the 2^{nd} order differencing for momentum terms for the RANS modelling, whereas the bounded central differencing was used for the DES simulation. Three different turbulence models, namely the SKE, RSM and DES were employed in the simulation.

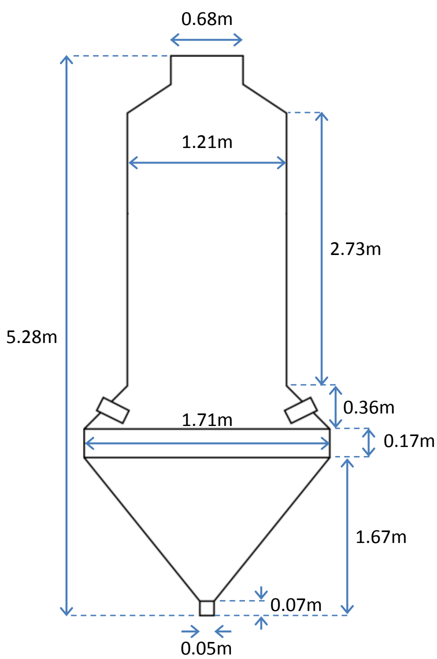


Fig. 1 Spray dryer geometry

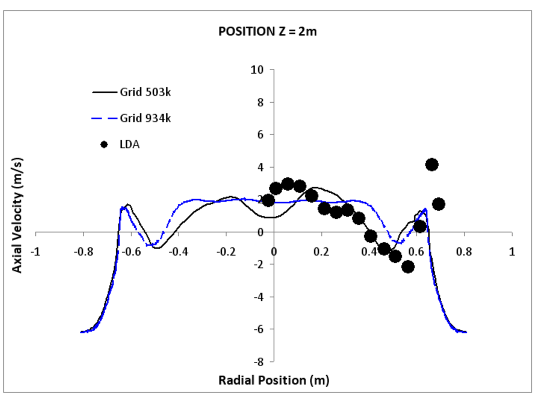


Fig. 2 Result from grid dependent study

The SKE model is a semi-empirical model based on transport equations for the turbulent kinetic energy and its dissipation rate. Transport equations for k and ε for all k-ε variant models can be generalised as follows:

$$\underbrace{\frac{\partial(\rho k)}{\partial t}}_{\text{time derivative}} + \underbrace{\frac{\partial}{\partial x_i}(\rho u_i k)}_{\text{convection}} = \underbrace{\frac{\partial}{\partial x_i}\left(\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right)}_{\text{diffusion}} + \underbrace{\rho P_k}_{\text{production}} - \underbrace{\rho\varepsilon}_{\text{destruction}} \tag{1}$$

and

$$\underbrace{\frac{\partial(\rho\varepsilon)}{\partial t}}_{\text{time derivative}} + \underbrace{\frac{\partial}{\partial x_i}(\rho u_i \varepsilon)}_{\text{convection}} = \underbrace{\frac{\partial}{\partial x_i}\left(\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right)}_{\text{diffusion}} + \underbrace{S_\varepsilon}_{\text{source term}} \tag{2}$$

The turbulent (eddy) viscosity, μ_t, is obtained from:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{3}$$

The relation for production term P_k, for the k-ε variant models (i.e. k-ε, RKE and RNG) is given as:

$$P_k = \mu_t\left(\frac{\partial u_j}{\partial x_i}+\frac{\partial u_i}{\partial x_j}\right)\frac{\partial u_j}{\partial x_i} \tag{4}$$

For the SKE model the source term, S_ε, is given by:

$$S_\varepsilon = \rho\left(C_{1\varepsilon}\frac{\varepsilon}{k}P_k - C_{2\varepsilon}\frac{\varepsilon^2}{k}\right) \tag{5}$$

The model constants are[11]: $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $C_\mu = 0.09$, $\sigma_\varepsilon = 1.3$ derived from correlation of experimental data.

RSM abandons the assumption of the isotropic eddy-viscosity hypothesis, to close the RANS equations, by solving transport equations for the individual Reynolds stresses, together with a transport equation for the dissipation rate. RSM has a greater potential to give accurate predictions for complex flows, as it takes into accounts the effects of streamline curvature, swirl, rotation, and rapid changes in strain rate in a more rigorous manner than two-equation models such as k-ε. The foundation of RSM is the exact set of transport equations:

$$\begin{aligned}
&\underbrace{\frac{\partial}{\partial t}\left(\rho\overline{u_i'u_j'}\right)}_{\text{Time derivative}} + \underbrace{\frac{\partial}{\partial x_k}\left(\rho u_k\overline{u_i'u_j'}\right)}_{C_{ij}=\text{Convection}} = \underbrace{-2\rho\Omega_k\left(\overline{u_j'u_m'}\,\varepsilon_{ikm}+\overline{u_i'u_m'}\,\varepsilon_{jkm}\right)}_{F_{ij}=\text{Production by system rotation}} \\
&\underbrace{-\frac{\partial}{\partial x_k}\left[\rho\overline{u_i'u_j'u_k'}+\overline{p\left(\delta_{kj}u_i'+\delta_{ik}u_j'\right)}\right]}_{D_{T,ij}=\text{Turbulent diffusion}} + \underbrace{\frac{\partial}{\partial x_k}\left[\mu\frac{\partial}{\partial x_k}\left(\overline{u_i'u_j'}\right)\right]}_{D_{L,ij}=\text{Molecular diffusion}} \\
&\underbrace{-\rho\left(\overline{u_i'u_k'}\frac{\partial u_j}{\partial x_k}+\overline{u_j'u_k'}\frac{\partial u_i}{\partial x_k}\right)}_{P_{ij}=\text{Stress production}} + \underbrace{\overline{\frac{p}{\rho}\left(\frac{\partial u_i'}{\partial x_j}+\frac{\partial u_j'}{\partial x_i}\right)}}_{\phi_{ij}=\text{Pressure strain}} \underbrace{-2\mu\overline{\frac{\partial u_i'}{\partial x_k}\frac{\partial u_i'}{\partial x_k}}}_{\varepsilon_{ij}=\text{Dissipation}}
\end{aligned} \tag{6}$$

The Ω_k is an angular velocity and both ε_{ikm} and ε_{jkm} are permutation tensors. Of the various terms in these exact equations, C_{ij}, $D_{L,ij}$, P_{ij}, and F_{ij} do not require any modelling. However, $D_{T,ij}$, ϕ_{ij}, and ε_{ij} need to be modelled to close the equations. The reason is simply because the averaging procedure of $\overline{u_i'u_j'u_k'}$ generates a lot of unknown variables and it becomes impossible to solve them directly.

The turbulent diffusivity transport term is modelled using a simplified form of the generalized gradient diffusion hypothesis as:

$$D_{T,ij} = \frac{\partial}{\partial x_k}\left[\frac{\mu_t}{\sigma_k}\frac{\partial\left(\overline{u_i'u_j'}\right)}{\partial x_k}\right] \tag{7}$$

The pressure strain term is modelled as:

$$\phi_{ij} = \overline{\frac{p}{\rho}\left(\frac{\partial u_i'}{\partial x_j}+\frac{\partial u_j'}{\partial x_i}\right)} = -C_1\frac{\varepsilon}{k}\left[\overline{u_i'u_j'}-\frac{2}{3}\delta_{ij}k\right] - C_2\left[P_{ij}-\frac{2}{3}\delta_{ij}P\right] \tag{8}$$

where $P = 0.5P_{ij}$ is the turbulence production due to shear, and the constants are $C_1 = 1.8$ and $C_2 = 0.6$.

The dissipation term is assumed to be isotropic and is approximated by:

$$\varepsilon_{ij} = 2\mu\overline{\frac{\partial u_i'}{\partial x_k}\frac{\partial u_i'}{\partial x_k}} = \frac{2}{3}\delta_{ij}\varepsilon \tag{9}$$

The scalar dissipation rate is computed with a model transport equation similar to the one in the SKE model.

As introduced earlier, the DES model belongs to a class of a hybrid turbulence model which blends LES away from boundary layer and RANS near the wall. This combination (RANS-LES) model was introduced by Spalart et al.[9] in an effort to reduce the overall computational of LES modelling by allowing the coarser grid at the boundary layer. The DES employed for the turbulence modelling in this work is based on SA model[12] and has never been previously used for modelling of spray drying.

The SA one-equation model solves a single partial differential equation for a variable $\tilde{\nu}$ which is related to the turbulent viscosity. The variable $\tilde{\nu}$ is identical to the turbulent kinematic viscosity except in the near-wall (viscous-affected) region. The model includes a wall destruction term that reduces the turbulent viscosity in the log layer and laminar sub-layer. The transport equation for DES is:

$$\frac{\partial}{\partial t}(\rho\tilde{\nu}) + \frac{\partial}{\partial x_i}(\rho\tilde{\nu}u_i) = G_\nu + \frac{1}{\sigma_{\tilde{\nu}}}\left[\frac{\partial}{\partial x_j}\left\{(\mu+\rho\tilde{\nu})\frac{\partial\tilde{\nu}}{\partial x_j}\right\} + C_{b2}\rho\left(\frac{\partial\tilde{\nu}}{\partial x_j}\right)^2\right] - Y_\nu \tag{10}$$

The turbulent viscosity is determined via:

$$\mu_t = \rho\tilde{\nu}f_{v1}, \quad f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3}, \quad \chi \equiv \frac{\tilde{\nu}}{\nu} \tag{11}$$

where $\nu = \mu/\rho$ is the molecular kinematic viscosity. The production term, G_ν, is modelled as:

$$G_\nu = C_{b1}\rho\tilde{S}\tilde{\nu}, \quad \tilde{S} \equiv S + \frac{\tilde{\nu}}{k^2 d^2} f_{v2}, \quad f_{v2} = 1 - \frac{\chi}{1+\chi f_{v1}} \tag{12}$$

S is a scalar measure of the deformation rate tensor which is based on the vorticity magnitude in the SA model. The destruction term is modelled as:

$$Y_\nu = C_{w1}\rho f_w\left(\frac{\tilde{\nu}}{d}\right)^2, \quad f_w = g\left[\frac{1+C_{w3}^6}{g^6+C_{w3}^6}\right]^{1/6}, \quad g = r + C_{w2}\left(r^6 - r\right), \quad r \equiv \frac{\tilde{\nu}}{\tilde{S}k^2d^2} \tag{13}$$

The closure coefficients for SA model[11] are $C_{b1} = 0.1355$, $C_{b2} = 0.622$, $\sigma_{\tilde{\nu}} = \frac{2}{3}$, $C_{v1} = 7.1$, $C_{w1} = \frac{C_{b1}}{k^2} + \frac{(1+C_{b2})}{\sigma_{\tilde{\nu}}}$, $C_{w2} = 0.3$, $C_{w3} = 2.0$, $k = 0.4187$.

Results and discussion

The prediction from the CFD simulation was compared to the Laser Doppler Anemometer (LDA) measurement by Bayly et al.[8] at various positions of the spray drying chamber. Data from CFD simulation were taken as a statistical average from up to 1000 time steps after the pseudo convergence was achieved, which mimic the data collected in experimental measurement. Generally, all CFD models tested in this work can predict the flow pattern in counter-current spray drying reasonably well (Figs. 3 and 4). However, ultimate agreement was not achieved. The Rankine vortex feature due to the swirling flow also reproduced correctly.

Prediction of the DES model is by far the best among the model tested. This is attributed by the fact that DES employs LES in the bulk flow, which in turn provides much better predictions of the turbulence flow. Around the boundary layer (i.e. the wall) the DES turn to a single equation SA turbulence model which provides a fair approximation of the flow near the wall without necessarily having to resolve the small eddies. This method of hybrid LES-RANS model employed in DES reduces the overall computational demand of a full LES solution while at the same time maintaining the prediction accuracy.

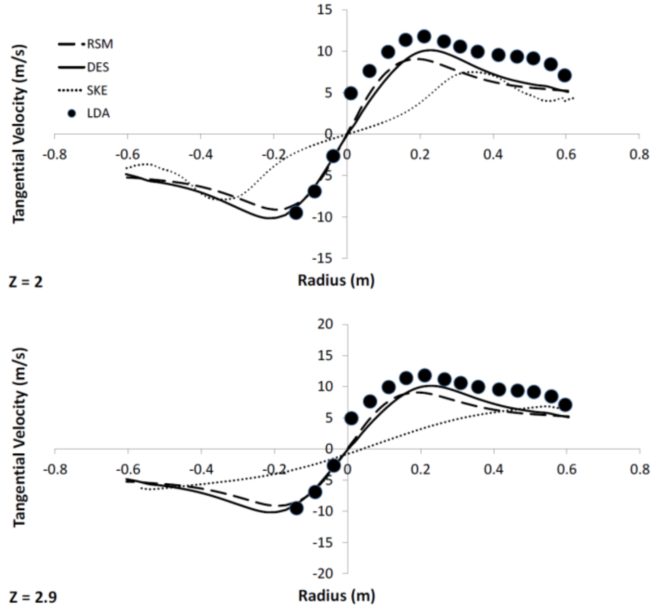


Fig. 3 Prediction of tangential velocity inside the counter-current spray dryer chamber using various turbulence models. Data points adapted from Bayly et al.[8]

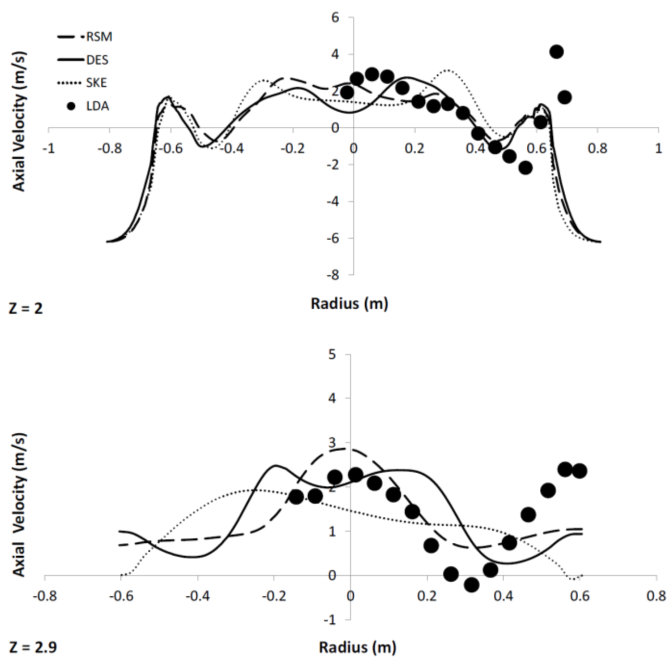


Fig. 4 Prediction of axial velocity inside the counter-current spray dryer chamber using various turbulence models. Data points adapted from Bayly et al.[8]

The RSM model outperformed the other RANS based turbulence models (SKE) tested in this work. This is attributed by the anisotropic eddy viscosity model in RSM model, which is known for its excellent prediction for swirling and strong anisotropic turbulence flow such as in cyclone[13,14,15]. As it is mentioned in the previous section, the flow pattern inside the counter-current spray dryer studied in this work exhibits some swirling flow due to the position and design of the inlet gas around the tower hip, hence requiring a more complicated turbulence model for accurate prediction of the mean flow field inside the chamber.

SKE is proven to give a good prediction of co-current spray drying chamber (e.g. Anandharamakrishnan et al.[3], Huang et al.[6]), owing to the absence of a strong curvilinear and swirl flow. However, prediction of SKE for counter-current spray dryer in this work is rather poor in comparison to either DES or RSM, which can be attributed to the anisotropic assumption of eddy viscosity and lacking the feature to model swirl flows. It is therefore, not advisable to use SKE model for counter-current spray dryer tower, especially for swirling flow dominated region.

The axial flow pattern exhibits a single peak pattern similar to those normally seen for a reverse flow cyclone (e.g. Fraser et al.[16]). All models predict the axial flow pattern reasonably well compared to the experimental measurement. The DES and RSM models again provide a much closer agreement with the experimental data similar to the trend seen for the prediction of tangential velocity. In most cases, DES predictions are marginally better than that of RSM. The SKE model was the worst among all the turbulence models tested, and hence should be avoided for modelling of a counter-current spray dryer.

Conclusions

CFD predictions obtained using various turbulence models has uncovered a great potential of DES for modelling the flow field of the counter-current spray dryer. The Rankine vortex features due to the swirling flow are also reproduced correctly by both the DES and RSM turbulence model. The prediction from the DES is more accurate than those obtained using both SKE and RSM model. Results from this simulation may be useful for development of a more comprehensive and accurate model for counter-current spray dryer in the future.

Notes and references

1. K. Masters, (1991). Spray Drying Handbook. Longman Scientific and Technical, Harlow.
2. F.G. Kieviet, (1997). Modelling Quality in Spray Drying. Ph.D. Thesis, Endinhoven University of Technology, Netherlands.
3. C. Anandharamakrishnan, J. Gimbun, A.G.F. Stapley, C.D. Rielly, (2010). A Study of Particle Histories during Spray Drying using Computational Fluid Dynamics Simulations. Drying Technology, 28: 566-576.
4. D.B. Southwell, T.A.G. Langrish, (2001). The Effect of Swirl on Flow Stability in Spray Dryers. Chemical Engineering Research and Design, 79: 222-234.
5. D.J.E. Harvie, T.A.G. Langrish, D.F. Fletcher, (2001). Numerical Simulations of Gas Flow Patterns within a Tall-Form Spray Dryer, Chemical Engineering Research and Design, 79: 235-248.
6. L. Huang, K. Kumar, A.S. Mujumdar, (2004). Simulation of Spray Dryer Fitted with a Rotary Disk Atomizer using a Three-Dimensional Computational Fluid Dynamic Model. Drying Technology, 22: 1489-1515.
7. P. Wawrzyniak, M. Podyma, I. Zbicinski, Z. Bartczak, J. Rabaeva, (2012). Modeling of Air Flow in an Industrial Countercurrent Spray-Drying Tower, Drying Technology, 30: 217-224.
8. A.E. Bayly, P. Jukes, M. Groombridge, C. McNally, (2004). Airflow Patterns in a Counter-Current Spray Drying Tower-Simulation and Measurement. IDS2004, B: 775-781.
9. P.R. Spalart, W.H. Jou, M. Strelets, S.R. Allmaras, (1997). Comments on the Feasibility of LES for Wings, and on a Hybrid RANS/LES Approach, Advances in DNS/LES, 1st AFOSR International Conference on DNS/LES, 4–8 Aug, Greyden Press, Columbus, OH.
10. J. Gimbun, N.I.S. Muhammad, W.P. Law, (2015). Unsteady RANS and Detached Eddy Simulation of the Multiphase Flow in a Co-current Spray Drying. Chinese Journal of Chemical Engineering, http://dx.doi.org/10.1016/j.cjche.2015.05.007
11. B.E. Launder, D.B. Spalding, (1974). Numerical Computation of Turbulent Flows. Computer Methods in Applied Mechanics and Engineering, 3: 269-289.
12. P.R. Spalart, S.R. Allmaras, (1992). A One-Equation Turbulence Model for Aerodynamic Flows. American Institute of Aeronautics and Astronautics, 92-0439.
13. J. Gimbun, T.G. Chuah, A. Fakhru’l-Razi, T.S.Y. Choong, (2005). The Influence of Temperature and Inlet Velocity on Cyclone Pressure Drop: A CFD Study. Chemical Engineering and Processing, 44: 7-12.
14. J. Gimbun, (2008). CFD Simulation of Aerocyclone Hydrodynamics and Performance at Extreme Temperature. Engineering Applications of Computational Fluid Mechanics, 2: 22-29.
15. J. Gimbun, T.S. Choong, A. Fakhru’l–Razi, T.G. Chuah, (2004). Prediction of the effect of dimension, particle density, temperature, and inlet velocity on cyclone collection efficiency. Jurnal Teknologi, 40: 37-50.
16. S.M. Fraser, A.M.A. Razek, M.Z. Abdullah, (2000). Computational and Experimental Investigations in a Cyclone Dust Separator. Proceedings of the Institution of Mechanical Engineering Part E, 211: 241-257.