IDS'2018 – 21st International Drying Symposium
València, Spain, 11-14 September 2018
DOI: http://dx.doi.org/10.4995/ids2018.2018.7686

CFD study of air flow patterns and droplet trajectories in a vortex chamber spray dryer

Rahman, U.J.U.*; Baiazitov, I.; Pozarlik, A.K.; Brem, G.
ThW group, Department of TFE. University of Twente, Enschede, The Netherlands

**E-mail of the corresponding author: u.jamilurrahman@utwente.nl

Abstract
In order to develop an alternative spray drying technology, a high drying rate in a smaller volume must be achieved. In this paper, results of CFD study are presented, carried out to investigate the possibility of spray drying in a novel design vortex chamber. The model is validated against experimental data, that makes a good agreement with an average error of 7% with only air and 24% with water spray. Results of temperature fields and droplet impact positions are discussed. The computations demonstrate that vortex chamber spray dryer can be an attractive solution for drying technology.

Keywords: *CFD; spray drying; vortex chamber; atomization.*

1. Introduction

Spray drying is a widely used process for converting liquid feed into dry powder [1]. However, the technology has only little improved over time. This comes with an inherent weakness; conventional spray dryers need great volumetric flow rates of air and thus high capital costs [2]. Additionally, low inlet temperatures and small gas-solid slip velocities lead to low drying rates. In order to develop an alternative spray drying technology, high inlet temperatures whilst maintaining a small residence time must be accomplished [3]. Therefore, a new technology for spray drying called vortex chamber spray dryer (VC) is investigated here. Additionally, high-G fluidization in VC, leads to intensification of interfacial heat, mass and momentum transfer [4, 5].

The high temperatures and turbulent multiphase flow in spray drying, make it difficult to design and optimize the dryer. For this purpose, Computational Fluid Dynamics (CFD) modeling has emerged as a vital tool in understanding the flow fields and gas-particle dynamics [6, 7]. In this research, a CFD model is developed to investigate air flow patterns and droplet trajectories for spray drying in a novel multi-zone VC dryer. The predicted results are compared against available experimental data. An overall drying performance is evaluated based on temperature profiles, particle's moisture content, impact positions and product quality obtained at the outlets.

2. Methodology

2.1. CFD methodology

The simulations have been performed in the commercial CFD package FLUENT 16.0. The flow in a spray dryer is a dispersed multiphase flow, therefore, the Eulerian-Langrangian approach was used. A three dimensional steady state model with k-epsilon turbulence model is used to simulate the gas phase. A two way coupling is applied to consider the heat, mass and momentum exchange between the continuous and discrete phase. The discrete phase turbulence is modeled using stochastic tracking model. The details of the CFD methodology and the governing Navier-Stokes, turbulence and discrete phase equations can be found in the literature and FLUENT user guide [8-10]. In order to simply the computations, the droplet-droplet interactions, agglomeration and coalesce is not considered. Also, only the evaporation model is used for droplet drying i.e. droplets are not porous and contains only surface moisture.

2.2. Geometry and boundary conditions

The VC geometry simulated in this study is shown in Figure 1. presenting the inlets and outlets (due to confidentiality reasons the exact dimensions are not presented). Such setup of VC offers two separate temperature zones: axial hot zone in the cylinder and relatively cold zone in VC. Hot air enters the dryer through a honey comb flow distributor hence diminishing small turbulences in the cylinder. The VC is made up of six wheels, each wheel consisting of 36 inlet slots through which air enters the chamber tangentially, creating a high-G vortex flow. Droplets are injected, counter flow to hot air using a pressure nozzle. The position of the atomizer is depicted in Fig.1.

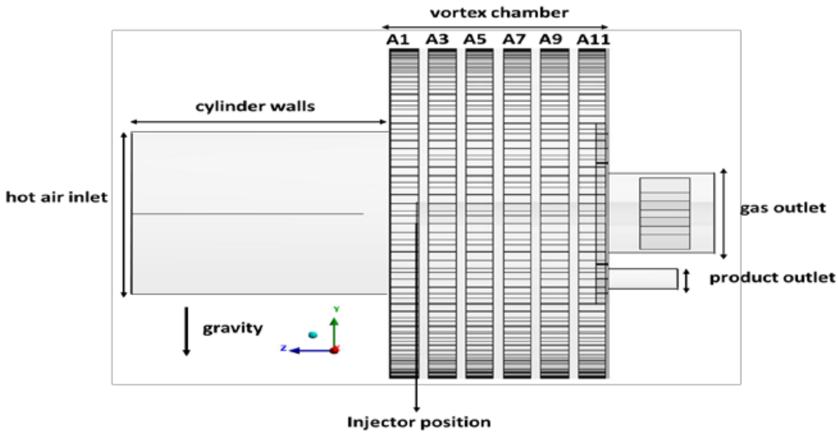

Figure 1. Geometry of vortex chamber spray dryer

The hot air inlet temperature is 350 °C and mass flow rate is 682 kg/hr while cold air average temperature to VC is 132 °C and mass flow rate of 615 kg/hr. The outlets are set to pressure of 5000 Pascal. The chamber walls are treated as adiabatic i.e. no heat loss to the environment is considered. A spray feed rate of 23 kg/hr is sprayed at a temperature of 65 °C with an injection velocity of 70 m/s. Droplet size distribution is modeled using Rosin-Rammler method. A spread parameter of 2.05 as reported by Kievet was used [11]. The maximum and minimum droplet size is set to 90 and 20 microns respectively while the mean droplet size of 52 microns [12] is used. Furthermore, a restitution coefficient of 0.2 is applied on VC walls.

3. Results and discussion

This section presents the predicted results of the CFD model. The model is validated against the available experimental data. Furthermore, an overall drying performance is evaluated based on particle impact positions and product obtained at outlets.

3.1. Grid independency study

For the grid independency study, three grid sizes; 1.5M, 3M and 5.5M elements were used. By comparing temperature profiles at 3 different locations, an average error of less than 1% was found between the coarsest and finest mesh. Since, the difference was not significant, the grid of 1.5M elements is applied for further research.

3.2. Validation of CFD model against experimental data

The experimental data used for the model validation is obtained from the work of Thomas et al. [13]. It should be pointed out that, during experiments, it was not possible to operate the VC in steady-state, due to the clogging of the nozzle. The thermocouples, V1 and V4 are located in the outer region of VC at top and bottom respectively, V6 and V8 are located in the middle transition region of VC at left and right respectively while the thermocouples V12 and V13 are located in the central core of the VC. The thermocouples are positioned on each wheel in the axial direction.

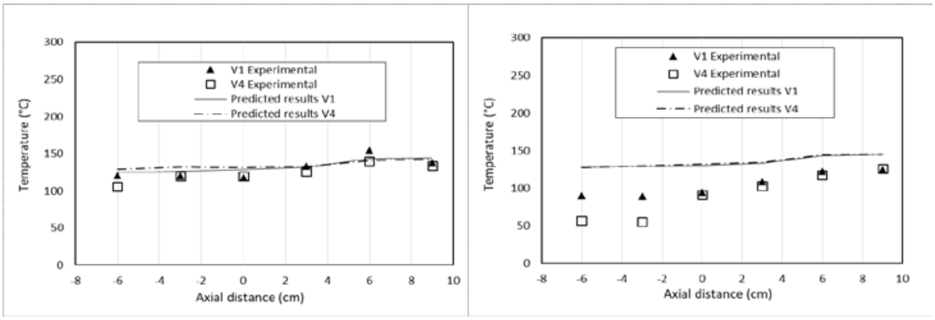

Figure 2. Comparison of results for V1 and V4 probes a) only air (left) b) with water spray (right)

The Figure 2a shows a comparison of predicted temperature profiles against experimental data for probes V1 and V4. The model makes a good agreement with an average error of 2% and 9% for V1 and V4, respectively. It can be seen that model slightly over estimates the temperature. The error can be caused by no heat loss assumption in the model whereas in the experiments heat loss can occur. Figure 2b, gives comparison with water spray. A relatively bigger error; 30% and 60% is seen for V1 and V4 respectively. Much lower temperatures; 50-80 °C for wheels A1 and A3 are observed in the experiments. However, these can be due to water impinging and evaporating on the thermocouple. Another reason could be that in the model spray penetrates into the hot cylinder where major evaporation takes place, and thus predicted results with only air and with spray do not show significant difference.

The Figure 3a shows the comparison of predicted air temperature against experimental data for thermocouples V12 and V13. The predicted results correspond well with the experimental data with an average error 1% and 7% for V12 and V13 respectively. Moreover, low temperature values in experimental measurements suggests, stronger radial mixing of hot air with the rotating cold air. Figure 3b, shows the comparison with spray. The experimental measurements show a strong asymmetry between V12 and V13. The dissymmetry might be induced by the product outlet being, too close to the gas outlets causing the spray to instantly deflect back on one side giving much lower temperatures for V13. The effect is less pronounced in the predicted results. An average error of 6% and 20% is obtained for V12 and V13 probes, respectively.

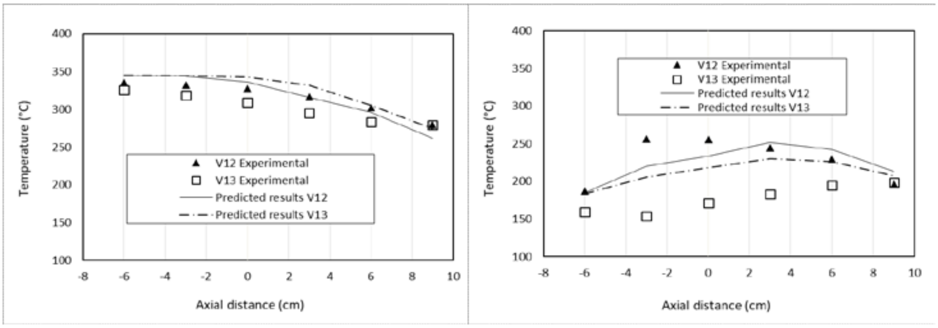

Figure 3. Comparison of results for probes V12 and V13 a) only air (left) b) with water spray (right)

Figure 4a, shows the comparison of predicted results for thermocouples V6 and V8. An opposing trend with an under prediction of temperature is seen for the middle-transition region. An average error of 11% for V6 and 13% for V8 is seen. The results imply a similar trend to that observed for the core region i.e. a stronger radial mixing of hot air with rotating cold air, that gives higher temperatures in the middle region of VC in the experiments.

Figure 4b, shows the predicted results against experimental measurements with water spray. The results shows a good agreement between numerical and experimental data with average error of 12% and 20% for V8 and V6 respectively. Plots show a similar trend of low temperature at the front two wheels where the biggest discrepancies are seen. This can again be explained with water droplets, which in experiments, do not penetrate into the hot cylinder but instead moves to the front of VC (A1 and A3 wheels).

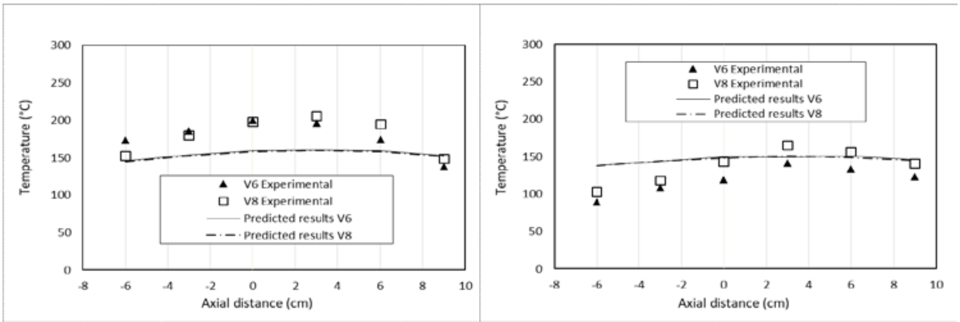

Figure 4. Comparison of results at probes V6 and V8 a) only air (left) b) with water spray (right)

3.3. Influence of multi component spray injection

In this section, the discussion has been extended by using 56 kg/hr of feed with 40% solid content and feed density of 1080 kg/m^3. A performance analysis of VC dryer is drawn based on particles impact positions, separation efficiency and mass of product recovered at the outlets. Figure 5, presents the temperature profiles on two axial planes for air flow only. A small reflux of cold air, from VC to cylinder can be seen on both planes. The reflux is caused mainly by two reasons; low pressure in the cylinder and the influence of product outlet's back pressure. Such recirculating zones can be problematic, since it can lead to product deposition on cylinder walls. Furthermore, two temperature regions are visible in the dryer: a hot zone in the cylinder and around the nozzle and cold zone on the outer region of the VC.

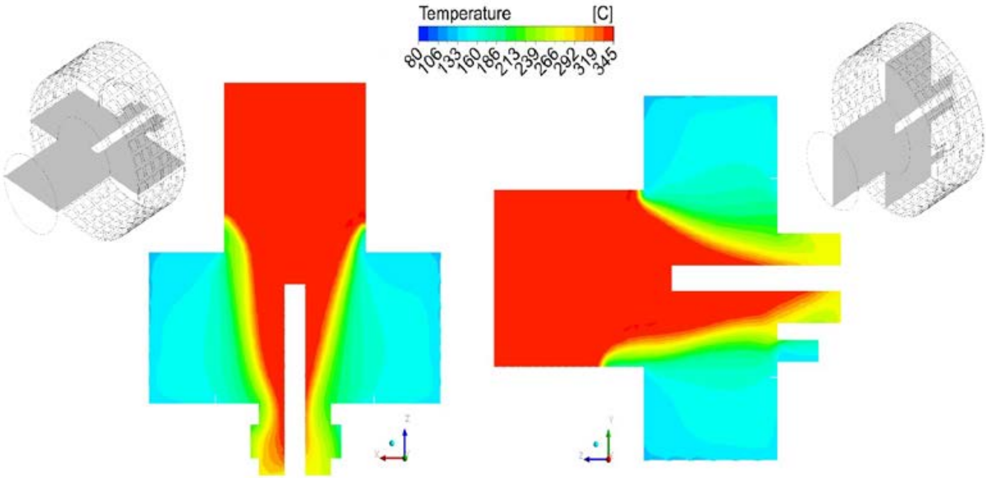

Figure 5. Temperature patterns on two axial planes, only air

Figure 6, illustrates the temperature patterns with spray (40% solid content). It can be seen that temperature profiles are asymmetric. The reason for this is mainly, the strong back pressure caused by the product outlet, forcing the droplets to instantly deflect back. Moreover, it can be seen that significant evaporation takes place in the hot zone while in the VC temperature drop is mostly near wheels-A1 and A3.

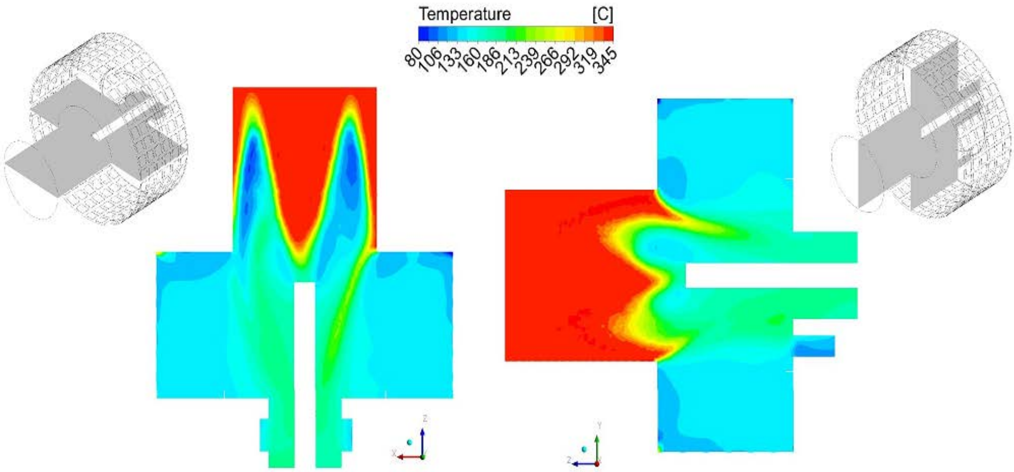

Figure 6. Temperature patterns on two axial planes, with water spray

3.3.1.Particle trajectories

In order to design and optimize a dryer, it is of prime interest to predict the trajectories of different sizes particles. Figure 7, reveals the particle trajectories for different particle size ranges, as a function of water mass fraction. These figures indicate that particles below 45 µm are instantly deflected back and leave via gas outlets. The particle sizes between 45-80 µm, are very uniformly distributed over VC. These particles mostly leave via product outlet. A mean particle size of 30 and 40 µm and a mean residence time of 0.48 and 1 seconds is found at gas and product outlet, respectively. Particles bigger than 80 µm are mostly impinging the honey comb air inlet or cylinder walls and only a few return to VC. Thus, it can be concluded that a maximum of 80 µm particle size, can be successfully dried without any risk of deposition on the walls. Moreover, all particles impinging VC walls are dried with zero moisture content.

Figure 7. Particle trajectories as a function of water mass fraction

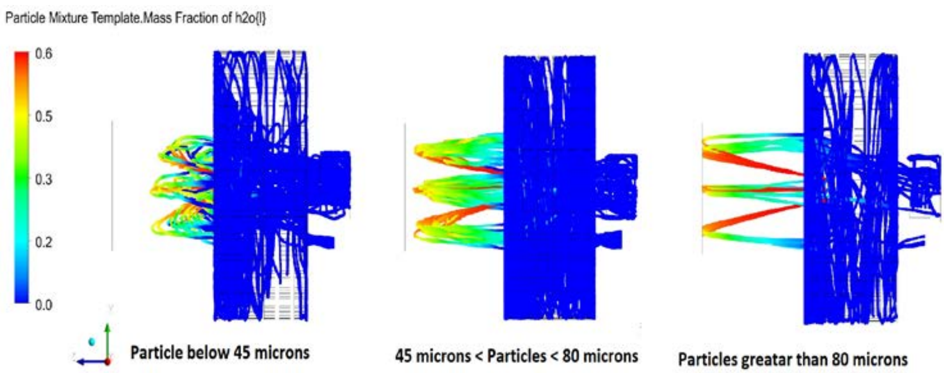

Table 2, shows an overall drying performance as discussed by Huang et al [14]. Some of the particles are trapped in circulating zones and do not escape. These are removed from calculations after 10 seconds.

Table 1. Overall drying performance of vortex chamber spray dryer

	Percentage (%)	Deposit rate (kg/hr)
Gas outlets	56	15
Product outlet	18	4.5
Cylinder walls	3.5	0.7
Hot air inlet	12.5	0.8
Incomplete	10	1.4
Gas outlet temperature (°C)	155	
Energetic Specific Consumption (KJ/Kg.H_20)	10,000	

4. Conclusions

A three-dimensional CFD model for spray drying in a VC was developed. A good agreement was made with the experimental data, depicting an overall average error; 7% and 24% with only air and with water spray respectively. The results depict small recirculation zones of cold air in cylinder, causing asymmetric spray patterns. It was found that a large proportion of particles leave via the gas outlets. In order to have an efficient VC dryer, separation of smaller particles from air is necessary. Based on the CFD research, it can be concluded that redesign of current VC configuration is necessary in order to harvest its full potential.

5. Acknowledgements

This research is conducted within RVO-Vortex chamber II Project-TEEI115007, in collaboration with ISPT (Drying and Dewatering cluster), Royal FrieslandCampina, Université Catholique de Louvain, Unilever and Energy Research Centre of the Netherlands (ECN). Authors would like to express their gratitude to RVO for the financial support and to all project members for the fruitful discussions during the meetings. Furthermore, authors would like to convey their special thanks to Prof. Juray De Wilde, Mr. Axel de Broqueville and Mr. Thomas Tourneur *(Université Catholique de Louvain)* for providing the geometry, operating conditions and experimental data.

6. References

[1] Masters, K., *Spray drying handbook*. 1991: Longman Scientific & Technical.
[2] Mujumdar, A.S., *Handbook of industrial drying*. 2015.
[3] Kudra, T. and A.S. Mujumdar, *Advanced Drying Technologies*. 2001: Taylor & Francis.
[4] De Wilde, J., *Gas–solid fluidized beds in vortex chambers.* Chemical Engineering and Processing: Process Intensification, 2014. **85**: p. 256-290.
[5] Eliaers, P., et al., *Modeling and simulation of biomass drying in vortex chambers.* Chemical Engineering Science, 2015. **123**: p. 648-664.
[6] Jin, Y. and X.D. Chen, *A Three-Dimensional Numerical Study of the Gas/Particle Interactions in an Industrial-Scale Spray Dryer for Milk Powder Production*. Drying Technology, 2009. **27**(10): p. 1018-1027.
[7] Kuriakose, R. and C. Anandharamakrishnan, *Computational fluid dynamics (CFD) applications in spray drying of food products.* Trends in Food Science & Technology, 2010. **21**(8): p. 383-398.
[8] Fletcher, D., *Computational Fluid Dynamics Simulation of Spray Dryers—An Engineer's Guide.* Drying Technology, 2017. **35**(7): p. 903-903.
[9] Woo M.W., H.L.X., Mujumdar A.S., Daud W.R.W, *CFD simulation of spray dryer*. Spray Drying Technology. Vol. one. 2010.
[10] Frydman, A., et al., *Simulation of Spray Drying in Superheated Steam Using Computational Fluid Dynamics.* Drying Technology, 2007. **17**(7-8): p. 1313-1326.
[11] Kieviet, F.G., *Modeling quality in spray drying.* 1997.
[12] M.Klaassen, *Near filed atomization in pressure swirl nozzle. Experimental investigation regarding inpact of fluid properties and nozzle specifications on spray characteristics,*, in *TFE (Thermal Engineering Group)*. 2016, University of Twente: Enschede.
[13] Tourneur, T., A.d. Broqueville, and J.D. Wilde, *Experimental and CFD study of multi-zone vortex chamber spray dryers*, in *International Symposium on Chemical Reactor Engineering-ISCRE 25*. 2018.
[14] Huang, L.X., K. Kumar, and A.S. Mujumdar, *A comparative study of a spray dryer with rotary disc atomizer and pressure nozzle using computational fluid dynamic simulations.* Chemical Engineering and Processing: Process Intensification, 2006. **45**(6): p. 461-470.