

Prediction of Air Flow, Temperature and Humidity Patterns in a Pilot Plant Spray Dryer

Saad Nahi Saleh
Chemical Engineering Department,
College of Engineering,
University of Tikrit
E mail: saad_nahi68@yahoo.com

Abstract

This paper presents the prediction of air flow, humidity and temperature patterns in a co-current pilot plant spray dryer fitted with a pressure nozzle using a three dimensional model. The modeling was done with a Computational Fluid Dynamic package (Fluent 6.3), in which the gas phase is modeled as continuum using the Euler approach and the droplet/ particle phase is modeled by the Discrete Phase model (Lagrange approach). Good agreement was obtained with published experimental data where the CFD simulation correctly predicts a fast downward central flowing core and slow recirculation zones near the walls.

In this work, the effects of the air flow pattern on droplets trajectories, residence time distribution of droplets and deposition of the droplets on the wall also were investigated where atomizing of pure water was used.

Introduction

Spray dryer is an essential unit operation for the manufacture of many products with specific powder properties, e.g. chemical, ceramic, food; pharmaceuticals etc. In spite of the wide uses of the spray dryers, they are still designed mainly on the basis of experience and pilot experiment [1]. One of the big problems facing spray dryer designer and operators is the complexity of the spray/air mixing process in spray chamber [2] where the air flow patterns existing inside the spray dryer is considered as one of the primary factors that influence the residence time of droplet / particle , in turn the equality of the product produced by the dryer such as moisture content , size distribution , and bulk density. The particle residence time and surrounding air temperature are particularly important in the spray drying of thermal sensitive products, such as milk, where product degradation can occur if the particles remain in an air stream for too long, or experience an air stream is too hot [3].

A very important phenomenon of spray dryer operability is the particle wall deposition

which is affected by the temperature and humidity patterns inside the dryer when moist particles contact the spray dryer wall. Such depositions can lead to a build up large amounts of product on the wall. These depositions may be dangerous, as they can fall and cause damage to the chamber walls, or they can char, resulting in a potential explosion hazard [4].

Application of computational fluid dynamic (CFD) techniques in analyses of spray dryers have been carried out successfully and reported by Oakley et al. [5] , Kieviet [6] and others. Most of these earlier works assume the flows in the dryers are two-dimensional and axisymmetric in order to reduce the demand on computational resources. There is clear experimental evidence to demonstrate three dimensional behavior in this type of equipment suggesting that numerical simulations of spray dryers need to include the three-dimensional nature of the flows[7]. Previous two-dimensional, axisymmetric simulations can only be regarded as indicative, at best, since they do not reproduce the basic physics that are involved [8].

As explained above, it is apparent that in order to avoid wall buildup and insufficient residence time of particle which influences the product quality, we must able to model of the complexity of the spray/air mixing and understand what happened inside the spray dryer?

The evaporation of pure liquid droplets has been extensively studied for many years, e.g., Ranz and Marshall [9];Manning [10] and Crowe[11]. Part of the reason why these systems were among the first to be studied is that they are considerably simpler than those involving a higher content of dissolved/suspended solids. Understanding such processes is not only commercially important in itself, e.g., for modelling fuel atomisation in engines, but conclusions drawn from their study also form the basis for understanding more complicated drying mechanisms [1,12] .

Modelling Approach

The flow in a spray dryer is turbulent and two-phase (gas and droplets or gas and particles). There are two commonly used approaches for modeling two-phase flow [13]. Firstly, one can treat the disperse phase as an extra fluid with its own flow field (Euler/Euler approach). In the case of spray drying, with rather concentration of particles, one usually use the second approach, the Euler/ Lagrange approach. In this approach the gas field is calculated first (Euler). This is done by calculating solutions of the Navier-Stokes and continuity equation on a grid of control volumes. Subsequently the particles are tracked individually (Lagrange). Along the particle trajectories the exchange of mass, energy and momentum with the continuous phase is calculated. These transfer terms are added to the source terms of the Navier-Stokes equations of the gas flow calculation. After the particle tracking, the air flow calculation pattern is recalculated, taking the transfer terms into account. This cycle of airflow calculation followed by particle tracking is repeated until convergence is reached. This scheme is called the Particle-In-Cell model [14].

The droplet field is established by integrating the differential equations for droplet motion to determine droplet velocities and, with further integration, droplet trajectories. At each time step along the trajectory, droplet size and temperature history are calculated using the equations for droplet mass and heat transfer rates. These equations can be found in [15, 16]. Since space is limited, they are not repeated here.

The evaporation rate of the spray is modeled using simple drying kinetics where

$$\frac{dm}{dt} = k_c A_d M_w (C_{w,s} - C_{w,\infty}) \quad 1$$

Where the evaporation rate is governed by gradient diffusion, with the flux of droplet vapour into the gas phase related to the gradient of the vapour concentration between the droplet surface and the bulk gas.

The concentration of vapour at the droplet surface is evaluated by assuming that the partial pressure of vapour at the interface is equal to the saturated vapour pressure, P_{sat} at the droplet temperature, T_d :

$$C_{w,s} = \frac{P_{sat}}{RT_d} \quad 2$$

The concentration of vapour in the bulk gas is calculated by:

$$C_{w,\infty} = X_w \frac{P}{RT_\infty} \quad 3$$

Where X_w is the local bulk mole fraction of water vapour, P is the local absolute pressure, and T_∞ is the local bulk temperature in the gas. The mass transfer coefficient in Eq. (1) is calculated from the Sherwood number correlation [9]:

$$Sh_{AB} = \frac{k_c d_d}{D_{i,m}} = 2.0 + 0.6 Re_d^{1/2} Sc^{1/3} \quad 4$$

The effect of turbulence on the droplet motion is modeled by the turbulent stochastic model. Turbulent stochastic tracking of droplets admits the effect of random velocity fluctuations of turbulence on droplet dispersion to be accounted for in prediction of droplet trajectories [17].

Case study

For CFD simulation, the spray dryer used in this article is a co-current pilot plant spray dryer by Niro Atomizer as shown in Fig.1. The geometry and air inlet size are the same as those used by Kieviet [6]. The nozzle atomizer is located at the top of the drying chamber; hot drying air enters the chamber through an annulus with the nozzle as its centre. The outlet of the spray dryer is a pipe mounted through the wall of the cone section of the chamber, bent downwards in the centre of the chamber. This type of spray dryers is a more complex geometry than the simple box configuration, which requires an unstructured mesh for accurate representation using 740000 tetrahedral mesh elements (Fig. 2). To check whether the solution was dependent on the mesh which had chosen, the mesh was refined to 1100000 elements. For each element of original mesh, the value of axial, radial and tangential velocities was compared with the corresponding values in refined mesh. The differences were smaller than 4%, therefore we can say that solution is mesh independence. As a first step, the modeling of the air flow without spray and swirl is performed. A second step will be the modeling the temperature and humidity when pure water droplets are tracked through the air under operational conditions with 5 swirl degree.

For modeling the air flow without spray, the velocity components (axial, radial and

tangential) of inlet air were: 6.03,-4.22 and 0.0 m/s respectively.

The nozzle was a hollow-cone-type centrifugal pressure nozzle (Spraying Systems Co: SX-type, spray angle 76°C) with initial velocity of 49 m/s. A spray is represented by a

10 droplet sizes, ranging from 10 to 138 μm . Rossin Rammler distribution parameters for 42 kg/hr of feed pure water are mean diameter and power of 68.6 μm of 2.45 respectively

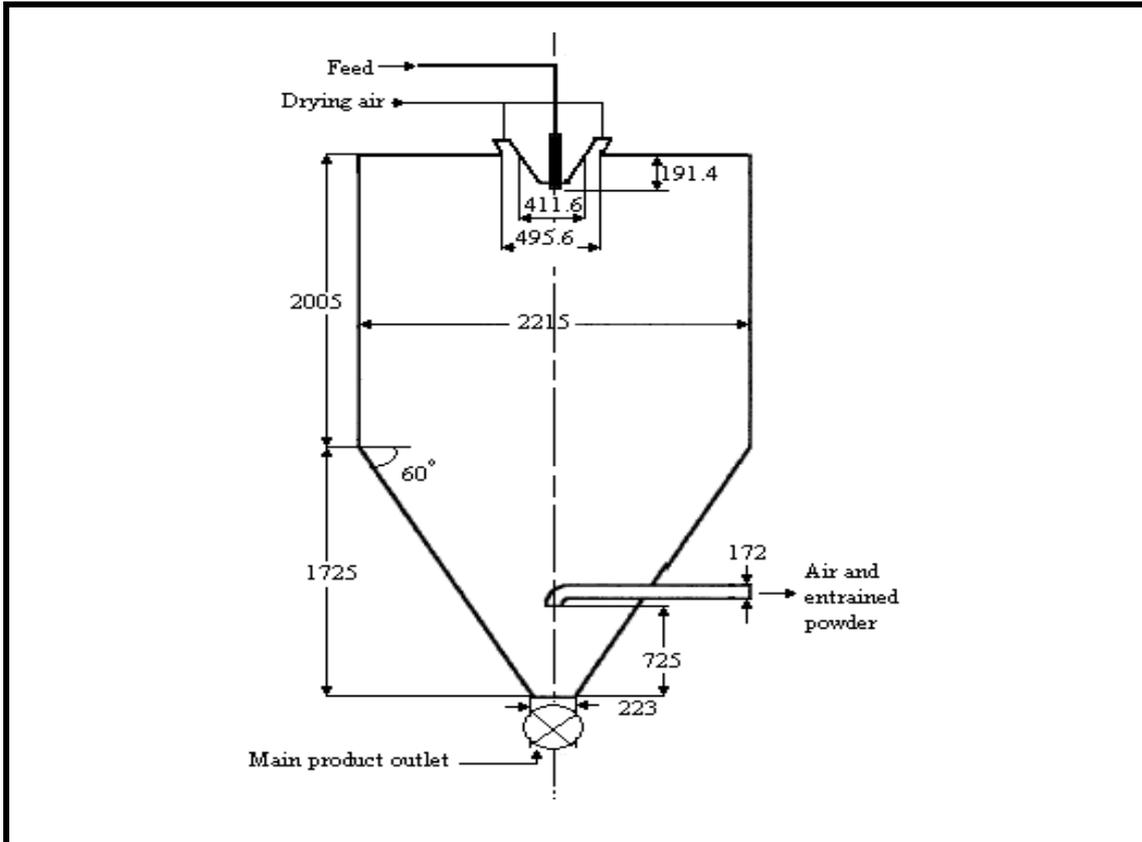


Fig 1: The geometry of the pilot plant spray dryer (the dimensions in mm)

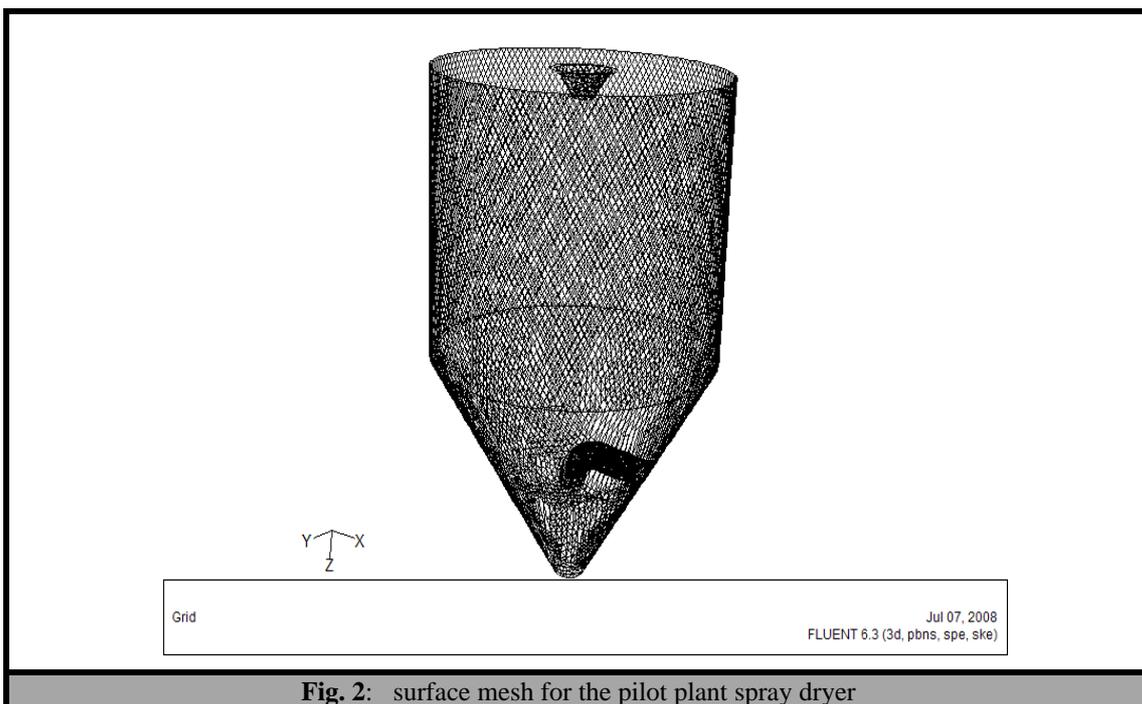


Fig. 2: surface mesh for the pilot plant spray dryer

Table 1. Boundary conditions used for case study			
Air flowrate (m ³ /s)	Air temperature (°C)	Air humidity(kg _w /kg _a)	Feed temperature (° C)
0.42	195	0.009	27
Air axial velocity (m/s)	Air radial velocity (m/s)	Air tangential velocity (m/s)	Swirl Degree
7.42	-5.19	0.649	5
Pressure at outlet (Pa)	Turbulence - k - value (m ² /s ²)	Turbulence- ε --value (m ² /s ³)	Chamber wall thickness (m)
-150	0.027	0.37	0.002
Wall material	Wall heat transfer coefficient (W/m ² .K)	Air temperature outside wall (° C)	Wall boundary condition
Steel	3.5	25	Escaped

Boundary Conditions

A $k - \varepsilon$ model was chosen to model the turbulence. The $k - \varepsilon$ model is the most commonly used in engineering practice because it converges considerably better than the algebraic stress model (ASM) and Reynolds stress model (RSM) and require less computational effort [6].

To determine the fate of the droplet trajectory when hit the wall of the drying chamber, Fluent [15] submit multi options that could be selected for the present work. The possible fates for a droplet trajectory are as "escaped", "trapped", "evaporated", "reflected" and "coalesced". In this work, 300 droplet trajectories were calculated and the "escaped" boundary condition is used, where the droplets are lost from calculation at the point of impact with the wall.

Of each droplet trajectory, the time of flight and the location of the end -point were recorded.

In order to compare the CFD predicted results with experimental results of Kieviet [6], the same conditions were used as tabulated in Table 1.

Results and Discussion

Analysis of the CFD simulation of air velocity profile without spray in the spray dryer, as shown the contour of air velocity (Fig.3) and the vector plot of the simulated air flow field (Fig.4), showed that the flow field consists of a fast flowing downward core with a slow recirculation around that core near the upper section of the conical part of the chamber. The core broadens as going down to the outlet.

The predicted and measured velocities at different levels are depicted in Fig.5. One note that the central core is of the radius of about 0.25m and the remainder of the chamber is at very low velocity (0.2 m/s). The highest

velocity magnitude in the core is about 7.3m/s at the 0.3m level. The sharp descent of velocity magnitude at the axis of the chamber is reduced as the air goes into the cone section of the chamber as shown in Fig.6 which shows the predicted and measured velocity for at different levels for region of radius 0.25m. This trend is agreed very well with the measured results. Only the predictions of velocity magnitudes at 1.0 m level are somewhat higher than the measured values. We expect the reason is that the air flow reveals periodicity in the velocity magnitude at several locations in the chamber and the 1.0 m level is one of its. This nature of air flow was noticed by Kieviet[6] when measuring the velocity signal in these locations. Therefore we can say that the air flow in this pilot plant spray dryer is transient in nature and we need a transient CFD simulation to consider this behavior.

In Fig. 7and 8, the predicted counters of temperature and humidity of air with spray are depicted. From these figures, it can be seen that a large volume of the chamber has almost constant temperature and humidity(107 °C, 0.04 kg_w/kg_a). It appears that most of the evaporation takes place in the fast flowing core.

It is practical to compare the predicted values with the measured values at certain points like outlet. As shown in Fig. 7 and 8 , the predicted value of temperature and humidity at outlet are 99 °C and 0.04 kg_w/kg_a but the measuring values at outlet are 86 °C and 0.04 kg_w/kg_a. This difference in temperature due to under estimation of wall heat transfer coefficient which is used to consider the heat loss from wall. Although of this, the predicted results agree very well with the measured results.

In Fig. 9, the vector plot of the predicted air velocity showed that the center of recirculation flow moved upward and toward the right side of the chamber in the case with spray and consequently the droplets flow in a large circle in this region resulting in an increase of the residence time (3.45 s) and promote the wall deposition as shown in Fig 10. This phenomena because of the coupled effect of momentum transfer between the air and spray, therefore we can say that effects of droplets / particles on the gas flow pattern are strong, although that the hypothesis which ignoring the effects of droplets on the gas flow pattern if the feed rate is less than 10% of the gas mass flow rate [18,19].

In Fig. 10, a 300 droplet trajectories which represent the spray of 10 droplet sizes are predicted. As can be seen, a bigger droplets (greater than $95 \mu m$) appear to be able to penetrate the fast flowing core into the slow recirculation zone whereas the smaller droplets are trapped in this core where the massive evaporation occur for this smaller droplets in the core region due to high air temperature as shown in contour of temperature pattern (Fig.7).

We find that a large fraction of the droplets is evaporated (65%), 26% of the droplets hit the conical part of the chamber wall, 7.5% of the droplets hit the cylindrical part of the chamber wall, and 1.5% of the droplets hit the roof. It is expected that this large deposition at the conical part is because of the air flow pattern effects in this region where the air approaches

the wall at right angles, the air velocity is zero, and this location can be considered as stagnation point (Fig.11) which reveal higher deposition. This explanation is consistent with the measured results of Kota [20].

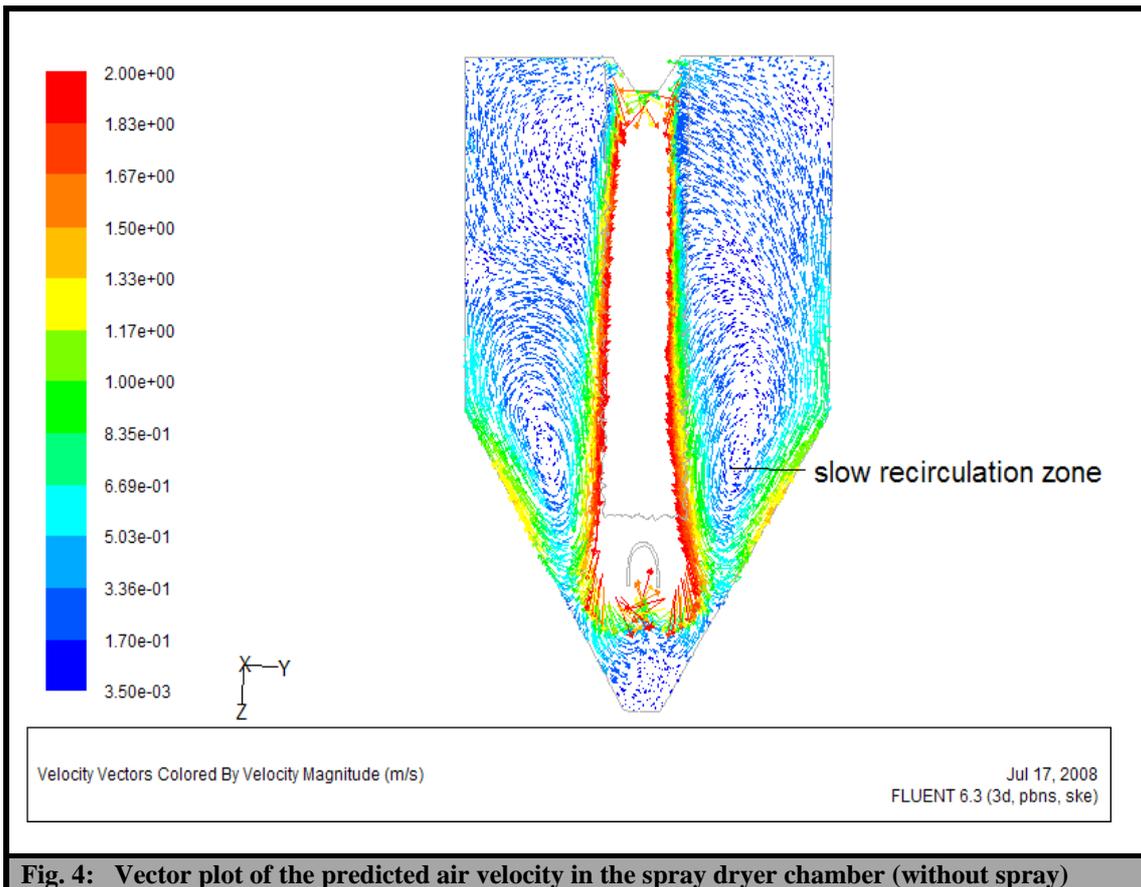
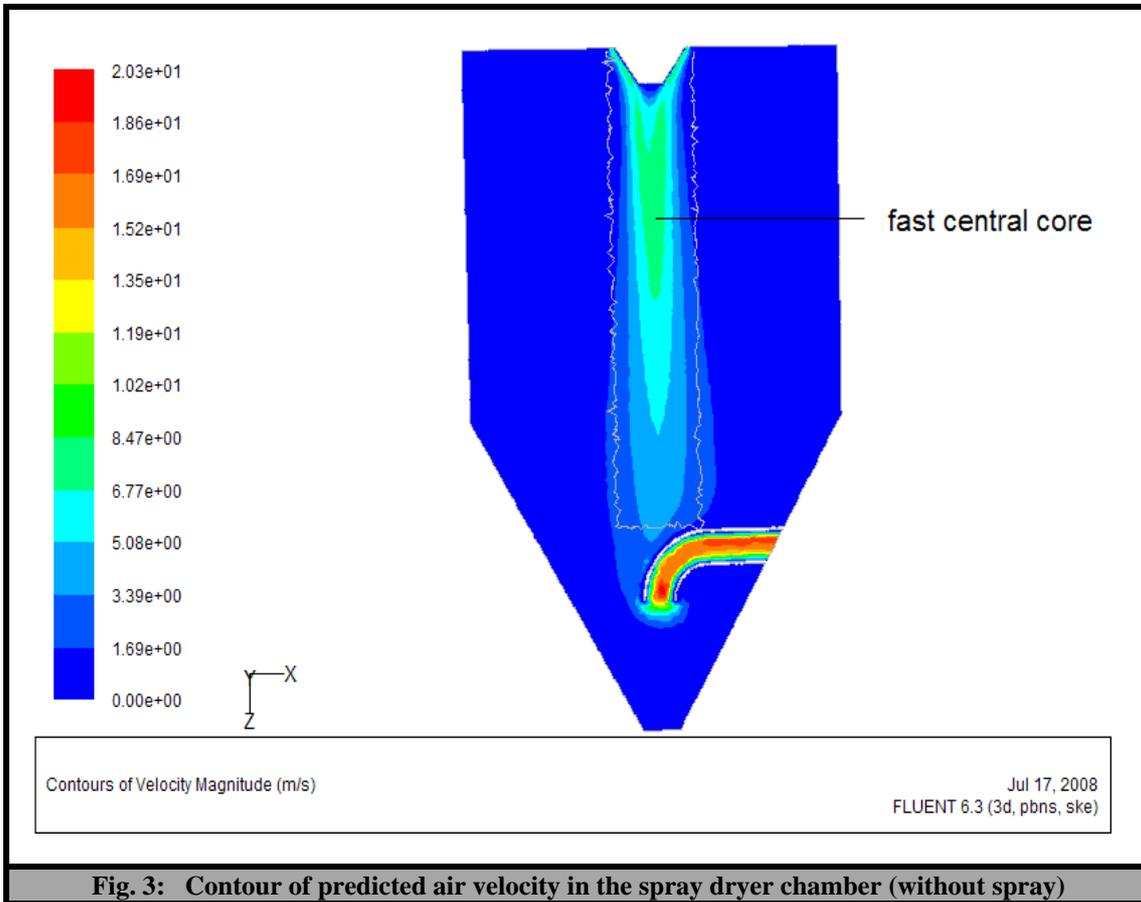
It is generally assumed that the residence time distributions of the particles/droplets are equal to those of the air [1]. Little research has been published on the actual residence times of particles in spray dryers [6, 21], with air flow rate $0.421 \text{ m}^3/\text{s}$, a mean residence time of air is (24.2 s) in this dryer [6].

Analysis of CFD simulation shows that mean droplets residence time in the dryer (2.4 s) is shorter than mean air residence time due to high initial droplets velocity.

Conclusions

The CFD simulations correctly predict the internal behavior of the spray dryer. It discerned that the air flow at specified conditions consist of fast flowing core and slow recirculation zone around it, and the air flow reveals a periodicity in some locations in the dryer which can simulated by a transient CFD model. The evaporation of droplet take place in the core region where the smaller droplets evaporated due to high air temperature.

There is a the coupled effect of momentum transfer between the air and spray resulting in a modifying of the droplet trajectory and affect the evaporation of droplets , further it relating to some phenomena such as wall deposition which is considered a serious problem when drying a real material.



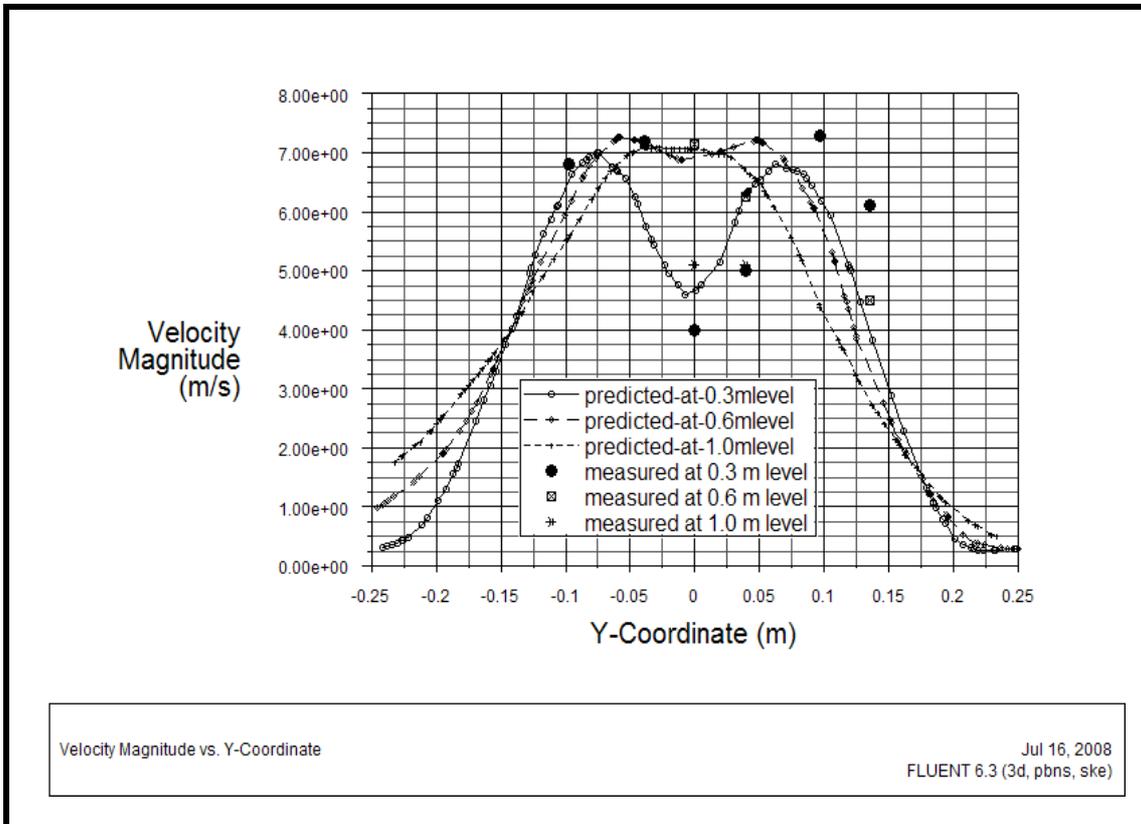


Fig. 6: Predicted and measured velocities at different levels measured from the ceiling (0.3, 0.6, 1.0 m) in the region of 0.25 m of spray dryer chamber

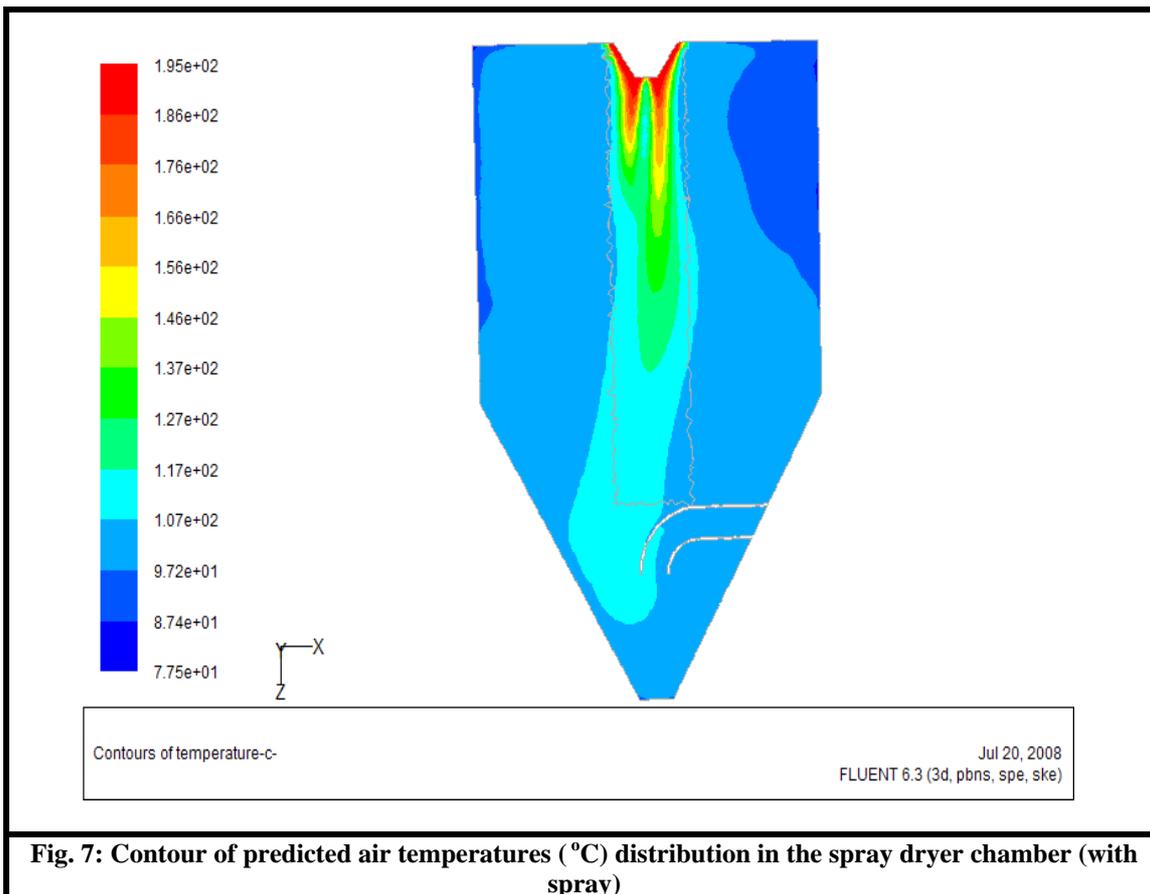


Fig. 7: Contour of predicted air temperatures ($^{\circ}\text{C}$) distribution in the spray dryer chamber (with spray)

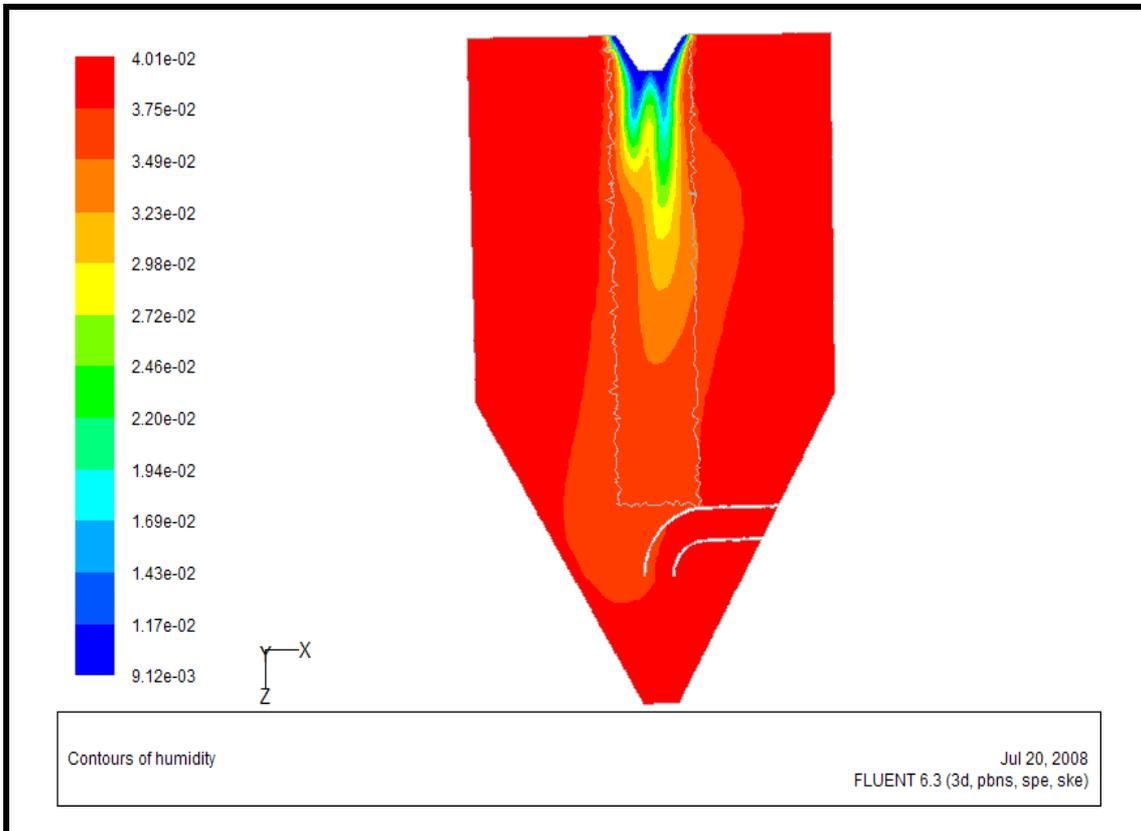


Fig. 8: Contour of predicted air humidities (kg_w/kg_a) distribution in the spray dryer chamber (with spray)

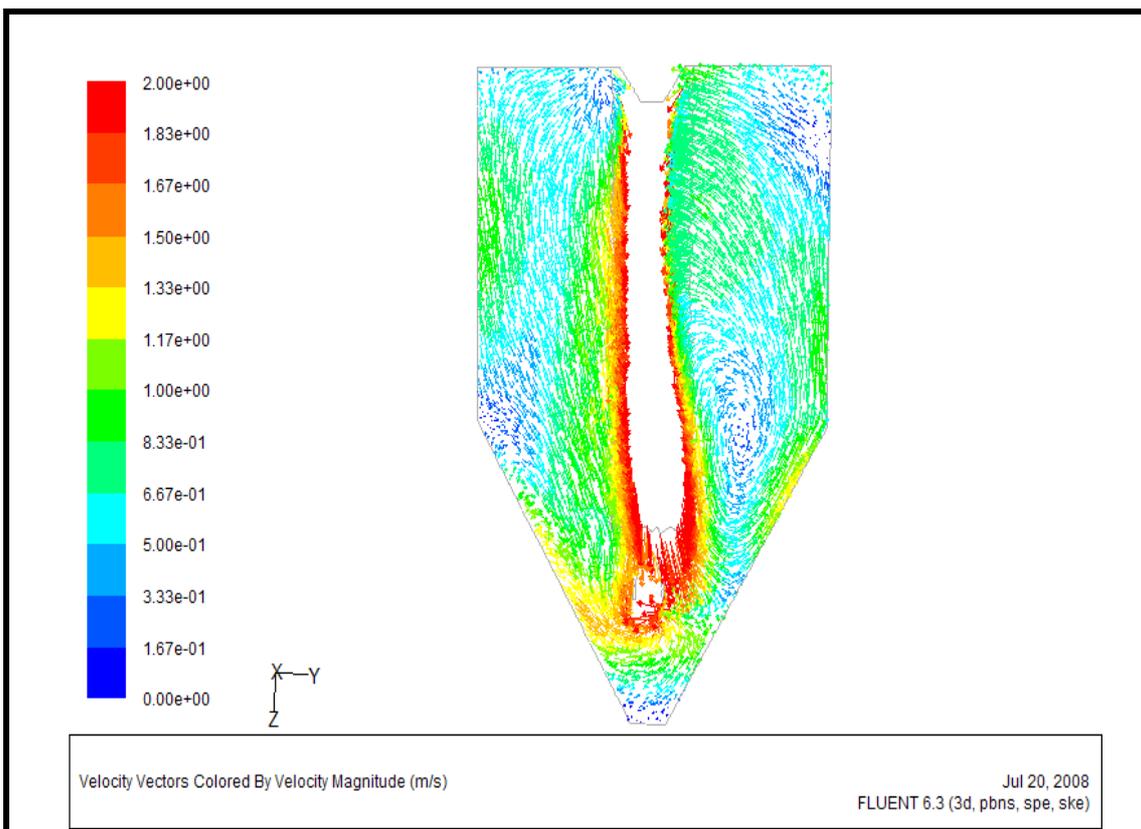


Fig. 9: Vector plot of the predicted air velocity in the spray dryer chamber (with spray)

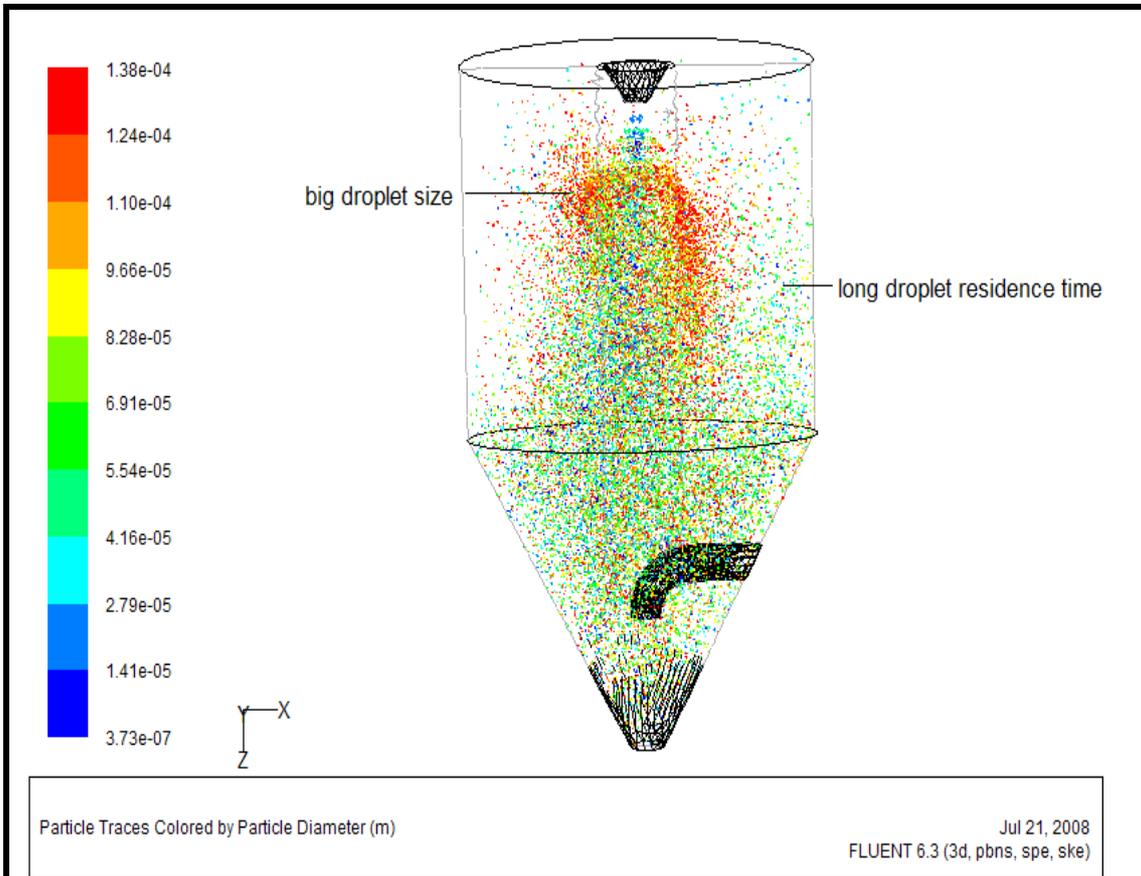


Fig. 10: Predicted droplet trajectories (300 droplet tracks represent a 10 droplet size distribution) in the spray dryer chamber

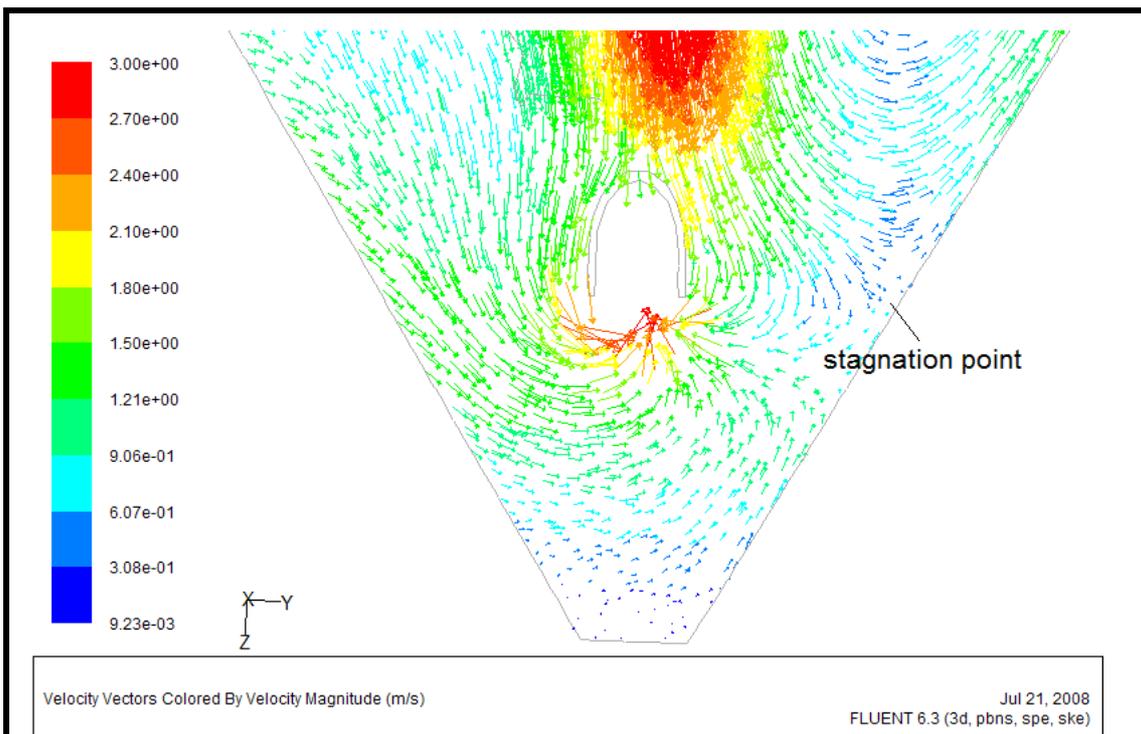


Fig. 11: Vector plot of the predicted air velocity in the lower conical of the spray dryer chamber

Nomenclature

A_d	droplet surface area	m^2
$C_{w,s}$	concentration of vapour	$kmol/m^3$
$D_{w,m}$	diffusion coefficient	m^2/s
d_d	droplet diameter	μm
k_c	mass transfer coefficient	m/s
M_w	molecular weight of vapour	$kg/kmol$
m	droplet mass	kg
P	pressure	pa
R	universal gas constant	
T	temperature	K
t	time	s
Re	Reynolds number	-
Sc	Schmidt number	-
Sh	Sherwood number	-
X_w	mole fraction	-
subscript		
d	droplet	
s	droplet surface	
sat	saturation	
w	water	
∞	bulk air	

References

- 1- Masters, K., Spray Drying Handbook, 4th ed., John Wiley and Sons, New York, 1985.
- 2- Oakley, D., "Produce uniform particles by spray drying", Chem. Eng. Prog., October, pp. 48-54, 1997.
- 3- Harvie, D.J.E., Langrish, T.A.G and Fletcher D.F., "A computational fluid dynamics study of a tall-form spray dryer". Trans. IChemE C, 80(3), 163-175, 2002.
- 4- Harvie, D.J.E.; Langrish, T.A.G.; Fletcher, D.F., "Numerical simulations of gas flow patterns within a tall-form spray dryer". Transactions of the Institution of Chemical Engineers, 79 (3) A, 235-248, 2001.
- 5- Oakley, D.E.; Bahu, R.E., "Computational modeling of spray drying, Comput". Chem. Eng. 17 (1993) 493-498.
- 6- Kieviet, F.G. Modelling Quality in Spray Drying. Ph.D. thesis, Eindhoven University of Technology, the Netherlands, 1997.

- 7- Southwell, D.B.; Langrish, T.A.G., "Observations of flow patterns in a spray dryer". Drying Technology, 18 (3), 661-685, 2000.
 - 8- Langrish, T.A.G.; Fletcher, D.F., "Prospects for the Modelling and Design of Spray Dryers in the 21st Century". Drying Technology, 21 (2), 197-215, 2003.
 - 9- Ranz, W.; Marshall, W., "Evaporation from drops". Chem Eng. Prog., 48(141), 173, 1952.
 - 10- Manning, W. P. and Gauvin, W. H., "Heat and Mass Transfer to Decelerating Finely Atomized Sprays", AIChE J., 6, 2, pp. 184-190, 1960.
 - 11- Crowe, C.T., "Modelling spray-air contact in spray-drying systems. In Advances in Drying", Mujumdar, A.S., Hemisphere New York, Vol. 1, 63-99, 1980.
 - 12- Handscomb, C., Kraft, M., Bayly, A., "A new model for the drying of droplet containing suspended solids", University of Cambridge, 2007
 - 13- Sommerfeld, M., "Review on numerical modeling of dispersed two phase flows", Proc. 5th Int. Symp. on Refined Flow Modelling and Turbulence Measurements, Paris, 1993.
 - 14- Crowe, C.T., Sharma, M.P., Stock, D.E., "The particle-source-in-cell (PSI-Cell) model for gas-droplet flows", Journal of Fluid Engineering, 99, 325-332, 1977.
 - 15- Fluent Manual, "Chap. 23: Discrete phase models", www.fluent.com, 2006.
 - 16- Al-Hemiri, A. A., Saad N. Saleh, "Computer Aided Design for Spray Dryers", Iraqi J. of Chem. And Petro. Eng., 8 (1), 27-34, 2007
 - 17- Haung, L.; Kumar, K.; Mujumdar, A. S., "A Parametric Study of the Gas Flow Patterns and Drying Performance Of Co-current Spray Dryer", Drying Technology, 21, 6, pp. 957-978, 2003.
 - 18- Reay, D., "Modelling continuous convection dryers for particulate solids-progress and problems", Drying '85, 5, pp. 67-74, 1985.
 - 19- Elghobashi, S., "On predicting particle-laden turbulent flows", Applied Scientific Research, 52, 309-329, 1994.
 - 20- Kota, K., Langrish, T., "Prediction of Deposition Patterns in a Pilot-Scale Spray Dryer Using CFD Simulations", Chem. Prod. and Proc. Modeling, 2(3), 26, 2007.
- Pham, Q. T., Keey, R., B., "Some experiments on the residence-time distribution of droplets in a cocurrently worked spray chamber", Can. J. Chem. Eng., 55(44), pp. 466-470, 1977.

حساب أنماط الجريان ودرجة الحرارة والرطوبة للهواء في وحدة المجفف الرذاذ

د. سعد ناهي صالح

قسم الهندسة الكيمياوية / كلية الهندسة / جامعة تكريت

E-mail : saad_nahi68@yahoo.com

الخلاصة :

يهدف هذا البحث إلى حساب أنماط توزيع جريان الهواء والرطوبة ودرجة الحرارة في وحدة المجفف الرذاذ ذو الجريان المتوافق باستخدام نموذج رياضي ثلاثي الأبعاد يعتمد على برنامج (Fluent6.3) (أحد تطبيقات CFD) حيث تمت معاملة طور القطره /الجسيمه باستخدام طريقة Lagrange approach
التطابق بين نتائج المحاكاة والنتائج المختبرية كان جيداً إذ تم تخمين جريان مركزي سريع للهواء في منتصف الوحدة محاطاً بمناطق إعادة دوران للهواء حول هذا الجريان المركزي.
كما تم بحث تأثيرات نمط جريان الهواء على مسارات القطرات ، توزيع زمن الاستبقاء للقطرات وعلى ترسيب القطرات على جدران المعدة بمحاكاة ترذيد الماء.

This document was created with Win2PDF available at <http://www.daneprairie.com>.
The unregistered version of Win2PDF is for evaluation or non-commercial use only.