Journal Homepage: -www.journalijar.com

INTERNATIONAL JOURNAL OF ADVANCED RESEARCH (IJAR)

ISSN NO. 2320-5407

Article DOI:10.21474/IJAR01/7185
DOI URL: http://dx.doi.org/10.21474/IJAR01/7185

RESEARCH ARTICLE

DESIGN AND MATHEMATICAL MODELING OF SPRAY DRYER USING CFD.

Ms. Utkarsha V. Malshe, Dr. K. R. Jethani and Dr. V. S. Vitankar.
Department of Chemical Engineering, AISSMS College of Engineering, Savitribai Phule Pune University FluiDimensions, Pune.

……………………………………………………………………………………………………....

Manuscript Info

……………………….

Manuscript History

Received: 02 April 2018
Final Accepted: 04 May 2018
Published: June 2018

Keywords:-
Spray dryer, Computational fluid dynamics, multiphase, rotary disc atomizer.

Abstract

……………………………………………………………………

This paper presents the prediction of air flow, temperature patterns and mass fraction of water vapour in a co-current spray dryer fitted with rotary disc atomizer. The multiphase CFD model was developed in which the gas phase is modeled with Eulerian approach and droplet/particle phase is modeled by Lagrangian approach. The transition K-Kl-Omega model is used to model the turbulence in spray dryer.
In this work the effect of cylinder height and diameter on air flow pattern has been studied**.**

……………………………………………………………………………………………………....

Introduction:-

Spray dryer is an essential unit operation for converting the liquid feed material in to the dried powder form.In powder form the product results with the good quality, low water activity, easier transport and storage, easy handling, high surface area etc. Spray drying is one of the best drying methods to convert directly the fluid materials into solid or semi-solid particles.[4] It is a unit operation in which a liquid product is atomized in a hot gas to directly gain a powder. The drying medium generally used is air or sometimes an inert gas such as nitrogen.The primary liquid feeds in spray dryer can be a solution, an emulsion or a suspension. The spray dryer is used for both the heat resistant and heat sensitive products. [9]

One of the big problems facing by spray dryer designer and operators is the complexity of the spray/air mixing process in spray chamber 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, 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. A very important phenomena of spray dryer operability is the particle wall deposition which is affected by temperature and humidity patterns inside the dryer when moist particles contact the spray dryer wall. These depositions can lead to build up large amounts of product on the wall. Such depositions may be dangerous, as they can fall and cause damage to the chamber wall.[11]

The objective of this paper is to study the flow patterns in a co-current spray dryer and to study the effect of change of the cylindrical height and diameter on flow patterns inside the dryer.

Corresponding Author:- Ms. Utkarsha V.
Address:- Department of Chemical Engineering, AISSMS College of Engineering, Savitribai Phule Pune University Flui Dimensions, Pune.

Mathematical Modeling:-

In the spray dryer, hot gas is modeled as a continuous phase and the liquid/droplet as dispersed phase. There are two commonly used approaches for modelling the two phase flow, one is Euler/Euler Approach and another is Euler/Lagrange Approach.[5]

In Euler/Euler Approach the concept of phasic volume fraction is introduced. These volume fractions are assumed to be the continuous functions of space and time and their sum is equal to one. The first approach is used when both phases are in same fraction. In spray dryer the droplet phase occupied very small fraction of total volume so Euler/Euler approach cannot be used. Therefore, the Euler/Lagrange Approach is used here.The gas flow field is calculated first using Euler approach and this is done by calculating the solution of continuity equation and Navier-Stockes equation on a grid of control volume. The droplet phase is calculated by tracking the number of individual particle through the gas flow using Lagrange approach.[1,2]

Governing Equations for the Continuous Phase (Gas mixture of Air and Water Vapor):-

Mass conservation or Continuity equation

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\,\vec{v}) = S_m \quad (1)$$

Momentum conservation equations

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F} \quad (2)$$

$$\bar{\bar{\tau}} = \mu[(\nabla\vec{v} + \nabla\vec{v}^T) - \frac{2}{3}\nabla.\vec{v}I] \quad (3)$$

Energy conservation Equation

$$\frac{\partial}{\partial t}[\rho E] + \nabla.[\vec{v}(\rho E + p)] = -\nabla.(\sum_j h_j J_j) \quad (4)$$

Governing Equations for Dispersed Phase (Droplets or Particles):-

The Euler-Lagrangian Approach is used to obtain particle trajectories by solving the force balance equation for the particles which are as follows

$$\frac{du_{pi}}{dt} = C_D \frac{18\mu}{\rho_p d_p}\frac{Re}{24}\left(u_i - u_{pi}\right) + g_i \frac{\rho_g - \rho}{\rho_g} + Fx_i \quad (5)$$

$$Re = \frac{\rho d_{p|u_p - u|}}{\mu} \quad (6)$$

$$C_D = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2} \quad (7)$$

Where, Fxi = additional forces, which can be Brownian force, Saffman's lift force, thermophoretic force, etc

Mass and Heat Transfer between the Two Phases:-

The mass transfer equation for evaporation is as follows

$$N_i = k_c\left(C_{i,s} - C_{i,\infty}\right) \quad (8)$$

The heat balance equation for the heat transfer between the droplet and the gas calculated at each time step is as follows

$$m_p c_p \frac{dT_p}{dt} = hA_p(T_\infty - T_P) + \frac{dm_p}{dt}h_{fg} \quad (9)$$

The boiling rate equation is applied as follows

$$\frac{d(d_p)}{dt} = \frac{4k_\infty}{\rho_p c_{p,\infty} d_p}\left(1 + 0.23\sqrt{Re_d}\right)\ln\left[1 + \frac{c_{p,\infty}(T_\infty - T_p)}{h_{fg}}\right] \quad (10)$$

Turbulence Models:-

Three equations for turbulent kinetic energy(K_T), laminar kinetic energy (K_L) and the inverse turbulent time scale (ω) are used to model the turbulence using transition K- K_L- ω model.

$$\frac{D_{KT}}{DT} = P_{KT} + R + R_{NAT} - \omega K_T - D_T + \frac{\partial}{\partial xj}\left[\left(v + \frac{\propto_T}{\propto_K}\right)\frac{\partial K_T}{\partial x_j}\right] \quad (11)$$

$$\frac{D_{KL}}{DT} = P_{KL} - R - R_{NAT} - D_L + \frac{\partial}{\partial xj}\left[\left(v\frac{\partial K_L}{\partial x_j}\right]\right. \quad (12)$$

$$\frac{D\omega}{Dt} = C_{\omega 1} * \frac{\omega}{K_T} * P_{KT} + \left(\frac{C_{\omega R}}{f_W} - 1\right)\frac{\omega}{K_T}(R + R_{NAT}) - (C_{\omega 2} * \omega^2) +$$

$$C_{\omega 3} * f\omega * \propto_T * \ f_W{}^2 * \frac{\sqrt{K_T}}{d^3}) + \frac{\partial}{\partial \mathrm{xj}}\left[(\upsilon + \frac{\propto_T}{\propto_K})\frac{\partial \omega}{\partial x_j}\right] \quad (13)$$

Evaporation Modeling:-

The liquid evaporation model is model for particles with heat and mass transfer. The model uses two mass transfer correlations depending on whether the droplet is above or below the boiling point.

Boiling point is determined by Antoine equation:

$$\ln \mathrm{P_{vap}} = \mathrm{A} - \frac{\mathrm{B}}{\mathrm{T+C}} \quad (14)$$

When the particle is above boiling point, the mass transfer is determined by

$$\frac{\mathrm{dm_p}}{\mathrm{dt}} = -\frac{\mathrm{Q_C+Q_R}}{\mathrm{V}} \quad (15)$$

Where, V is latent heat of evaporation of particle

Q_c and Q_R are the convective and radiative heat transfer rates

When the particle is below boiling point, the mass transfer is determined by

$$\frac{dm_p}{dt} = \pi d_p \ \rho \ D \ S_h \frac{W_c}{W_G} \ \ ln\left(\frac{1- x_s}{1-x_{vap}}\right) \quad (16)$$

Experimental Details and Boundary Conditions:-

Experimental Details:-

For CFD simulation the co-current pilot plant spray dryer fitted with rotary disc atomizer is used. In rotary disc atomization, the liquid feed is distributed centrally on the disc. The liquid extends over the rotating surface as thin film and then gets convert into liquid droplets.

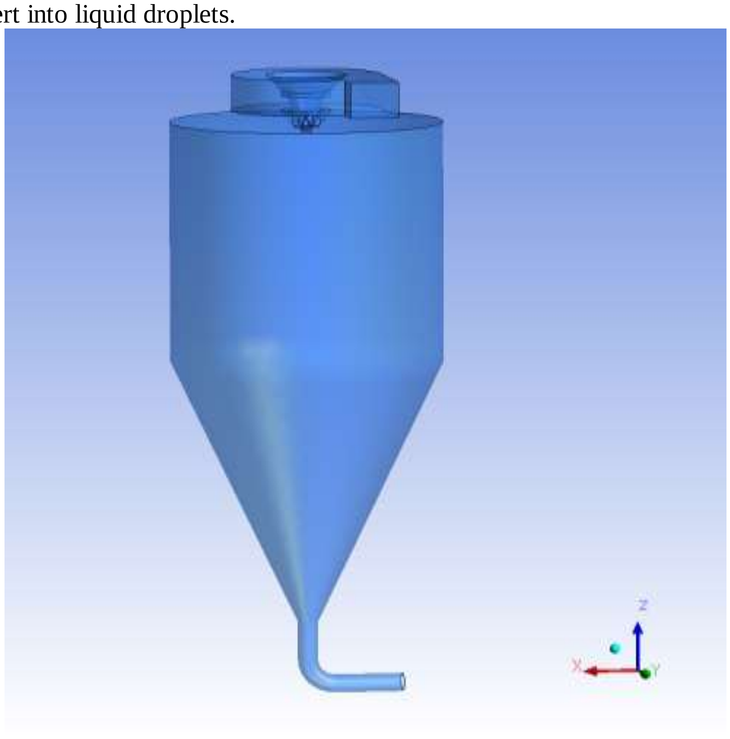


Figure 1:-Geometry of dryer drawn in Design Modular

Figure 1 shows the geometry of co-current spray dryer fitted with rotary disc atomizer with cylindrical diameter 4 m and cylindrical height 4m.

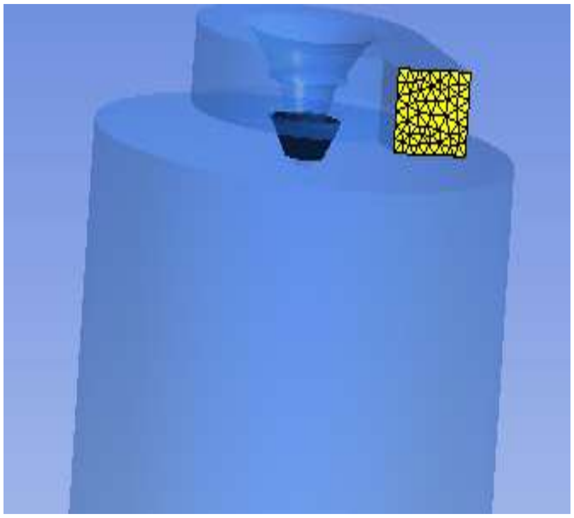


Figure 2:-Air Inlet

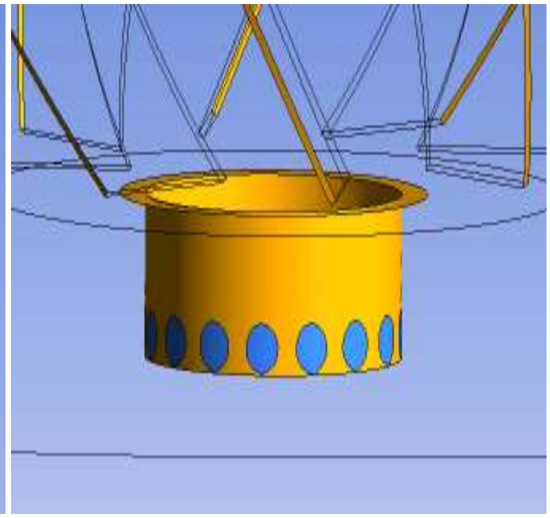


Figure 3:-Rotary Disc Atomizer

The yellow part in Figure 2 shows the inlet for air. Figure 3 shows the rotary disc atomizer. The blue colour circles are the small holes from where the liquid feed enters in the dryer in the form of small droplets.

Boundary Conditions:-

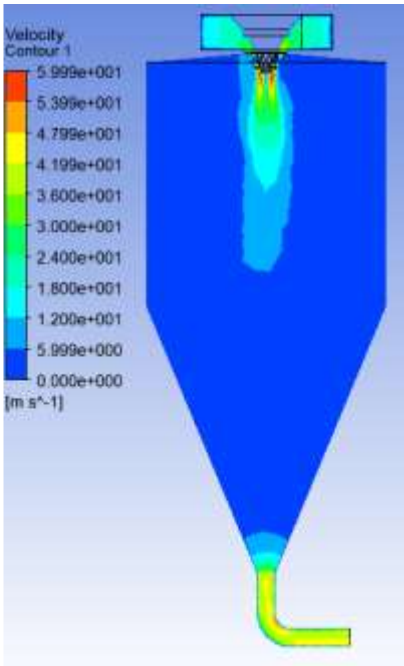
The following boundary conditions are used to simulate the spray dryer in ANSYS Fluent:

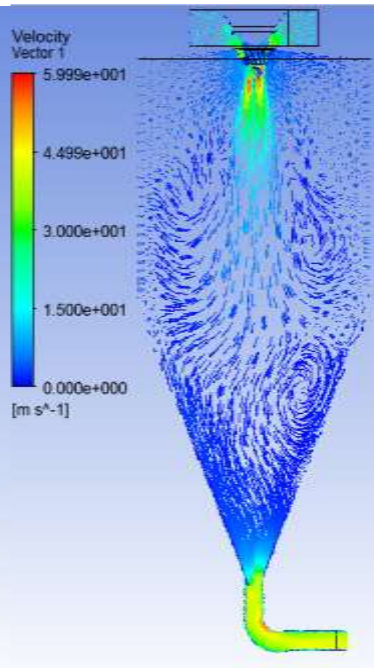
Table 1:-Boundary Conditions for air and feed

Inlet flow rate of air (Kg/Sec)	Inlet air Temperature (K)	Percentage of solids in slurry (%)	Slurry Feed rate (Kg/Sec)	Droplet Temperature (K)	Speed of rotation of disc (rpm)
2.1612	478	35	0.1291	303	14000

Results And Discussion:-

As a first step, CFD simulation is performed without adding the liquid feed in spray dryer. Figure 4 shows the velocity contour in the dryer. Due to the tangential inlet, the strong swirling flow is observed in the central core of the drier. The swirl gradually expands. Due to this, a central high velocity plume is seen. The rest of the cylindrical section shows very low velocity. Due to this gradient in the flow, the k-kl turbulence model has been used..





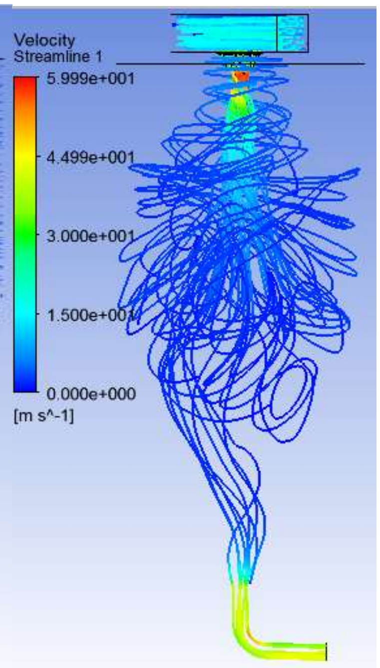


Figure 4:-Velocity Contour **Figure 5:-**Velocity Vector **Figure 6:-**Velocity Streamlines

Figure 5 shows the velocity vector. Recirculation in flow can be clearly seen in the cylindrical portion of dryer. Due to recirculation, air goes back upwards and gets mixed in the central core. Figure 6 shows the velocity Streamlines. The clear recirculation is seen from these streamlines.

Simulation with liquid feed:-

Feed is introduced in to the dryer with the help of atomizer. Due to high speed rotating wheel liquid feed is converted into small droplets. These droplets come in contact with hot air and transport phenomena takes place in both the phases. As droplets enters in to the dryer it distracts the air flow. Figure 7 shows the velocity contour after addition of feed. The air velocity increases here, this is because of the speed of rotation of disc which was given while adding the droplets in to the dryer.

Figure 8 shows the temperature contour after evaporation.Near the atomizer temperature is higher and as move further temperature drop is observed. Figure 9 shows the water mass fraction contour after addition of feed. The variable distribution of mass fraction of water is observed.

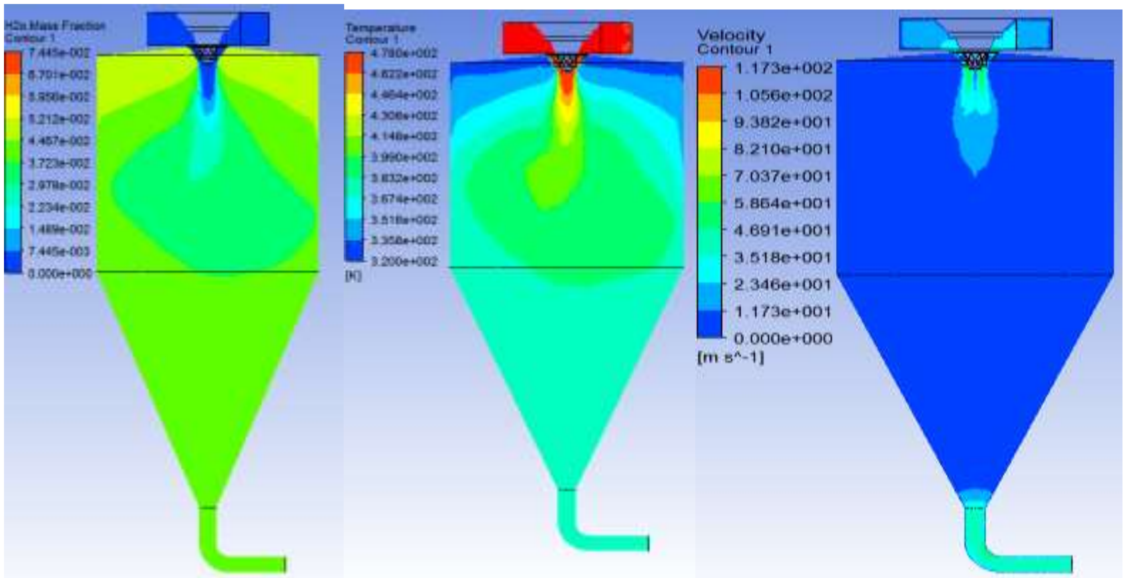
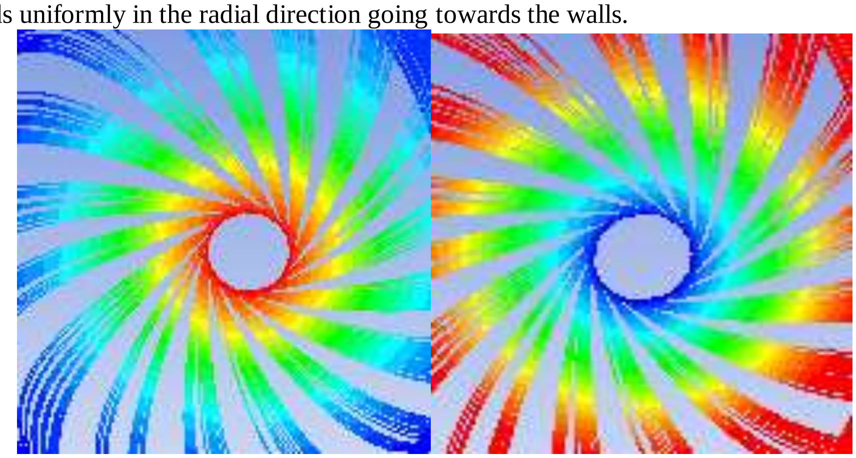


Figure 7:-Velocity Contour after **Figure 8:-**Temperature Contour **Figure 9:-**Water mass fraction addition of feed Contour

Figure 10 (a) shows the particle track coming out of the atomizer. These particle tracks are coloured by velocity.The droplets are formed due to the high speed rotation of the rotating disc. The droplets move radially outward at high velocity. It spreads uniformly in the radial direction going towards the walls.

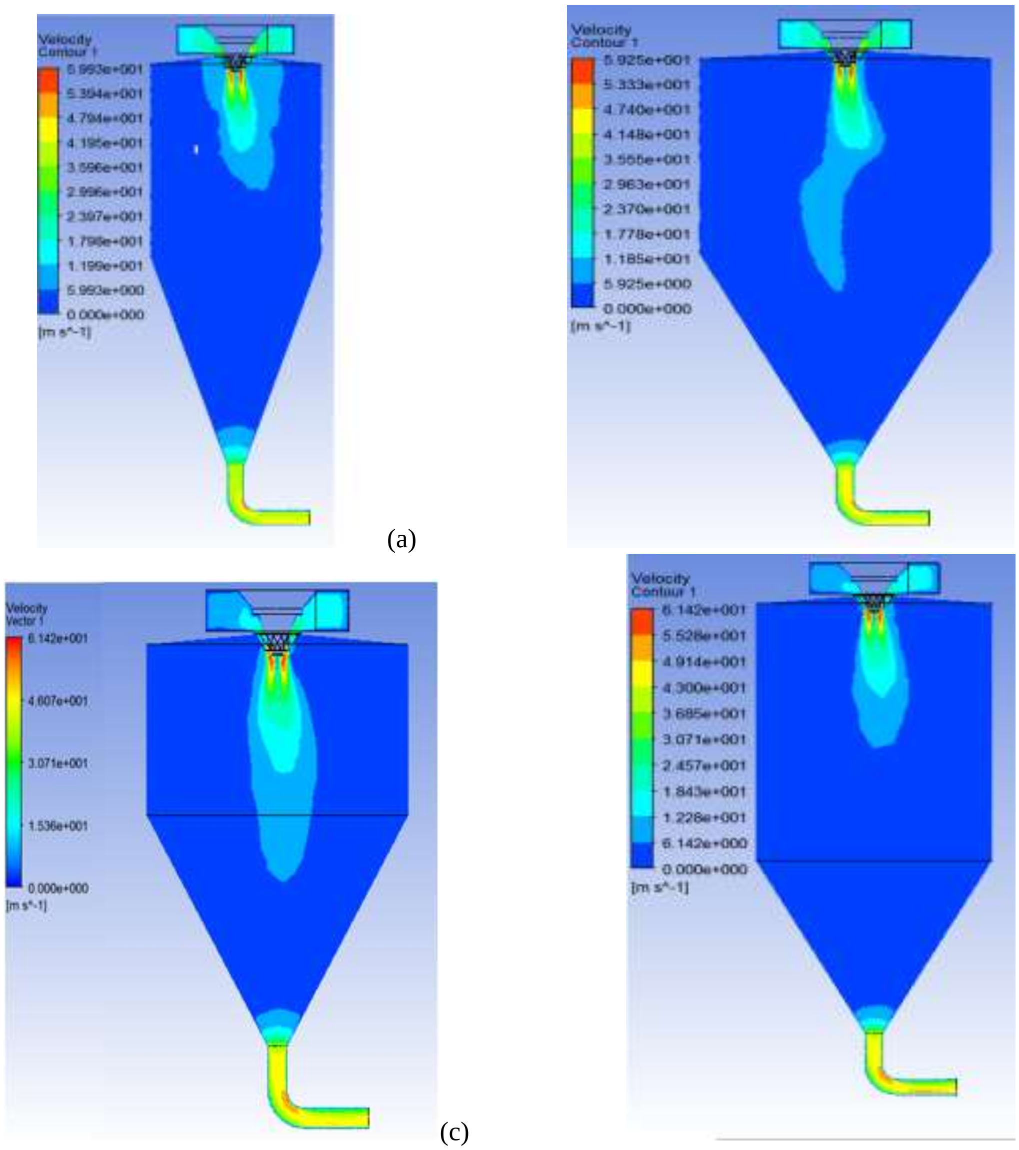


(a) (b)

Figure 10:-Particle tracks coloured by velocity and temperature

Figure 10 (b) shows the particle tracks coloured by temperature. When particle enters into the dryer, its temperature is relatively low as compared to the hot air. At the time of contact air gives heat to the droplet and it losses moisture and becomes dry.

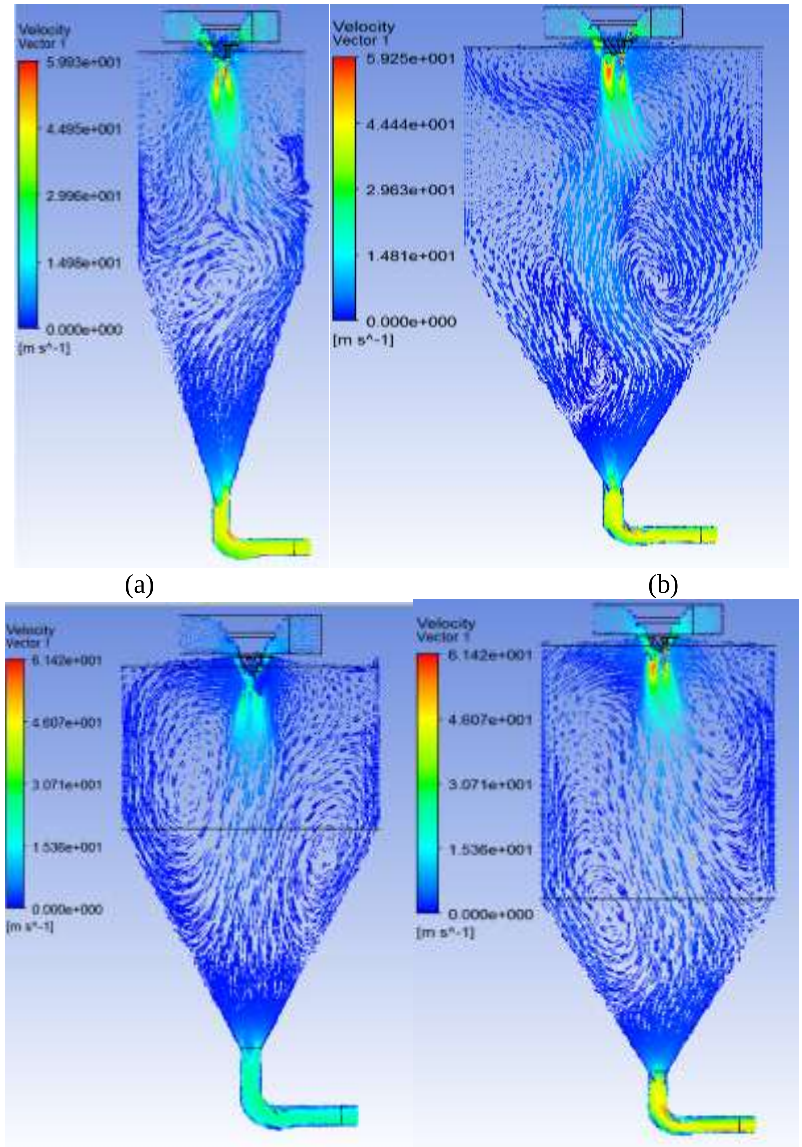
Because of the recirculation of the moist air, the spray drying operation faces the problem of wall deposition. In order to study the effect of change of cylindrical height and diameter on air flow pattern, recirculation, the CFD simulation was done without adding the droplets, by varying the cylindrical height to 2.5 m and 5 m. and the cylindrical diameter was changed to 3.2 m and 4.8 m.



(a) (b) (c) (d)

Figure 11:-Velocity Contour without adding feed after change in diameter and height

Figure 11(a) shows the velocity contour in the decreased diameter case having diameter 3.2 m. The velocity contour is centrally flowing core in this case. Figure 11(b) shows the velocity contour in the increases diameter case having diameter of cylinder as 4.8 m. This contour is the plum which is bended towards the left side is observed. Figure 11(c) shows the velocity contour in decreased cylindrical height case having the cylindrical height as 2.5m. Figure 11(d) shows the velocity contour in the increased cylindrical height case having cylindrical height as 5m.

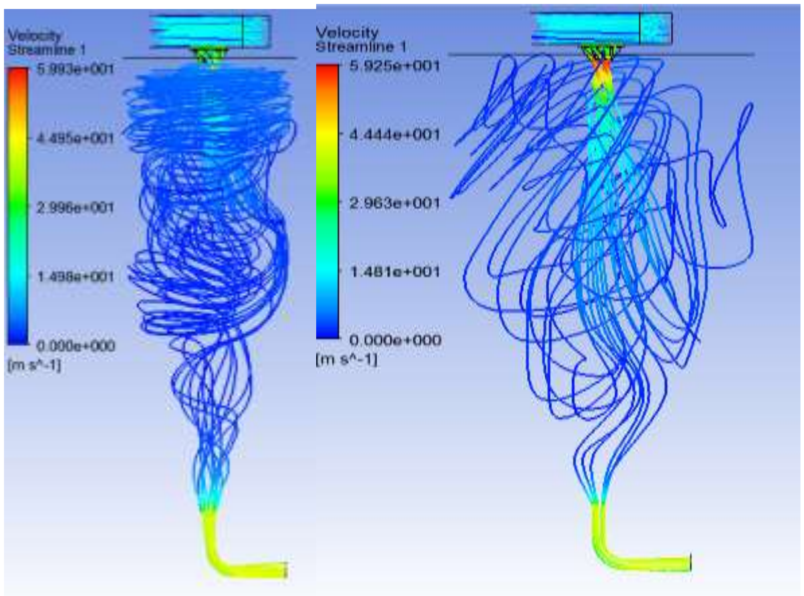


(a) (b)

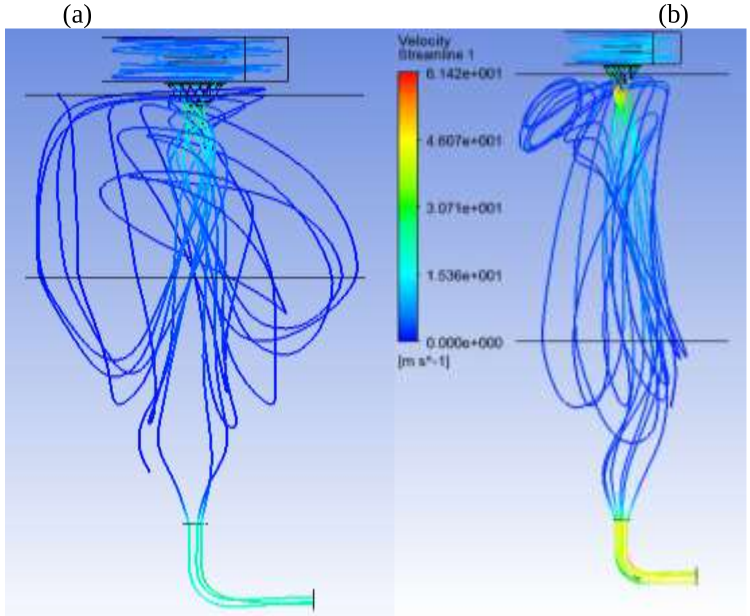
(c) (d)

Figure 12:-Velocity Vectors in new cases

Figure 12 shows the velocity vectors.Figure 12 (a) shows more recirculation because the diameter is decreased. In Figure 12(b) less recirculation is observed than 12(a). Figure 12(c) shows the velocity vectors in decreased height case. Figure 12(d) shows the velocity vectors in increasd height case, in which recirculation is minimum as compared to all other three cases.



(a) (b)



(c) (d)

Figure 13:-Velocity Streamlines

Figure 13 shows the velocity streamlines. We can easily observe the recirculation with the help of streamlines. In all above four cases the maximum recirculation is observed in the figure 13(a), that is in the case of decreased diameter. Minimum recirculation is observed in the figure 13(d), that is the case of increased height. Figure 13 (b),(c) shows the streamlines in the case of increased diameter and decreased height respectively.

Conclusion:-

A three-dimensional computational fluid dynamic model for rotary disc atomizer was developed and studied. The results obtained from simulation are presented in terms of velocity vectors, streamlines,velocity contour, temperature contour, water mass fraction. Results show that because of recirculation there will be a problem of wall deposition can occur.

Hence the another four cases were simulated for air flow by changing the diameter and height of cylinder. From the results it is observed that maximum recirculation is in the case of reduced diameter, having the diameter of cylinder as 3.2 m. The minimum recirculation is observed in the case of increased height of cylinder, having cylindrical height as 5 m.

From predicted results, it can be concluded that to minimize the problem of wall deposition which occurs because of recirculation, we can increase the cylindrical height of dryer.

References:-

1. Frank Geert Kieviet “Modelling quality in spray drying” Eindhoven university of technology,1997, ISBN-90-386-1008-4.
2. G. T. Crowe, M. P. Sharma, D. E. Stock, “The Particle-Source-ln-Cell (PSI-CELL) Model for Gas-Droplet Flows”, J Fluid Eng. 9 (1977) 325-332.
3. J.R. Gabitesa, J. Abrahamsona, J.A. Winchesterb , “Air flow patterns in an industrial milk powder spray dryer”chemical engineering research and design 8 8 (2010) 899–910.
4. Li Xin Huanga, K. Kumarb, 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 45 (2006) 461–470.
5. L. Huang and A.S. Mujumdar, “Development of a New Innovative Conceptual Design for Horizontal Spray Dryer via Mathematical Modeling” Drying Technology, 23: 1169–1187, 2005.
6. Li Xin Huang , M. L. Passos , Kurichi Kumar & A. S. Mujumdar, “ A Three-Dimensional Simulation of a Spray Dryer Fitted with a Rotary Atomizer’ Drying Technology, 23: 1859–1873, 2005.
7. Lixin Huang a , Kurichi Kumar b & A. S. Mujumdar, “A Parametric Study of the Gas Flow Patterns and Drying Performance of Co-current Spray Dryer: Results of a Computational Fluid Dynamics Study”,drying technology Vol. 21, No. 6, pp. 957–978, 2003.
8. Maksim Mezhericher, Avi Levy, Irene Borde, “Three-Dimensional Spray –Drying Model Based on Comprehensive Formulation of Drying Kinetics”,Drying Technology, 30:1256-1273, 2012.
9. Phisut, N. “ Spray drying technique of fruit juice powder: Some factors influencing the product properties” International Food Research Journal 19(4): 1297-1306, (2012)
10. Rajeshkhara M C, Raghvendra N, “CFD Simulation of Co-current Spray Dryer for Silica Powder Production” International Advanced Research Journal in Science, Vol 2, Issue 6, June 2015
11. S.N.Saleh, “CFD Simulations of a Co-current Spray Dryer” World Academy of Science, Engineering and technology, Vol 4.