The University of western Ontario
(Department of Mechanical and Material Engineering)

Applied Computational Fluid Mechanics and Heat Transfer
Subject Code: MME 9614

"Instructor: Prof. Chao Zhang"

Date: November 11, 2016

Assignment No. 3

"Numerical Modelling of Experimental Setup of coating booth to find out Coating Layer Thickness and Particle Transfer Efficiency"

Submitted to:
Prof. Chao Zhang
Department of MME

Submitted by:
Jagdeep Singh
Meng(MME)
250911825

APPLIED COMPUTATIONAL FLUID MECHANICS AND HEAT TRANSFER
Course MME 9614

Department of Mechanical and Materials Engineering
University of Western Ontario

1.PROBLEM DESCRIPTION

This problem involves numerical simulation of a coating booth to understand the two-phase flow field inside the coating booth. Numerical simulation is used to calculate the transfer efficiency and coating layer thickness. The flow inside the coating booth is modelled as 3-Dimensional turbulent air flow with continuous solid particle as a discrete phase. The continuous gas flow is predicted by solving navier- stokes equation with 2 equation model i.e. K-epsilon model with enhanced wall treatment. Gas and solid 2 phase flow considered to be dilute flow because solid volume fraction is less than 0.5%. A two-way coupling is used in this numerical modelling because solid volume fraction is greater than 0.01%. Particles are inert in nature therefore there is no reaction is considered in this scheme.

1.1Operating Conditions:

Air Mass Flow Rate	0.003 (kg/s)
Inlet Air Velocity	5 (m/s)
Suction Pressure	-100 pa(gage)
Particle Mass Flow Rate	0.002 (kg/s)
Inlet Particle Velocity	5 (m/s)

1.2Particle Parameters:

Density	1550 kg/m^3
Diameter	50 (micron meter)
Shape	Spherical

1.3Dimension of the 3-D coating booth and coating surface are presented in the given table:

Booth Dimensions	1 m (L) x 1 m (D) x 1.5 m (H)
Coating surface	0.5 m x 0.5 m
Powder spray gun location	Centre of the front surface (1 m x 1.5 m)

Diameter of the suction holes are given 0.15 meter.

Diameter of the Gun can be calculated from the below given formula:

Mass flow rate of air = Density of Air X Area x velocity of air

Radius = 0.0125 m
Diameter = 0.025 m

1.4Phases:

1. For the gas phase, air is chosen as the primary phase as given in the assignment problem.

2. For the solid phase, anthracite is chosen as the secondary phase as given in the assignment problem. Density of the anthracite is given as 1550 kg/m^3.

2.MATHEMATIC MODELS

To capture the multiphase flow physics inside the coating booth the following mathematical models are considered for gas and solid phase:

2.1For Gas phase:

$$Navier - Stokes\ equations + \rho * F_D (\overrightarrow{U_s} - \overrightarrow{U_g}) \qquad \text{Eq. 2.1}$$

Where,

$$\boldsymbol{Drag\ Force} = \boldsymbol{\rho * F_D (\overrightarrow{U_s} - \overrightarrow{U_g})} \qquad \text{Eq. 2.2}$$

If,

$$\overrightarrow{U_s} = \overrightarrow{U_g} \qquad \text{Eq. 2.3}$$

Then,

$\boldsymbol{Drag\ Force} = 0$ Eq. 2.4

2.2For Solid Phase:

Lagrangian Method

Particle-Particle Interaction is neglected

$\frac{d\overrightarrow{U_s}}{dt} = F_D\left(\overrightarrow{U_s} - \overrightarrow{U_g}\right) + \vec{g}(\rho_s - \rho_g)/\rho g$ Eq. 2.5

$\frac{d\vec{X}}{dt} = \overrightarrow{U_s}$ Particle Trajectories Eq. 2.6

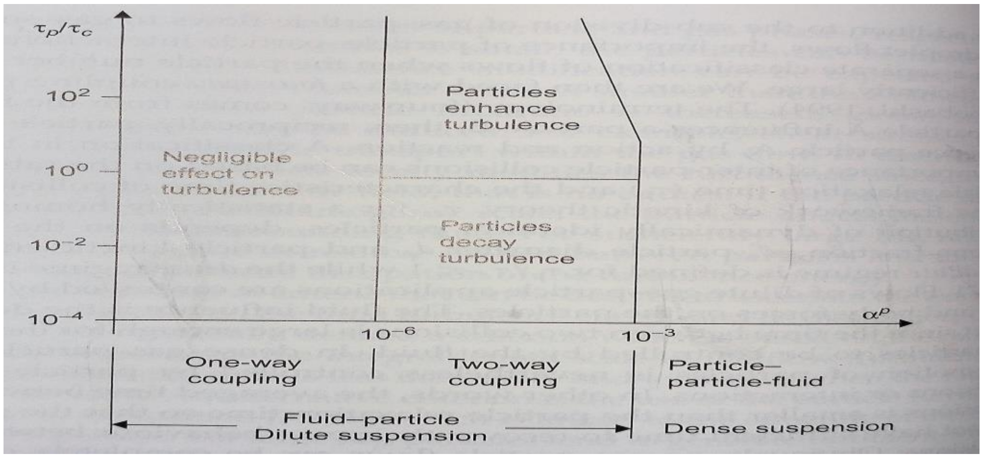
2.3Turbulence Effect:

Stochastic tracking approach is used to account for the random effects of the turbulence on particles dispersion.

For statistical accuracy, Large number of particles need to be Considered, 10,000 to 100,000 particles.

For determination whether it is a one, two or four-way coupling by particle-turbulence suggested by Elghobashi (1994), which is shown below in the figure. For particle volume fraction is less than 10e-6, particle will not have any influence on the turbulence of the gas phase there it is considered very-very dilute flow therefore it can be considered as one way coupling. If the volume fraction is in between 10e-6 to 10e-3 it can be adopted as two way coupling as shown in the figure below:

In our case the volume fraction is in the range of 10e-6 to 10e-3 there we must considered two-way coupling.



Proposed map for particle-turbulence modulation (after Elgobashi,1994). This figure is taken from the book "Computational Techniques for Multiphase Flows" Guan Heng Yeoh, Jiyuan Tu

Turbulent Modelling first equation for Kinetic Energy:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right) + \nabla p + 2\mu_t E_{ij} E_{ij} - \rho\varepsilon \qquad \text{Eq. 2.7}$$

Turbulent Modelling second equation for Dissipation:

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right) + \nabla p + C_{1\varepsilon}\frac{\varepsilon}{k} 2\mu_t E_{ij} E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \qquad \text{Eq. 2.8}$$

Where,

u_i represents velocity component in corresponding direction

E_{ij} represents second order tensor with two free indices of rate of deformation

μ_t represents eddy viscosity

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

3.NUMERICAL PROCEDURE

3.1Meshing:

Mesh	Properties	Coarse	Medium	Fine
Mesh Size	Level	0	0	0
	Cells	109507	126767	171072
	Faces	228206	264773	357146
	Nodes	23701	27839	37530
	Partitions	3	3	3
Mesh Quality	Minimum Orthogonal Quality	4.55407e-02	2.51898e-03	9.54600e-02
	Maximum Aspect Ratio	9.54459e-01	9.78206e-01	9.04540e-01
	Maximum Ortho skew	3.36583e+01	4.00042e+02	2.62207e+01

3.2Solver Description:

We use Fluent 17.1 commercial package to capture the multiphase physics of flow inside the coating booth for Geometry and meshing ICEM CFD is used. The coating thickness is captured with the help of prism layer meshing because it gives accurate results for boundary layer physics than hex meshing. On the other hand, the fluid domain will be captured by hexa- meshing.

For gas phase, turbulent modelling a two-equation model i.e. K-epsilon with enhanced wall function model is used and solid phase Discrete phase model is used with continuous phase interaction and particles are injected normal to the surface, spherical drag model and discrete Random walk model is used for physical and turbulent dispersion respectively.

[1]In the Discrete random walk model or “eddy lifetime” model, the interaction of a particle with a succession of discrete stylized Fluid phase turbulent eddies is simulated. Each eddy is characterized by

1. A Gaussian distribute random velocity fluctuations, u', v', w'
2. A time scale

The number of tries that satisfies at least 10,000 particles from inlet for better statistics predictions.

[2]The minimum number of tries = Number of Required Particles/Number of Faces at Inlet

The coupled Scheme is used in Numerical Simulation. While the Turbulent Equations are solved by Single Order Upwind Scheme due to gravity influence and it will create instabilities if we use second order upwind scheme. On the other hand, momentum equations are solved by using second order upwind scheme.

---

[1] Fluent Inc, December 3, 2001, Chapter 19. Discrete Phase Models, PP 19-16.

[2] As described in Lab manual 5 and 6.

3.3Boundary Conditions for the coating booth are given below:

For Gas Phase:
1. Inlet velocity of the Gas = 5 m/s (velocity inlet)
2. outlets = -100 pa (Pressure Outlets
3. All others are walls with no slip boundary conditions

For Solid Phase(DPM):
1. Inlet velocity of the Gas = 5 m/s (velocity inlet), DPM set to be Escape, Particle will escape or injected from inlet.
2. Outlets set be escape
3. Front coating set to be trapped because it will capture particles
4. Bottom wall set be trapped wall
5. All other walls set be reflective type

Parameter	Coarse Mesh	Medium Mesh	Fine Mesh
Number of Faces at Inlet	43	73	84
Number of Tries	240	140	130

3.4Solutions Methods and Controls:

The phase – coupled is applied for the pressure based solver because it is a multiphase physics flow, the phased momentum equations, and the shared pressure and the phase volume fraction equations are highly coupled. The discretization of Turbulent equation that is for k and epsilon set to be on first order upwind scheme because of gravity influence. It will create instability if we set to the higher order.

A surface monitor for area weighted average of coating layer thickness which we have created as custom field variables as given below;

Coating layer thickness = (Accretion rate/1550) * 10^6

A convergence criterion of none for each scaled residual component is applied.

3.5GRID/TIME STEP INDEPENDENT TESTS
From the given below table it is concluded that the percentage difference is less than 1% therefore the numerical simulation can be done with any one of the following grid size without any error in the results.

	Transfer Efficiency	Difference
Coarse	89.95	0.82%<1%
Medium	90.70	
Fine	91.45	0.80%<1%

Table. (1)

4.ANALYSIS OF RESULTS

Numerical Simulation of experimental coating booth has been performed in this problem to calculate the transfer efficiency of particle and coating thickness. The simulation is performed in ANSYS software and transfer efficiency of particle is 92% as calculated by using finer mesh with 0.06 global mesh seed size as shown in Table (2). The transfer efficiency of particle increases with the refinement of the mesh. While, the coating thickness is 228, 247 and 274 computed numerically in coarse, medium and fine mesh as shown in Fig. (3), Fig. (7) and Fig. (10) respectively. The air velocity vectors are computed by using mid plane method and are represented as shown in Fig. (4) and Fig. (8). The Fig. (2) and Fig (6) depicts the particle trajectories in a coating booth as per Eq. (2.6).

Efficiency, η= Trapped mass flow in coating front surface/0.002

The calculated efficiency and coating layer thickness in micron meter as given in Table. (2) and Table (3).

5.DISCUSSION OF RESULTS

It is very difficult to capture the boundary layer thickness but the numerical techniques presented in this problem are looks promising and prism layer shows good results for boundary layer physics. The efficiency shows large deposition of particle over the surface However, there is big difference in density of solid and fluid particle it is recommended to use smaller diameter powder to enhance the particle trajectories.

6.Conclusion:

The Discrete phase model shows great success in defining the particle tracking phenomena. The layer thickness is calculated with the Lagrangian model i.e. material coordinates. DPM is only limited to low volume fraction of particle phase means it is only limited to dilute flows and it cannot be modelled for any other dense flow regimes. Therefore, there is a need of another model which accounts for both dilute flows and dense flows like Eulerian- Eulerian frame work which treated both fluid and particle a continuum hypothesis.

	Coarse Mesh	Medium Mesh	Fine Mesh
Total Mass Flow Rate	0.002	0.002	0.002
Mass Deposited on the plate	1.799e-03	1.814e-03	1.829e-03
Transfer Efficiency (%)	89.95	90.70	91.45

Table. (2)

	Coarse Mesh	Medium Mesh	Fine Mesh
Coating Thickness	2.28e+2	2.47e+2	2.74e+2

Table. (3)

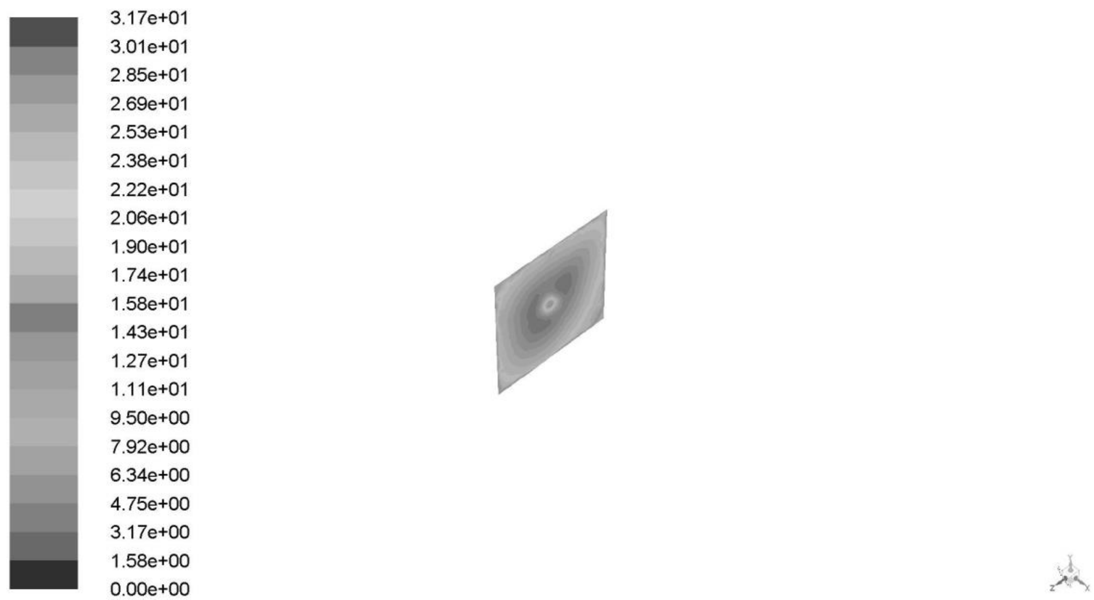


Fig. 1 (Y+ Contour Plot for Air Only Coarse Mesh)

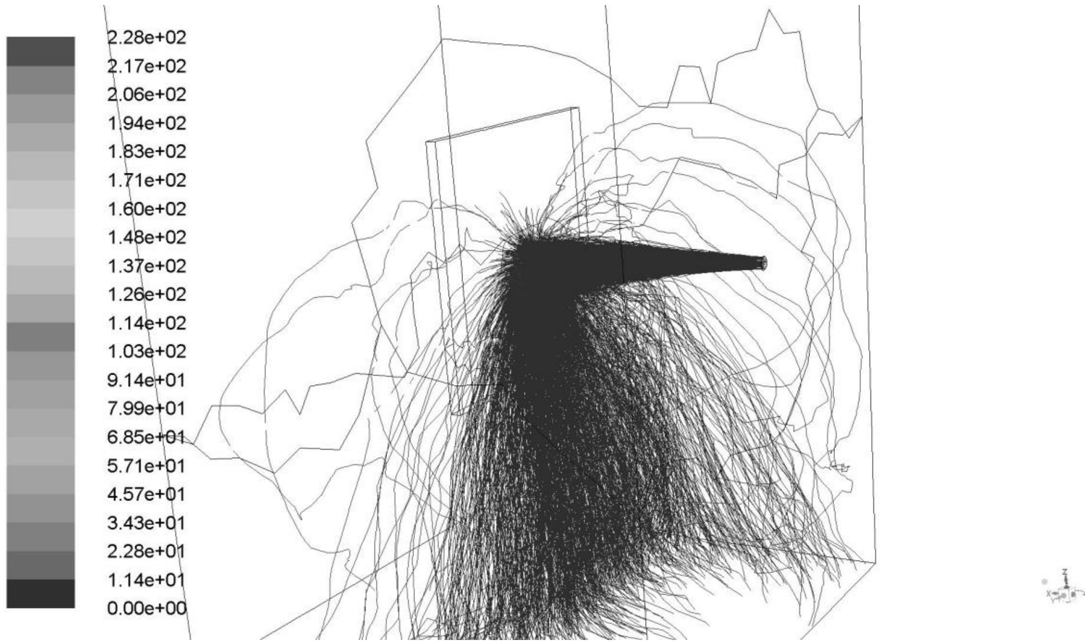


Fig.2 Particle Track by Velocity Magnitude for Coarse Mesh

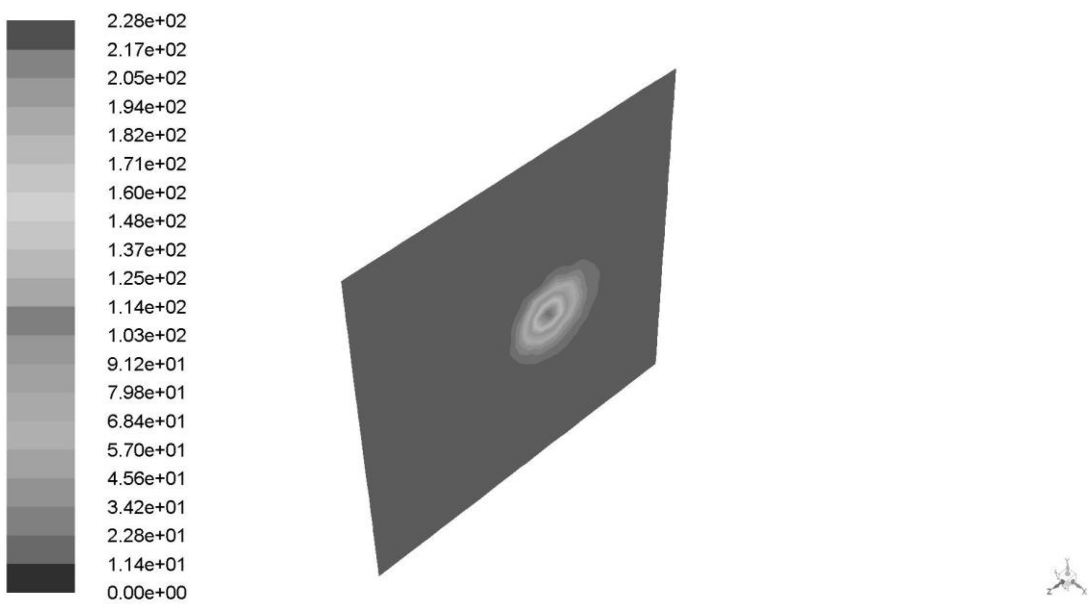


Fig.3 Coating Thickness for Coarse Mesh

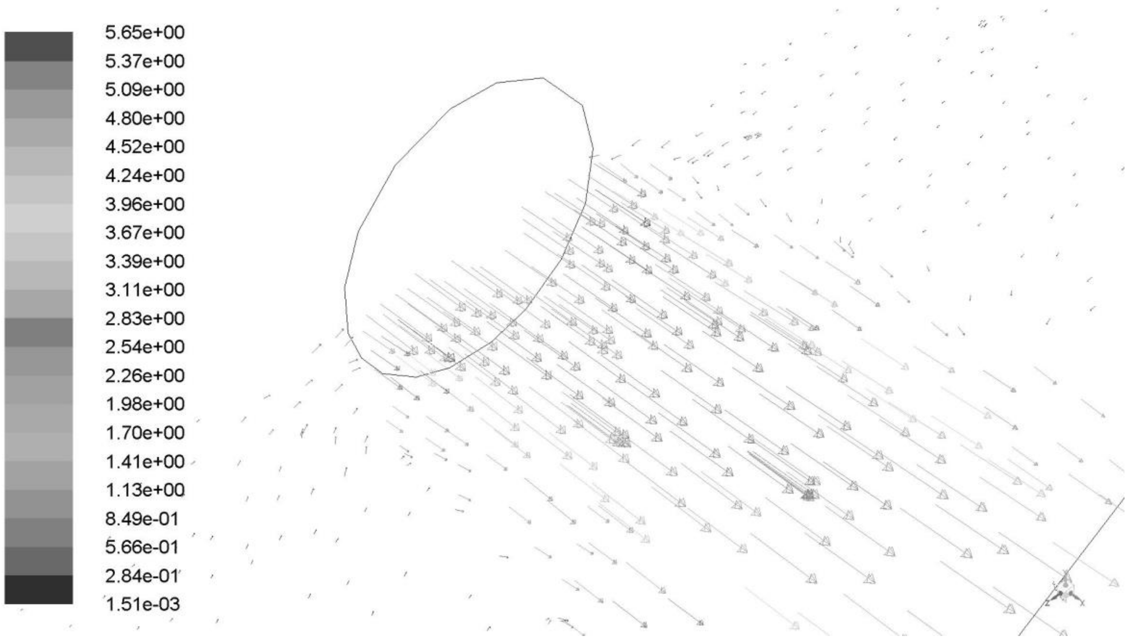


Fig. 4 Air Velocity Vector Field at Midplane passing through Y

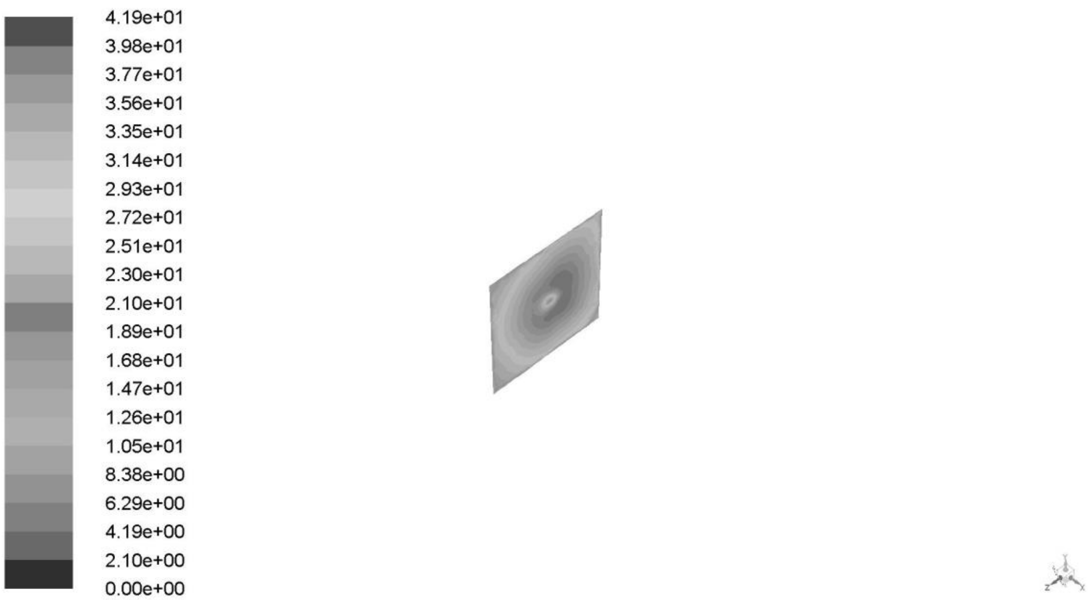


Fig. 5 (Y+ contour for DPM Medium Mesh)

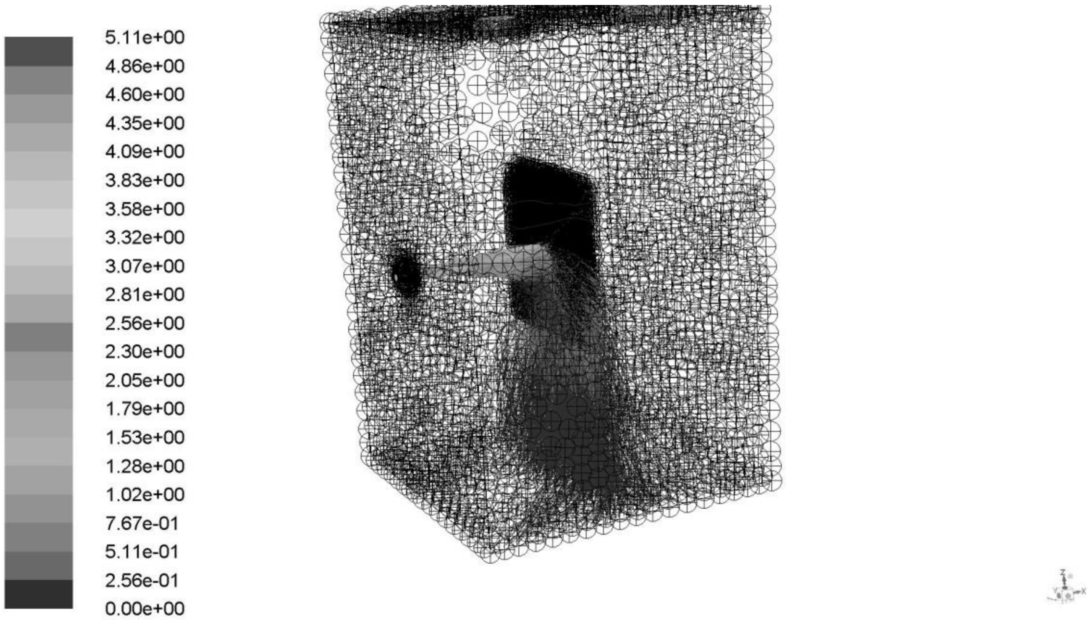


Fig. 6 (Particle Tracking by Velocity Magnitude-Medium Mesh)

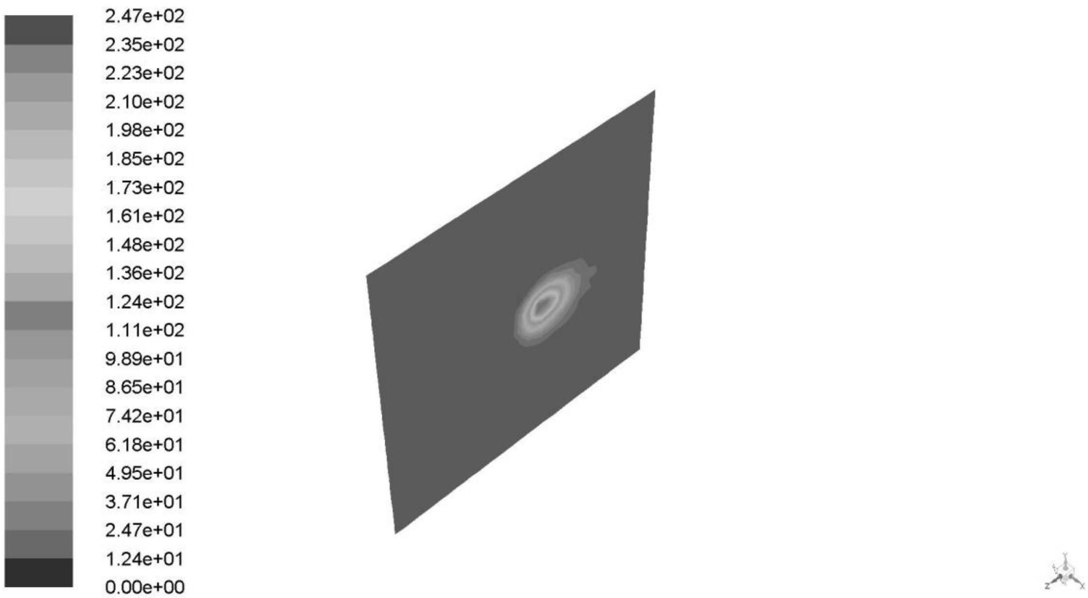


Fig. 7 (Coating Thickness on Front Coating Surface Medium Mesh)

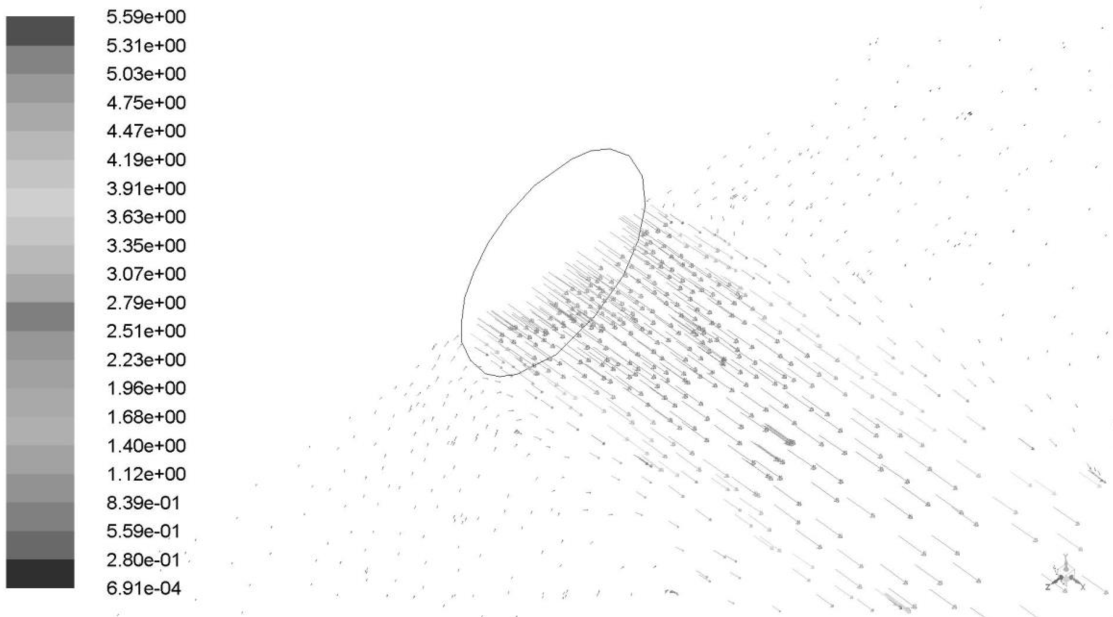


Fig. 8 (Air Velocity Vectors-Medium Mesh)

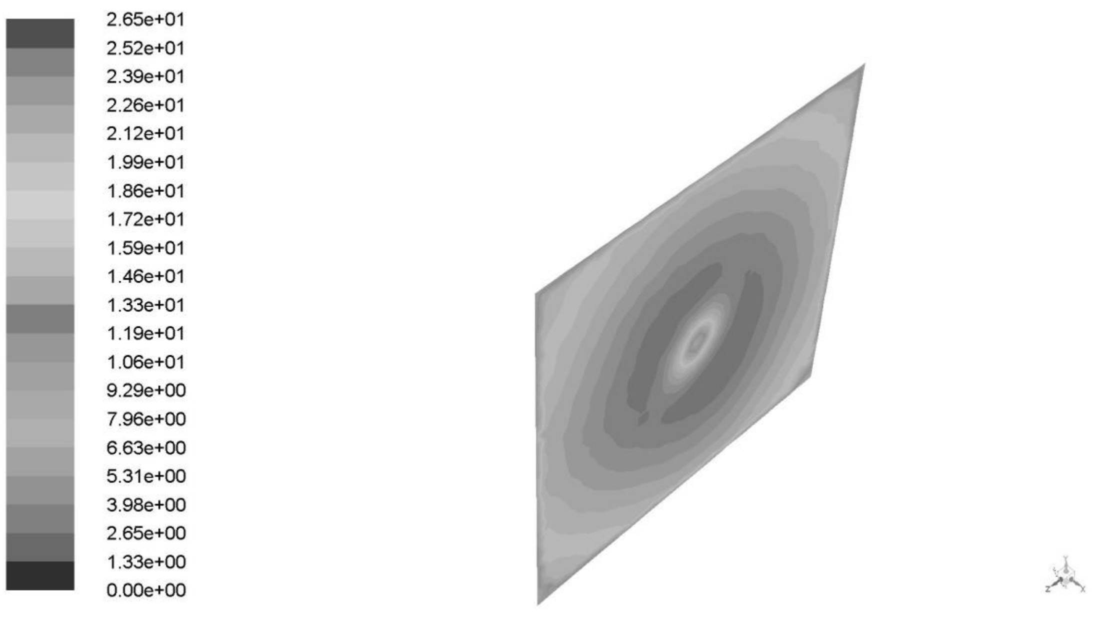


Fig. 9 (Y+ Contour Plot for Air Only Fine Mesh)

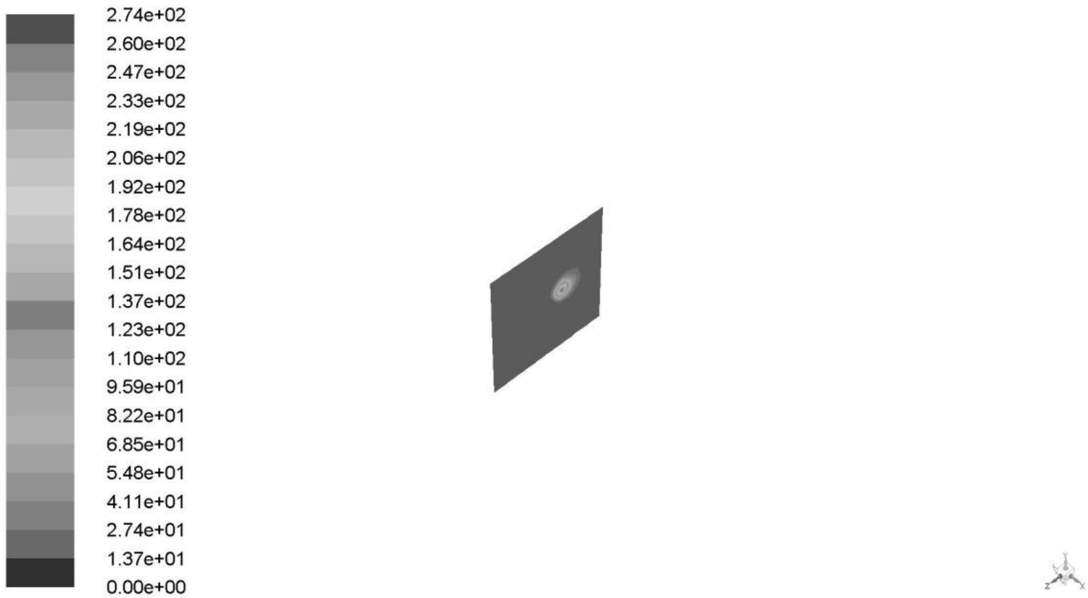


Fig. 10(Coating Thickness Fine Mesh)

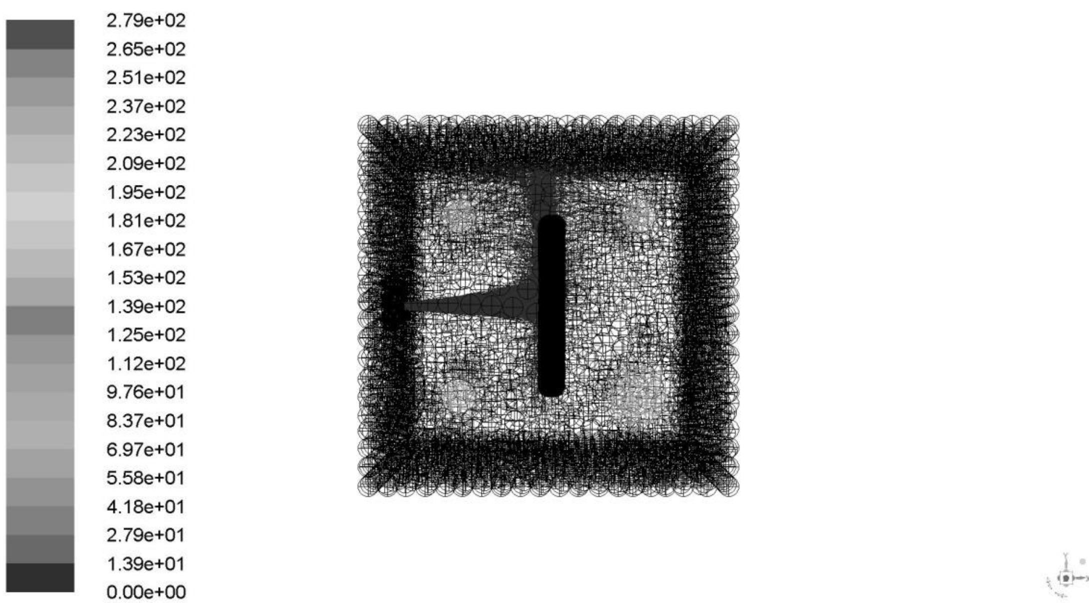


Fig. 11 Particle Trajectories using Fine Mesh

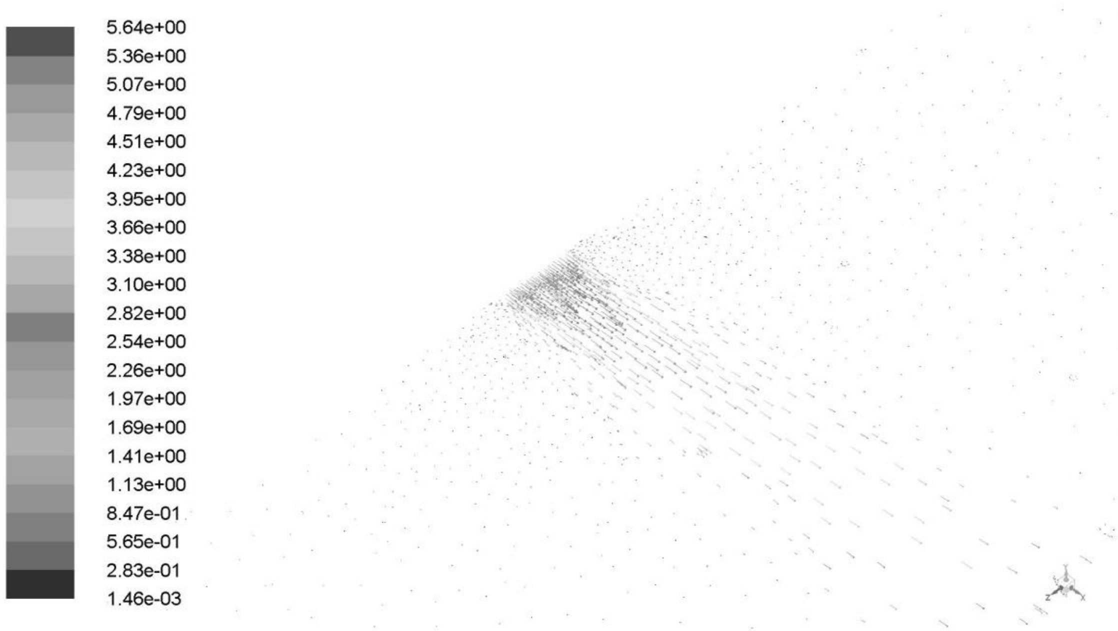


Fig. 12 Air Velocity Vectors Fine Mesh

References:

1. Guan Heng Yeoh, Jiyuan Tu, “Computational Techniques for Multiphase Flows”, PP 243-349.
2. Ansys FLUENT Theory Guide, 2012, Ansys, Inc.
3. Ansys FlUENT User Guide, 2012, ANSYS,Inc.
4. Jiyuan Tu, Guan Heng Yeoh, Chaoqun Liu “Computational Fluid Dynamics” A practical approach.

EVALUATION SHEET

	MARKS	**MAX**
General Organization		5
Quality of Presentation		5
Technical Writing		5
Problem Description		5
Mathematic Models		10

Numerical Procedure		10
Grid/Time Step Independent Tests		10
Analysis of Results		20
Discussion of Results		20
Conclusions		5
References		5
TOTAL		100

Name: ______________________________ Assignment NO. __________