Simulation of air velocity in a vertical perforated air distributor

T N W Ngu1, C M Chu1* and J A Janaun1
Chemical Engineering Programme, Faculty of Engineering, Universiti Malaysia Sabah (UMS), 88400, Sabah, Malaysia

E-mail: chrischu@ums.edu.my

Abstract. Perforated pipes are utilized to divide a fluid flow into several smaller streams. Uniform flow distribution requirement is of great concern in engineering applications because it has significant influence on the performance of fluidic devices. For industrial applications, it is crucial to provide a uniform velocity distribution through orifices. In this research, flow distribution patterns of a closed-end multiple outlet pipe standing vertically for air delivery in the horizontal direction was simulated. Computational Fluid Dynamics (CFD), a tool of research for enhancing and understanding design was used as the simulator and the drawing software SolidWorks was used for geometry setup. The main purpose of this work is to establish the influence of size of orifices, intervals between outlets, and the length of tube in order to attain uniformity of exit flows through a multi outlet perforated tube. However, due to the gravitational effect, the compactness of paddy increases gradually from top to bottom of dryer, uniform flow pattern was aimed for top orifices and larger flow for bottom orifices.

1. Introduction
Perforated pipes are often used in industry for chemical vapor deposition, ventilation system, and bubble column reactor [1]. In Asia, two major types of dryer are widely used by industries for paddy drying purpose: fluidized bed and inclined bed dryer [2]. The moisture content (MC) of paddy rice at harvest around 28-35% dry basis [3]. Paddy should undergo drying to reduce the moisture content to 12-15% to be suitable for storage purpose to prevent production of mycotoxins. However, air is known to be distributed unevenly in commercial dryers. Therefore, it is necessary to minimize the uneven air flow in dryer. Sangkyoo and team [4] discovered that the flow rate distribution between orifices of perforated pipes could be adjusted by varying aspect ratio of orifices. A new conceptual configurations for paddy drying, Laterally Aerated Moving Bed (LAMB) as proposed in Tan [5] with an inner vertical aerated tube was installed. The aerator is a vertical perforated tube which function as to distribute hot air to surrounding. The designed vertical inner tube reduces pressure drop hence power required is lesser than fluidized bed dryer [5]. In brief, LAMB drying will lead to more uniform drying of grain.

Drying is a common process in industrial field. Air flow or air velocity play important roles in drying rate. However, air flow is difficult to measure while process is operating. Therefore, computational fluid dynamics (CFD) was applied to model fluid flow situations.
2. Methodology

Schematic diagram of a LAMB dryer is shown in Figure 1. Air distribution through orifices of inner perforated tube in LAMB dryer was focused on this study by using Phoenics CFD and SolidWorks 2013. Phoenics CFD and SolidWorks 2013 were used to simulate the air flow and build up the geometry that to be simulated respectively. Phoenics is a CFD software package that able to analyze fluid flow by showing the prediction of fluid flow such as air, water and etc. Predictions are based on Navier-Stokes equation. The simulation can be viewed holistically by using color contour. Phoenics solves finite domain by using mass, momentum, energy conservation equations. In brief, CFD analysis come into the picture to replace the tedious and time consuming manual calculation. Besides, SolidWorks 2013 was used to build up a multi outlet pipe. SolidWorks is a three dimensional solid modelling program used to produce computer design models and has to be converted to .stl before import to Phoenics CFD.

![Figure 1. Schematic Diagram of LAMB for Paddy Drying [2]](Image1)

![Figure 2. Geometry imported to Phoenics CFD](Image2)

| Table 1. Specification Sheet of Vertical Perforated Pipe |
|---------------------------------------------------------|
| Perforated pipe length, L (mm)                          |
| Pipe inner diameter, ID (mm)                            |
| Pipe outer diameter, OD (mm)                           |
| Diameter of orifices from top to bottom of vertical perforated pipe, D<sub>i</sub> (mm) |
| At 0° and 180°                                         |
| At 90° and 270°                                        |
| 200                                                    |
| 22.46                                                  |
| 26.68                                                  |
| 4.62, 4.92, 5.35, 6.07, 8.00                           |
| 4.76, 5.11, 5.66, 6.72                                 |

2.1 Assumptions and Geometry Construction

Assumptions were made in the simulation of the flow in a perforated pipe: 1) it was one-dimensional where the main flows were not considered to be modified by secondary flows. If the flow was along the pipe axis, the radial flow would be neglected; and if the flow was a discharge from a perforated hole, only the flow perpendicular to the cross-sectional flow area of the hole was important; 2) the fluid was considered to behave as an ideal gas as the pressure would not exceed 20 bar, the limit of ideal gas behavior, 3) the system was isothermal since the maximum temperature difference would be
30K, with minimum heat loss, and 4) the flow was at steady state with a pump constantly running as fluid mover. The geometry of the flow domain was created in Cartesian coordinates form. The geometry consisted of a straight vertical pipe with 22.46 mm internal diameter and 26.68 mm outer diameter with height of 200 mm. In order to achieve uniform fluid distribution at top orifices, a module of vertical multi outlets perforated pipe was specially designed with various orifices diameter and located at uniform intervals. In this study, 4 orifices were located at uniform longitudinal intervals of 40mm along the pipe at both 90° and 270°, with the orifices at both ends measured at 40mm from either end of the pipe; while 5 orifices were located at 4 intervals of 40mm at both 0° and 180°, where the orifices at both ends were located at 20mm from either end of the pipe.

Specification sheet of a perforated tube is shown in Table 1, and the geometry imported from SolidWorks into the CFD simulation software is shown in Figure 2.

2.2 Simulation Set-Up
Simulations with different mesh/grid size are used in Phoenics CFD. Settings include dimension of geometry, grid size, boundary conditions, and number of iterations. Mesh size with in increment order are used in the simulation. In this study, as far as mesh size is concerned, three types of settings were done, namely coarse, fine, and finest.

Boundary conditions of simulations (Case 1) are shown in Table 2.

| Setting                          | Simulation       |
|---------------------------------|------------------|
| Fed air velocity (m/s)          | Coarse | Fine | Finest |
| 0.10                            | 0.10   | 0.10 | 0.10   |
| 59 (X)                          | 86 (X) | 80 (X)|        |
| 44 (Y)                          | 86 (Y) | 80 (Y)|        |
| 60 (Z)                          | 80 (Z) | 90 (Z)|        |
| Operating temperature (°C)      | 90     |      |        |
| # iterations                    | 1,000  |      |        |

2.3 Mesh Independence Study
For this investigation, 3 simulations were carried out. Three simulations with lesser numeric iterations (1,000) were carried out to analyze the influence of the mesh grid. Three simulations with different mesh grid; coarse, fine and finest mesh. A domain height of 0.2 m and domain length and width of 0.1 m was chosen. In this study, only inlet been set at bottom of perforated tube which function as to introduce hot air in z direction (upward). Contour plot (a 2D color maps that convey information and data) was applied in simulations to show the direction of fluid flow. The purpose of carry out simulations at different grid size was to discuss the meshing requirements for a valid result. The more accurate/fine the mesh, the more accurate the converged solution will be. Convergence is an important identification for all CFD users to ensure a valid solution. In brief, convergence with mesh independent study must be ensured for every simulation or else extra iterations have to be carried out. In this study, error of less than 1 % was set as targeted convergence percentage. Independent of the mesh solution study helps to avoid inaccurate results in CFD. Since the deviation in average value between fine grid and finest grid was only 0.3 %, therefore, simulation named case 2 with grid size of 80 × 80 × 90 (cells in x direction x cells in y direction x cells in z direction) (250 total cells) was simulated with 10,000 iteration. The simulation set up of a perforated tube for Case 1 is shown in Table 3.

From the result in Table 3, convergence was reached for all simulations. The monitored value on error percentage was used to judge whether convergence achieved through the simulation.
Convergence was fully achieved as far as steady state is concerned. As the data regarding monitor error (%) for pressure and velocity, it can be seen that the error percentage drop as mesh/grid size increases.

Table 3. CFD set-up for case 1

| Setting                  | Case 1 | Coarse | Fine | Finest |
|--------------------------|--------|--------|------|--------|
| Fed air velocity (m/s)   |        | 0.10   |      |        |
| Grid resolution (units)  |        | 59 (X) | 80 (X)| 118 (X)|
|                          |        | 44 (Y) | 60 (Y)| 116 (Y)|
|                          |        | 60 (Z) | 80 (Z)| 140 (Z)|
| # iterations             |        | 1,000  |      |        |

| Error (%)                |        |        |      |
|--------------------------|--------|--------|------|
| Pressure                 |        | $1.98 \times 10^{-1}$ | $2.49 \times 10^{-1}$ | $3.04 \times 10^{-1}$ |
| Velocity                 |        | $2.59 \times 10^{-1}$ | $3.12 \times 10^{-1}$ | $2.60 \times 10^{-9}$ |
| Temperature              |        | 0      | $1.73 \times 10^{-3}$ | $5.05 \times 10^{-3}$ |
| Average velocity at probe location of $0.048884 \times 0.048593 \times 0.016250$ | | 0.019110 | 0.020830 | 0.020891 |
| (X location x Y location x Z location) | | | |
| Time required (min)      |        | 145    | 114  | 558    |

2.4 Theoretical Calculations

In this study, the simulated results produced by Phoenics CFD were compared with the theoretical calculated values. The basic principles used by most researchers to model the flow behavior of hot air in the internal pipe were the conservation of mass and the conservation of energy.

Assuming that the density of hot air remains unchanged, the theoretical velocity at the orifices can be calculated by using equation (1) derived from the conservation of mass on a perforated tube as shown schematically in Figure 3 [6]:

$$V_i = \left(1 - \frac{i-1}{n}\right)V_1$$

![Figure 3. Expanded Schematic of Perforated Tube [6]](image)

Due to the loss of mass along the pipe from the inlet, the mass flow hence the velocity would progressively reduce but the conservation of energy ensured that pressure increased. This would predict that discharge rate at orifices further away from the inlet would increase.
3. Results and Discussions

3.1 Steady State Simulations

Velocity profile of a perforated tube for case 1 is shown in Figure 4. From velocity profiles below, air distribution increases from top to bottom of perforated tube. However, flow at the same level of orifice did not distribute evenly for large grid size.

![Figure 4. Velocity Profile for Case 1 a) coarse, b) fine, c) finest](image)

Table 4 was the set up for Case 2, where the mesh size was ‘fine’ from Case 1. The number of iterations was now set at 10,000. Velocity profile of the perforated tube for Case 2 is shown in Figure 5.

| Setting                  | Case 2          |
|--------------------------|-----------------|
| Fed air velocity (m/s)   | 0.10            |
| Grid resolution (units)  | 80 (X)          |
|                          | 80 (Y)          |
|                          | 90 (Z)          |
| # iterations             | 10,000          |
| Error (%)                |                 |
| Pressure                 | 2.23 x 10^1     |
| Velocity                 | 9.10 x 10^1     |
| Temperature              | 4.21 x 10^1     |
| Time required (min)      | 1,705           |
From velocity profiles above, fluid flows at the highest velocity at middle of the multi outlet perforated pipe which represent hot air has higher flow rate at middle of perforated pipe. Besides, uniform flow was achieved for top 3 orifices and the largest flow at bottom orifice.

3.2 Mean Values From Phoenics CFD and Theoretical Values

While the air flow of top orifices seemed to be uniform as shown in Figure 5, it is insufficient to justify whether uniform flow had been achieved by looking at the velocity plot that was simulated. Therefore, 6 data collection planes were installed at inlet and orifices (outlet) of perforated pipe as dummy objects. At these dummy objects mean velocity and mass flow rate were calculated by Phoenics CFD with a user-defined routine feature known as In-Form 13. The mean values show that uniform flow was achieved for top orifices.

As stated in section 2.4, theoretical values for velocity can be calculated by using the equations listed. Both mean values obtained from Phoenics CFD simulations and theoretical values were compared and converted into graphs. Both graphs show the same trend. To conclude, computational analysis was performed on set geometry and the analysis was similar to the theoretical calculated values. The two methods mutually corroborate each other. Mass flow rate and velocity comparisons are shown in Figures 6 and 7 respectively.

![Figure 5. Velocity Profile for Case 2](image)

Figure 5. Velocity Profile for Case 2

![Figure 6. Mass flow rate of simulated result and theoretical value](image)

Figure 6. Mass flow rate of simulated result and theoretical value
4. Conclusion
This research study has been carried out to investigate the performances of multi outlet perforated tube. CFD simulation of hydrodynamics of perforated tube has been carried out at different design configuration. Since geometry of multi-outlet perforated tube was not readily unavailable in Phoenics CFD, Solidworks 2013 was used for the geometry assembly before it could be imported into the CFD analysis software. Mesh independent study was carried out to ensure a reliable and converged result. Besides, fluid flow at different velocity in the same perforated pipe configuration was carried out. The purpose was to ensure that the designed perforated pipe actually distribute uniform fluid flow at top orifices within a range of inlet velocities.

From the designed perforated pipe, the relationship between the orifices diameter and the location of orifice with respect to the inlet as datum level $H$ was deduced by a correlation of the ratios between i-th and i+1-th hole with the ratio of the pipeline flow area to the total orifices area, $\frac{A_p}{\Sigma A_o}$ which was equal to 1.67. The diameters of adjacent orifices were found to follow the ratios relationship $\frac{D_{i+1}}{D_i} = \left( \frac{H_i}{H_{i+1}} \right)^{0.25}$.

5. References
[1] Melanie P-P 2012 Simulation of Perfusion Flow Dynamics for Contrast Enhanced Imaging, Thesis. University of Toronto.
[2] Sarker M S H, Ibrahim M N, Ab. Aziz N, and Mohd Salleh P 2014 Energy Conversion and Management 77 389–395 doi:10.1016/j.enconman.2013.09.038
[3] Wimberly J E 1983 Technical Handbook for the Paddy Rice Postharvest Industry in Developing Countries. Int. Rice Res. Inst.
[4] Sangkyoo L, Namsoo M, and Jeekouen L 2012 26 2751–2758. doi:10.1007/s12206-012-0721-z
[5] Tan KY 2014 Design and Fabrication of Single-Tube Laterally Aerated Moving Bed (LAMB) Dryer for Agricultural Biomass, Final Year Project Thesis, Universiti Malaysia Sabah
[6] Florham P N J 1968 IEC PROCESS Des Dev 7