Influence of elevator moving pattern and velocity on the airflow uniformity for an LCD panel delivery facility

Sheam-Chyun Lin, Bor-Jang Tsai, Cheng-Ju Chang

Abstract—Owing to the increasing LCD panel size, the difficulty on delivering the glass substrate has been enhanced dramatically and become a critical problem in the LCD manufacturing industry. Nowadays, most of panel fabrication factory utilize the fully-automated delivering technology instead of the traditional labor delivery for diminishing the possibility of polluted particles on the LCD board. Thus, this study intends to investigate on maintaining the air quality inside the delivering facility with a moving elevator. Also, special emphasis is focused on the influence on the moving pattern and velocity of the elevator via numerical technique. Firstly, CFD code Fluent is used to execute the transient flow simulation and evaluate the flow patterns inside this delivery equipment. From analyzing the calculated results, it is found that the inferior air is generated mainly by the increasing vortex inside the delivery equipment for an upward-moving elevator. On the contrary, the flow field becomes very smooth without obvious vortex phenomenon, and thus induces a better air quality when the elevator moves downward. However, a better uniform flow field occurred when the elevator is moving upward. In addition, the airflow uniformity is not effectively improved by reducing the elevator velocity and increasing the FFU airflow velocity. It is concluded that the moving pattern of elevator has an essential impact and can be utilized to improve the air quality inside the LCD delivery facility.

Keywords—Delivery Facility, LCD, Transient Simulation, Vortex

Sheam-Chyun Lin is with the Department of Mechanical Engineering National Taiwan University of Science and Technology Taipei, Taiwan. (Phone: 886-2-27333141#6453; fax:886-2-27376460; e-mail: sclynn@mail.ntust.edu.tw)

Bor-Jang Tsai is with Department of Mechanical Engineering Chung Hua University Taipei, Taiwan. (Phone: 886-3-5186478; fax:886-3-5186478; e-mail: bjtsai@chu.edu.tw)

Cheng-Ju Chang is with the Department of Mechanical Engineering National Taiwan University of Science and Technology Taipei, Taiwan. (e-mail: D9603301@mail.ntust.edu.tw)

I. Introduction

Recently, low-pollution process and superior quality product have drawn significant attention due to the increasing demands in semiconductor, biotechnology, and pharmaceutical industry. Due to the extra small size and rigorous purity requirement on the high-performance products, it could result in a vital damage if there is a 0.1 micro-meter pollute particle adhere to its surface during the process. Therefore, many alternatives are proposed to solve those contaminating challenges for reducing the individual damage and increasing product's yield rate. For example, dealing with multiple glass substrates in the same process is used frequently and proven effectively for the situations adopted the NEMI (National Environmental Methods Index) or JEDEC (Joint Electron Device Engineering Council) standard. Nevertheless, installing those substrates requires extra step and thus increases the damage probability.

This scratch problem becomes even worse in manufacturing the LCD panel due to the fragile TFT-LCD glass, which can be scraped because of a slight speed difference between two delivering motors in the transmission belt. Moreover, effectively controlling the cleanness level in the delivering facility turns into a crucial technology for ensuring a satisfactory yield rate. Therefore, maintaining the air quality inside the transport elevator becomes the topic of this research.

In 1996, Hu et al. [1] used an ultrasonic anemometer to measure the velocity distribution in a full-scale clean room equipped with the FFU (fan filter unit) system. The results showed that the non-uniformity and deflection angle of airstream inside a clean bench are 13% and 7.9 degrees, respectively. In 1999, Cheng et al. [2] used a finite-volume CFD code STAR-CD to simulate the flow field of clean room by solving the Reynolds-averaged Navier-Stokes equations. The numerical results demonstrated that the airflow uniformity increases along with the proper allocation of floor porosity or by controlling the distribution of inlet velocity profile.

In 2010, Shih et al. [3] investigated the particles deposition velocity (V_d) onto an upward moving 300mm–diameter wafer in a clean room with a 0.3 m/s downward velocity (V_0). The calculated simulation was performed by means of the dynamic mesh model of FLUENT CFD code. The results indicate that the particle deposition velocity was increased with the increase of wafer moving velocity. Later, Giannoulis [4] examined the airflow around a raising panel via experimental and numerical efforts. Numerical and experimental outcomes correlate well and illustrate a significant difference between the airflows around the panel covered by impermeable and permeable materials. Clearly, the airflow around the elevating panel becomes smoother when the plastic film is replaced by permeable nets.

In 2010, Lambert et al. [5] utilized CFD code to estimate the time-elapsed decay of contaminants within a chamber experiencing a high-Reynolds-number flow. They found that despite different flow rates, the measured contaminant washout took 12~13% longer than the numerically predicted value. Furthermore, deviation between computational and experimental data is as low as 5.32%, which implies that CFD is a useful tool for studying ventilation phenomena.

Kobayashi et al. [5] used experimental and CFD tools to analyze cross-ventilated flow through a single room. Their emphases are (a) to clarify and understand the airflow characteristics on the windward vortex and the leeward wake for various openings; (b) to explore the flow pattern above the ground from pressure measurements; (c) to identify the accuracy of CFD. The results showed that the pressure and vortex dropping decreased with an increased opening size. Recently, Saidi et al. [6] numerically evaluated the effectiveness of ventilation system in a full-scale clean room. The results show that the contaminant source motion and its path have a great influence on the contaminant dispersion through the room.

Based on the literature reviews, it is summarized that the polluted particles have a severe impact on the yield rate and CFD code employed dynamic mesh can provide accurate flow visualization for analyzing the physical phenomena in a cleanroom. Hence, this study utilizes CFD code Fluent [7] together with the dynamic-mesh technique to investigate the transient flow field inside the LCD delivering facility with a moving elevator. The main goals include maintaining the air quality of the facility and understanding the influence on the elevator moving pattern, elevator speed, and FFU discharge air velocity. Also, special emphases are focused on identifying the vortex location and airflow uniformity in the delivering equipment via the numerical flow visualization. Finally, an appropriate parameter setting can be found through the above-mentioned parametric study to ensure the yield rate of LED substrates.

II. Physical Model Description

To save delivering time and prevent panel damage during moving the glass substrate, this study employs a vertical laminar-flow facility to transport the LCD glass substrate into storage cabinets or next station. This vertical laminar-flow delivering facility is built based on standard of mini-environment, and is equipped with six FFUs installed on the top ceiling (see Fig.1) to serve as air source for driving the internal flow. The incoming airstream firstly hits the moving elevator and LCD panel; then partial flow is reflected and expelled to ambient atmosphere through the top exhauster while the rest of airflow passes through the channel between delivering facility and elevator. This bypass airstream divides into several small streams, flowing between glass substrates and generating vortex inside the moving elevator, and finally exhausts from the lower and bottom ventilating holes.

It is worthy to note that transporting glass substrate is accomplished by moving elevator upward and downward, which is the dominant factor for casting the internal flow pattern. The other driving force is the incoming air current which is generated by FFU and expelled through the ventilation openings at the top, lower, and bottom locations. The interaction between the aforementioned phenomena results in an unsteady and complicate flow field, which needs sophisticate CFD software to simulate and analyze. The proper assumptions and boundary conditions is set according to the practical conditions and described in the following subsections.

A. Assumptions

To capture the actual physical phenomena, several appropriate assumptions and boundary conditions are made to simulate the flow field. The following assumptions are enforced in order to simplify the flow complexity

(1) Incompressible flow is assumed because the fluid velocity is quite low for an internal flow;
(2) The flow is treated as Newtonian fluid, the density is constant, and viscosity is isotropic;
(3) Dynamic mesh is applied to simulate the moving patterns of elevator;
(4) the standard k-ε turbulent model is adopted for turbulence calculation;
(5) The influences of radiation heat and floatation terms are neglected;
(6) Spherical particles are assumed.

B. Boundary Conditions

Several boundary conditions are used in this work and are described as follows:

(1) Velocity inlet boundary condition

The vertical laminar-flow delivering facility is a typical mini-environment, which is usually driven out the polluting particles by the airflow. So as to control airflow velocity easily, FFU system is installed on the ceiling to ensure a uniform airflow direction.

(2) Pressure outlet boundary condition

In order to maintain the positive pressure in the clean room, the pressure is set to be 25Pa. Positive pressure represents to control the pressure difference in two spaces.

(3) Dynamic Mesh method

Instead of using relative velocity, this study utilizes dynamic mesh method to simulate moving elevator for an accurate calculation.

III. NUMERICAL SIMULATION

As stated in last section, the physical phenomena caused by the interaction of moving elevator and FFU discharge airflow is very complex and unsteady. Thus, the numerical tool is adopted to simulate and investigate this complicate flow field. This section describes several important numerical models and judging terminologies used in this work.

A. Numerical method and turbulent model

In the numerical algorithms, the second upwind differencing is adopted to solve convection terms and the SIMPLEC rule is used to solve velocity and pressure coupling iterations. Also, this study utilizes the k-ε turbulence model that developed by Launder [8] et al. to solve the Navier-Stokes equations. And the turbulent is computed as a function of turbulence kinetic (k) and turbulence dissipation rate (ε).

B. Dynamic mesh and moving patter of elevator

Dynamic mesh is known for its capability that could automatically change the grid number and shape for fitting with the moving rigid body. Therefore, this investigation employs the dynamic-mesh technology to examine the flow field inside the LCD delivering facility. Also, the moving patterns of elevator are planned as：

Pattern-1:

The elevator moves downward from top to bottom exports at a constant speed, and stays 5 seconds after reaching the lowest position.

Pattern-2:

The elevator moves upward at a constant speed, and stays 5 seconds after reaching the top position.

There are two moving patterns together with three moving speeds 0.08, 0.12, and 0.16 m/s considered in this

ISBN: 978-1-61804-020-6

investigation. In accordance with the different of moving patterns and speeds observed airflow patterns to obtain the best moving pattern and speed.

C. Index of airflow uniformity (λ)

It is essential to maintain high uniformity airflow in the delivering facili1ty with class-1 level. Also, the indication of the uniformity of airflow is defined as

$$\lambda = (1 - \frac{\sigma}{\mathrm{V}}) \times 100\% \qquad (1)$$

where σ is the standard deviation of the velocity distribution, and V is the average velocity of the cross-section area. Clearly, a higher λ value represents that airflow uniformity is better and flow pattern becomes smoother in the delivering facility.

A comprehensive simulation program is arranged to execute the systematic parametric investigation on the airflow uniformity and ability to exclude particles in the delivering facility. These parameters include FFU air velocity, moving pattern and velocity of elevator. Three FFU outlet velocity (0.54, 0.63, 0.71 m/s) and elevator speed (0.08, 0.12, 0.16 m/s) are designated for consideration in this work. Thereafter, the outcome can provide an important reference for building a panel plant. Also, an appropriate combination of the above parameters can be reached for keeping uniformity flow in the delivering facility.

IV. Numerical Results and Discussions

The cleanliness level of this clean room is set to be class 1 and the ceiling-installed FFU system has an outlet area of 6.5*12.5 m^2. CFD simulation indicates that, as illustrated in Fig. 2, FFU discharge air flows in the delivering facility and splits into two streams after directly striking the top of elevator. One stream is reflected and expelled via the top exhausting holes, and the remaining airflow successfully enters the enclosing channel between wall and elevator. This passing-through airflow is the driving force to eject the pollutant form the thin layer between LCD boards, and also is the reason to generate unfavorable vortex and recirculation. The distinctions on flow phenomena induced via different parameter settings are described and discussed in details under various elevator moving patterns.

Firstly, the FFU air velocity and the elevator moving speed are set at 0.63 m/s and 0.16 m/s for comparison under the same criterion. For moving pattern-1, as indicated in Fig. 3, the airflow above the elevator is induced to shift downward with a descending elevator at the beginning (t=1 sec). Also, since most of airflows are blocked by elevator, thus the flow beneath the elevator is quiet smooth except slight recirculation occurs near the bottom exhausting openings. At t=3s, the airflow around the elevator flows smoothly when the elevator locates at the middle point of stroke. Finally, the elevator reaches the lowest position (t=6s), the flow field near the elevator becomes more complex due to the strong interaction between the downward elevator and ventilating holes at the bottom. However, as shown in Table 1, the airflow uniformity recovers to 84.79% quickly after a short stop (5 seconds), which is needed for loading/unloading LCD panels.

As regards the upward pattern-2, the rising elevator compresses the air above it, therefore a low-velocity zone is formed immediately on the top of elevator (see Fig. 4). Besides, a steadily ascendant air movement is observed under the elevator due to this smaller driving force. However, this induced upward force is enhancing as the elevator is moving up. It follows that an extensive recirculation zone appears under the elevator when t=3s. Later, at the end of this moving stroke (t=6s), these compressed and induced airflows above and beneath elevator speed up to maximum strength and quickly expel out through the top ventilating openings. Similar to the patter-1, after a 5-second stop, the airflow uniformity restores to 97.43% (see Table I).

Consequently, after carefully comparing above results, it is obvious that the airflow uniformity and flow field in the moving pattern-2 is superior to that of pattern-1; nevertheless, recirculation phenomenon is identified around the corner of delivering facility in both patterns. This conclusion can serve as an essential reference for arranging the working procedures and layouts of LCD panel factory.

V. RESULTS AND DISCUSSIONS

In this work, a comprehensive CFD investigation incorporated with dynamic mesh is carried out for the parametric study on the airflow uniformity and ability to exclude particles in the delivering facility. As a result, moving pattern of the elevator has an obvious affect on the flow field and airflow uniformity. Also, occurrences of vortex and recirculation influence the general trend of flow field and downgrade the airflow uniformity. Based on the above results and discussions, the following conclusions are obtained.

(1) Regardless of any moving speed of elevator, better airflow uniformity is presented in the delivering facility when the elevator is moving with pattern-2.

(2) Enhancing the discharge velocity of FFU and reducing the speed of moving elevator illustrate insignificant impact on airflow uniformity in the delivering facility.

(3) Since decreasing the elevator speed requires extra time and cost to transport LCD panel, thus it is helpful to increase the elevator speed and FFU air velocity as long as no serious airflow uniformity appears.

In conclusions, an accurate numerical model together with dynamic mesh is established and used for understanding the flow field and air quality inside the delivering facility at different elevator speeds and FFU airflows. Consequently, an appropriate parameter combination is obtained for keeping uniformity flow in the delivering facility. Also, this outcome can serve as an important reference for transporting the LCD panel.

REFERENCE

[1] S. C. Hu, Y. Y. Wu, and C. J. Liu, “Measurements of Air Flow Characteristics in a Full-Scale Clean Room”, Building and Environment, Vol. 31, No. 2, pp. 119-128, 1996.

[2] M. Cheng, G. R. Liu, K. Y. Lam, W. J. Cai, and E. L. Lee, ”Approaches for Improving Airflow Uniformity in Unidirectional Flow Cleanrooms”, Building and Environment, Vol. 34, pp. 275-284, 1999.

[3] A. Mistriotis Giannoulis and D. Briassoulis, “Experimental and Numerical Investigation of the Airflow around a Raised Permeable Panel”, Journal of Wind Engineering and Industrial Aerodynamics, pp. 808-817, 2010.

[4] R. Lambert, C. L. Lin, E. Mardorf, and P. O’Shaughnessy, ”CFD Simulation of Contaminant Decay for High Reynolds Flow in a Controlled Environment”, Behalf of the British Occupational Hygiene Society, Vol. 54, No. 1, pp. 88-99, 2010.

[5] T. Kobayashi, M. Sandberg, H. Kotani, and L. Claesson, “Experimental Investigation and CFD Analysis of Cross-Ventilated Flow through Single Room Detached House Model”, Building and Environment, Vol. 45, pp. 2723-2734, 2010.

[6] M. H. Saidi, B. Sajadi, and G. R. Molaeimanesh, ”The Effect of Source Motion on Contaminant Distribution in the Cleanrooms”, Energy and Buildings, Vol. 43, pp. 966-970, 2011.

[7] Fluent 6.3 documentation. FLUENT INC.

[8] B. E. Launder and D. B. Spalding, Lectures in Mathematical Models of Turbulence. Academic Press, London, England, 1972.

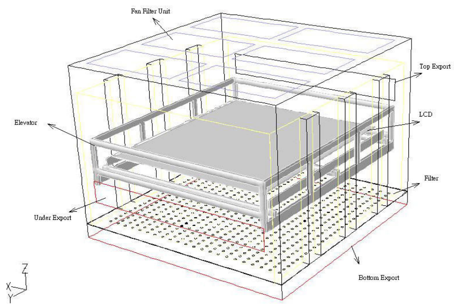


Fig.1 Sketch of delivering facility with a moving elevator.

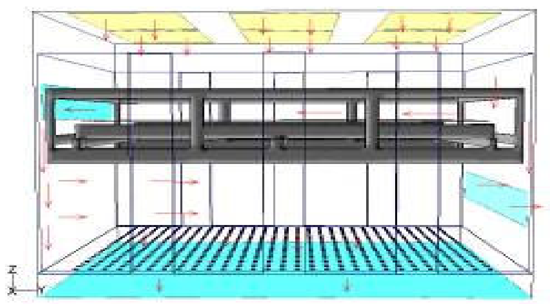
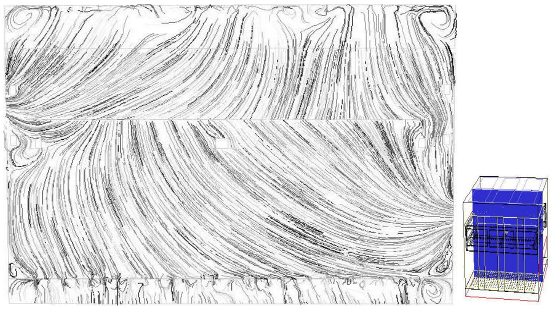
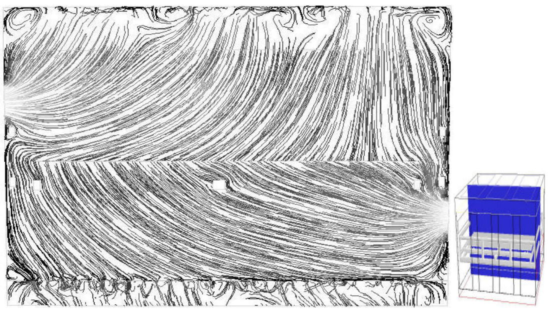


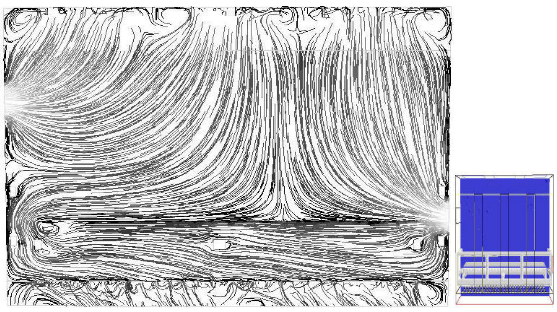
Fig. 2 Flow field inside the LCD delivering facility



(a) t = 1 second

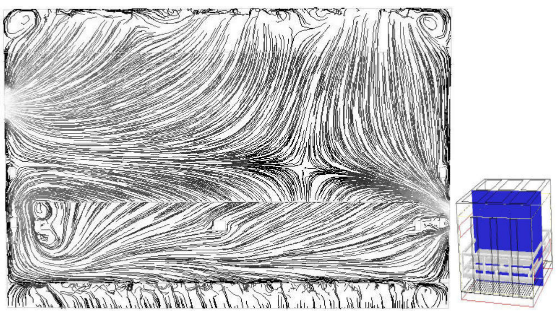


(b) t = 3 seconds

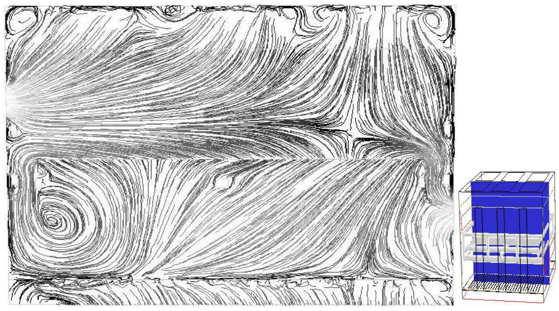


(c) t = 6 seconds

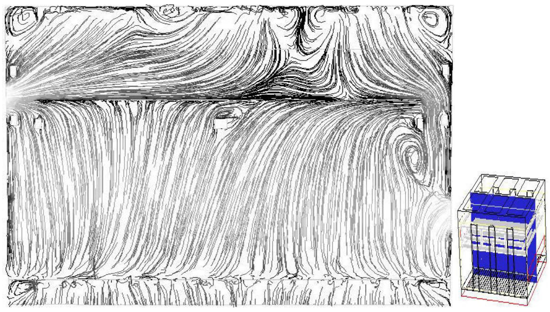
Fig.3 Velocity distribution in the delivering facility (Pattern-1)



(a) t = 1 second



(b) t = 3 seconds



(c) t = 6 seconds

Fig.4 Velocity distribution in the delivering facility (Pattern-2)

Table I Airflow uniformities for different moving speeds of elevator

Moving elevator speed is set 0.16 m/s						
Velocity inlet of FFUs system(m/s)	0.54		0.63		0.71	
Moving pattern	Pattern-1	Pattern-2	Pattern-1	Pattern-2	Pattern-1	Pattern-2
Airflow uniformity (λ)	85.53	96.76	84.79	97.43	84.34	97.55
Velocity inlet of FFUs system is set 0.63 m/s						
Moving speed of elevator (m/s)	0.08		0.12		0.16	
Moving pattern	Pattern-1	Pattern-2	Pattern-1	Pattern-2	Pattern-1	Pattern-2
Airflow uniformity (λ)	82.85	97.43	82.85	97.6	84.79	97.43

Dr. Sheam-Chyun Lin is currently Professor of Mechanical Engineering at National Taiwan University of Science and Technology in Taiwan, Republic of China. Dr. Lin earned his Ph.D. from the School of Aerospace and Mechanical Engineering, University of Oklahoma in 1987, and had his M.S. and B.S. in Mechanical Engineering from Tennessee Technological University and Tatung University in 1983 and 1980, respectively. Professor Lin's research interests cover: (1) Thermal Management on Electronic Devices, such as PC CPU, notebooks, power supply, video projector, LED TV, missile control system, and IC simulator; (2) LED lighting R&D, such as street lamp, motorcycle and automobile LED headlamps, Vehicle Daytime Running Light (DRL); (3) Fan Design and its Applications, such as FFU, small PC cooling fan, vacuum cleaner, Auto ventilation fan, kitchen Range, Left Ventricular Assist Device, and Continuous Positive Airway Pressure (CPAP); (4) CFD Simulation associated with LCD Manufacture Facility, such as LCD cleaning, drying and cooling facilities, LCD panel delivery device, and the clean room; (5) Vertical Axis Wind Turbine (VAWT) R&D, such as the aerodynamic and structural analyses, the novel VAWT design, and the on-site test of a small wind turbine.