Evaluation and Improvement of Ventilation System Inside Low-Cost Automation Line to Reduce Particle Contamination

Lamai Puangburee*, **Wutthikrai Busayaporn****, **Mongkol Kaewbumrung*****, and **Jatuporn Thongsri***†, Non-members

ABSTRACT

A Low-Cost Automation (LCA) line, a group of machines to manufacture hard disk drive's components located inside a clean room of factory, faces the problem of particle contamination caused by an improper ventilation system. To solve this problem, Computational Fluid Dynamics (CFD) has been implemented to evaluate airflow and simulate solutions to improve the ventilation system of the LCA. By using actual operating conditions collected at the factory and Fluent CFD software, the simulation showed that airflow patterns in such areas were substandard. For example, the large areas of recirculation zone with air velocities lower than 0.2 m/s such as Fan Filter Units' conveyor and Work Area. The low velocity of the airflow can cause particle contamination and leads to low-quality production. To reduce the particle contamination, we suggested novel solutions based on the CFD results by increasing the momentum source (S_m) and/or redesigning the LCA's model especially extending its height of cover. The increasing of S_m can be simply implemented by increasing the air-condition power to the optimal values according to the calculation leading to a reduction in recirculation areas. In addition, extending the height of the LCA's cover also improved the air velocities in the critical areas to meet the factory's standard.

Keywords: Airflow, ANSYS Fluent, Computational Fluid Dynamics, Manufacturing Process, Particle Contamination, Ventilation

Manuscript received on August 21, 2019 ; revised on November 28, 2019 ; accepted on December 5, 2019. This paper was recommended by Associate Editor Chanon Warisarn.

*The authors are with the Computer Simulation in Engineering Research Group, College of Advanced Manufacturing Innovation, King Mongkut's Institute of Technology Ladkrabang, Bangkok, 10520, Thailand.

**The author is with the Synchrotron Light Research Institute (Public Organization), Nakhon Ratchasima, 30000, Thailand.

***The author is with the Department of Mechanical Engineering, Faculty of Engineering and Architecture, Rajamangala University of Technology Suvarnabhumi, Phranakhon Si Ayutthaya, 13000, Thailand.

†Corresponding author. E-mail: jatuporn.th@kmitl.ac.th

Digital Object Identifier 10.37936/ecti-eec.2020181.210333

Nomenclature

A	Cross-sectional area that the air flowed through (m^2)
i, j	1, 2 ,3 correspond to the components of x, y and z respectively
E	Internal energy (J)
V	Volume of the element (m^3)
S_m	Source terms of momentum (N/m^3)
ρ	Fluid density (kg/m^3)
h	Height of wall-side cover (cm)
u	Velocity (m/s)
U_i	Mean velocity component in i^{th} direction (m/s)
P_k	Production of turbulent kinetic energy ($kg/m.s^3$)
μ	Molecular dynamics viscosity ($kg/m.s$)
μ_t	Eddy viscosity ($kg/m.s$)
k	Turbulent kinetic energy (m^2/s^2)
x_i	Cartesian coordinate in i^{th} direction (m)
T	Temperature (K)
$\dot{m}$	Mass flow rate (kg/s)
ω	Specific dissipation rate (s^{-1})

1. INTRODUCTION

Thailand is the biggest hard disk drive (HDD) manufacturer, creating income of more than 16,000 million USD each year. Along with the HDD manufacturing, more than 2,000 components have been assembled. Dust and small-scale particles can be contaminated into the production line causing those HDDs to be unable to perform their intended function. Therefore, it is necessary that the manufacturing of HDDs must be completed inside the clean room of a factory that is highly concerned with protecting the production line from particle contamination [1]. Commonly, the factory obeys US FED-STD-209E standard for clean rooms. For example, the standard of the clean room for effective manufacturing of electronics components is "class 100", that is only 100 particles/ft^3 larger than 0.5 μm are permitted. However, a class 100 clean room is expensive to establish and maintain. Therefore, class 1000 clean room is used in practical manufacturing instead which means only 1,000 particles/ft^3 larger than 0.5 μm are permitted [2]. Therefore, to compensate for the lower "cleanliness" standard, most of the factories focus to make only the ventilation system to be in "class 100" with the lowest particle contamination. To evaluate and maintain the level of contamination, one of the effective, low cost and popular methods is using Computational Fluid Dynamics (CFD) to simulate and evaluate the situation.

Simulation of the ventilation system using CFD has been widely applied in many fields of research. Results from the CFD can be useful for clean room design or improvement of the ventilation to gain a higher level of cleanliness and to be in "class 100" that is suitable for the production line. Examples of CFD usages are in Jai-Ngam and Tangchaichit [3], in which CFD was applied to eliminate the contaminated particles in the Head Stack Assembly (HSA) process. The work figured out the suitable air velocity and pressure running through the HSA process to exclude the contaminating particles from the area. The result showed that the suitable air velocity was 2.5 m/s at the pressure of 35 psi. These values will obtain the minimum particle contamination and have no effect to the HSA process. Hwang et al. [4] applied the CFD to simulate the air ventilation to investigate sources of Airborne Molecular Contamination (AMC) in the clean room of semiconductor materials production. This was used instead of installation of a large number of particle sensors which would drastically raise the cost of production. Moreover, particle detection from the sensors also consumed a lot of time for detection and signal processing which would delay the time to solve the problem in the production line. The CFD was accepted to be an effective method to reduce the sources of AMC in this work. Sadrizadeh et al. [5] investigated effects of a mobile laminar airflow screen to bacterial contamination in an operating room using CFD. The results showed that operating with a mobile laminar airflow screen could reduce the number of bacteria in the air. Noh et al. [6] studied the control of contaminations in a small area of clean room to improve the efficiency of production. Concentration of particles and ventilation of airflow were simulated using CFD. The study was completed to investigate the contamination flow in LCD production of a clean room in Korea. The simulation revealed the accumulation of particles caused contamination and proposed solutions by reducing the power of the ventilating fan and also installing additional Fan Filter Units (FFUs). The result showed improvement of the air velocities distribution and reduction of the contaminations. Thongsri et al. [7–8] also used CFD to find a solution for particle contamination in automated machines by investigation of the air speed of Fan Filter Units (FFU) installed above the machines. The work suggested that the proper air speed could reduce the accumulation of particles. The reason was that the proper air speed causes laminar flow of the air. In contrast, improper air speed will create a recirculation zone and vortex flow which leads to a higher level of particle accumulation. Khaokom et al. [9] investigated the airflow in the clean room of an HDD production line to find out solutions of contamination. The CFD results revealed that one of the exhaust fans caused the contamination into the system. The solution from this discovery led to installation of a guide box to cover the exhaust fan to protect the contaminated air and deliver it to the floor level beneath the machines which has no effect to the production. After implementation, the contaminated air in the machine was significantly decreased.

As recently reported, a hard disk drive factory was faced with a problem of particle contamination in the Low-Cost Automation line (LCA) line. Engineers who designed the clean room and controlled the ventilation system of the LCA line anticipated that the problem rooted from improper ventilation system inside the LCA line; therefore, an urgent attempt to solve this problem must be encountered. This article reports our successful solution to solve the problem by using the CFD simulation.

In this article, study and solutions of particle contamination in the LCA line of an HDD factory were reported. The CFD was implemented to simulate airflow of the ventilation system in the LCA line and find out causes of the particle contamination. The simple experiment was employed to validate the CFD results and its outcomes were reported. Finally, the suggested methods to improve efficiency of the ventilation system in this LCA line and maintenance of the standard to achieve air quality were concluded. The most benefit of this work is the proposed solution which has been proved to successfully develop HDD manufacturing and solve actual problems by using the simple methodology with instruments readily available at the factory.

2. THEORETICAL BACKGROUND

2.1 Governing equations

Airflow behaviour in the LCA line, automated machines and clean rooms is considered to be turbulent flow which can be calculated using ANSYS Fluent software [10], the CFD software widely qualified for industrial applications. The airflow can be calculated via solutions of 2$^{\text{nd}}$ derivative equations which contain two parts; conservation and turbulence equations. The conservation equation consists of three equations of mass (1), momentum (2) and energy (3) conservation equations described as;

$$\frac{\partial}{\partial x_j}\left(\rho U_j\right)=0 \tag{1}$$

$$\frac{\partial}{\partial x_j}\left(\rho U_i U_j\right)=-\frac{\partial p'}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\right]+S_m \tag{2}$$

$$\frac{\partial}{\partial x_j}\left(\rho E U_j\right)=\frac{\partial}{\partial x_j}+\left[\mu U_j\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}+\frac{2}{3}\delta_{ij}\frac{\partial U_i}{\partial x_j}\right)\right] \tag{3}$$

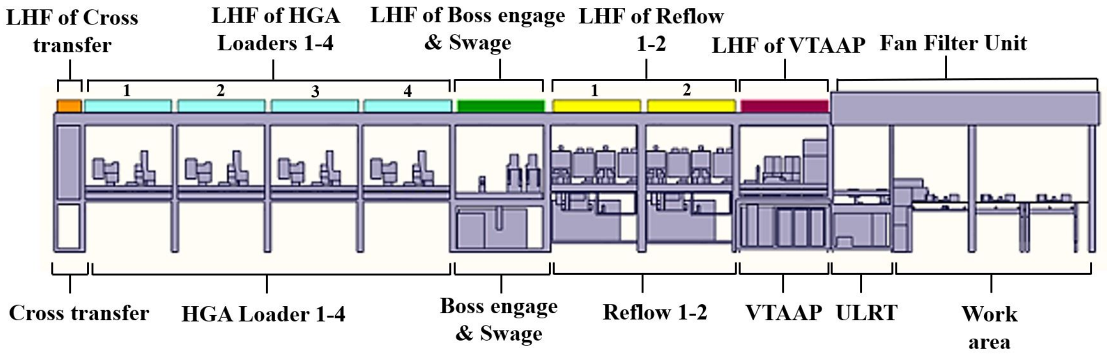

Fig.1: *A solid model of LCA line.*

In this research, the Reynold number is around 40,000 which is turbulence flow. For the turbulent equation, Shear Stress Transport (SST) $k-\omega$ turbulence model developed by Mentor [11] was used. The SST $k-\omega$ model was shown to have higher accuracy than a common $k-\varepsilon$ model. An example of successful usage of the SST $k-\omega$ to simulate the airflow in a small volume jet nebulizer was reported in [12] and aerodynamics of trailing blade in a gas turbine was reported in [13]. The SST $k-\omega$ model consists of two equations as in Eqs. (4) – (5) as [11, 13];

$$\frac{\partial}{\partial x_j}\left(\rho U_j k\right)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_{k3}}\right)\frac{\partial k}{\partial x_j}\right]+P_k-0.09\rho k\omega \tag{4}$$

$$\frac{\partial}{\partial x_j}\left(\rho U_j \omega\right)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_{\omega 3}}\right)\frac{\partial \omega}{\partial x_j}\right] +1.71\left(1-F_1\right)\frac{\rho}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}+\alpha_3\frac{\omega}{k}P_k-\beta_3\rho\omega^2 \tag{5}$$

Full forms and descriptions of related parameters in Eqs. (1)–(5) can be found in [11, 13]. The airflow behavior is derived from the Eqs. (1)–(5). In Fluent software, the fluid model is divided into a number of small elements while each element is related as in Eqs. (1)–(5). According to the Finite Volume Method included boundary conditions, solutions will be calculated and the numerical results displayed via color graphics. The simulation results of the graphical color will be analyzed in Section 4, results and discussion.

2.2 Momentum source

Momentum source (S_m), a key parameter in this article is a user-defined function in Eq. (2) defined in boundary condition. The S_m causes acceleration of the airflow as there is an external source to increase the speed of the air in the defined area. Practically, increasing power of a fan or an air conditioner enhances the S_m and affects the airflow. The S_m is

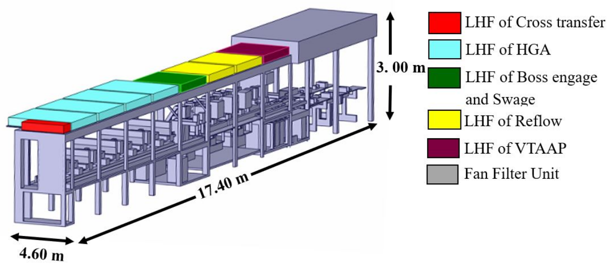

Fig.2: *The LCA line and LFHs.*

commonly defined when the air has been simulated after passing the exhaust fan or Laminar Flow Hood (LFH) in the LCA line. The LFHs can be installed in such places as in Figs. 1 and 2. As for each LFH the S_m are different, the precise S_m defined in the simulation are highly correlated to the precision of result. The S_m can be found out from Eq. (6) as

$$S_m = \frac{\rho A u_j^2}{V} \tag{6}$$

In this work, the S_m was determined from the actual speed of the air released from the LFH in the factory. The S_m for the boundary condition of each machine in the LCA line will be described later in the next section.

3. METHODOLOGY

3.1 Low-Cost Automation line

A Low Cost Automation (LCA) line comprises of 7 types of machine which are 1) cross transfer, 2) HGA Loader 1-4, 3) Boss engage and Swage, 4) Reflow 1-2, 5) VTAAP, 6) ULRT, and 7) Work Area as presented in Fig. 1. In the LCA line, each machine is installed with an overhead LFH whose function is to deliver the air from above and release it into the machines. This design has an exception in only the Work Area where an FFU has been installed. The FFU will directly deliver the air from the air conditioner located outside of the clean room to the Work Area. With this design, airflow inside the machines was claimed to be in class

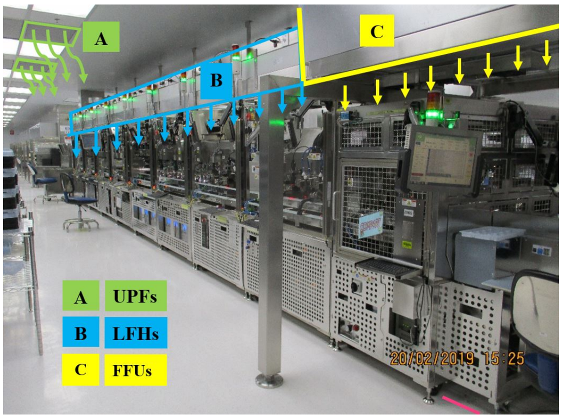

Fig.3: *The actual LCA line and ventilation system.*

100 clean room, while the outside was in class 1,000 [1, 7–9]. Fig. 2 illustrates LCA line with LFHs with different colours representing magnitudes of S_m from Eq. (6). These S_m was derived by using the actual air speed reported from the factory. The accurate S_m for each machine will be reported next in Section 3.4. The mentioned machines are cooperating in the LCA line to produce the HDD's components from 1)–7), sequentially.

3.2 Ventilation system

Fig. 3 presents the actual LCA line in the factory and a summary of the ventilation system. The model of the LCA line was simplified to reduce the computational time and resources. The simplified model was adjusted carefully so it contained only parts that had influence on the ventilation system in the critical areas. From Fig. 3, the Ulpa Filters (UPFs) notified as "A" were installed on the ceiling of the clean room to deliver the air from air conditioners located outside into the large clean room, totally covering the LCA line. The air was filtered through the LFHs denoted by "B" which were situated above the machines to eliminate the particles occurring from the production line and shields the particles from outside the clean room as reported in [1, 7–9]. After that, the air was guided to flow through the machines and released aside. Inside the machines, air cleanliness was expected to be equal to clean room class 100. For the area inside the Work Area denoted by "C", the air must be less contaminated compared to other areas. Therefore, instead of using UPFs and LFHs, FFUs would be operated directly to acquire highly clean air from outside the LCA line and release it into the Work Area. Arrows in Fig. 3 represent the directions of the airflow. There were reports from technicians indicating that airflow inside the LCA line was in class 1000 instead of class 100. Therefore, this is a topic and challenge of this research.

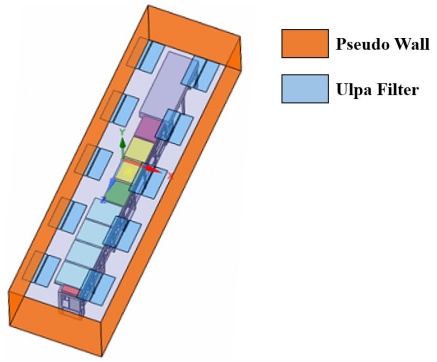

Fig.4: *Fluid model and layouts of Pseudo Wall and UPFs.*

3.3 3D fluid and mesh models

To simulate the airflow based on CFD, the solid model and layout of clean room from the factory's data were adopted to create a fluid model as shown in Fig. 4. Sizes and dimensional parameters of UPFs were input from the layout of the clean room. As the LCA line in this problem is just a part of the large clean room, other production lines will not be taken into the simulation. Connecting areas between the LCA line and other production lines will be defined by Pseudo Wall. By the Pseudo Wall, the air can freely flow to other areas based on boundary conditions. These mentioned methods lead to less complicity of calculation and reduce the resource of computation as it is not necessary to take all other areas into the simulation. Similar methods have been successfully used as in [1, 9] to simulate air ventilation to solve the problem of water condensation. The fluid model as in Fig. 4 was created to be the mesh model. After that, independent mesh analysis was carried out. Results showed that the 3D mesh model with 4,995,215 nodes of tetrahedron at 26,712,873 elements were suitable numbers to provide enough scale of elements representing the area in the machines. The 3D mesh model is demonstrated as in Fig. 5 as the side view of the production line.

3.4 Boundary conditions and Fluent setting

From Subsections 3.1 and 3.2, UPFs including FFUs are inlets and Pseudo Wall is an outlet of the ventilation system. The LFHs operation in the LCA line are the momentum source (S_m) of the air ventilation. With measurable parameters taken into the simulation, the boundary conditions, momentum sources and other parameters in Fluent settings can be defined as in Tables 1–2.

For ANSYS Fluent release 19.1 settings, the air was defined with density at 1.225 kg/m^3 with incompressible flow. Pressure-velocity was set to coupling mode. Furthermore, momentum, turbulence kinetic energy and turbulence dissipation were assigned to be the second order upwind. All of the calculation

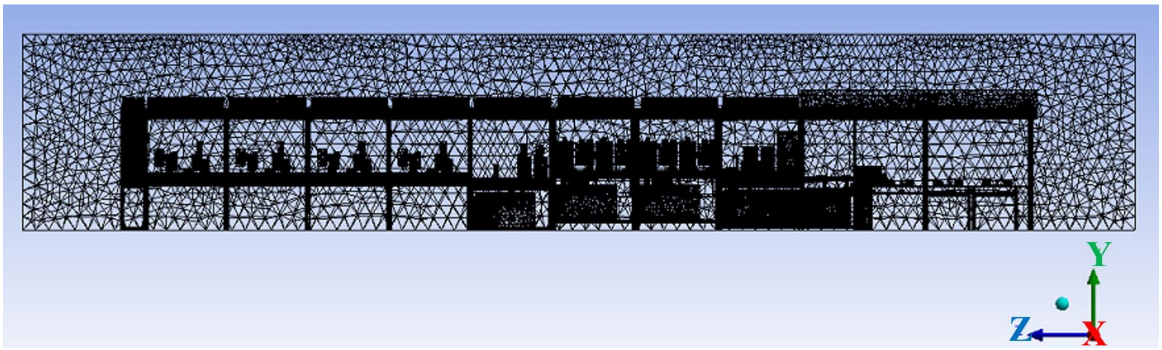

Fig.5: *Mesh model for simulation.*

Table 1: *Boundary conditions.*

Name	Number	Size (m^2)	Type	Value (m/s)
UPFs	20	0.65 × 0.25	velocity inlet	0.55
FFUs (ULRT)	1	1.40 × 1.80	velocity inlet	0.51
FFUs (Work Area)	1	3.00 × 17.40	velocity inlet	0.51
Pseudo Walls	1	3.00 × 4.60	pressure outlet	0

Table 2: *Momentum sources in the LFHs.*

Name	Number	Size (m^2)	S_m*
Cross transfer	1	1.27 × 0.30	2.57
HGA Loader 1-4	1	1.27 × 1.72	2.04
Boss Engage and Swage	1	1.27 × 1.72	1.78
Reflow 1-2	1	1.27 × 1.72	1.74
VTAAP	1	1.27 × 1.72	2.21

*Obtained from the actual measurement and calculation using Eq. (6)

was carried out as in steady state which applied SST $k-\omega$ turbulence model into the calculation. Convergent criterion was fixed at 10^{-4}. The calculation was limited to 1,000 iterations before exporting the result of the simulation.

4. RESULTS AND DISCUSSION

4.1 Validation

To validate the results of the simulation, airflow directions and air velocities were examined. For the airflow directions, 11 planes at the middle of the machines were established to be the areas of observation as in Fig. 6. However, in this article, only two planes as critical areas at HGA Loader3 and Work Area will be presented. Generally, the validated method of airflow direction that is accepted as accurate validation is a smoke visualization test. Unfortunately, steam or dry ice vapors from the smoke visulization might condense on the conveyor and ruin the HDD's production; therefore, the mentioned methods are not suitable for the factory. To avoid this problem, an alternative validation was proposed by using a special thread which has small diameter and light weight. This method was effectively employed to test the airflow direction in the clean room in [1, 8–9]. Fig. 7 shows comparison of the airflow direction between the simulation and the actual measurement. In Fig. 7 centre, the velocity vectors of the airflow at the HGA Loader3 plane was presented using CFD. Figs. 7 left and right represent investigation of the airflow by using the special thread method. As shown in Fig. 7, the thread flowed outward to the left-and right-hand sides of the machines, respectively.

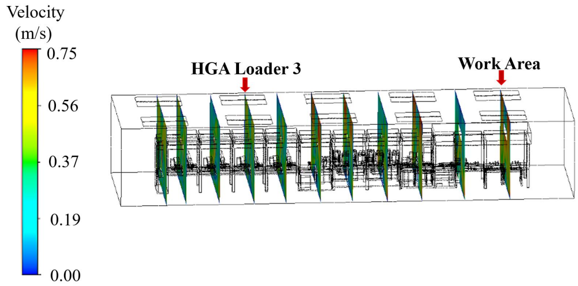

Fig.6: *Section planes for investigation of velocity vectors.*

The flows of these threads are corresponding to the velocity vectors in the marked areas of the simulation. This result exhibits consistency of the airflow direction between the CFD's simulation and the actual measurement. Noted that to emphasize the visibility of the small threads in Fig. 7, red solid lines were graphically amplified and included into the pictures. Fig. 7 centre also shows that the airflow passing FFUs and LFHs were consistent to the principle of ventilation system mentioned in Section 3.2. The air velocities on the left-hand side of the machine was faster than the right-hand ones because this area was closer to the UPFs. The air velocities released from UPFs was equal to 0.55 m/s then passing through LFHs at HGA Loader3 with higher speed at 0.75 m/s. For

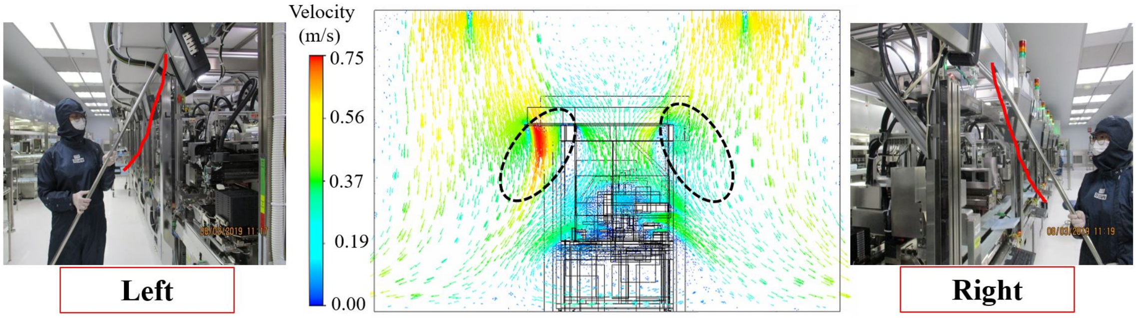

Fig.7: *Validation of airflow direction using lightweight sewing thread in HGA loader3's plane.*

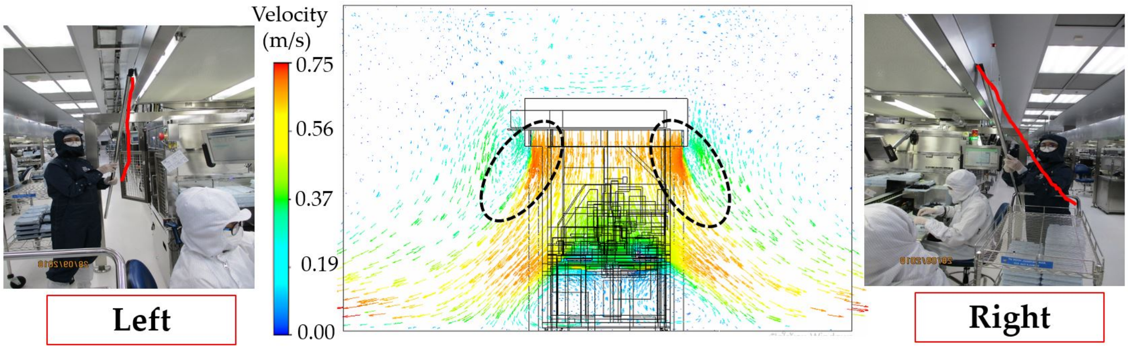

Fig.8: *Validation of airflow direction using lightweight sewing thread in Work Area.*

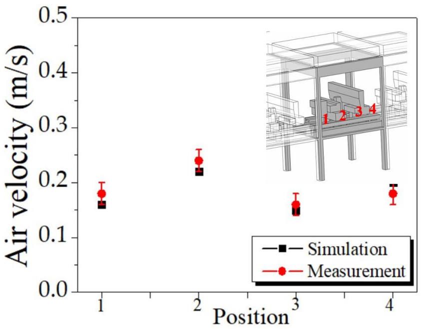

Fig.9: *Comparison between the simulated and measured air velocities in 4 positions at HGA Loader3's conveyor.*

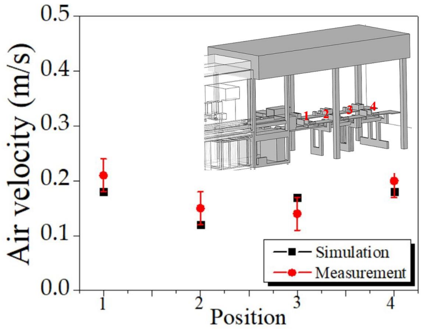

Fig.10: *Comparison between the simulated and measured air velocities in 4 positions at Work Area's conveyor.*

further understanding, readers should consider the actual production line in Fig. 3 alongside with Fig. 7. Likewise, in Fig. 8, the plane of Work Area acquires high air velocity flow because operators who work in this area are the main source of particle contamination. Figs. 8 left and right show the special thread directions representing the wind of the air corresponding to the directions of velocity vectors from the simulation in the marked areas. In this picture, the air ventilation released from FFU has speed of 0.51 m/s. The air with higher velocity passed into the machines and was released to the sides. The other 9 planes in the simulation gave consistent trends with the planes presented. Figs. 9 and 10 show the comparison of air velocities between the results from simulation and the actual data from the conveyor collected at HGA loader3 and the Work Area, respectively.

In the experiment, the air velocities were measured in positions 1–4, 20 times at each position as shown in the inset with a VelociCalc®anemometer that has accuracy of ±0.015 m/s. In Fig. 9, the maximum discrepancy between the results from the simulation and the actual measurement was not over 13% at the position 1. Also, in Fig. 10, the maximum discrepancy was lower than 19% at the position 2. The discrepancy might have occurred because we used a single average value of 0.55 m/s as velocity inlet for the air velocity coming from UPFs (0.51 m/s for FFUs)

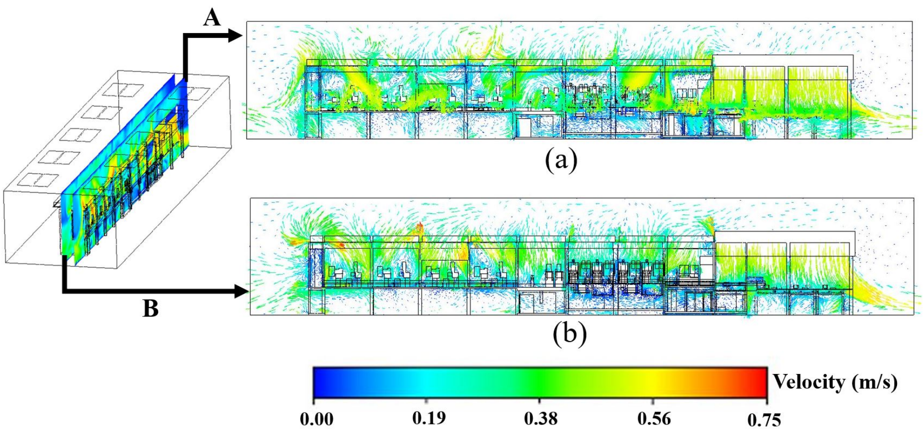

Fig.11: *Airflow pattern inside the LCA line for (a) plane A and (b) plane B.*

in the simulation instead of the actual unsteady air velocity which was in a range of 0.47–0.58 m/s. However, all of the simulation results were within the error bars, indicating that the simulation was consistent with the measurement. The positions 1–4 at the conveyors were chosen to be in consideration because the products were placed here. The air at these areas must be considered to be clean and had the minimum velocity at least 0.2 m/s. All simulated result from Figs. 7–10 were consistent to the measured data; therefore, the results from the simulation are shown to be reliable, credible and precise results.

4.2 Evaluation and improvement

Fig. 11 shows plots of velocities distribution representing the airflow vectors inside the LCA line for (a) plane A and (b) plane B both perpendicular to the floor and situated along the LCA line. Both planes were placed at the critical areas with plane A cut at the center of the average LCA line while plane B was cut at the center of conveyor. The results of the airflow agreed well with the discussion of the velocity vectors explained in Figs. 7 and 8, and show that the air flowed down from the FFUs over the LCA line, moved into the machines, and then out through the outlet at all sides. The air flowed continuously in all areas of the LCA line. In the beginning and end of the LCA line, the air would flow down and out of the domain with higher velocity than the air that flowed from above. With this ventilation system, if there were contaminated particles in the manufacturing process, the proper air velocity would push them below and released out of the LCA line. All mentioned parameters above were consistent with the conceptual design of the LCA line reported by engineers of the factory to effectively eliminate particles by the ventilation system as mentioned in [1, 9]. As shown in Figs. 9 and 10, air velocities at 3 cm above conveyors of HGA Loader3 and Work Area were 0.15–0.24 m/s. In positions 1, 3, 4 of HGA loader 3's conveyor and positions 2, 3 of Work Area, they were lower than the factory's standard value at 0.20 m/s. These low-air velocities might cause accumulated particles at the conveyor which correspond to the understandard products.

To improve the air velocities, we redesigned the LCA line by increasing the S_m in HGA Loader3 and extending the height of wall-side cover (h) of both HGA Loader3 and Work Area. An idea to redesign these areas by extending h was referenced from the work by Thongsri et al. [1, 7–9] that reported that the air from FFUs will flow through the side of the machines more than moving straight to the bottom part of the production line.

Therefore, to solve the problem of low air velocities, the h between the FFUs and the top of the machines and also S_m must be increased. Incrementing both S_m and h will amplifly the speed, and then force a larger amount of air flowing into the machines. Fig. 12 shows solid models before and after redesigning for (a) HGA Loader3 and (b) Work Area. For HGA Loader3, the h was extended from an original size of 16.23 cm to be 46.23 cm. Similarly, for the Work Area, h was extended from 39.20 cm to be 59.20 cm. The simulation was further carried out by using the latter models.

The CFD results confirmed that increasing S_m and extending h can increase the air velocities to the acceptable standard values as presented in Table 3. In the problematic factory, increasing S_m in HGA Loader3 from 2.04 N/m^3 to 2.10 N/m^3 was equal to increasing around 5% of air conditioners' power. From our liturature review, no previous research has used the solutions to improve ventilation system by increasing S_m and h as mentioned in this article; therefore, this idea is simple, effective and novel. From Table 3, increasing of S_m and h enhanced air velocity above the HGA Loader3's conveyor and Work Area around 15.79%–75.00% which maintained the air velocity to be higher than 0.20 m/s (factory's standard). Fig. 13 presents planes of the airflow (a) before and (b) after redesigning of HGA

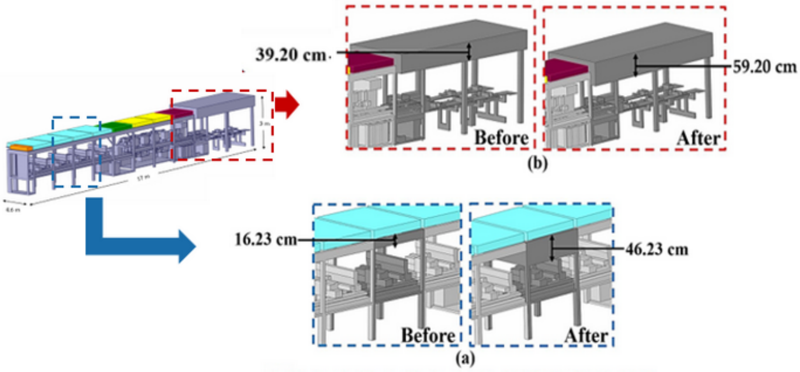

Fig.12: *Solid model of LCA line before and after redesigning for (a) HGA Loader3 and (b) Work Area.*

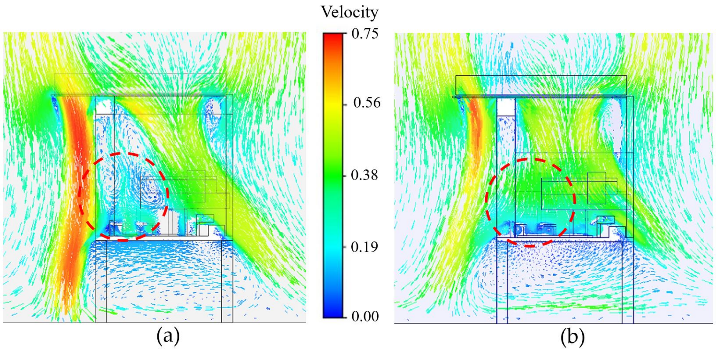

Fig.13: *Airflow in critical areas of HGA Loader3 (a) before and (b) after redesigning.*

Table 3: *Simulated air velocities before and after improvement by increasing S_m and extending h.*

Position	Air velocity (m/s)		
	Before	After	Increment
HGA Loader 3			
Position 1	0.19	0.25	31.58%
Position 2	0.16	0.28	75.00%
Position 3	0.17	0.29	70.59%
Position 4	0.16	0.24	50.00%
Work Area			
Position 1	0.18	0.24	33.33%
Position 2	0.18	0.25	38.89%
Position 3	0.19	0.22	15.79%
Position 4	0.14	0.24	71.43%

Loader3. After redesigning, the recirculation areas (dashed line) were decreased while the air velocities were increased. This phenomenon occurred in several planes of simulation; however, only one plane was presented here because of a limited space of the article. These changes of the recirculation areas and the air velocity improved the quality of ventilation system to be suitable for the HDD's component manufacturing. These parameters were applied into the actual factory setting and reported to be a significantly effective method to reduce the particle contamination.

For further discussion, the increasing of S_m by increasing the air conditioner power will lead to higher electricity consumption. However, the increasing of h is practical for the machines that require lower routine of maintenance as the cover would obstruct the way of the engineers getting into the site of maintenance. Therefore, to effectively improve the ventilation system, an optimization of both S_m and h simultaneously is the suitable way to redesign the LCA line to maximize efficiency of production and minimize operational cost as previously reported in [1, 7–9]. In this research, we found that increasing 0.1 N/m^3 of S_m and 30 cm of h for HGA Loader3 and extending 20 cm of h for Work Area were the optimal conditions to effectively improve the ventilation system for the LCA line. In addition, this work was established by CFD calculation with only steady state of air velocities to evaluate and improve the ventilation system. This method is convenient and uses less time of calculation while the result exhibits the satisfactory level. However, for further and precise results of simulation, particle traces calculation should be integrated with the airflow simulation by using Discrete Phase Model (DPM). The DPM will be utilized to predict traces of the particles and find a method to sustainably eliminate them as reported in [12–14]. Therefore, a combination technique of CFD and DPM for the LCA line modelling is promising work in the future.

5. CONCLUSION

This article reports evaluation and improvement of the ventilation system in LCA line for the HDD factory. The particle contamination in the production line may lead to low-quality products. This work proposed the methods to develop the efficiency of the ventilation system and LCA line to reduce the particle contamination. By using CFD, the results showed that the airflow patterns inside 7 areas in the LCA line are all connected in the large clean room. The contaminated areas are in the HGA loader3 and Work Area. All of the CFD results were consistent with the actual measurements conducted at the factory. This consistency provides confidence in the CFD result and the research methodology. The result also leads to the evaluation of the ventilation system. The two specific areas, HGA loader3 and Work Area, exhibited the recirculation zone with the air velocities lower than 0.2 m/s which is substandard for the factory. The under-standard velocities lead to particle contamination in the production line. To solve the problems, a new method for improvement of the ventilation system was proposed by increasing the momentum sources and redesigning the height of the wall-side cover of FFUs. The CFD results showed that these two methods would reduce the recirculation zone and increase the air velocities about 15.79%–75.00% to be consistent with the factory's standard. The methods were applied to the factory's modifications and have been approved to be effective ways to reduce the particle contamination in the actual LCA line. Finally, the solutions and the methodology proposed in this article were reported as it could reduce the cost of defect products and the cost of LCA lines maintenance more than 0.2 million USD/year.

ACKNOWLEDGEMENT

This research was supported by College of Advanced Manufacturing Innovation, King Mongkut's Institute of Technology Ladkrabang, and Seagate Technology (Thailand) Ltd. The fund was sponsored by Research and Researchers for Industry (RRI), grant number MSD60I0125.

References

[1] J. Thongsri, "A successful CFD-based solution to a water condensation problem in a hard disk drive factory," *IEEE Access*, vol. 5, pp. 10795–10804, 2017.

[2] ISO No. 14644-1, *Cleanrooms and associated controlled environments-Part 1: Classification of Air Cleanliness*, 1999.

[3] N. Jai-Ngam and K. Tangchaichit, "Simulation of airflow inside a computer hard disk drive to develop an impinging air jet particle detachment system for cleaning head stack assemblies," *IEEE Trans. on Magn.*, vol. 54, no. 6, Article 7401808, 2018.

[4] J. J. J. Hwang, K. Chou, C. M. Yang, J. Lin, et al. "Innovative approach to identify location of AMC source in cleanroom by inverse computational fluid dynamics modeling," in *Proc. 23rd Annual SEMI Advanced Manufacturing Conference (ASMC)*, Saratoga Springs, New York, USA, 15–17 May 2012.

[5] S. Sadrizadeh and S. Holmberg, "Effect of a portable ultra-clean exponential airflow unit on the particle distribution in an operating room," *Particuology*, vol. 18, pp. 170–178, 2015.

[6] K.-C. Noh, H.-S. Kim, and M.-D. Oh, "Study on contamination control in a minienvironment inside clean room for yield enhancement based on particle concentration measurement and airflow CFD simulation," *Building and Environment*, vol. 45, pp. 825–831, 2010.

[7] J. Thongsri and M. Pimsarn, "Optimum airflow to reduce particle contamination inside welding automation machine of hard disk drive production line," *Int. J. Precis. Eng. Manu.*, vol. 16, no. 3, pp. 509–515, 2015.

[8] J. Thongsri, "A problem of particulate contamination in an automated assembly machine successfully solved by CFD and simple experiment," *Math. Prob. Eng.*, vol. 2017, Article 6859852, 9 pages, 2017.

[9] A. Khaokom, J. Thongsri and P. Kaewkhaw, "A CFD investigation of airflow in a hard disk drive production line to detect cause(s) of contamination and its mitigation," in *Proc. 3rd International Conference on Engineering Technologies and Social Sciences*, Bangkok, Thailand, 7–8 August 2017.

[10] Ansys Inc., *Ansys Fluent 17.1, User's guide*, Ch. 4, 2016.

[11] F. R. Mentor, "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA Journal*, vol. 32, pp. 1598–1605, 2007.

[12] S. Santati, J. Thongsri and P. Sarntima, "Modified small volume jet nebulizer based on CFD simulation and its clinical outcomes in small asthmatic children", *J. Healthc. Eng.*, vol. 2019, Article 2524583, 2019.

[13] M. Kaewbumrung, W. Tangsopa and J. Thongsri, "Investigation of the trailing edge modification effect on compressor blade aerodynamics using SST $k-\omega$ turbulence model," *Aerospace*, vol. 6, 48, 2019.

Lamai Puangburee received B.Sc. in Chemistry from Department of Chemistry, Faculty of Science and Technology, Thammasat University, Thailand, in 2015. She is currently studying for M.Eng. in Advance Manufacturing System Engineering program at College of Advanced Manufacturing Innovation, King Mongkut's Institute of Technology Ladkrabang, Thailand, since 2017. Her research interest includes Computational fluid dynamics, particle contamination, and cleaning process.

Wutthikrai Busayaporn received his PhD. in Materials from The University of Manchester, UK, in 2010. He has expertise in first principle calculation and simulation of surface structure determination. Currently, he is a beamline scientist at Synchrotron Light Research Institute (SLRI), Thailand and head of Computational Materials Physics (CMP) project at SLRI.

Mongkol Kaewbumrung received his Ph.D. in Mathematics from Mahidol University, Thailand, in 2018. He has expertise in Finite Element Analysis (FEM), Computational Fluid Dynamics (CFD), and Archaeological Innovation. Currently, he is a lecturer and researcher at Department of Mechanical Engineering, Rajamangala University of Technology Suvarnabhumi, Thailand. Also, he is a consultant of several simulation software, who has a lot of experience in applying his expertise to develop the research & innovation for industries.

Jatuporn Thongsri received his B.Sc. in Physics from Khon Kaen University, Thailand, in 2002, and the M.Sc. degree and the D.Sc. degree in Physics from Chulalongkorn University, Thailand, in 2006 and 2011, respectively. He has expertise in computer aided design (CAD), computational fluid dynamics (CFD), finite element analysis (FEA), and computer simulation (CS). He is currently a lecturer at College of Advanced Manufacturing Innovation, King Mongkut's Institute of Technology Ladkrabang, Thailand. Also, he is a researcher and a consultant of several factories, who has a lot of experience in applying CAD, CFD, FEA and CS to solve various problems and improve the manufacturing processes.