Feasibility Study for Installing Machine in Production Line to Avoid Particle Contamination Based on CFD Simulation

Adisorn Khaokom\textsuperscript{a}, Jatuporn Thongsri\textsuperscript{b}\textsuperscript{*}

Computer Simulation in Engineering Research Group, College of Advanced Manufacturing Innovation, King Mongkut's Institute of Technology Ladkrabang, Bangkok, 10520, Thailand  
\textsuperscript{a}A.khaokom@gmail.com, \textsuperscript{b}Jatuporn.th@kmitl.ac.th

\textbf{Abstract.} Ventilation system inside production line for electronic component production needs to meet the factory standard. Because it can eliminate small particles which may cause of human or machine in production as well as it can distribute the circulating air temperature uniformly. CFD is used in this research in order to study the feasibility and plan for machine layout in production line before actual installation. The simulation shows the airflow in every area inside production line. From simulation with releasing the particles from human and machine is found that this ventilation system generates airflow that makes most particles float out of the machines and no particle downs to the conveyor, it results to contamination. In addition, the simulation also shows the range of 19-26 \textdegree C air temperature that meets the factory standard. The results of this research are the parts of the data to renovate the production line to get more efficiency and proper on the production.

1. Introduction
At present, the electronics industry has developed modern production technology for qualified and good quality products. The component production is essential to work in clean rooms with low particle contamination. Most of particles in clean room are always caused by friction of machine at work or any procedures, as well as humans that including the working suit, hair, skin even breathes itself. The amounts of particles occur depends on each individual human activity \cite{1}. Generally, the sizes of any contaminant, dust and small particles are from 0.001-100 microns. The particles larger than 5 microns are typically found on various surfaces and have the potential to float in the air for short time.

This research is cooperated project with one of electronics components manufacturing factories in Thailand, which plans to improve the production line installing the new machines with new higher efficiency and technology production. Machine layout in production line inside clean room will undoubtedly affect to airflow pattern and temperature distribution \cite{2}. Therefore, if study the feasibility and plan in advance before install machine in production line actually, it makes production quality good and reduces production cost.

From studying, it was found that Computational Fluid Dynamics (CFD) has been applied for simulating in order to solve any problems related to airflow, temperature and contamination in buildings and environment. For example, CFD is applied for simulating the airflow in the clean room of the LCD industry \cite{3} and in the classroom \cite{4} that the simulation results can improve airflow

\textsuperscript{*}Corresponding author.
quality using CFD. CFD is applied for studying the cooling by simulating the airflow in large-scale factories by CFD that the simulation results can be used for cooling generated in the factories efficiently [5]. CFD is applied for simulating the airflow in large-scale factories by CFD that the simulation results can be used for cooling generated in the factories efficiently [5]. CFD is applied for monitoring the particle transportation in the office with ceiling ventilator attaching [6]. CFD is applied for studying the emergence and the distribution of human particles in the indoor environment [7]. As well as CFD has been used to simulate the airflow in small environments of clean rooms for hard disk drive factories [8] and in automatic machinery for hard disk drives production [9]. That the simulation results can be used to determine the best air conditions and can be really used in the factory to reduce the contamination of particles that may occur in the automatic machinery for hard disk drive production.

So CFD is used in this research to simulate airflows, temperature and particle traces in production line under actual ventilation system of factory in order to assess the air quality that if the machines have been installed whether the efficiency of the airflow will meet the factory standard. The results of this research will be used to improve the production line for further production.

2. Methodology

2.1. Production Line and Machinery Models
The production line including machines to be installed used for the simulation is in the Class 1000. The model must be a simple model in order to simplify the simulation setting any environmental conditions as clean room in the actual factory [8]. The air from air conditioning flows into the FFU and Ulpa Filter on the ceiling and flows out the return shaft, return ceiling and all 3 sides of wall of the specific simulation. The laminar is also installed above the machine that the laminar sucks the air inside the room from the top of the laminar through the HEPA filter, can eliminates the particles floating in the air. Then release the clean air into the machine for driving the particles to outside machine in order to minimize particle contamination from the machine and human.

To solve the problem with CFD, aims to find the direction of airflow occurred, heat transfer in a clean room, and particle behavior caused by machinery and human. The model is created with decreasing the complexity in the irrelevant parts in order to simplify for simulation by measuring the size of the real machine. The machine consists of three main parts: 1) Machinery is function of producing the products; 2) Conveyor is function of transporting the components to the machinery; 3) Human is responsible for bringing in and out of production as shown in Fig. 2.

2.2. Fluid and Mesh Models
The FLUENT program is used based on the models shown in Figs. 1 and 2 to create the fluid and mesh models respectively. Then the mesh model created will be used in further simulation. The fluid and mesh models used in this work are shown in Fig. 3, respectively. The type, size and quality of the mesh affect to the time for calculation and the result in the process of this mesh model creation.

The mesh model created, has 1.41 million nodes, 7.3 millions elements of tetrahedron and 0.93 of the maximum skewness. Based on the mesh analysis, it was found that this mesh model is suitable for both quality of the result and the time for calculation.

![Figure 1. Production line](image1.png)

![Figure 2. Machinery Model with human inside](image2.png)
2.3. Mathematical Modeling

The conservation equations consist of mass (1), momentum (2), and energy (3) conservations.

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m, \] (1)

\[ \frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \rho \vec{g} + \vec{F}, \] (2)

\[ \frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho \vec{v} (\rho E + p)) = \nabla \cdot \left( \sum J_j \right) + S_h. \] (3)

Where \( \rho \) is the static pressure and \( \rho \vec{g} \) and \( \vec{F} \) are the gravitational body force and external body forces, respectively. \( S_m \) and \( S_h \) are user-defined source terms.

The Transition Shear Stress Transport (Transition SST) [10] is used for turbulence model in this paper because of its high reliability. Transition SST consists of turbulence kinetic energy \( k \), specific dissipation rate \( \omega \), transition momentum thickness Reynolds number \( Re_{\theta t} \), and intermittency \( \gamma \) equations. This paper is limited space, therefore the equation cannot be expressed and described at all. You can study more information in the Reference [11]. The computer must solve 3 conservation equations and 4 turbulence equations of total 7 equations based on the principle of CFD to find the solution in the calculation of turbulence model.

2.4. Fluent Settings

Based on environmental conditions measurements from the real factory that affect to small specific simulations, it was found that the air flows from 2 ways: FFU and Ulpa Filter on the ceiling area with temperature of 20.5\(^{\circ}\)C and 13.6\(^{\circ}\)C respectively. The conditions of inlet air are shown in Table 1.

When the air flows into his specific environmental simulation, laminar installed on machine will suck the air in the room from the top of the laminar itself through the HEPA filter. Then release the clean air to the machine for driving the particles to outside machine and flow out to this environment in 3 ways: return shaft, return ceiling and 3 sides of wall of the specific simulation. The conditions of out-flow air are shown in Table 2. By the position specified in Figure 4.

In addition, laminar with velocity of 0.45 m/s is installed on machine to blow air into the machine in order to remove the particles out. The shear stress transport of turbulence model with pressure-velocity as coupling, momentum, turbulence kinetic energy and turbulence dissipation values determined as second order upwind, human temperature as 37\(^{\circ}\)C, machinery temperature as 27\(^{\circ}\)C, and environment temperature and pressure as 24.5\(^{\circ}\)C and 101,250 Pascal, respectively. For the other values are set automatically by the program as appropriate. This calculation is a steady state of 1,000 cycles for accurate convergence.

| Table 1. Condition of In-flow Air. |
|-----------------------------------|
| FFU (Position) | Mass Flow Rate (kg/s) | Ulpa Filter (Position) | Mass Flow Rate (kg/s) |
|----------------|------------------------|------------------------|------------------------|
| 1              | 0.449                  | 1                      | 0.384                  |
| 2              | 0.568                  | 2                      | 0.398                  |
| 3              | 0.449                  | 3                      | 0.36                   |
| 4              | 0.449                  | 4                      | 0.36                   |

| Table 2. Condition of Out-flow air |
|-----------------------------------|
| Return Shaft (Position) | Outflow (%) | Wall (Position) | Outflow (%) | Return Ceiling (Position) | Outflow (%) |
|-------------------------|-------------|---------------|-------------|---------------------------|-------------|
| 1                       | 7.56        | 1             | 1.90        | 1                         | 6.07        |
| 2                       | 7.56        | 2             | 0.46        | 2                         | 5.02        |
| 3                       | 12.62       | 3             | 2.17        |                           |             |
| 4                       | 12.32       | 2             |             |                           |             |
| 5                       | 14.61       | 2             |             |                           |             |
| 6                       | 12.06       | 2             |             |                           |             |
3. Results and Discussion

Airflow in every area of ventilation system was found from this simulation. And this condition may result in vortex in some areas as shown in Fig. 5 that it occurs in the areas away from the conveyor and it does not affect to production line. Fig 6 shows the flow directions that the air flows through the machine then flow out at Return Shaft. The air velocity average at the conveyor line is 0.24 m/s that it's more than the velocity of 0.2 m/s. Therefore, it's considered as the air circulates at the conveyor is effective.

By simulating to release 400 particles within the defined time as shown in Fig. 7 found that the particles caused of human are blew out and eliminated by Return Shaft and laminar, so it's no contamination at conveyor line area. Most particles caused of machine are eliminated by the same way however, there is potential risky for contamination. In the worst case, 3.17% contaminated particles caused of machine is found from testing releasing 4000 particles.

The simulated results show the temperature distribution of the Ventilation system as shown in Fig. 8. The temperature average at the conveyor is 20.71°C; all points are less than 22°C that meet the factory requirement. However, the figure shows that the temperatures coherences are possible to have humidity, so it's interesting for further study.

![Figure 5. Airflow directions in XZ plane](image)

![Figure 6. Airflow direction in ZY plane](image)
4. Conclusion

In this study, CFD has been used to simulate the airflow, particle traces and temperature distribution in production line with planning to install in clean room of factory. The simulation result with various conditions including actual ventilation system of factory was found that the air flow in production line has efficiency to avoid the contamination caused of machine and human. In addition, the temperature in production line is proper and meets the factory requirement. The information from this research will be used for renovating production line of industrial factory.

Acknowledgment

The authors would like to thank College of Advanced Manufacturing Innovation (AMI) and King Mongkut's Institute of Technology Ladkrabang (KMITL) for supporting this research.

References

[1] McDonagh A and Byrne M 2014 J Environ Radioact. 127 119-26
[2] Thongsri J, Pimsarn M and Tangsopha W 2016 KKV Eng. J. 43 250-53
[3] Noh K C, Kim H S and Oh M D 2010 Build. Environ. 45 825-31
[4] Noh K C and Yook S J 2016 Build. Environ. 119 111-18
[5] Tanasic N, Jankes G and Skistad H 2011 Build. Environ. 43 1200-06
[6] Makhoul A, Ghali K, Ghaddar N and Chakroun W 2013 Build. Environ. 63 97
[7] Tao Y, Inthavong K, and Tu J 2016 Indoor Built Environ. 26 185–98
[8] Montrisavatkul P, Kaewkhaw P, Tangsopha W and Thongsri J 2016 The 4th Academic Science and Technology Conference 2016 (Thailand: ASTC2016) pp 295-301
[9] Thongsri J and Pimsarn M 2015 Int. J. Precis. Eng. Manuf. 16 509–15
[10] Menter F R 2009 Int. J. Comput. Fluid Dyn. 23 305-16
[11] Ansys Inc 2013 Ansys fluent 15 user’s guide (USA: Ansys Inc)