AJCHE 2011, Vol. 11, No. 2, 8 – 21

Numerical Simulation of Air Flow and Temperature Distribution in Yarn Drying Room

Jindarat Pimsamarn*[1]
Panit Kitsubun[2]
Rabin Tongruk[3]
[1]126 Pracha-U- Thit Road, Bangmod, Thugnkru, Bangkok, 10140
[2]126 Pracha-U- Thit Road, Bangmod, Thugnkru, Bangkok, 10140
[3]9 I-4 Road, Map Ta Phut, Muang, Rayong, 21150 Tel : 03892-2000, Fax : 03892-5600
**E-mail:* jindarat.pim@kmutt.ac.th

This research studied the recovery of wasted heat from exhaust air releasing from the compressor to increase the drying efficiency. The yarn drying room model was developed using Computational Fluid Dynamics (CFD) so as to study the distribution of air flow and temperature numerically. The most suitable design of the drying room considering the decrease of drying time such as the locations of hot air inlet and outlet were investigated. At the exhaust air outlet position, the average exhaust air temperature is 56 °C, while the average flow rate and relative humidity are 1.75 m/s and 16.2 percent, respectively. The hydraulic calculation revealed that the appropriate duct size was 0.412 x 0.412 m^2 width and height. The hot air temperature after transferring through air duct was decreased from 56°C to 52°C. The simulation results showed that the appropriate inlet position is at the ceiling and split into 4 inlet positions which had 0.152 x 0.152 m^2 width and height. The most appropriate outlet position was 3 m above the ground in order to achieve the highest distribution of moisture content with standard deviation of 1.9×10^{-4}. The locations which had the high accumulation of moisture were at the center and both sides of the drying room due to the low air turbulence. From this configuration, the drying time was decreased from 2 days to 89 minutes. Furthermore, after the 2 air circulators were installed in the drying room, the distribution of moisture content, represented in term of standard deviation, was about 1.5×10^{-4}. As a consequence, the drying time could be reduced to 78 minutes, but the monthly electricity cost of air circulators was around 590 baht.

Keyword: CFD, Yarn, Drying

INTRODUCTION

Background Problem

The selected textile company produces 3 types of yarns i.e. nylon-6, nylon-6,6 and polyester.During the production process, the moist yarns are dried naturally under room temperature. The yarn drying step takes time as long as 2 days. This quite too long period of time allow the fungi to grow on the yarn product and also waste time for productivity. From this reason, it is desired to design a more efficient drying room. Heat used in drying room can possibly be obtained from hot exhaust air releasing from the compressors which is recently wasted to the surrounding. It flows through pipeline from exhaust air outlet to the drying room as shown in Figure 1.

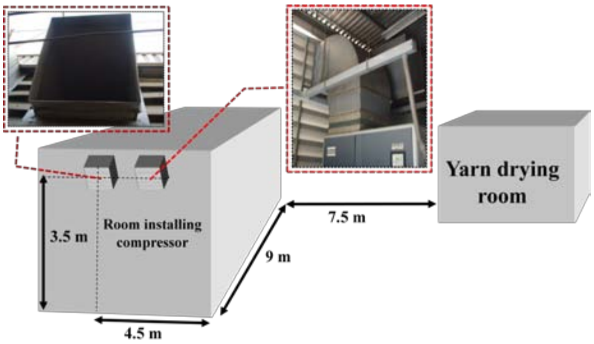


Fig. 1: Distance between compressors and yarn drying room

Objective

To design the yarn drying room using wasted heat from air compressor

THEORY

Computational Fluid Dynamics (CFD)

CFD stands for the computational fluid dynamics which is used as a computer-based tool for simulating the behaviour of fluid flow. CFD is widely used in the various industries that involve the fluid dynamics such as aerospace, automotive and chemical industries. Nowadays, there are many software packages of CFD used to predict the fluid behaviour such as FLUENT, CFD-ACE+, AVL FIRE and CFX. In this research, CFX program is used [1].

Governing equation

To predict the fluid behavior, the governing equations consisting of mass, energy and momentum balance equations were shown in Equations 1, 2 and 3, respectively.

$$\frac{\partial(\rho)}{\partial t} + \nabla \bullet (\rho U) = 0 \qquad (1)$$

$$\frac{\partial(\rho h_{total})}{\partial t} - \frac{\partial(\rho)}{\partial t} + \nabla \bullet (\rho U h_{total}) = \nabla \bullet (\lambda \nabla T) + \nabla (U \tau) + U \bullet S_M + S_E \qquad (2)$$

$$\frac{\partial(\rho U)}{\partial t} + \nabla \bullet (\rho U x U) = -\nabla p + \nabla \bullet \tau + S_M \qquad (3)$$

To solve these equations, the solver uses the finite volume method as standard numerical solution technique to convert all of these partial differential equations to algebra equations and subsequently solve these algebra equations numerically.

Drying [2]

The removal rate of the drying was controlled by the rate of heat transfer to the evaporating surface and provided the latent heat of evaporation for the liquid. To derive the equation for drying, the radiation heat transfer on the solid surface is negligible and also the heat conduction from metal pans is not taken into account. The equations of heat and mass transfer for this drying process can be written as followed:

The rate of convective heat transfer

$$q = h(T - T_w) A \qquad (4)$$

The flux of water vapor from the drying material to the hot gas

$$N_A = k_y (y_w - y) \qquad (5)$$

As the rate of mass transfer balances with the rate of heat transfer at the steady state, the amount of heat needed to vaporize the water from drying material is shown below.

$$q = N_A \lambda_w A \quad (6)$$

From Equations 4, 5 and 6, the rate of drying in the period of constant-rate can be derived and shown in Equation 7.

$$R_c = \frac{h(T\text{-}T_w)}{\lambda_w} \quad (7)$$

METHODOLOGY

The methodology consisting of 3 main steps i.e. study and collect data, perform the hydraulic calculation including wall design and model simulation.

Study and collect data

In the first step, some theories involving with this research such as hydraulic calculation and CFD were studied so as to understand the fundamental knowledge. Meanwhile, the necessary information for the hydraulic calculation and simulation were collected from the selected textile company. The data could be collected from 2 parts which were the data relating with the effluent air from the air compressor such as temperature, velocity, and relative humidity and the characteristic of yarn such as geometry , size and weight.

Perform the hydraulic calculation including wall design

The appropriate duct size was determined in this step so as to give the proper transportation of exhaust air from exhaust air outlet position to the yarn drying room. For example, if the duct size is too large, it will enhance the heat loss from hot air to the surrounding. It implied that the larger duct size, the higher temperature drop. On the other hand, if it is too small, the pressure drop in duct will be so high that the exhaust air cannot flow into the yarn drying room, and the vibration of duct will also occur. For the appropriate design, both temperature and velocity of hot air are maintained as high as possible because these two variables directly affect the drying rate.

After finishing the determination of temperature and pressure drops in duct, the wall of drying room must be designed to resist the heat loss from drying room as much as possible. In order to design the yarn drying room, the double layers of brick which has the thickness of 6.5 cm in each side was used for the construction and there was a gap of trapped air in the middle of the wall which was about 5 cm width to resist the heat transfer rate.

Model Simulation

Create and mesh the model

In this research, the size of drying room was designed by considering the volume of yarn and the minimum space between drying material, 30 cm. The geometry of drying room was 3 m width, 5.5 m length and 3.5 m height. For the turbulence model, the SSG Reynolds stress model was used because of high accuracy for predicting the turbulent flow in bulk and also good for buoyant flow. Moreover, in order to ensure the numbers of meshes and nodes were high enough to accurately solve the governing equation, the mesh independence test was performed by varying the number of nodes and gathering the average velocity in drying room. As the results, it was found that although the number of node was increased over 350,000 nodes, the average velocity in drying room was remained the same. Owing to this reason, the number of elements and nodes used in this work was 350,306 nodes and 2,142,612 elements.

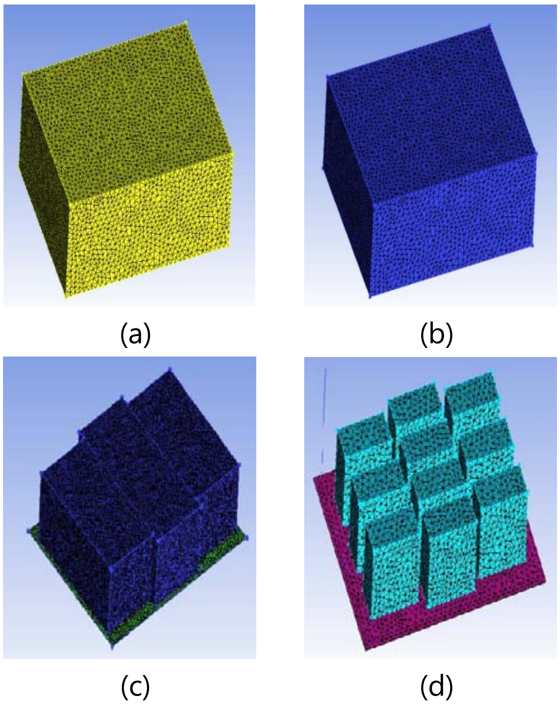


Fig. 2: The Discrete Domain
(a) fluid domain (b) solid domain
(c) yarn domain (d) sub-domain

In the simulation, the computational domain can be divided into 4 domains which were fluid domain, solid domain, yarn domain and sub-domain as shown in Figure 2. The sub-domain was created for the determination of proper entrance location of the hot air. It was defined by dividing the fluid domain into 2 volumes. One was the inner volume and another was the outer volume. The inner volume was the volume of free space which had the distance in all directions from yarn less than 15 cm. This was the minimum distance between materials which can enhance air circulation according to the literature [3]. The outer volume was the volume of the drying room subtracting with inner and yarn volumes. In this research, the criterion to select the most appropriate inlet position was the volume of free space having the velocity higher than 1.4 m/s in the inner volume.

Simulate model

Find the appropriate inlet position of hot air

- Inlet position installed on the wall

  In this configuration, the different inlet positions that were installed on the wall can be categorized into 3 scenarios. In Scenario I, the inlet position was laid on the corner of the wall and risen 25 cm from the ground. The outlet position was laid across another top corner. In Scenario II, the inlet position was installed at the middle of wall and the level was the same as Scenario I. The outlet position was levitated 2 m from the ground at the opposite wall. Lastly, in Scenario III, the inlet position was spit into 2 positions and levitated 25 cm from the ground while the outlet position was the same as Scenario II.
- Inlet position installed at the ceiling

  The inlet position was installed at the ceiling of the drying room by using air diffuser. In this research, 3 types of air diffuser, i.e. square, round and slot diffusers were used. In the square-diffuser case, the inlet position was shared into 2 positions so as to maintain the velocity as high as possible. However, in round and slot diffuser cases, the inlet position was split into 4 positions, in order to reduce the inlet velocity. It was because the high inlet velocity can cause the resonance of air in diffuser.

Optimize the inlet cross sectional area

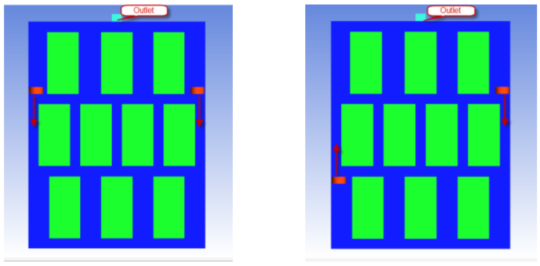
In order to make highest turbulence in the drying room, the velocity of hot air at the entrance of the drying room was increased by reducing the inlet cross sectional area. To decrease the cross sectional area as much as possible, the outlet pressure was treated at least 1 bar. The optimization of inlet cross sectional area was performed using Aspen plus simulation program to determine the cross sectional area that the outlet pressure was equal to 1 bar. As the results, the inlet cross sectional area was around 0.024 m^2. After achieving the minimum cross sectional area, in order to achieve the most appropriate number of inlet position, the configuration was divided into 3 categories which were 2, 4 and 6 inlet positions.

Find the appropriate outlet position of hot air

The investigations of proper outlet position of hot air including the number of outlet position were performed. The criterion used to make decision was the distribution of moisture content in the drying room. To find the proper outlet position, the outlet position was investigated by varying between 0.25 m and 3 m above the ground. The distribution of moisture content can be calculated from the standard deviation. After achieving the proper outlet position, the number of outlet position was varied between one and two outlet positions.

Find the proper air circulator type and its location

In this step, the investigated locations were divided into 2 scenarios, demonstrated in Figure 3. All of them were different in the direction of airflow from the circulator. In order to select the best scenario, the distribution of moisture content was used as the criterion. The most appropriate air circulator location was considered from the case which gave the lowest value of standard deviation.



(a) (b)

Fig. 3: The location and direction of air circulator dividing into
(a) Parallel-counter current flow
(b) Across-co-current flow

RESULT AND DISCUSSION

Hydraulic Calculation

From the design using the same criterion as the engineering firm, the appropriate size of duct was designed at 0.412 x 0.412 m^2.

To determine the conditions of exhaust air at the entrance of drying room, Aspen Plus simulation program was used. As a result, it was revealed that the pressure and temperature of exhaust air at the entrance of drying room were 1.012 bar and 52.63°C, respectively (referred as inlet position installed at the wall). After changing the inlet position to the ceiling, it was found that the pressure and temperature drops were slightly increased because of the longer length of duct (around 5 m) and higher static pressure. Hence, the pressure and temperature of exhaust air at the inlet position were installed at ceiling were 1.009 bar and 52.06°C, respectively.

Wall design.

In order to minimize heat loss from the drying room, the multi layers wall with air gap in the middle was used because of the very low thermal conductivity of air. From

Newton's law of cooling and Fourier equations, the heat loss releasing from this drying room was estimated to be 0.898 kW.

CFD Post

Inlet position installed at the wall

Temperature distribution in drying room

In the drying room, the temperature distributions of Case I and II were shown in Figure 4. Case I referred as the empty drying room while drying room putting 10 blocks of arranged yarn were shown in Case II. Due to the assumptions which are the neglect of moisture physical properties and steady state system, the temperature distribution in yarn drying room of all scenarios in Case I and II were the same which were around 52°C. Consequently, based on these assumptions, it could be concluded that the different inlet positions had no effect on temperature distribution at steady state.

Volume of free space having the velocity higher than 1.4 m/s

The volumes of free space having the velocity higher than 1.4 m/s in Case I and II were shown in Figure 5. In Case I, the volumes of free space having the velocity higher than 1.4 m/s in all scenarios were higher than Case II. After the 10 blocks of yarn were put in the drying room, the volumes of free space having the velocity higher than 1.4 m/s, in all scenarios, were obviously decreased. This was because when the hot air crashed the yarn, it lost the momentum which related to the velocity. Therefore, the velocity of hot air was decreased.

To select the most appropriate inlet position, the comparison of volume of free space having the velocity higher than 1.4 m/s in all scenarios were shown in Table 1. From Table 1, even though Scenario I had the highest volume of free space having the velocity higher than 1.4 m/s in the total volume, it was not selected as the most appropriate scenario. This is because the criterion used to select the most appropriate scenario was the volume of free space having the velocity higher than 1.4 m/s in the inner volume instead of total volume. It was because 10 blocks of yarn were in the inner volume. Therefore, the inlet position in Scenario III was selected as the best inlet position because this scenario gave the highest volume of free space having the velocity higher than 1.4 m/s in the inner volume.

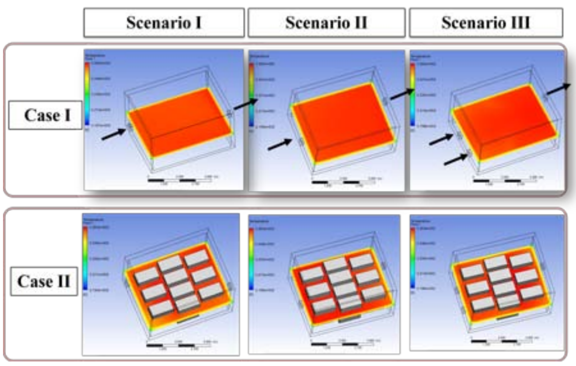


Fig. 4: The temperature distribution in drying room of Case I and II

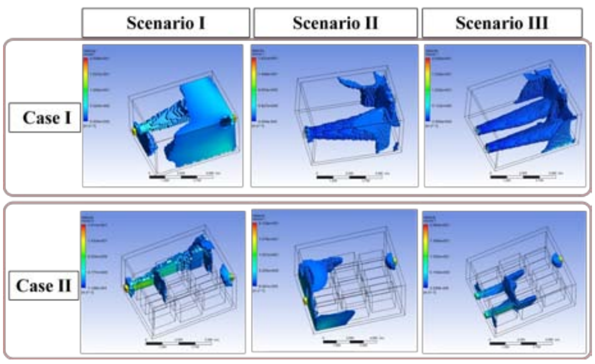


Fig. 5: Volume of free space having the velocity higher than 1.4 m/s in all scenarios of Case I and II

Table 1. Volume of free space having the velocity higher than 1.4 m/s in all scenarios after putting yarn into drying room

Scenario	Volume of free space* (m^3)		
	Total volume	Inner volume	Outer volume
I	7.92	2.76	5.17
II	6.44	1.20	5.24
III	5.03	4.07	0.96

*having the velocity higher than 1.4 m/s

Inlet position installed at the ceiling

Planes of air velocity in the drying room

The results of CFD simulation when hot air flows from the ceiling were presented in this section. The hot air velocities which flew through any types of diffuser were obviously different as shown in Figure 6. Furthermore, the effect of diffuser at the entrance region was shown in Figure 7.

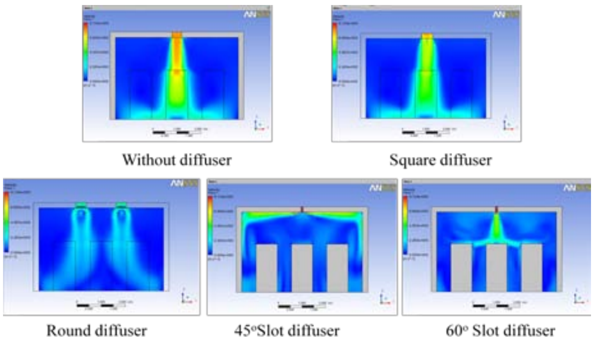


Fig. 6: The planes of air velocity in drying room

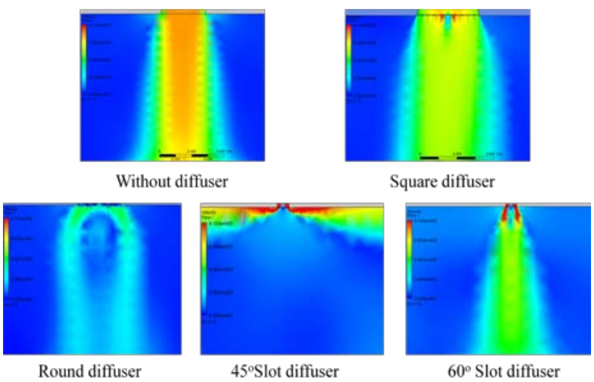


Fig. 7: The effect of diffuser at the entrance

As considering between without-diffuser and square-diffuser cases, the velocity of hot air in without-diffuser case was greater than square-diffuser case. It was because when the hot air flew and hit the diffuser, the momentum was transferred to the diffuser so it caused the decrease of hot air velocity. Considering the distribution of air flow, the flow of hot air was slightly spread after flowing through the square-diffuser. This was the effect of air diffuser. It is different from without-diffuser case that there was no spread of air flow. In the round-diffuser case, in order to avoid the air resonance problem, the air flow rate must be split into 4 inlet positions. The split of air flow was directly affected on the decreasing of inlet velocity. Consequently, the velocity of hot air in round-diffuser case was significantly different from the first two cases. In slot-diffuser case, the results could be divided into 2 cases which were 45° and 60° slot-diffusers. Even though both of them were resulted in the same velocity of the hot air, they were obviously different in the direction of air flow. For 45° slot-diffuser, the hot air flew and then attached the ceiling after flowing from the diffuser. On the other hand, when the 60° slot-diffuser was used, the hot air can flow in the downward direction.

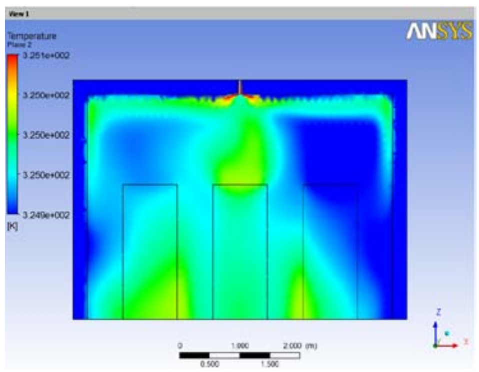


Fig. 8: The plane of temperature in yarn drying room

The reason that the directions of air flow from both cases of slot-diffuser were different was the free convection effect. Considering Figure 8, the hot air temperature at the entrance was about 0.2°C higher than the temperature of drying room. From this reason, the upward buoyancy force was occurred, and it enabled the hot air to float in upward direction. In case of the 45° slot-diffuser, the flow of hot air attached to the ceiling instead of flow in the downward direction because the upward buoyancy force was greater than the downward inertia force of hot air. On the other hand, when the 60° slot-diffuser was used, the upward buoyancy force was less than the downward inertia force of hot air. This caused the hot air to flow directly in the downward direction. As observing between these two cases, the downward inertia force of hot air in the 60° slot-diffuser was greater than 45°slot diffuser. This was because the 60° slot-diffuser had greater magnitude of force when the vector quantity of force was divided into components parallel to the z coordinate axis.

Volume of free space having the velocity higher than 1.4 m/s

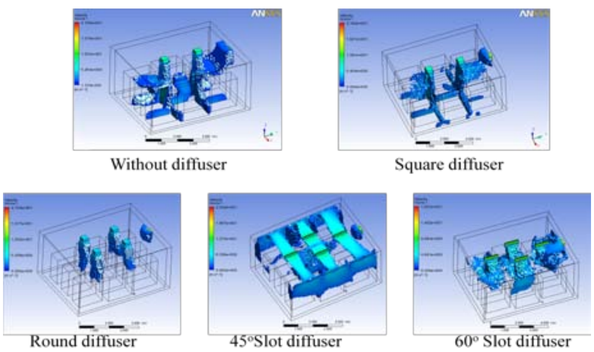


Fig. 9: Volume of free space having the velocity higher than 1.4 m/s when flowing from ceiling

Table 2. Volume of free space having the velocity higher than 1.4 m/s in any type of diffuser

Case	Volume of free space* (m^3)		
	Total volume	Inner volume	Outer volume
Without	7.80	4.77	3.03
Round	6.11	4.37	1.75
Square	5.62	3.80	1.82
Slot 45°	12.86	2.46	10.40
Slot 60°	8.43	3.26	5.17

Figure 9 showed the volume of free space having the velocity higher than 1.4 m/s of all cases in order to make a decision which type of diffuser was the most appropriate. As considering between without-diffuser and square-diffuser cases, the volumes of free space having the velocity higher than 1.4 m/s in both cases were obviously different. However, when the round-diffuser was used, the volume of free space having the velocity higher than 1.4 m/s was less than the first two cases. It was because the inlet position in round diffuser case must be split into 4 positions to avoid the noise problem. This split of air flow rate caused the decreasing of velocity. When considering on 45° and 60° slot-diffuser cases, the volumes of free space having the velocity higher than 1.4 m/s in the total volume were greater than the other types of diffuser as demonstrated in Table 2. However, the reason that the slot diffuser type was not selected as the most appropriate case could be divided into two issues. The first one was because the criterion used to select the most appropriate scenario was the volume of free space having the velocity higher than 1.4 m/s in the inner volume instead of total volume. Moreover, when considering in the economic aspect, the increase of volume of free space having the velocity higher than 1.4 m/s in total volume when the slot-diffuser was used, may be not worthwhile when comparing with the investment and maintenance costs of slot-diffuser. Therefore, the without diffuser case which had the highest volume of free space having the velocity higher than 1.4 m/s in the inner volume was selected as the most appropriate case.

Optimization of inlet cross sectional area

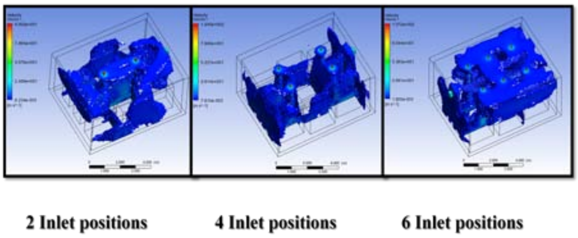


Fig. 10: Volume of free space having the velocity higher than 1.4 m/s after optimization of inlet cross sectional area

As shown in Figure 10, the volume of free space having the velocity higher than 1.4 m/s was dramatically increased after optimizing the inlet cross sectional area. In addition, the volume of free space having the velocity higher than 1.4 m/s increased when the number of inlet positions were increased. However, although the six inlet positions case had the greatest volume of free space having the velocity higher than 1.4 m/s in the total volume, the volume of free space having the velocity higher than 1.4 m/s in the inner volume was less than that of four-inlet-position case. It was because the pressure loss due to the fitting was occurred. Moreover, considering the economic aspect, it may be not worthwhile to install six inlet positions because the higher number of inlet positions, the higher installation and maintenance costs. Consequently, the inlet position of four was the most appropriate number of inlet position.

Investigation of the appropriate outlet position

Effect of outlet position on the direction of air flow

Figure 11 showed the direction of air flow, represented by the vector, when the outlet position was 0.25 m above ground whereas the direction of air flow, when the outlet position was 3 m above ground, was illustrated in Figure 14.

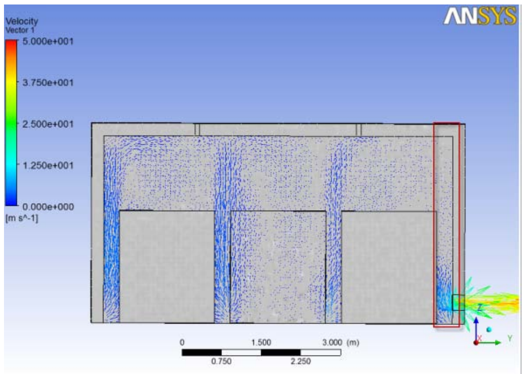


Fig. 11: The direction of air flow when the outlet position was 0.25 m above ground

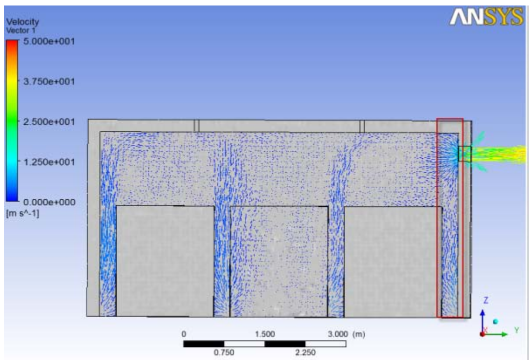


Fig. 12: The direction of air flow when the outlet position was 3 m above ground

Both of them were almost resulted in the same direction of air flow except the zone in red box that the air turbulence in Figure 13 was less than Figure 14. The reason why both cases gave the different air turbulence in the red box zone was the vortex zones as shown in Figure 13. These vortex zones were generated when the hot air flew into the drying room. The sheer stress between the hot air inlet and adjacent surrounding air in the drying room caused the upward spiral movement of air. Consequently, if the air was removed at 0.25 m above ground, it would be retarded by the air movement from the vortex zone. On the other hand, if the hot air was removed at 3 m above ground, it would be enhanced by the air movement from the vortex. Therefore, it indicated that greater air turbulence in the red box zone was achieved, if the outlet position was 3 m above ground.

Effect of outlet position on the distribution of moisture content

The moisture content distributions when the outlet positions were 0.25 m and 3 m above the ground were shown in Figure 14 and 15, respectively. The distributions of moisture content in both cases were obviously different in every location, particularly the red box zone. This was because the difference of air turbulence directly affected on the accumulation of moisture. In case that the outlet position was 0.25 m above ground, the moisture generated from yarn was accumulated and gradually increased because of the low air turbulence in red box zone. As a consequence, the moisture accumulation at red box zone in this case was greater than the case that the outlet position was 3 m above ground. In addition, considering on the greatest moisture accumulation zone, it was in the same locations which was around the center of the room. It was because at this location, the turbulence due to the vortex cannot spread out to this location

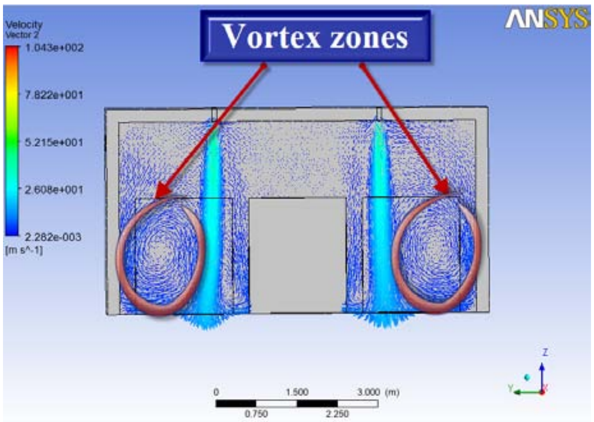


Fig. 13: The generation of vortex zones

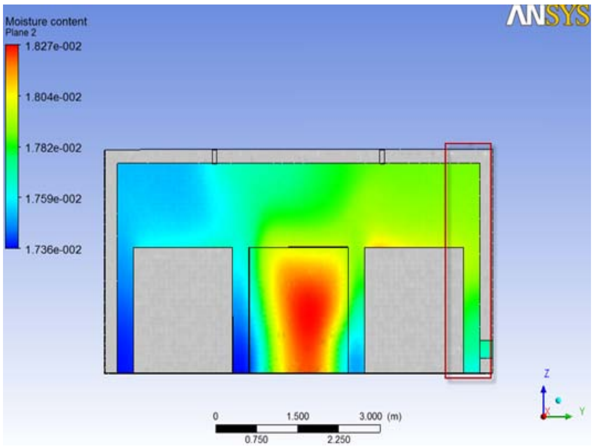


Fig. 14: The distribution of moisture content when the outlet position was 0.25 m above ground

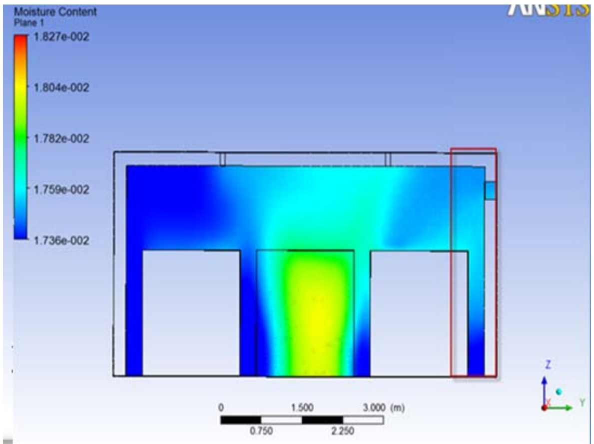


Fig. 15: The distribution of moisture content when the outlet position was 3 m above ground

Effect of the number of outlet positions on the moisture content distribution

Figures 16 and 17 showed the direction of air flow and distribution of moisture content when one and two outlet positions were used, respectively. Both cases were insufficiently different in the direction of air flow and the distribution of moisture content. It indicated that the number of the outlet position insignificantly different on the direction of air flow and the distribution of moisture content. Considering the regions which had the moisture accumulation, in both cases, they were resulted the same regions which were regions A, B and C.

The case which had the least standard deviation was the most appropriate case. From the calculation, the standard deviation in the case that the outlet was 3 m above ground and one outlet position was the least. Consequently, this case was selected as the most appropriate case.

Investigation of the appropriate location of air circulator

Effect of air circulator on the direction of air flow

The results of the direction of air flow were discussed. Figure 18 showed the direction of air flow when there was no air circulator while the direction of air flow of Parallel-counter-current flow and Across-co-current flow cases were demonstrated in Figures 19 and 20, respectively.

In the case of no air circulator, the air turbulence at regions A and C was obviously different from Parallel-counter-current flow and Across-co-current flow cases. This was the effect of air circulator. Considering between Parallel-counter-current flow and Across-co-current flow cases, the air turbulence at any locations in the drying room were the same except the region B.

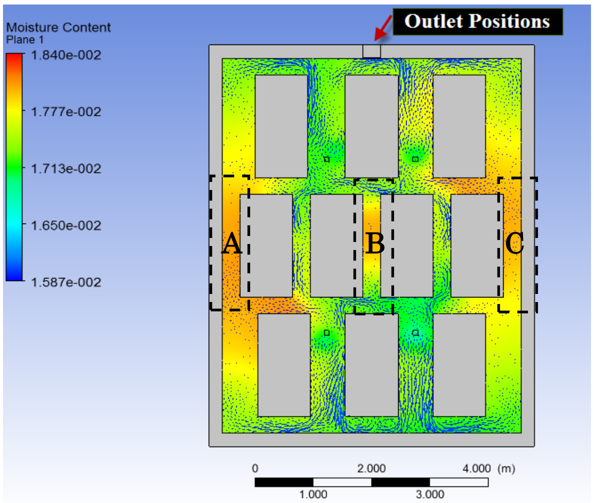


Fig. 16: The distribution of moisture content and direction of air flow when one outlet position was installed

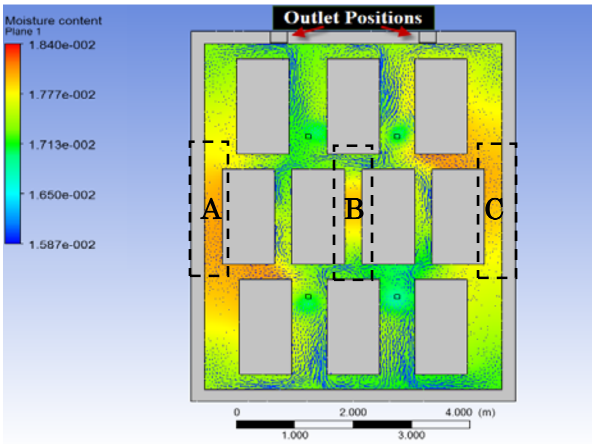


Fig. 17: The distribution of moisture content and direction of air flow when two outlet positions were installed

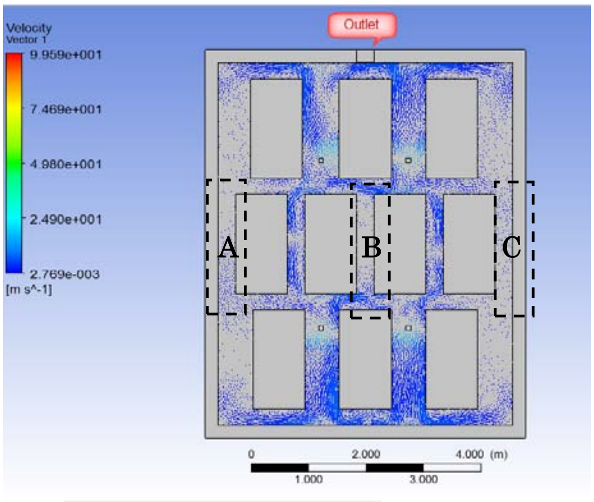


Fig. 18: The direction of air flow when there are no air circulator

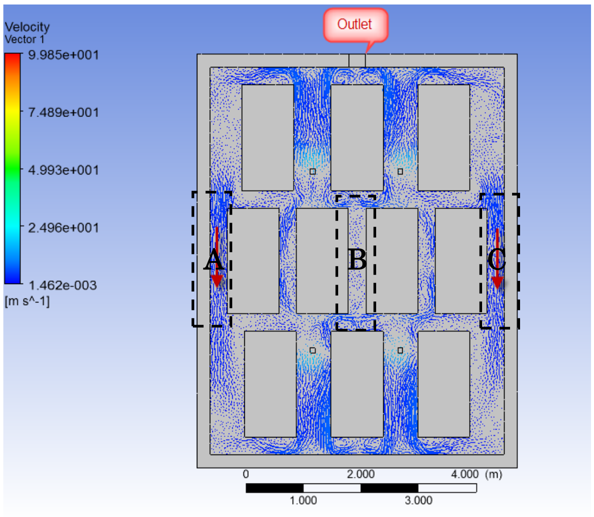


Fig. 19: The direction of air flow in Parallel-counter-current flow case

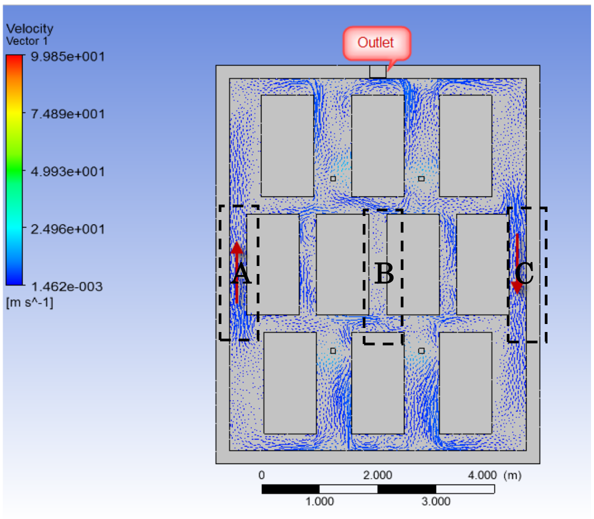


Fig. 20: The direction of air flow in Across-co-current flow case

In this region, the air turbulence in the Parallel-counter-current flow case was greater than Across-co-current flow case. It was because, in Parallel-counter-current flow case, the air from the circulator flew in the same direction that cause the air stream to crash each other in the middle and then flew into the center of the room. As a consequence, the air turbulence at the region B due to the air circulator was occurred. It was different from Across-co-current flow case that the air from circulator flew in the opposite direction resulted in the air flow without crashing. Therefore, in this case, there was no turbulence due to air circulator in the region B.

Effect of air circulator on the distribution of moisture content

Table 3 Standard deviations of moisture content of all cases

Case	S.D.
Without air circulator	0.000189
Parallel-counter-current flow	0.00015
Across-co-current flow	0.000212

In order to determine the most appropriate location of air circulators, the effects of air circulator on the distribution of moisture content in all cases were shown in this section. Figure 21 showed the distribution of moisture content before adding the air circulator. Meanwhile, the distributions of moisture content of Parallel-counter-current flow and Across-co-current flow cases were illustrated in Figures 22 and 23, respectively. As demonstrated in Figure 24, the distribution of moisture content in this case was better than the case without air circulator. The moisture accumulations at the region A and C were eliminated while the moisture accumulation at the region B was reduced because of greater turbulence in this region. Hence, the standard deviation of this case was lower than that of the case without air circulator as shown in Table 3. When considering Across-co-current flow case, although the air circulator could get rid of the moisture accumulations at the region A and C, the moisture accumulation at the region B was not decreased as found in Parallel-counter-current flow case. This was the reason why the standard deviation in this case was greater than the Parallel-counter-current flow case. Thus, it was indicated that the location and direction of air circulation should be assembled as the Parallel-counter-current flow case.

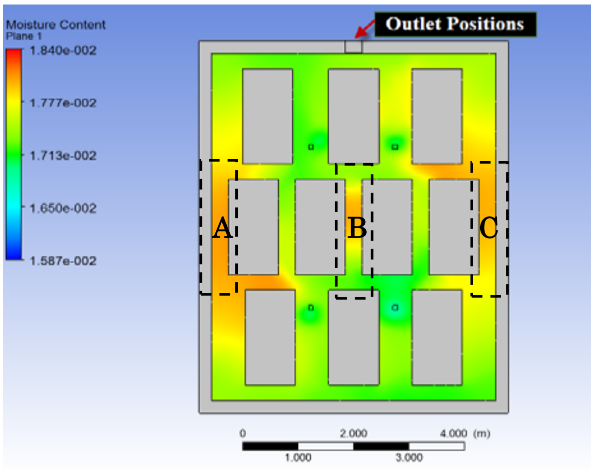


Fig. 21: The distribution of moisture content before adding air circulator

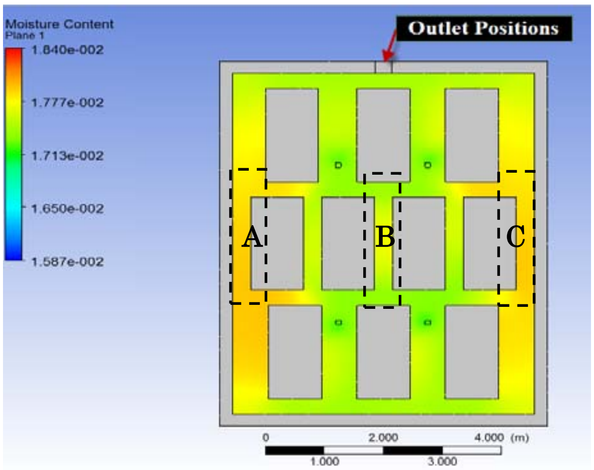


Fig. 22: The distribution of moisture content in Parallel-counter-current flow case

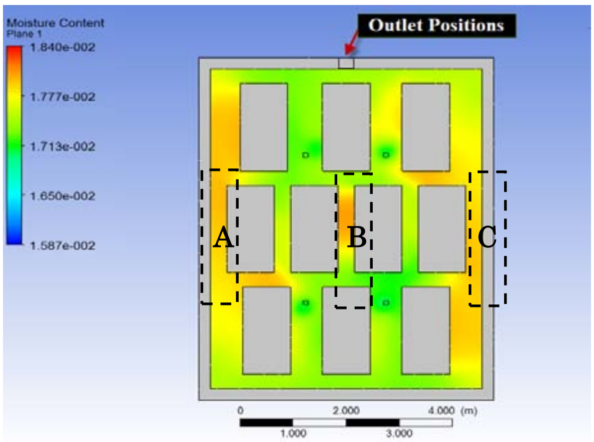


Fig 23: The distribution of moisture content (top view) in Across-co-current flow case

CONCLUSION

From the calculation, the appropriate duct size used to transport the exhaust air from the exhaust air outlet position to yarn drying room was 0.412 x 0.412 m^2. The galvanize steel with 0.71 mm thickness was selected as fabrication material. From this configuration, the temperature and pressure drop of exhaust air due to the transportation were about 5 x 10^{-3} bar and 4^oC, respectively. Therefore, the condition of exhaust air at the entrance of drying room was around 1.01 bar and 52^oC. The multi layers of wall which had the 6.5 cm thickness of brick and 2.5 cm thickness of concrete were used as construction material for the wall of drying room. In addition, there was 5 cm gap of air in the middle of composite wall. By using this construction, the heat loss from this drying room was only about 0.9 kW.

From the CFD simulation, it was revealed that the most appropriate inlet position was at the ceiling. Any types of air diffusers, square, round and slot diffusers were not necessary to be installed, but the inlet position must be spitted into 4 positions in order to increase the distribution of air flow. The cross-sectional area of each inlet was reduced from 0.412 x 0.412 m^2 to 0.156 x 0.156 m^2 so as to make the turbulence in drying room as much as possible. Meanwhile, a proper outlet position was installed at 3 m above ground. This was enough to remove the moisture from the room. As considering on this most appropriate configuration, it gave the volume of free space having the velocity higher than 1.4 m/s in inner and total volumes around 16 m^3 (58% of total inner volume) and 25 m^3, respectively. The average velocity and temperature in the drying room were 1.51 m/s and 51.67^oC, respectively whereas the average moisture content was 0.0178 kg H_2O/kg dry air and 0.000189 of standard deviation (1.06% deviation). From the calculation using these results, the drying time was decreased from 2 day to 89 minutes. Lastly, the requirement of air circulator was based on the satisfaction of the company when considering between the drying time which was faster only about 11 minutes (decreasing from 89 minutes to 78 minutes) and electricity cost of air circulators which was about 570 baths per month.

REFERENCE

1) Jiyuan, T., Guan, H.Y. and Chaoqun L., 2008, **Computational Fluid Dynamics (a practical approach),** UK, pp. 29-63.
2) Pimsamarn, J., et al, 2008**, Efficiency Improvement of Yarn Drying Process Using Wasted Heat from Air Compressor,** Senior Project, Bachelor Degree of Chemical Engineering Department, King Mongkut's University of Technology Thonburi, Thailand, pp. 1-65.
3) **Wood drying** [Online], Available: http // www.wikipedia.org/wiki/Wood_drying [May 8, 2009].