

Numerical Simulation of the Airflow Characteristics of Drying by Ground Air Film

Zheng Yu-lan¹, Peng Xiao-yong¹

¹ School of civil engineering, University of South China, Hengyang Hunan China

1170218114@qq.com, pengxiaoyong@126.com, huwei@126.com

Abstract. According to the interference and coupling characteristics among sub-systems of the chassis of the forklift truck, the the neutral network inverse system method of the non-linear system is applied for the decoupling control of the active rear steering (ARS) and the direct yaw moment control (DYC) of the forklift truck. Based on the analysis of the reversibility of the chassis system, a back propagation neutral network reverse system model is set to decouple the chassis system into two independent pseudo-linear system; A PD closed loop controller is designed to comprise a composite controller with the neutral network reverse system and the simulation is performed for verification. The simulation result shows that the decoupling control strategy with the neutral network reverse system can eliminate the interference and coupling among sub-systems of the chassis and thus enhances the status tracking and operating stability of forklift trucks.

1. Introduction

Indoor water easily cause personnel fall and conducive to breeding mold[1]. It seriously affects the indoor environment and the health of workers. The ground dryer is a device used to dry the surface of the water and improve the indoor wet environment. The traditional ground drying machine uses large air jet to dry the ground. Only a small part of the air flow is used to dry the ground, and most of the air flow is wasted. This kind of ground drying machine has the characteristics of quick drying ground and convenient use. However, with the greater noise, and the utilization rate of air is very low, it is not conducive to save energy .

Coanda effect is also called attaching effect. It points out that the fluid (water or airflow) has a tendency to move away from the original flow direction to the surface of the protruding object[2]. At present, the effect of Coanda is widely used in the field of aviation, building ventilation and industry. Zhang Shiyu et al. studied a supercritical circulation control airfoil of dual-radius, and optimized its shape, and obtained the optimal model of lift resistance performance. This circulation control technology is derived from the Coanda effect, it can solve the traditional high lift system for aerodynamic noise problems. Bladeless fan is a typical application of the Coanda effect, and the air volume can be increased by 15 times of the air intake rate of the initial air inlet, which solves the problem of strong wind blowing and noise. In a similar study, Ye Yaling used the numerical simulation method to study the airflow characteristics of the airfoil air inducer, and verified the numerical simulation results from the experimental point of view. In view of the fact that the traditional range hood is not easy to be cleaned, the noise is big and the energy consumption is big, Huang Jun you proposed a new range hood based on the effect of Coanda, which has the advantage of



energy saving and noise reduction, and its theoretical displacement can reach 24.3m³/h, much higher than the national standard (7m³/h).

The Coanda effect was adopted by this paper in the application of ground dryer. The surrounding air was induced by small air volume to flow closely on the Coanda curved surface, resulting in the formation of an air film attached on the ground. The numerical simulation technique was used to simulate the segment length and inclination angle of Coanda surface, as well as the air film formation and airflow characteristics when the curvature of Coanda surface changed.

2. Governing equations and numerical methods

2.1. Control Equations and Turbulence Models

The air flow of the ground dryer is low-speed incompressible turbulent flow, satisfying the N-S equation, and the control equation is as follows:

continuity equation

$$\frac{\partial u_j}{\partial x_j} = 0 \quad (1)$$

momentum equation

$$\frac{\partial u_i u_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\mu}{\rho} \frac{\partial}{\partial x_i} \left(\frac{\partial u_i}{\partial x_j} \right) + f_i \quad (2)$$

Among them, ρ is the air density, f_i for the volume force, u for the air velocity, p for the gas pressure, μ for the dynamic viscosity coefficient, two-dimensional simulation, i, j, k value of 1 and 2. Three-dimensional simulation, i, j, k value of 1, 2 and 3, u_i for the average wind speed.

The turbulence model uses the standard k- ϵ model, and the turbulent kinetic energy k and turbulent dissipation rate ϵ equation are as follows:

$$\frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon \quad (3)$$

$$\frac{\partial(\rho u_i \epsilon)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} \quad (4)$$

Among them, the turbulent viscosity $\mu_t = \rho C_\mu k^2 / \epsilon$. The universal constants of the model are $C_{1\epsilon} = 1.44$, $C_{2\epsilon} = 1.92$, and $C_{3\epsilon} = 0.09$ respectively. The turbulence of the turbulent kinetic energy and the dissipation rate ϵ is $\sigma_k = 1.0$, $\sigma_\epsilon = 1.3$, respectively.

2.2. Numerical Methods

The numerical simulation was conducted based on the SIMPLE algorithm and the standard pressure compensation equation [3]. The discretization was performed by finite volume method. The first-order upwind spatial difference discrete scheme was applied in the continuity equation, the momentum equation, the turbulent kinetic energy equation and the dissipation rate equation. Ye Yaling et al. used the same numerical method as described above to numerically simulate the airflow characteristics of the airfoil-shape air induction unit based on Coanda effect. The relative deviation between the numerical simulation result and the experimental result of the airfoil-shape unit's air flow velocity was 11.4%. This has proved the reliability of numerical method.

3. Numerical simulation of air film and air flow characteristics of ground dryer

3.1. Physical Model, Model Parameters and Mesh Generation

The physical model of the ground dryer is shown in Figure 1, consisting of an arc-shaped cover plate, a slit outlet, an air inlet and a Coanda surface. Among them, the angle between the Coanda surface and the floor is Coanda surface inclination angle (α), The air inlet is v_1 , The air outlet is v_2 and Coanda curvature is k . Specific parameters of physical model are shown in table 1. The computational domain, shown in Figure 2, is 4000mm * 1030mm. FLUENT software is used for numerical simulation, in order to achieve the accuracy of simulation calculation, mesh encryption is carried out in the inlet and outlet and solid wall regions, The other parts of the grid are distributed gradually, and the surface mesh is divided by Quad (quadrilateral), which ensures the accuracy of calculation and meets the requirements of computational efficiency. The grid number is about 120 thousand. The surface and ground boundary conditions are set as wall, the narrow slot outlet and inlet are set as velocity inlet, and the flow field boundary is set as pressure outlet[4].

Table 1. The parameters of 1,2,3,4,5,6 model

model	Size (length mm × width mm × height mm)	α	V_1 (m/s)	V_2 (m/s)	k
1	666×330×85	11°	5.56	20	0.076
2	516×330×85	17°	5.56	20	0.076
3	446×330×85	23°	5.56	20	0.076
4	400×330×85	30°	5.56	20	0.076
5	446×330×85	23°	5.56	20	0.059
6	446×330×85	23°	5.56	20	0.091

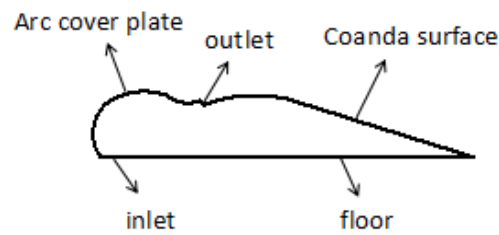


Figure 1. Physical model of ground dryer

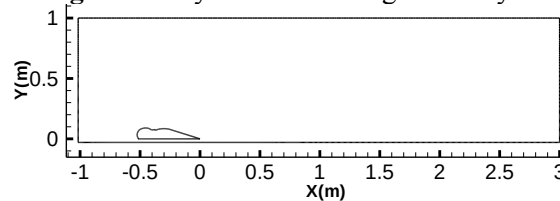


Figure 2. Computational domain

3.2. Simulation Results and Analysis

3.2.1 Effects of The Coanda's Surface'S Inclination Angle and the Length on the Formation of Air Film and Airflow Characteristics

The selection of the Coanda's surface's inclination angle and model length has an important effect on the performance of the ground dryer[5]. In this paper, the model 1, 2, 3 and 4 in Table 1 were simulated with the outlet size being 8.4 mm. The influences of the model change on the airflow field and the formation of ground air film were observed and investigated in this paper.

(1) Analysis of velocity field

Figure 3 is the flow chart for model 1, which is similar to Models 2, 3 and 4. The air flow goes into the ground dryer through the air inlet, then sprays out from the narrow outlet, flows on the Coanda surface, induces the surrounding air to flow to the ground area, and thus continues to drive the surrounding air to flow forward. This is because the air flows on the Coanda surface flow, forms a negative pressure in the vicinity of the surface, then the surrounding air continues to supply the negative pressure area, in order to achieve the induction of small air volume.

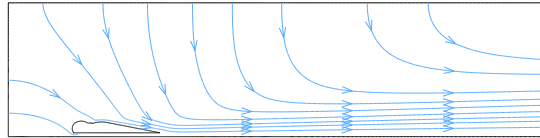
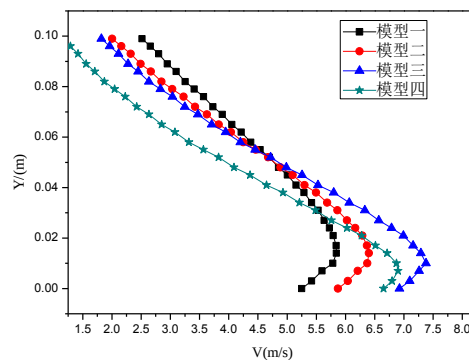


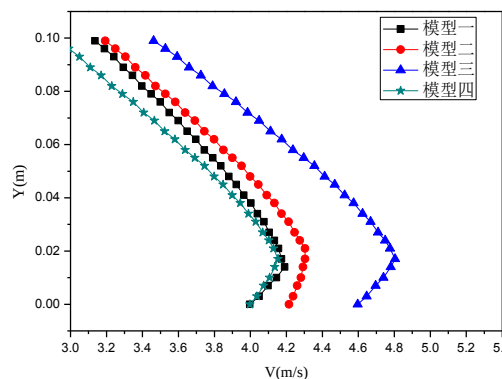
Figure 3. Model 1 streamline diagram

(2) Velocity analysis of equal cross section

The airflow that can effectively dehumidify the ground is the airflow near the ground. So the velocity distribution along the Y axis from the ground to the height of 0.1 m was analyzed. The distance in the positive direction along the X-axis from the ground to the tail of the dryer is set as d. The distance from the ground to the height of 0.1 m in the positive direction along the Y axis at d=0.33 m, 1.33 m and 2.33 m is shown in Figure 4. It can be seen from Figure 4 that when d=0.33 m, the velocity near the ground of model 3 is the largest, followed by model 4, model 2 and model 1. With the height increasing 0.06 m, the velocity of model 1 becomes the largest while the velocity of model 4 becomes the smallest. With the distance d increasing, the difference among the velocities of different models get more obvious, and the best velocity is ordered by model 3, 2, 1 and 4. Therefore, the ground air film of model 3 has the best effect.



(a) d=0.33m



(b) d=1.33m

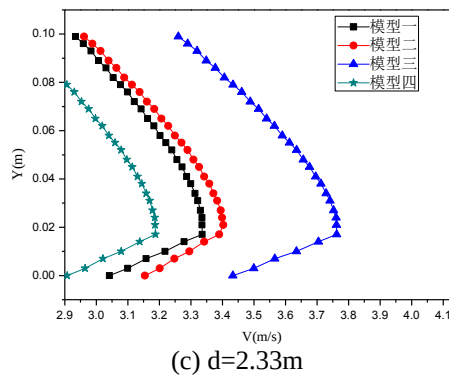


Figure 4. The velocity distribution of different models in different cross-sections along the positive Y-axis

3.2.2 Effects of Different Coanda Surface's Curvatures on the Formation of Air Film and the Characteristics of Airflow

In order to observe the influence of Coanda's surface curvature on the flow field characteristics of the ground dryer, the air film formation and air flow characteristics of model 5 ($k=0.059$), model 6 ($k=0.091$) and model 7 ($k=0.13$) were calculated based on model 3 with the curvature being 0.076.

(1) Analysis of velocity field

Figure 5 shows the near-ground velocity skew of the ground dryer with the Coanda surface's curvature being $k=0.076$, and models 5, 6, 7 are similar to that. Figure 6 is the local airflow field map of the ground dryer under different Coanda surface curvatures. From Figure 5 and Figure 6, it can be seen that the velocity attached to Coanda surface is relatively higher; that the velocity becomes larger when getting closer to the Coanda surface; and that the stratification of the velocity region is obvious, with no irregularity in the high and low velocity regions. It can be seen from Figure 6 that the velocity distribution of the Coanda surface near the ground is related to the curvature of the Coanda surface. The velocity region of 14-18 m/s becomes wider when the curvature increases from $k=0.059$ to $k=0.091$, and then the region is narrowed when increased to $k=0.13$.

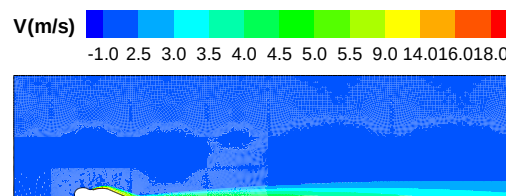
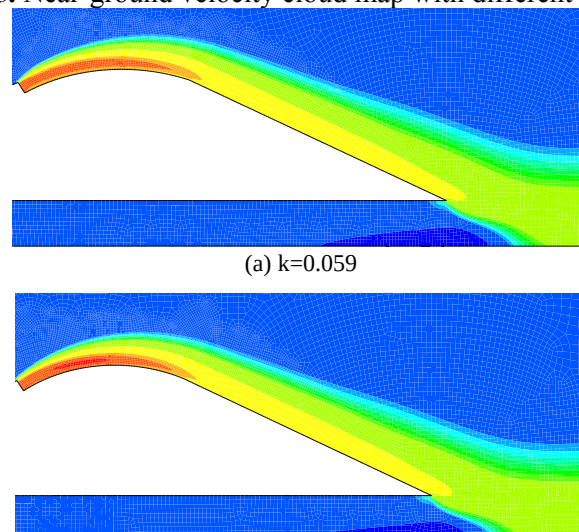


Figure 5. Near-ground velocity cloud map with different curvatures



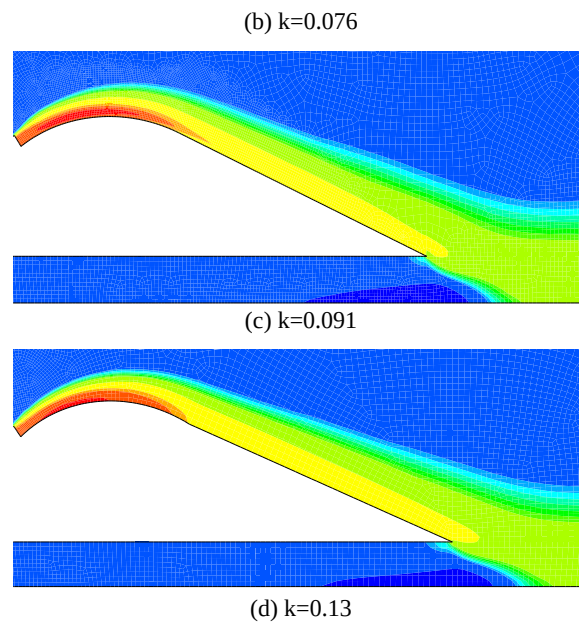
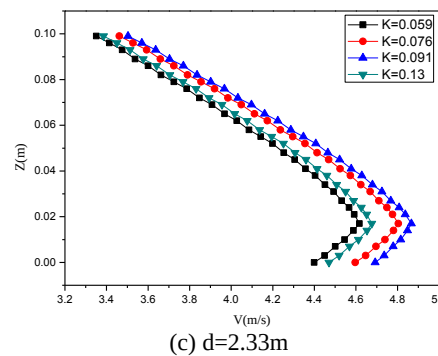
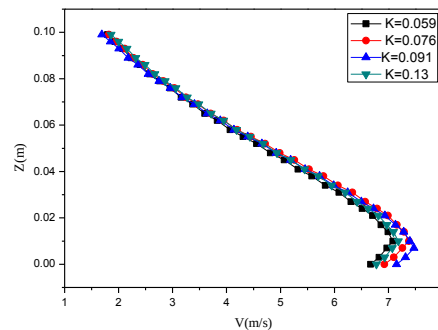


Figure 6. Enlarged local velocity cloud map with different surface curvatures

(2) Analysis of cross-section velocity

The airflow that can effectively dehumidify the ground is the airflow near the ground. So the velocity distribution along the positive Y axis from the ground to the height of 0.1 m was analyzed. Figure 7 shows the velocity distribution of the cross sections at $d=0.33$ m, 1.33 m and 2.33 m from the ground to the height of 0.1 m along the positive Y axis. It can be seen from Figure 7, no matter how the height or distance increases, the near-ground velocity of model 6 has the maximum velocity, and the ground air film of model 6 has the best effect.



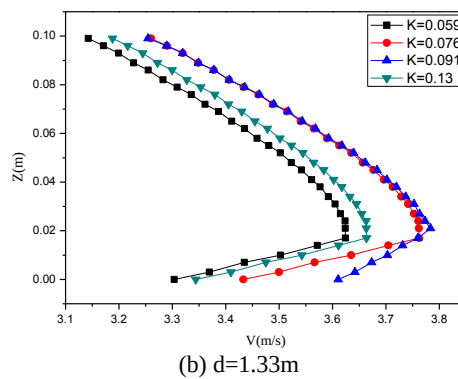


Figure 7. The velocity distribution of different cross sections along the positive Y-axis under different Coanda surface curvatures

4. Conclusion

The CFD technique was adopted to simulate the airflow characteristics of ground air film in a two-dimensional pattern. The air film was formed by ground dryer where the small air volume was used to induce the airflow to flow closely on the Coanda surface.

The two-dimensional numerical simulation results show that the change of the segment length, the inclination angle and the surface curvature of Coanda surface have great influence on the air film effect. As the segment length of the Coanda surface decreases and the inclination angle increases, the maximum velocity in the air film increases first and then decreases. In the four models, model 3 exhibits the best performance. With the increase of Coanda surface's curvature, the film effect becomes stronger and then weaker, and model 6 shows the best performance among the four models..

5. Reference

- [1] Charpin D 2007 Moulds in indoor environments and respiratory diseases. *J. Rev mal respire* **24** 246-247
- [2] Coanda H 1936 Device for deflecting a stream of elastic fluid projected into an elastic fluid. *J. U.S. Patent* **1936** 205-286
- [3] Guoq Li 2014 Influence of coanda surface curvature on performance of bladeless fan. *J. Journal of thermal science* **23** 422-431
- [4] Junyou Huang and Huaming Yang 2013 Research of lampblack machine optimization. *J. Journal of mechanical & electrical engineering* **11** 1388-1392
- [5] Shiyu Zhang, Yicheng Zhong and Pengzhe Fu 2011 Numerical investigation of dual-radius in circulation control airfoil. *J. Aircraft design* **4** 1-6