AI translation · View original & related papers at
chinarxiv.org/items/chinaxiv-202602.00009

Drag Reduction Design and Chamber Flow Field Characteristics of High

Authors: Chen Hongyong

Date: 2026-01-30T23:31:13+00:00

Abstract

Wind resistance power is an important basis for the drive power design of large boom-type geotechnical centrifuges, whose chamber flow field is characterized by with-the-flow development, temperature rise, and high flow velocity. To reduce aerodynamic drag, streamlining and drag-reduction designs must be applied to the rotating arm or the basket. For two types of centrifuges that respectively adopt rotating-arm streamlining and shroud-based drag reduction, CFD (computational fluid dynamics)-based aerodynamic simulations were carried out. By optimizing the simulation parameters, the internal flow-field characteristics of the chamber during high-speed operation and the aerodynamic power loss of the centrifuge were obtained, and the aerodynamic drag coefficient of the centrifuge shroud was optimized through profile design. The study shows that, when the high-speed centrifuge operates under low air pressure, the frictional torque on the lower wall of the chamber is dominant and exceeds that on the circumferential wall; when no shroud is used for the basket, flow separation occurs at the shoulder of the windward end face, leading to a relatively large pressure drag; after adopting a shroud, the separation at the leading-edge shoulder of the basket is weakened, and the total drag coefficient is significantly reduced. The results indicate that conducting streamlining design for high-speed centrifuges is necessary: by optimizing the outer shape of the rotating arm and the design of the shroud, wind resistance power can be effectively reduced, which is more conducive to controlling the temperature rise in the chamber.

Full Text

Study on Drag Reduction Design and Chamber Flow Field Characteristics of High-Speed Rotating-Arm Centrifuges

Chen Hongyong[1], Yang Xin[1], Li Qisheng[1], Gong Zhibin[1], Song Qiong[1], Li Xinyao[1], Yin Yihui[1]

[1] Institute of Systems Engineering, China Academy of Engineering Physics, Mianyang 621999, China
[2] Sichuan Key Laboratory of Impact and Vibration of Engineering Materials and Structures, Mianyang 621999, China

Abstract

Wind resistance power serves as a critical basis for designing the driving power of large-scale rotating-arm geotechnical centrifuges. The chamber flow field exhibits characteristics of co-rotation, temperature rise, and high flow velocity. To reduce aerodynamic drag, rectification and drag reduction designs must be implemented for the rotating arm or basket. This study investigates the aerodynamic characteristics of two centrifuge configurations—one employing arm fairing and the other using basket fairing—through computational fluid dynamics (CFD) simulations. By optimizing simulation parameters, the internal flow field characteristics and wind resistance power at high operating speeds were obtained. The aerodynamic drag coefficient of the centrifuge fairing was further optimized through profile design. The results demonstrate that for high-speed centrifuges operating at low pressure, the frictional resistance moment on the lower chamber wall dominates, exceeding that on the circumferential wall. Without fairing, flow separation occurs at the shoulder of the windward face of the basket, creating substantial pressure drag. With fairing installation, separation at the basket’ s leading-edge shoulder is significantly weakened, and the total drag coefficient is substantially reduced. These findings confirm the necessity of rectification design for high-speed centrifuges. Through rotating arm shape optimization and fairing design, wind resistance power can be effectively reduced, which is more beneficial for controlling chamber temperature rise.

Keywords: rotating-arm centrifuge; high-speed rotation; flow field; fairing

1. CFD Analysis Method for Centrifuge Chamber Flow Field

Common estimation methods for centrifuge wind resistance power include semi-analytical approaches based on linear flow field distribution assumptions. Yin Yihui et al. derived the conversion relationship between driving power and wind resistance heat generation from moment equilibrium conditions by dividing the moment between air and the centrifuge arm into windward, leeward, and backwind components. Wang Yongzhi et al. established simplified models and calculation methods for large-scale geotechnical centrifuge wind resistance power, validating them against measured data from two centrifuges. Hao Yu et al. performed CFD analysis on a medium-low speed geotechnical centrifuge, achieving less than 10% deviation between simulated and measured wind resistance power. Guo Yinan et al. conducted numerical modeling of the flow field and temperature field for the ZJU400 geotechnical centrifuge, comparing results with experiments. However, these studies focused on relatively low rotational speeds and did not require special computational strategies.

For high-speed centrifuges, the flow field around the basket and rotating arm exhibits turbulence and large-scale separation phenomena. The physical model characterizing fluid motion consists of the continuity equation, momentum equation, and energy equation. Due to the high Reynolds number, direct simulation of turbulent fluctuations would be computationally prohibitive. Therefore, turbulence models are employed to account for turbulent effects. The governing equations for the centrifuge flow field are the Reynolds-averaged Navier-Stokes (RANS) equations, which include the time-averaged continuity equation, momentum equation, and scalar transport equation. Compared with standard equations, the RANS formulation contains an additional Reynolds stress term.

The simulation of the chamber flow field essentially involves solving the compressible ideal gas equations. The rotational speed is typically high, and the flow field around the centrifuge basket and arm frequently exhibits turbulence and large-scale separation. Turbulent flow velocity and pressure exhibit temporal and spatial fluctuations. In the RANS equations, fluctuating terms are introduced to represent turbulent motion.

CFD analysis methods for centrifuge chamber aerodynamic characteristics include the rotating reference frame method and dynamic mesh method. The rotating reference frame method provides a steady-state approximation solution for each cell at a given angular velocity, representing the time-averaged flow field after full development. To obtain the instantaneous flow field at different times, sliding mesh or dynamic mesh methods must be employed for unsteady calculations.

For high-speed centrifuges, the basket tip linear velocity often exceeds 200 m/s, necessitating a compressible gas model. A specialized computational strategy was developed: first, the Multiple Reference Frame (MRF) method is applied at lower rotational speeds until the flow field converges, then the speed is gradually increased to the target value. Subsequently, the steady-state flow field obtained from MRF is used as the initial condition for sliding mesh calculations to capture unsteady flow field characteristics. This approach avoids large axial pressure gradients that occur during direct sliding mesh initialization and improves computational stability.

Using the sliding mesh technique, the chamber flow field is divided into a rotating region near the centrifuge and a stationary region near the chamber wall. Both regions are concentric circular domains centered on the rotation axis, with a grid interface formed between them. The rotating and stationary domains rotate relative to each other along this interface, where numerical data exchange occurs.

Table 1: Calculation Methods and Parameter Settings

Method/Parameter	Setting
Equation Solver	Density-based implicit solver
Rotating Flow Algorithm	Multiple Reference Frame (MRF) + Sliding Mesh
Turbulence Model	k-ω SST with Scalable Wall Functions
Spatial Discretization	Second-order upwind for flow, QUICK for turbulence
Gas Model	Compressible ideal gas
Rotational Speed	70 rad/s
Wall Conditions	Adiabatic for centrifuge and lower wall, isothermal (300 K) for circumferential wall
Boundary Layer Grid	$y^+ = 20$, thickness 5 mm, first layer 0.01 mm, 12 layers
Total Grid Count	1.2\$×10^{6}\$

---

2. Flow Field Analysis of High-Speed Centrifuge with Arm Fairing

Based on the aforementioned simulation methodology, flow field simulations were conducted for a high-speed centrifuge with arm fairing. Due to excessively high g-values and operational speeds, this centrifuge could not adopt an additional fairing approach due to strength and connection stiffness concerns. Therefore, the rotating arm itself was designed with an aerodynamic shape for drag reduction.

The centrifuge arm has an outer radius of 3.5 m, width of 0.8 m, and height of 1.2 m, operating at 70 rad/s. To prevent excessive temperature rise, the chamber initial pressure was set to 3 kPa. The influence of chamber wall roughness was considered in the simulations.

Figure 1 shows the model of the high-speed centrifuge with arm fairing. The pressure distribution along the radial direction is illustrated in **Figure 2**, while **Figure 3** presents the pressure distribution curve. **Figure 4** displays the pressure distribution nephogram on the circumferential wall, and **Figure 5** shows the schematic of angular positions. The pressure distribution at different heights on the circumferential wall is shown in **Figure 6**.

At the chamber center, the pressure is 2.85 kPa, increasing to approximately 3.65 kPa near the chamber wall, demonstrating a radial pressure gradient. The pressure distribution pattern is consistent with results from Guo Yinan et al. The pressure difference between the zenith and nadir directions is minimal, though the region between the windward side and chamber wall exhibits slightly higher pressure.

The velocity distribution nephogram under 3 kPa low-pressure conditions is shown in **Figure 7**, with the radial velocity distribution curve presented in **Figure 8**. Similar to the pressure distribution, velocity increases gradually along the radial direction, with maximum velocities occurring near the fairing outer edge and wake region. The maximum velocity reaches approximately 310 m/s.

Under these conditions, the wind resistance power of the high-speed centrifuge is about 134.2 kW. The aerodynamic moments on the centrifuge and chamber walls are detailed in **Table 2**. The moment on the lower chamber wall is 1.1195 kN·m, while the circumferential wall moment is 0.762 kN·m, indicating that lower wall friction dominates, being approximately 1.5 times that of the circumferential wall.

Table 2: Aerodynamic Forces and Wind Resistance Power Under 3 kPa Pressure

Component	Moment (kN · m)
High-speed centrifuge	1.1195
Lower chamber wall	0.762
Circumferential chamber wall	-

---

3. Aerodynamic Fairing Design

For another high-speed, heavy-load integrated large centrifuge with a centrifugal acceleration of 400g and rotational speed of 16.7 rad/s, a basket fairing design was developed. The maximum incoming flow velocity at the basket is about 200 m/s, placing it in the low subsonic regime. Therefore, airfoil leading-edge arcs were adopted for the fairing’ s windward and leeward surfaces.

The fairing profile employs a Bernstein polynomial expression:

$$\psi(t) = \sum_{i=0}^{n} \binom{n}{i} (1-t)^{n-i} t^{i} \eta_i$$

where ψ and η are airfoil coordinates, and the coefficients represent airfoil shape parameters.

Based on the basket geometry, a fully enclosed fairing approach was used. The side fairing aerodynamic shape is symmetric along the zenith-nadir direction, with the side surface closely aligned with the co-rotating airflow direction. **Figure 9** illustrates the aerodynamic design of the fairing shape.

---

4. Aerodynamic Drag Analysis of Fairing

To analyze the drag reduction effectiveness, the aerodynamic performance of the basket was compared for both faired and unfaired configurations. **Figure 10** shows the pressure distribution contour and velocity isolines for the unfaired basket cross-section. When airflow passes the basket, flow separation occurs at the shoulder of the windward face, with large-scale separation also appearing on the leeward side, creating substantial pressure drag.

Figure 11 presents the pressure distribution on the centrifuge arm and fairing surfaces. **Figure 12** shows the pressure and velocity contours for the circumferential basket cross-section after fairing installation. With the fairing, separation at the basket' s leading-edge shoulder is weakened, and no large-scale separation occurs on the leeward side.

Table 3 compares the drag coefficients before and after fairing installation. Although the viscous drag coefficient increases slightly with fairing, the pressure drag coefficient is significantly reduced, resulting in a substantial decrease in the total drag coefficient. Consequently, fairing design for high-speed centrifuge baskets can greatly reduce operational wind resistance power, benefiting temperature control design.

Table 3: Aerodynamic Coefficient Comparison for Centrifuge Basket

Configuration	Pressure Drag Coefficient	Viscous Drag Coefficient	Total Drag Coefficient
Without fairing	[value]	[value]	[value]
With fairing	[value]	[value]	[value]

---

5. Conclusions

This study conducted flow field characteristic simulations for two types of high-speed centrifuges with arm fairing and basket fairing drag reduction designs. The key findings are:

1. Chamber pressure increases radially outward, with minimal variation along the zenith-nadir direction at the same radial position. The region between the windward side and chamber wall shows slightly elevated pressure.

2. Velocity distribution follows a similar radially increasing pattern as pressure. Maximum velocities occur at the basket outer edge and wake region.
3. For high-speed centrifuges operating at low pressure, frictional resistance moment on the lower chamber wall dominates, exceeding that on the circumferential wall.
4. Without fairing, flow separation occurs at the windward face shoulder of the basket, with large-scale separation on the leeward side, resulting in high pressure drag.
5. With fairing installation, separation at the basket’ s leading-edge shoulder is weakened, and the total drag coefficient is substantially reduced.

These results demonstrate that rectification design is essential for high-speed centrifuges. Through rotating arm shape design and fairing optimization, wind resistance power can be effectively reduced, which is more beneficial for controlling chamber temperature rise.

References

[1] Chen Yuhu, Zhao Zhenyu, Liu Ming. Overview of geotechnical centrifuge development[J]. Equipment Environmental Engineering, 2015, 12(5): 19-27 (in Chinese).

[2] Wang Ming. New progress in development of geotechnical centrifuges and specialized test devices in China[J]. Journal of Changjiang River Scientific Research Institute, 2012, 29(4): 80-84 (in Chinese).

[3] Li Zhenhua, Wang Yuzhen. Review of geotechnical dynamic centrifugal model test development[J]. Chinese Journal of Mechanics and Engineering, 2006, 25(S2): 4026-4033 (in Chinese).

[4] Zhou Yifeng, Zhang Yuhu. Application of geotechnical centrifuge at Changjiang River Scientific Research Institute[J]. Journal of Changjiang River Scientific Research Institute, 2011, 28(10): 141-147 (in Chinese).

[5] Dou Yikang. Development of geotechnical centrifuge 400g-t and its shaking table performance assessment[J]. Chinese Journal of Geotechnical Engineering, 2011, 33(12): 1887-1894 (in Chinese).

[6] Yin Yihui, Chen Shaorong, Wang Xiaojun. Aerodynamic power of geotechnical centrifuges with ventilated chamber[J]. Journal of Mianyang Teachers’ College, 2010, 29(5): 1-5 (in Chinese).

[7] Wang Yongzhi, Zhuo Shi, Li Yuanheng. Review of development technique for scientific test centrifuges[J]. Equipment Environmental Engineering, 2015, 12(5): (in Chinese).

Figure 1

Figure 1: Figure 1

Figure 7

Figure 2: Figure 7

[8] Yin Yihui, Chen Shaorong, Wang Xiaojun. Aerodynamic power of geotechnical centrifuges with closed chamber[J]. Journal of Mianyang Teachers' College, 2010, 29(2): 1-5 (in Chinese).

[9] Huang Peng, Yin Yihui, Zhang Shunli. Simplified calculation method for steady-state resistance power of geotechnical centrifuge and cooling design optimization[J]. Earthquake Engineering and Engineering Dynamics, 2014, 34(S1): 909-914 (in Chinese).

[10] Wang Yongzhi, Zhuo Shi, Li Long. Simplified model and calculation method for wind resistance power of large geotechnical centrifuge[J]. Chinese Journal of Geotechnical Engineering, 2011, 33(12): (in Chinese).

[11] Hao Yu, Yin Yihui, Zhang Qiang. Analysis of wind resistance power and starting process of ZJU400 geotechnical centrifuge[J]. Equipment Environmental Engineering, 2015, 12(5): (in Chinese).

[12] Guo Yinan, Zhang Yalin. CFD simulation of flow field and temperature field for ZJU400 geotechnical centrifuge[J]. Equipment Environmental Engineering, 2018, 15(2): 52-56 (in Chinese).

[13] Guo Yinan, Zhang Yalin. Simulation method for wind resistance and flow field characteristics of geotechnical centrifuges based on CFD[J]. Equipment Environmental Engineering, 2020, 17(11): 85-89 (in Chinese).

[14] Cheng Yonghui, Zhang Qingyun, Liu Xibao. Wind resistance optimization of precision centrifuge based on CFD[J]. Journal of Engineering Design, 2014, 21(6): 572-577 (in Chinese).

Figures

Source: ChinaXiv —Machine translation. Verify with original.

Figure 9

Figure 3: Figure 9

Figure 11

Figure 4: Figure 11