Numerical investigation for coupled rotor/ship flowfield using two models based on the momentum source method

Zhao Dingxuan*a, Yang Haojiea, Yao Shuangji b and Ni Tao b

aSchool of Mechanical Engineering, Yanshan University, Qinhuangdao, People’s Republic of China; bSchool of Vehicles and Energy, Yanshan University, Qinhuangdao, People’s Republic of China

ABSTRACT
The coupled rotor/ship flowfield is investigated using the Navier–Stokes solver when the helicopter rotor hovers in the ship’s airwake. The solid rotor model (SRM) and virtual rotor model (VRM) are employed, considering the different types of rotor simulation by the momentum source method. Qualitative and quantitative rotor/ship coupled flowfield analyses are conducted when the rotor hovers at different locations and with different wind-over-deck conditions. The results show that both models effectively reflect the different vortex interactions and velocity distributions in the rotor/ship coupled flowfield, and the discrepancy of velocity magnitudes between the SRM and VRM is kept within 1 m/s under most conditions. Moreover, a comparison is made of computational costs between the SRM with the moving overset mesh method and the VRM during a helicopter vertical shipboard landing. It is shown that the computational time of the VRM is smoother than the SRM by about 10 times. Considering the computational cost, it is more appropriate to use the VRM to conduct coupled flowfield studies in the helicopter/ship dynamic interface.

ARTICLE HISTORY
Received 23 March 2021
Accepted 25 October 2021

KEYWORDS
Helicopter; coupled flowfield; momentum source method; ship airwake; rotor model

1. Introduction
Helicopter shipboard landing and launch are extremely dangerous and challenging tasks for helicopter pilots. This is not only because of the complex recirculation flow regime (Oruc, Horn, et al., 2016) formed by the slender ship and narrow flight deck space, but also because of the strong-vortex coupled flowfield formed by the rotor downwash and the ship airwake, which adversely causes an imbalance in the rotor forces (Crozon et al., 2018). Furthermore, the large velocity gradient and chaotic flow direction in the coupled flowfield increase the operational difficulty for helicopter pilots (Kaeaeiae et al., 2012; Lee & Horn, 2004; Lee et al., 2003). Therefore, to ensure the safe take-off and landing of shipboard helicopters, it is of great significance to study the rotor/ship coupled flowfield.

In the early years of the twenty-first century, many researchers carried out simple isolated ship flowfield studies (Hodge et al., 2009; Sezer-Uzol et al., 2005; Sharma & Long, 2001). The inviscid flow results under both steady and time-accurate solutions have been thoroughly analyzed (Sezer-Uzol et al., 2005; Sharma & Long, 2001), and the inviscid unsteady flowfield result has been incorporated into a flight simulator to study the disturbance effect of the ship wake on maritime helicopters (Hodge et al., 2009). Although the main flow characteristics have been captured, the turbulent flow details have been lost in ignoring the viscous effects of the ship flowfield. Then, considering the complex structures of the ship flowfield in both steady and unsteady cases, some papers use various methods to analyze the velocity and vortex distributions. Zhang et al. (2009) employed the Cobalt solver to study the unsteady flows over the simplified frigate shape 2 (SFS2) ship model under three different wind-over-deck (WOD) conditions. To capture the large vortex flows over the ship model, a one-equation Spalart–Allmaras turbulence model was employed. Based on two different ship models, Thornber et al. (2010) used high-order large eddy simulation (LES) to capture the time-accurate flow data under 14 WOD angles, although the calculation of the high-order turbulence model needs more time. Forrest and Owen (2010) used detached-eddy simulation (DES) to simulate the flows through the SFS2 ship, and compared the results with wind tunnel data under different WOD angles. The simulated results were consistent with the experimental data, and could capture large turbulent structures. To obtain a more accurate air vortex structure, Weixing et al. (2018) used delayed detached eddy simulation (DDES) to analyze the flowfield over the SFS2 ship, and...
the computed unsteady airwake data were in agreement with the experimental data. Although the LES or DES method can obtain more flow details, the cost of calculation is also higher. Compared with LES, DES or other methods, the Reynolds-averaged Navier–Stokes (RANS) solver with a k-ω turbulence model has higher computational efficiency and can also achieve satisfactory flow simulation results (Ghalandari et al., 2019; Mosavi et al., 2019; Muijden et al., 2013; Polsky, 2002). Therefore, RANS with a k-ω turbulence model has often been used to calculate the steady and unsteady ship airwake flowfield (Chen et al., 2020; Su, Shi, et al., 2019; Su, Xu, et al., 2019).

When analyzing the coupled helicopter/ship flowfield, because of the large computational domain and severe coupling between the rotor downwash and ship airwake, the simulation of the helicopter/ship coupled flowfield has a high computational cost. To simplify helicopter/ship coupled flowfield calculation, through analysis, Su et al. (2017) found that the fuselage has little influence on main characteristics of helicopter/ship coupled flowfield, so it is reasonable that only the main rotor downwash and ship airwake are considered in helicopter/ship coupled flowfield simulation (Shi et al., 2017; Su, Xu, et al., 2019).

For rotor/ship coupled flowfield analysis, the momentum source method is commonly used in the literature. Shi et al. (2017) compared the vorticity and velocity of a rotor/ship coupled flowfield between a steady rotor model based on the momentum source method and an unsteady rotor model with the moving overset mesh method. The result showed that the main characteristics of the shedding vortex from the rotors are similar and both models reflect the flowfield coupling phenomena well. To analyze the operational behaviors for a helicopter pilot during a shipboard landing when there are two helicopters in the coupled flowfield, Shi et al. (2020) developed a numerical method based on the DES and momentum source method, and the results showed that a large vortex in the headwind condition is the main factor affecting the landing operation. Under unsteady cases, Su, Xu, et al. (2019) used the moving overset mesh method and momentum source method to analyze the rotor loads during a coaxial-rotor helicopter vertical landing on the SFS2 ship. The result suggests that the collective control input and differential collective pitch are the two main reasons for stabilizing the heading of the helicopter, and the moving overset mesh method provides a good solution for rotor simulation under unsteady simulation. Based on the momentum source method, Oruc et al. (2015) developed a virtual fully coupled helicopter/ship dynamic interface using the actuator disk model (ADM), and then a comparison was made with the one-way coupled helicopter shipboard simulation. The results showed that the fully coupled simulation could reasonably capture the trailing vortices generated from the main rotor and the aerodynamic coupling effect of the main rotor downwash and ship airwake. In this paper, the rotor forces are distributed spatially with a one-dimensional Gaussian distribution and then the forces are interpolated into the grids using source terms. This has been shown to help in stabilizing and speeding up the numerical calculations. This method was also used in a real-time helicopter shipboard flight simulation and showed good results in reducing computational costs (Oruc et al., 2016).

As mentioned above, the momentum source method is widely used in rotor/ship coupled flowfield analysis, and there are different ways to add source items to the computational domain. Although the applications for the momentum source method have varied forms, the discrepancies between the different momentum source models in simulating the rotor/ship coupled flowfield and saving computational cost are not clear. These problems need further study.

In this paper, the solid rotor model (SRM) and virtual rotor model (VRM) are employed based on the momentum source method, and the numerical models are validated with experimental data. Then, using the SRM and VRM, the rotor/ship coupled flowfield is investigated during a helicopter shipboard landing, and the qualitative and quantitative comparison of velocity distribution is carried out under different WOD angles and at different locations on the deck. The results show that both models can capture a similar velocity distribution, and the velocity difference is kept within 1 m/s in most cases. To further understand the differences between the two models, a computational cost analysis during helicopter vertical shipboard landing is conducted. It is shown that the VRM is 10 times smoother than the SRM in computational efficiency.

2. Numerical methodology

2.1. Computational fluid dynamics (CFD) calculation methods

The commercial CFD solver FLUENT is used to perform the rotor/ship coupled flowfield calculations. In simulation, the uniform velocity inlet boundary condition is used to describe the rotor/ship flowfield environment. The location of the ship remains constant relative to the computational domain, and the three-dimensional (3D) unsteady RANS equations with source terms are used as
governing equations, the integral forms of which can be written as

$$\frac{\partial}{\partial t} \int_V \int W \, dV + \int_S (F_C - F_V) \cdot ndS = \int_V S \, dV$$

(1)

$$W = \begin{bmatrix} \rho & \rho u & \rho v & \rho w & \rho E \end{bmatrix}^T$$

$$F_c = \begin{bmatrix} \rho(q - q_b) \\ \rho u(q - q_b) + nxp \\ \rho v(q - q_b) + nyp \\ \rho w(q - q_b) + nzp \\ \rho H(q - q_b) + pq_b \\ 0 \\ n_x \tau_{xx} + n_y \tau_{xy} + n_z \tau_{xz} \\ n_x \tau_{yx} + n_y \tau_{yy} + n_z \tau_{yz} \\ n_x \tau_{zx} + n_y \tau_{zy} + n_z \tau_{zz} \\ n_x \Phi_x + n_y \Phi_y + n_z \Phi_z \end{bmatrix}$$

$$F_v = \begin{bmatrix} 0 \\ S_x \\ S_y \\ S_z \\ 0 \end{bmatrix}$$

$$S = \begin{bmatrix} C_l \, d\psi (\frac{-F}{2\pi}) \end{bmatrix}$$

(2)

where \( W \) is the conservative vector, \( F_c \) and \( F_v \) are the convective and viscous vector, respectively, \( S \) is the source term vector and \( n = [nx, ny, nz]^T \) is the unit vector.

Considering that the ship airwake has the characteristics of low velocity and high Reynolds number, the ship airwake is regarded as a 3D incompressible fluid in this paper, and the 3D double-precision pressure-based solver is adopted. Polsky (2002) used a two-equation shear stress transport (SST) \( k-\omega \) turbulence model to study the unsteady ship airwake, which shows that the SST \( k-\omega \) turbulence model can effectively capture the unsteady phenomena of the flowfield. Thus, the SST \( k-\omega \) turbulence model is used in this paper (Menter, 1994). To speed up the simulation process, the SIMPLEC pressure–velocity segregated algorithm (Doormaal & Raithby, 1984) with the second-order spatial discrete scheme is adopted. The Courant–Friedrichs–Lewy (CFL) number is set to 5 in unsteady cases.

2.2. Momentum source method for the SRM and VRM

The momentum source method is a simplified modeling solution for the rotor in the process of helicopter shipboard landing. In this method, the rotor is regarded as an infinitely thin actuator disk model, and the classical blade element theory is used to calculate the aerodynamic forces of the rotor. According to the blade element theory (John & Simon, 1990/2002), the calculation of lift force \((dL)\) and drag force \((dD)\) can be written as

$$\begin{align*}
    dL &= \frac{1}{2} \rho V^2 C_l \, d\psi \\
    dD &= \frac{1}{2} \rho V^2 C_d \, d\psi
\end{align*}$$

(2)

where \( \rho \) is the density of the fluid, \( V \) is the total inflow velocity at rotor radius \( r \), and \( C_l \) and \( C_d \) are the lift coefficient and drag coefficient of the airfoil, respectively.

If the total aerodynamic force acting on the blade element is recorded as \( dF \), then according to Newton’s third law, the force of the blade segment on the airflow can be written as \(-dF\). The source term vector \( S \) in Equation (1) can be written as

$$S = N_b \, d\psi (-F) / 2\pi$$

(3)

where \( N_b \) is the number of rotor blades and \( \psi \) is the blade azimuth angle. The SRM and VRM are developed later based on the momentum source method discussed above; the distinction between them is made according to the different processing of the source terms.

In the SRM, the source terms need to be loaded in a prebuilt surface mesh at the plane of the rotor (Figure 1a). In other words, the SRM needs to generate a surface mesh at the rotor position in advance to represent the rotor disk, and at least one layer of volume grid is needed in the region adjacent to the rotor disk to load the momentum source terms. In contrast, the VRM does not need to generate the surface mesh at the rotor position in advance; it needs only to load the source terms into corresponding mesh points (Figure 1b). Because of the existence of a solid rotor disk, it is more convenient for the SRM to add source items in the process of execution, which is also the reason why SRM is widely for modeling rotors (Kim & Park, 2013; Li et al., 2020; Stalewski & Surmacz, 2019). However, when using the SRM to simulate the dynamic shipboard landing of a helicopter, it is necessary to use the dynamic grid to update the position of the rotor disk in real time. In contrast, the VRM only adds source terms to the virtual mesh points, so it does not need to regenerate the grid when the rotor position changes. At the same time, the location where the source term is loaded becomes difficult to determine, and a grid with enough density is required to simulate the effect of the helicopter rotor on the surrounding airflow.

2.3. Validation of numerical methods

Since experimental data on the rotor/ship coupled flowfield are unavailable, the isolated ship flowfield and rotor/fuselage aerodynamic interaction are simulated to demonstrate the proposed numerical methods.
2.3.1. Isolated ship simulation

The SFS2 ship is widely used by researchers owing to its representative shape and available experimental data. The National Research Council of Canada (NRC) has investigated the SFS2 ship using wind tunnel tests, and the wind tunnel data can be obtained by referring to Zhang et al. (2009). To simulate the ship airwake more accurately, the inlet velocity is set to 15 m/s at 0° WOD angle, and four slices are plotted to show the distribution of streamwise velocity along the flight deck (Figure 2). As shown in Figure 2, the airstream flows through the ship and forms a low-pressure zone behind the hangar, and the recirculation flow regime is formed behind the hangar. Thus, the streamwise velocity of the airwake along the flight deck is greatly reduced, and gradually increases along the sides of the ship. All these phenomena are reflected by the experimental and CFD results (Figure 2).

To determine the appropriate grid size for the SFS2 ship airwake simulation, the corresponding coarse grid, basic grid and refined grid are used, with grid sizes of $4.6 \times 10^6$, $7.5 \times 10^6$ and $9.5 \times 10^6$, respectively. As shown in Figure 3, at the location of 50% deck length and 10.66 m above the flight deck, a quantitative comparison of streamwise velocity between the experimental data and

![Figure 1. Schemes of two different rotor models: (a) solid rotor model (SRM); (b) virtual rotor model (VRM).](image1)

![Figure 2. Streamwise velocity distributions over the SFS2 flight deck.](image2)
CFD simulation results has been conducted. The resulting discrepancies among the models with three different grid sizes are negligible; all of the models can reasonably reflect the variation in ship airwake velocity, and no significant improvement can be observed in the refined mesh model. This similarity can also be seen in Figure 4. Therefore, from the perspective of grid independence and computational cost, the basic grid is used in the ship airwake simulation.

### 2.3.2. Rotor/fuselage aerodynamic interaction calculation

A simple rotor/fuselage model is adopted to investigate the ability of the SRM and VRM to simulate the aerodynamic interaction. The original experiments were conducted by researchers at Georgia Institute of Technology (Nam et al., 2006). The rotor/fuselage model consists of a simplified cylindrical fuselage and a two-bladed flutering rotor, which has an airfoil of NACA0015 and a 6° pitch angle. In the process of simulation, the rotor is kept at an advance ratio of 0.1, with a blade tip Mach number of 0.295.

In Figure 5, three vertical slices have been used to investigate the vorticity distribution along the center axis of the fuselage. It is shown that when the rotor downwash
collides with the fuselage surface, the flow is blocked and a high-pressure zone is formed on the fuselage surface, and because of the tilt of the rotor shaft, the part at the back of the fuselage is subjected to more intense vortex disturbance. It can be seen that the fuselage and the rotor downwash are strongly coupled. As shown in Figure 5(a,b), the main characteristics of the aerodynamic interaction phenomena are captured effectively by both models. Nevertheless, for the deflection of the rotor shaft, the loading position of the source terms is difficult to determine, and fewer flow details are presented in the VRM. The pressure distribution at the top and port side of the fuselage is calculated in Figure 6. As can be seen, the pressure distribution of both models can be reasonably captured along the fuselage. However, in Figure 6(a), the pressure distribution at the front of the fuselage is slightly different from the experimental data, which may be due to the inaccuracy of the experimental measurement device and the in-house principal error of the momentum source method. This is also reflected in Shi et al. (2017).

3. Numerical simulation and results

3.1. Numerical simulation set-up

For more accurate analysis of the coupling phenomenon between the helicopter and the ship, a 1/2 landing platform dock-17 (LPD-17) ship with a modified flight deck is adopted as the landing environment for the helicopter (Sezer-Uzol et al., 2005; Sharma & Long, 2001) (Figure 7). The dimensions of 1/2 LPD-17 are as follows: the ship length is 100 m, the ship width is 16 m, the hangar height is 8 m, the deck length is 32 m, the deck height is 5 m and the whole height of the ship is 25 m. A previous study showed that the influence of the fuselage on the main
characteristics of the helicopter/ship coupled flowfield is small, and the rotor/ship combined group can be used in helicopter shipboard landing simulations to save time (Su et al., 2017). This solution has also been adopted by Shi et al. (2017) and Su, Xu, et al. (2019). Thus, to simulate the interaction phenomena of the helicopter/ship flowfield, the main rotor of a ‘Dolphin’ helicopter is considered. The main parameters of the rotor are as follows: the rotor radius is 5.965 m with four blades, the rotor chord is 0.385 m, the blade collective angle is 12.82°, the blade twist angle is −10° and the rotor speed is 36.6 rad/s.

To ensure the full development of the rotor/ship coupled flowfield, a \(10L \times 8L \times 6L\) (where \(L\) is the length of the ship) computational domain of the flowfield is created (Figure 8). The velocity inlet and pressure-outlet boundaries are adopted as the inlet and outlet boundaries, respectively. The ship and water are specified as no-slip and slip boundaries, respectively, and the other far-field boundaries are symmetric boundary conditions for ensuring that the coupled flowfield is fully developed. Since a uniform structural grid is needed to determine the center of the rotor hub and calculate the momentum source terms for VRM under unsteady cases, the hybrid mesh topology model is drawn using ICEM CFD software. A \(132 \times 100 \times 50\) m rectangular zone is covered by the structured grid, and the total number of grid cells is 8.9 million. Because of the existence of the rotor disk in the SRM, the mesh should be refined near the rotor disk model to adapt to changes in geometry, with the total number of grid cells being 8.9 million. The detailed structures of the hybrid grid are shown in Figure 9.

Using the SRM and VRM, the rotor/ship coupled flowfield is analyzed under different cases. To enable a comparison between the two models, the velocity and vorticity distributions are discussed when the rotor is located at 12, 9 and 6 m height relative to the center of the flight deck, and variation in the flowfield at the height of 9 m under 0° and 30° WOD angles is also analyzed.

### 3.2. Analysis of results

#### 3.2.1. Qualitative and quantitative analyses

Figure 10 shows the vorticity distributions at 0° WOD angle when only an isolated ship, or a rotor with the SRM or the VRM exists in the computational domain, where the rotor is located at 9 m above the flight deck. As shown in Figure 10(a), in the case where only the isolated ship exists in the flowfield, because of the complex structure of the ship, the separation of airflow occurs.
**Figure 9.** Topological structures of hybrid grid in computational domain: (a) solid rotor model (SRM); (b) virtual rotor model (VRM).

**Figure 10.** Contour maps of vorticity at 9 m and 0° wind-over-deck (WOD) angle, colored according to velocity magnitude: (a) isolated ship; (b) coupled flowfield with solid rotor model (SRM); (c) coupled flowfield with virtual rotor model (VRM).
near the hangar when the wind blows through the ship towers, and little vortex is distributed above the flight deck. However, when rotor downwash is coupled with the backward ship airwake in the SRM and VRM (Figure 10b,c), the distribution of the vortex changes dramatically. This is reflected in the fact that near the rotor, the vorticity increases greatly, and the existence of the ship airwake makes the rotor downwash move backwards. Conversely, the downwash of the rotor also intensifies the downward deflection of the vortex. This makes the vortex interactions more intense above the deck.

A detailed comparison of the vorticity distribution between SRM and VRM was undertaken. As shown in Figure 11(a) and (b), the velocity streamlines of the coupled flowfield deflect obviously when passing through the helicopter rotor, then the rotor downwash collides with the deck, deflects again and affects the recirculation flow regime behind the hangar. Moreover, the existence of the coupled flowfield accelerates the formation of the recirculation flow regime behind the hangar, and the tip vortices from the rotor blades are captured effectively by both models. Through comparison, it can be found that both the SRM and VRM capture the main characteristics of the rotor/ship flowfield. However, the flow details around the rotor downwash dominant region are not clear; in particular, the shedding vortex from the rotors has not been captured fully, and this is a limitation of the momentum source method (Shi et al., 2017).

To analyze the velocity distributions of the coupled flowfield with the rotor at different heights, Figures 12 and 13 present the velocity streamlines when the rotor is located 12 and 6 m above the deck, respectively. As shown in Figure 12, the separated flow from the ship tower interacts with the rotor downwash and aggravates the velocity variation above the deck. Compared with the coupled flowfield for the rotor at the height of 9 m above the deck (Figure 11), it can be seen that the coupling degree of the flowfield is lower, as the flowfield is less affected by the recirculation flow regime behind the hangar and the shipboard effect near the deck. However, as the rotor is moved closer to the deck (Figure 13), the downwash flow interacts with the flowfield above the deck. At this time, the
flowfield is affected to a greater extent by the separated flow from the ship surface, the rotor downwash and the shipboard effect. Compared with the flowfield when the rotor is 12 and 9 m above the deck, the recirculation flow regime is further compressed in Figure 13.

The vorticity contour slices of the coupled flowfield are shown in Figure 14, where the rotor is located at 9 m above the deck. Four slices are used above the deck to analyze the vorticity distribution, which are named s1, s2, s3 and s4, and their positions are shown in Figure 14(a). As shown in Figure 14(b,c), the downwash flow of the rotor flows downwards and interacts with the airflow on both sides of the deck, forming a strong vortex–vortex interaction near the rotor zone. At the same time, the
airflow above the deck interacts with the rotor downwash and moves to the rear of the deck, and then forms strong vortex–vortex interactions again on both sides of the deck. Comparing the two models, it can be found that, whether from the intensity of vortex distributions or the details of flow, both the SRM and the VRM can capture the interactions between rotor downwash and ship airwake, and obtain similar results. However, owing to the lack of information on the geometric characteristics of the blade, the details of the flowfield around the blades, for instance, the distribution of the shedding vortex, cannot be completely captured (Shi et al., 2017).

To further illustrate the differences between the SRM and VRM, quantitative analyses of the two models are conducted. In Figure 15, three planes are selected to analyze the velocity flowfield, where the rotor is located at the height of 9 m above the deck, and the corresponding monitoring points are also identified in this figure.
Figures 16–18 show the velocity distributions in different planes at 0° WOD angle. In plane A (Figure 16), the x-direction velocity along \( y = 4 \) m fluctuates more smoothly than along \( y = 0 \) m. This is because that the separated flows are developed when the ship airwake flows through the ship tower, and the existence of a low-pressure area behind the ship tower retards the flow of the flowfield. This is also reflected in plane B (Figure 17a), where the x-direction velocity along \( y = 0 \) m has similar characteristics. However, because the flowfield is increasingly influenced by rotor downwash, the velocity along \( y = 4 \) m fluctuates slightly. In plane C (Figure 18a), the x-direction velocities along both curves are close to 0 m/s at \( x = -14 \) m. This can be explained by the ship airwake flowing through the hangar and a low-pressure area forming behind the hangar, which is also called the recirculation flow regime (Oruc, Horn, et al., 2016). Then, the magnitudes of the x-direction velocities begin to increase under the influence of the rotor downwash flow. This phenomenon is also described in Shi et al. (2017). From the above analyses, it can be seen that both the SRM and VRM could accurately predict the changes in velocity in the x-direction and achieve similar results.

Compared with the x-direction velocity, the variations in the y-direction and z-direction velocities are more complicated. Because of the interaction of ship airwake and rotor downwash, as well as the symmetry of the flowfield, the y-direction velocities in planes A and B change sharply along \( y = 4 \) m, while the magnitudes of the y-direction velocities change slowly along \( y = 0 \) m (Figures 16b and 17b). Moreover, in plane C, since the rotor downwash cracks the deck surface and develops the shipboard effect above the deck, the coupled flowfield behind the rotor becomes further complicated. This is reflected in Figures 18(b) and 14, where the y-direction velocity changes irregularly and the flow direction is considerably deflected. In the comparison of the y-direction and z-direction velocities in the three monitoring planes, the SRM and VRM obtain similar results, and the discrepancy in the velocity magnitudes is less than 1 m/s in most areas. Even at \( x = -7 \) m in plane C, where the flowfield is dominated by the recirculation flow regime, the discrepancy of velocity magnitudes is less than 1.5 m/s (Figure 18b). Overall, the SRM and VRM achieve similar results in capturing the flow details of the rotor/ship coupled flowfield at 0° WOD angle.

To analyze the detailed discrepancy between the SRM and VRM under different WOD angles, the 30° WOD angle of the coupled flowfield is also investigated. Figure 19 shows the contour maps of vorticity at 30° WOD angle,
which are colored according to the velocity magnitudes. It can be seen that with the increase in the WOD angle, the coupled flowfield is further compressed. In particular, behind the hangar, because the separated flow from the ship tower deflects to the leeward side, the recirculation flow regime is further reduced and more susceptible to the downwash flow of the rotor. Moreover, as shown in Figure 20, there is more vorticity distributed on the windward side of the deck, which enlarges the low-pressure region on the leeward side of the deck. The larger divergence of the vorticity between the two sides of the rotor is not conducive to the smooth landing of the helicopter. Comparing the simulation results of the SRM and VRM in Figures 19 and 20, it can be found that SRM and VRM can effectively simulate the main characteristics of the coupled flowfield, and have similar simulation results.

To further compare the simulation results of the two models, Figure 21 shows the velocity distributions
in plane C at 30° WOD angle, where the ship airwake interacts with the deflected rotor downwash and forms a complicated flowfield. In particular, under the rotor, the downwash flow at the windward side of the rotor deflects and collides with the deck, forming a shipboard effect (Figure 20), so that the magnitude of the difference in $z$ velocity between the windward side and the leeward side of the rotor reaches the maximum (Figure 21c). It further demonstrates the difficulty for the helicopter to land on the ship smoothly at a large WOD angle. Nevertheless, it can be found that the magnitude of the difference in velocity is within 2 m/s, while in most places the difference remains within 1 m/s.
### Table 1. Comparison of computational costs of the solid rotor model (SRM) and virtual rotor model (VRM) under unsteady conditions.

| Rotor model | SRM          | VRM          |
|-------------|--------------|--------------|
| Grid magnitude | $9.7 \times 10^6$ | $8.9 \times 10^6$ |
| Time steps   | 91,000       | 82,000       |
| Computation time (h) | 1400         | 139          |

#### 3.2.2. Calculation cost analysis

To compare the computational efficiency of the SRM and VRM under unsteady conditions, the moving overset mesh technology and SRM are used to update the rotor position in the flowfield in real time, and the results are compared with those of the VRM. The simulation condition is that the helicopter hovers at a height of 12 m above the deck, then slowly descends at 1 m/s, and finally lands at a height of 4.5 m above the deck. Before the unsteady calculation, 7000 iterations are performed to speed up the simulation process.

As shown in Table 1, the calculation time steps of the SRM and VRM are 91,000 and 82,000, respectively. Compared with the SRM, smaller grid sizes are required for the VRM in simulating the helicopter shipboard landing, and the computing time of the VRM is much shorter than the SRM, by about 10 times. This is because there is no dynamic mesh motion problem for the VRM during the CFD simulation. Considering the calculation cost requirement of the helicopter/ship dynamic interface calculation, especially in the process of real-time simulation, the VRM is more conducive to saving the calculation cost than the SRM with the moving overset method. A near-real-time helicopter/ship dynamic interface based on the momentum source method has been developed by Oruc et al. (2016), which is similar to the VRM. In their method, a simplified ship model was established to capture the shedding wake, and the overall grid was controlled at 0.38 million. To speed up the calculation, the authors adopted a multiple-program, multiple-data message passing interface. Finally, a near-real-time fully coupled pilot-in-the-loop helicopter dynamic shipboard landing simulation has been realized. This is a meaningful attempt at a momentum source method in dynamic interface simulation, although the accuracy and real-time performance of the coupled flowfield simulation need to be improved.

In summary, the solution of the VRM is very meaningful in dynamic interface modeling and simulation, especially in fully coupled pilot-in-the-loop simulation with high real-time requirements. This will help the pilot to carry out shipboard landing operations more effectively.

#### 4. Conclusions

A numerical investigation on the rotor/ship coupled flowfield was conducted based on two different models of the momentum source method. The qualitative and quantitative comparisons of vorticity and velocity distributions for the two models were carried out under different WOD conditions and rotor locations. Finally, the computational efficiency of the two models was analyzed during helicopter vertical shipboard landing. The main research outcomes and constructive suggestions for landing operations are summarized as follows:

1. Qualitative analysis of the coupled flowfield shows that whether from the intensity of vortex–vortex interactions below the rotor or the flow details behind the hangar, the SRM and VRM could obtain similar results in terms of the vorticity and velocity distributions, and both could capture the main characteristics of the flowfield, except for the flow details around the rotor downwash dominant region. This is because, on the one hand, the simulation accuracy of the momentum source method itself is relatively low compared with the solid blades. On the other hand, the deflection of loading positions for source terms in the VRM will also reduce the simulation accuracy. This is also reflected in simulating rotor/fuselage aerodynamic interactions.

2. Through quantitative comparison, it is found that the discrepancy in the velocity magnitudes between the SRM and VRM are mostly kept within 1 m/s under different monitoring locations and WOD angle conditions. This shows that the differences in velocity between the two models are small. At a large WOD angle, the magnitude of the difference in $z$ velocity between the windward side and the leeward side of the rotor is further enlarged. The enlarged velocity difference in the $z$-direction on both sides of the rotor is not conducive to helicopter shipboard landing.

3. The computational costs were compared using the two models during helicopter vertical shipboard landing. Combined with the overset mesh method, the SRM takes about 10 times more computing time than the VRM. It can be seen that the VRM is more suitable for computational work that emphasizes efficiency, especially in a pilot-in-the-loop real-time dynamic interface simulation. This will help the pilot to carry out shipboard landing operations more effectively.

In this paper, the uniform velocity inlet boundary condition was used to simulate the rotor/ship coupled
flowfield. To reflect the aerodynamic interactions of the rotor/ship flowfield more realistically, the atmospheric boundary layer conditions should be considered in future work; for instance, the uniform velocity profile used in this study could be changed to a power law velocity profile.

Disclosure statement
No potential conflict of interest was reported by the authors.

Funding
This work was supported by the Foundation for Innovative Research Groups of the Natural Science Foundation of Hebei Province [grant number E2020203174], the Natural Science Foundation of Hebei Province titled with â€œResearch on Multi-Field Coupling Dynamics Modeling for Rescue Helicopter Simulatorâ€​ [grant number E2019203431] and the Key Research and Development Program of Hebei Province [grant number 21351802D]. Thenumerical calculations in this paper were carried out on the supercomputing system in the High Performance Computing Center of Yanshan University.

References
Chen, H., Xu, G., & Shi, Y. (2020). Aerodynamic load analysis of helicopter rotor in dynamic landing’s ship air wake. Journal of Nanjing University of Aeronautics & Astronautics, 52(2), 171–180. http://dx.doi.org/10.1635/j.1005-2615.2020. 02.008
Crozon, C., Steijl, R., & Barakos, G. N. (2018). Coupled flight dynamics and CFD – Demonstration for helicopters in shipborne environment. The Aeronautical Journal, 122(1247), 42–82. https://doi.org/10.1017/aer.2017.112
Doormal, J., & Raithby, G. (1984). Enhancements of the SIMPLE method for predicting incompressible flow. Numerical Heat Transfer, 7, 147–163. https://doi.org/10.1080/0149528408961817
Forrest, J. S., & Owen, I. (2010). An investigation of ship airwakes using detached-eddy simulation. Computers & Fluids, 39(4), 656–673. https://doi.org/10.1016/j.compfluid.2009.11.002
Ghalandari, M., Mirzade, E., Mohamadian, F., Shamshirband, S., & Chau, K. (2019). Numerical simulation of nanofluid flow inside a root canal. Engineering Applications of Computational Fluid Mechanics, 13(1), 254–264. https://doi.org/10.1080/19942060.2019.1578696
Hodge, S. J., Zan, S. J., Roper, D. M., Padfield, G. D., & Owen, I. (2009). Time-accurate ship airwake and unsteady aerodynamic loads modeling for maritime helicopter simulation. Journal of the American Helicopter Society, 54(2), 22005–2200516. https://doi.org/10.4050/JAHS.54.022005
John, S., & Simon, N. (2002). Basic helicopter aerodynamics. Blackwell Science Ltd. (Original Work Published 1990).
Kaeaeiae, C. H., Wang, Y., Padfield, G. D., Forrest, J. S., & Owen, I. (2012). Aerodynamic loading characteristics of a model-scale helicopter in a ship’s airwake. Journal of Aircraft, 49(5), 1271–1278. https://doi.org/10.2514/1.C031535
Kim, Y. H., & Park, S. O. (2013). Unsteady momentum source method for efficient simulation of rotor aerodynamics. Journal of Aircraft, 50(1), 324–327. https://doi.org/10.2514/1.C031934
Lee, D., & Horn, J. (2004). Analysis of pilot workload in the helicopter/ship dynamic interface using time-accurate and stochastic ship airwake models. AIAA atmospheric flight mechanics conference and exhibit, Providence, RI, 16–19 August 2004.
Lee, D., Horn, J., Sezer-Uzol, N., & Long, L. (2003). Simulation of pilot control activity during helicopter shipboard operations. Journal of Aircraft, 42(2), 448–461. https://doi.org/10.2514/1.6786
Li, H., Li, B., Chen, J., Cui, P., & Zhang, Y. (2020). Validation of momentum source method in the analysis and prediction of the rotor finite ground effect in hover. Journal of Physics: Conference Series, 1600, 012057. https://doi.org/10.1088/1742-6596/1600/1/012057
Menter, F. R. (1994). Two-equation Eddy-viscosity turbulence models for engineering applications. AIAA Journal, 32(8), 1598–1605. https://doi.org/10.2514/3.12149
Mosavi, A., Shamshirband, S., Salwana, E., Chau, K. W., & Tah, J. (2019). Prediction of multi-inputs bubble column reactor using a novel hybrid model of computational fluid dynamics and machine learning. Engineering Applications of Computational Fluid Mechanics, 13(1), 482–492. https://doi.org/10.1080/19942060.2019.1613448
Muijden, J. V., Boelens, O., Vorst, J., & Gooden, J. (2013). Computational ship airwake determination to support helicopter/ship dynamic interface assessment. 21th AIAA computational fluid dynamics conference. San Diego, 24–27 June, 2013.
Nam, H., Park, Y. M., & Kwon, O. J. (2006). Simulation of unsteady rotor-fuselage aerodynamics interaction using unstructured adaptive meshes. Journal of the American Helicopter Society, 51(2), 141–149. https://doi.org/10.4050/JAHS.51.141
Oruc, I., Horn, J., Polsky, S., Shipman, J., & Erwin, J. (2015, May 5–7). Coupled flight dynamics and CFD simulations of the helicopter/ship dynamic interface. AHS 71st annual forum, Virginia Beach, VA.
Oruc, I., Horn, J., & Shipman, J. (2016, January, 4–8). Coupled flight dynamics and CFD simulations of rotorscraft/terrain interactions. AIAA modeling and simulation technologies conference, San Diego, CA.
Oruc, I., Shenoy, R., Shipman, J., & Horn, J. (2016, May 17–19). Towards real-time fully coupled flight dynamics and CFD simulation of the helicopter/ship dynamic interface. AHS 72nd annual forum, West Palm Beach, FL.
Polsky, S. A. (2002, 14–17). A computational study of unsteady ship airwake. 40th AIAA aerospace sciences meeting & exhibit, Reno, NV.
Sezer-Uzol, N., Sharma, A., & Long, L. (2005). Computational fluid dynamics simulations of ship airwake. Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, 221(5), 369–392. https://doi.org/10.1243/095441005X30306
Sharma, A., & Long, L. (2001, June 11–14). Airwake simulations on an LPD 17 ship. AIAA paper 2001-2589, AIAA computational fluid dynamics conference, 15th, Anaheim, CA.
Shi, Y., Li, G., Su, D., & Xu, G. (2020). Numerical investigation on the ship/multi-helicopter dynamic interface. Aerospace
1918

Science and Technology, 106, 1270–9638. https://doi.org/10.1016/j.ast.2020.106175
Shi, Y., Xu, Y., Zong, K., & Xu, G. (2017). An investigation of coupling ship/rotor flowfield using steady and unsteady rotor methods. Engineering Applications of Computational Fluid Mechanics, 11(1), 417–434. https://doi.org/10.1080/19942060.2017.1308272
Stalewski, W., & Surmacz, K. (2019). Helicopter flight simulation based on URANS solver and virtual blade model. Journal of KONES Powertrain and Transport, 26(3), 211–217. http://dx.doi.org/10.2478/kones-2019-0075
Su, D., Shi, Y., Xu, G., & Zong, K. (2017). Numerical simulation of coupled flow field of helicopter/ship. Acta Aeronauticaet Astronautica Sinica, 38(7), 520853–520853. http://hkxb.buaa.edu.cn/CN/10.7527/S1000-6893.2017.120853
Su, D., Xu, G., Huang, S., & Shi, Y. (2019). Numerical investigation of rotor loads of a shipborne coaxial-rotor helicopter during a vertical landing based on moving overset mesh method. Engineering Applications of Computational Fluid Mechanics, 13(1), 309–326. https://doi.org/10.1080/19942060.2019.1585390
Su, D. C., Shi, Y. J., & Xu, G. H. (2019). Numerical study of the rotational direction effect on aerodynamic loading characteristics of shipborne helicopter rotor. The Aeronautical Journal, 123(1263), 635–657. https://doi.org/10.1017/aer.2019.20
Thornber, B., Starr, M., & Drikakis, D. (2010). Implicit large eddy simulation of ship airwakes. The Aeronautical Journal, 114(1162), 715–736. https://doi.org/10.1017/S000192400004218
Weixing, Y., Alanna, W., & Richard, L. (2018). Combined numerical and experimental simulations of unsteady ship airwakes. Computers & Fluids, 172, 29–53. https://doi.org/10.1016/j.compfluid.2018.06.006
Zhang, F., Xu, H., & Ball, N. G. (2009, January 5–8). Numerical simulation of unsteady flow over SFS 2 ship model. 47th AIAA aerospace sciences meeting including the new horizons forum and aerospace exposition, Orlando, FL.