AN APPROXIMATION METHOD FOR THE NUMERICAL SIMULATION ON UNSTEADY FLOWS AROUND SMALL-DISPLACEMENT MOVING OBJECTS

Li Dun, Chen Bingyan
China Academy of Aerospace Aerodynamics, Beijing 100074

Keywords: *Cartesian mesh, relative movement, numerical simulation, moving boundary*

Abstract

The static-mesh/dynamic-boundary method for the simulation of unsteady flows around moving objects using 3-D viscous Cartesian mesh was introduced. The physical conservation laws and the geometric conservation law are fully satisfied in the spatial mesh. Approximation treatment was introduced at the moving boundary under the small-displacement assumption. Numerical results suggest that the introduced method is able to reflect the dynamic tendency of the moving object. The aerodynamic characteristics predicted by this method well agree with the experimental data.

1 Introduction

The movement of objects can be classified into two categories, i.e. large-displacement movement and small-displacement movement. Mesh generation methods suitable for small-displacement movement includes interpolation method, mesh stretching method, elastic body method, etc.

The numerical simulation of the unsteady flow around moving objects requires huge computation capacity. For practical application, it is necessary to introduce approximation treatments. One of the commonly used approximations is the steady calculation method, which divides the unsteady process into a series of steady state moments. The object is assumed to be static during the calculation at a certain moment. This method converts the simulation of an unsteady process into a series of steady calculation, which greatly reduced the complexity of the calculation and is easy to implement. However, the dynamic characteristics of the unsteady process would lose in this approximation treatment. In this paper, a numerical scheme using 3-D unstructured viscous Cartesian mesh was proposed to approximate the unsteady flow around moving objects. Taking advantage of the characteristics of Cartesian mesh, this method adopts the strategy that the spatial mesh remains static while the boundary is allowed to move. The advantage of this strategy is that the computation cost is greatly reduced by using static spatial mesh while the dynamic characteristics of the unsteady process can be reflected through the dynamic boundary.

The static-mesh/dynamic-boundary method for the simulation of unsteady flows around moving objects was introduced and studied in the following sections of this paper. Numerical analysis suggests that this approximation method is able to simulate the dynamic characteristics of the flow field. It provides a possible solution for the numerical simulation of the unsteady flow around small-displacement moving objects, e.g. morphing body.

2 Numerical simulation on moving objects using 3-D viscous Cartesian mesh

2.1 Control equations

In Cartesian coordinates, the velocity of the relative coordinates on the moving object is U. The object relative coordinates is unstationary. A control volume $\Omega(t)$, with small-displacement moving boundary, is in motion. There is physical flux, e.g. mass, momentum and energy, passing through the boundary of the control volume $\Omega(t)$. Considering the motion of the control volume, the integrated form of the N-S equation for the mesh and the control volume, which is in motion relative to the mesh, is shown as follow:

$$\begin{cases} \dfrac{d}{dt}\iiint\limits_{\Omega(t)} \rho dV + \iiint\limits_{\partial\Omega(t)} \rho(U-U_d)\cdot n dS = 0 \\ \dfrac{d}{dt}\iiint\limits_{\Omega(t)} \rho U dV + \iiint\limits_{\partial\Omega(t)} [\rho U(U-U_d)+P]\cdot n dS = \iint\limits_{\partial\Omega(t)} \tau\cdot n ds \\ \dfrac{d}{dt}\iiint\limits_{\Omega(t)} \rho dV + \iiint\limits_{\partial\Omega(t)} [E(U-U_d)+(P\cdot U)]\cdot n dS = \iint\limits_{\partial\Omega(t)} [(\tau\cdot U)-q]\cdot n ds \end{cases} \quad (1)$$

where P, q is the fluid static pressure tensor and heat flux, respectively. Ud is the velocity of the cell boundary. Having known the boundary velocity, the relative velocity of the fluid passing through the boundary is giving as U-Ud. Define the conservation variable as $W=(\rho, \rho u, \rho v, \rho w, E)^T$, the cell boundary $U_d=(u_d, v_d, w_d)$.

Considering the relative motion of the object, the nondimensionalized N-S equation can be rewritten as:

$$\frac{\partial}{\partial}\iiint\limits_{\Omega} W_D dv + \oiint\limits_{\partial\Omega} F_D\cdot\vec{n}\, dS = \oiint\limits_{\partial\Omega} G_D\cdot\vec{n}\, dS \quad (2)$$

where

$$W_D=\begin{bmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ E\end{bmatrix},\quad F_D=\begin{bmatrix}\rho(u-u_d) & \rho(v-v_d) & \rho(w-w_d)\\ \rho(u-u_d)u+p & \rho(v-v_d)u & \rho(w-w_d)u\\ \rho(u-u_d)v & \rho(v-v_d)v+p & \rho(w-w_d)v\\ \rho(u-u_d)w & \rho(v-v_d)w & \rho(w-w_d)w^2+p\\ (E+p)(u-u_d) & (E+p)(v-v_d) & (E+p)(w-w_d)\end{bmatrix} \quad (3)$$

The viscous term takes the following form:

$$G_D=\frac{\sqrt{\gamma}M_\infty}{Re_\infty}\begin{bmatrix}0 & 0 & 0\\ \tau_{xx} & \tau_{xy} & \tau_{xz}\\ \tau_{xy} & \tau_{yy} & \tau_{yz}\\ \tau_{xz} & \tau_{yz} & \tau_{zz}\\ \phi_x & \phi_y & \phi_z\end{bmatrix} \quad (4)$$

where

$$\begin{bmatrix}\phi_x\\ \phi_y\\ \phi_z\end{bmatrix}=\begin{bmatrix}k_r\dfrac{\partial T}{\partial x}+u\tau_{xx}+v\tau_{xy}+w\tau_{xz}\\ k_r\dfrac{\partial T}{\partial y}+u\tau_{xy}+v\tau_{yy}+w\tau_{yz}\\ k_r\dfrac{\partial T}{\partial z}+u\tau_{xz}+v\tau_{yz}+w\tau_{zz}\end{bmatrix}$$

$$E=\frac{P}{\gamma-1}+\frac{\rho}{2}(u^2+v^2+w^2)$$

$$k_r=\left(\frac{\mu_l}{Pr_l}+\frac{\mu_t}{Pr_t}\right)(\gamma-1)M_\infty^2$$

Comparing to the equations for static objects flow field simulation, the velocity of the cell boundary is introduced in the equations for moving objects, i.e. equations (2), (3) and (4). The equations for static objects can be viewed as a special case of the equations for moving objects.

2.2 Numerical scheme

The numerical simulation on 3-D flow around moving objects is in fact a problem of seeking the numerical solution of the unsteady N-S equation under dynamic mesh. Correspondingly, the numerical discretization method must consider the discretization of grid cell velocity. The geometric conservation law must be obeyed in solving the grid cell velocity:

$$\frac{\partial}{\partial}\iiint\limits_{\Omega} dV = \oiint\limits_{\partial\Omega} U_d\cdot\vec{n}\, dS \quad (5)$$

Meanwhile, the control volume must be closed during the process of movement or deformation:

$$\oiint\limits_{\partial\Omega}\vec{n}\, dS = 0 \quad (6)$$

Therefore, the numerical simulation on the unsteady flow around moving objects must satisfy not only the physical conservation laws (equation (2), (3) and (4)), but also the geometric conservation laws (equation (5) and (6)).

The steady calculation method assumes that the object is static during the calculation time interval. The position and direction of the object at the next time interval is determined based on the calculation of the steady flow.

The static-mesh/dynamic-boundary method can approximately satisfy equation (2), (3), (4), (5) and (6). Obviously this method is more accurate since the movement tendency of every point of the moving object has been taken into consideration.

The least square restructuring method was adopted in seeking the numerical solution of equation (2), (3) and (4). The viscous condition needs to be modified on moving grid boundaries. The no-slip condition on solid surface must be replaced by the relative velocity of the object.

Hence the velocity boundary condition on the moving object surface becomes:

$$U - U_d = 0 \tag{7}$$

Correspondingly the constraint equation on the moving object boundary takes the form as follow:

$$f_{de} = q_c + q_x(x_e - x_c) + q_y(y_e - y_c) + q_z(z_e - z_c) - q_{de} = 0 \tag{8}$$

For the whole flow field, the constraint equation is:

$$F_{de} = \sum_i (q_c + q_x(x_i - x_c) + q_y(y_i - y_c) + q_z(z_i - z_c) - q_i)^2 - \lambda f_{de} = 0 \tag{9}$$

The gradient of physical variables of each cell in the entire computation domain can be obtained by solving equation (9) using the least square method. The subscript "de" in the equation denotes the relative physical quantity on the solid surface.

For a general dynamic mesh, all of the grid cells vary with the motion of the object. The variation, due to the movement of the object, of each cell must be followed in every time step. The geometric characteristic of the mesh is a function of time when dynamic mesh is used in solving the N-S equation. Therefore, the calculation is very complicated and costly.

For an unstructured Cartesian mesh, only the grid cells near the object surface vary with the movement of the object. All the other cells in the computation domain remained unchanged. This feature is a great advantage when unstructured Cartesian mesh is applied into the dynamic mesh calculation.

Most of the Cartesian cells away from the object surface naturally satisfy the geometric conservation laws (4) and (5). Only the cells near the object surface need to be modified.

When using Cartesian mesh for the simulation of the flow around moving objects, a basic problem needs to be resolved is that a fluid cell may disappear into the interior of the object, or a new fluid cell may generate from the interior of the object, as the object moves. The conservation laws cannot be satisfied for the flux calculation in the next time step due to the increasing, or decreasing, of the grid cells.

In order to simplify the problem, giving the situation that the 3-D Cartesian mesh is static, the movement of the object relative coordinates can be categorized into 3 cases:

(1) In the time interval that is concerned, the displacement of the moving object is so large that it has exceeded the grid cell size. The object leaves its original position. All of the initial surface cells totally disappear and become fluid cells or interior cells of the object, i.e.

$$U_d >> dt \cdot L_{\min} \tag{10}$$

(2) In the time interval that is concerned, the object does not entirely leave its initial position. Part of the surface cells become fluid cells or interior cells, i.e.

$$U_d \approx dt \cdot L_{\min} \tag{11}$$

(3) In the time interval that is concerned, the object does not leave its initial position at all. None of the surface cells become fluid cells or interior cells, i.e.

$$\mathrm{U_d} << \mathrm{dt} \cdot \mathrm{L_{min}} \tag{12}$$

Where dt, Lmin are time interval and grid cell size, respectively.

For case (1), the mesh reconstruction quasi-steady method can be used for the numerical calculation.

For case (2), in which case the distance of movement exceeds the grid size, the problem can be solved by letting the moving Cartesian cells "appearing" or "disappearing". The algorithm of this method is complicated since it needs to identify the "appearing" and "disappearing" Cartesian cells. And it is difficult to apply in the case of viscous mesh.

For case (3), in which case the distance of movement does not exceed the grid size, the static-mesh/dynamic-boundary method can be used. This method is able to approximately simulate the unsteady process of the flow around moving objects.

Thus, the distance of the object movement can be classified into two categories, i.e. large displacement and small displacement. The case (1) and (2) defined above belong to large displacement; the case (3) belongs to small displacement.

The movement of an object can be always considered as a combined process of large displacement movement and small displacement movement. The large displacement moving process can be calculated using the mesh reconstruction method. The small displacement moving process can be simulated using the static-mesh/dynamic-boundary method.

For the case of small displacement, in order to assure the stability of the numerical calculation, how small the displacement needs to be is a topic that is to be further discussed.

One of the requirements for the numerical stability in the case of static object is that within a time step, the moving distance of the fluid that travels with the speed of maximum wave speed ($|U|+C$) must be less than the minimum grid size, i.e.

$$\Delta t_f \le \frac{CFL \cdot L_{\min}}{|U|+C} \qquad (13)$$

The maximum time step size in the case of moving object must consider the velocity of the object surface. For a moving object, the moving distance of the surface of the object, which travels with the speed of U_d, within a time step must also be less than the minimum grid size, i.e.

$$\Delta t_d \le \frac{CFL \cdot L_{\min}}{|U_d|+C} \qquad (14)$$

Therefore, to assure the calculation stability in the case of moving objects, equation (13) and (14) must be both satisfied, i.e.

$$\Delta t \le \Delta t_f \text{ , and } \Delta t \le \Delta t_d \qquad (15)$$

Or:

$$\Delta t \le \frac{CFL \cdot L_{\min}}{|U_d|+|U|+C} \qquad (16)$$

In the above equations, are the velocities of the fluid, moving object surface and sound, respectively.

In order to accelerate the convergence of calculation, the global time step in equation (16) can be replaced with local time step. The unsteady process is approximated by simplified unsteady method.

3 Analysis on the numerical method for flows around moving objects using 3-D unstructured viscous Cartesian mesh

The effectiveness of the static-mesh/dynamic-boundary method on the simulation of flows around moving objects in the case of small displacement was verified by specific cases.

3.1 Numerical simulation on cowlings with different relative velocities

The configuration used in the numerical simulation is shown in Fig. 1. The cowling has a velocity relative to the computation coordinates, which is fixed on the airframe.

Fig. 1 3-D moving cowling configuration

The ambient Mach number is 2.5. The Mach number of the cowling relative to the airframe is 1.5, 1.0, 0 and -1.0. Fig. 2 shows the variation of the axial force and normal force of the cowling with numerical integration time, for different relative velocities. It can be noted from the figure that for different relative velocities, the whole process of the numerical integration, from beginning to convergence, is totally different, although the ambient Mach number is the same, i.e. 2.5. Obviously, these numerical results are more reasonable comparing to the results of steady calculation, which would be the same for the same ambient Mach number.

From the numerical results it can be seen that if movement of the cowling has the same direction with the ambient flow, in which case the Mach number of the ambient flow relative to the cowling increases, the pressure on the cowling increases. On the contrary, if movement of the cowling has the opposite direction with the ambient flow, in which case the Mach number of the ambient flow relative to the cowling decreases, the pressure on the cowling decreases. This result indicates that under free state the cowling tends to be move with the ambient flow.

Numerical simulations on a large number of supersonic cases for different Mach numbers give similar results. These results suggest that using the static mesh/dynamic boundary method, the simulation on flows around moving cowlings is a better approaching to the real physics.

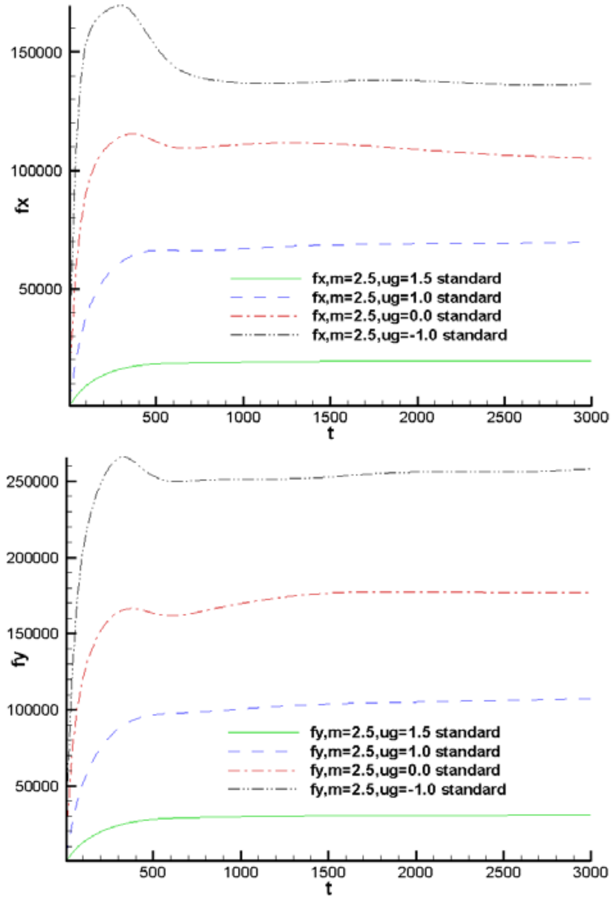
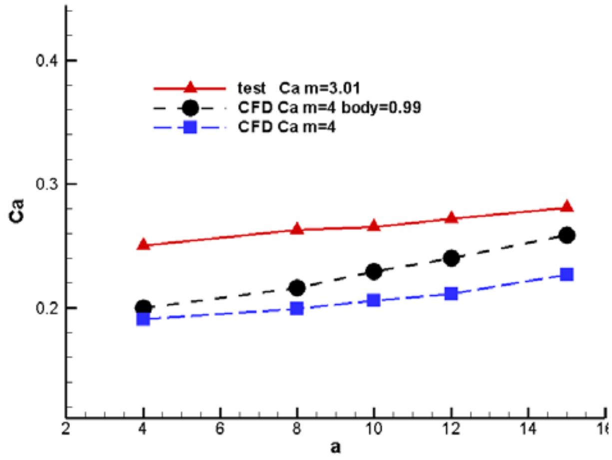


Fig. 2 Variation of axial force and normal force with integration time

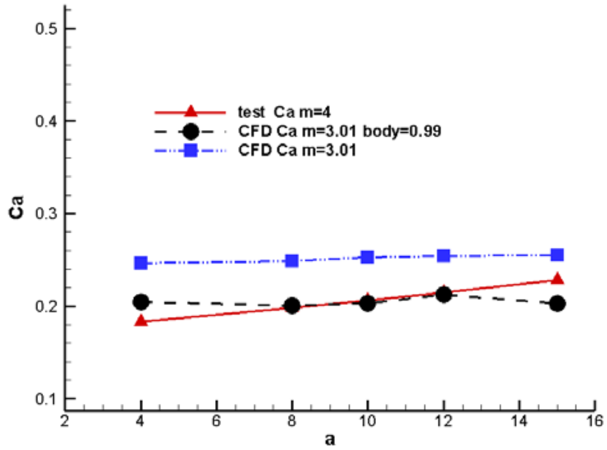
3.2 Verification calculation on moving blunt cones

The verification calculation on the flow around blunt cones was conducted in two cases. In the first case, the ambient flow Mach number is 4 and the Mach number of the blunt cone is 0.99. The movement of the object has the same direction with the ambient flow. In another word, the effective ambient flow Mach number is 3.01 relative to the blunt cone. In the second case, the Mach number of the ambient flow and the cone is 3.01 and -0.99, respectively. Their velocities have the opposite directions. The effective ambient flow Mach number is 4.0 in this case.

In both cases, numerical simulations were performed using two numerical schemes, i.e. the steady calculation method and the static-mesh/dynamic-boundary method. The first method does not take the velocity of the blunt cone into account while the latter does, as has been discussed above. Wind tunnel tests were also conducted on both cases in the form that the blunt cone was fixed and the flow was set at the effective ambient flow Mach number.



(a) case 1



(b) case 2

Fig.3 Variation of axial force coefficient with angle of attach

Fig. 3 shows the comparison between the computation results and experimental data of the variation of axial force coefficient with angle of attack, where the red line denotes the experimental data; the blue line is the computation result using the steady calculation method; the black line shows the calculation result using the static-mesh/dynamic-boundary method. The comparison result for the first case is shown in Fig. 3a. It can be seen from the figure that the black line is closer to the line, comparing to the blue line, especially when the angle of attack increases. Fig. 3b shows the comparison result for the second case. Similarly, the calculation results using the static-mesh/dynamic-boundary method is much closer to the experimental data.

These results suggest that the static-mesh/dynamic-boundary method shows great advantage over the steady calculation method in the numerical simulation of moving objects in terms of accuracy and reflecting the dynamic characteristics of the moving object.

4 Conclusion

Taking advantage of the characteristics of Cartesian mesh that the spatial grid remains unchanged, the static-mesh/dynamic-boundary method for the simulation of unsteady flows around moving objects was introduced in this paper. The strategy that the spatial mesh remains static while the boundary is allowed to move was adopted in this method.

Under this numerical scheme, the physics conservation laws and the geometry conservation law are fully satisfied in the spatial mesh. Approximation treatment was introduced at the moving boundary under the small-displacement assumption.

Verification calculations were conducted on single-moving-object and multi-moving-object. The simulation results suggest that the introduced method is able to reflect the dynamic tendency of the moving object. The comparison between numerical results and wind tunnel tests data indicates that aerodynamic characteristics predicted by the static-mesh/dynamic-boundary method show better agreement with the experimental data comparing to those predicted by the steady calculation method. The static-mesh/dynamic-boundary method provides a possible solution for the numerical simulation of unsteady flows around small-displacement moving objects, such as morphing body.

Copyright Statement