Numerical simulation of dynamic multi-body separation flowfields

Bing Han, Zhenxun Gao*, Chongwen Jiang

National Laboratory for Computation Fluid Dynamics, School of Aeronautic Science & Engineering, Beihang University, Beijing 100191, China

*E-mail: gaozhenxun@buaa.edu.cn, Tel.: +86-82338105

Abstract. In this paper, three-dimensional unsteady Navier-Stokes equations coupled with six-degree-of-freedom equations are solved numerically to simulate the process of multi-body separation in the condition of high speed and high altitude. Overset grids are adopted to deal with the relative motion between two different parts. In order to study the security of separating process, the trajectories and attitudes of different parts are calculated and the characteristics of flow field during separation are analyzed carefully. Different initial angles of attack are considered during the simulation and its influence on the separation process is analyzed. The result shows that the initial angle of attack has significant impact on the flow field. Large angle of attack would cause shock/shock interactions between the two parts, threatening the safety of the separating process.

1. Introduction
Multi-body separation and interaction widely exist in the fields of aeronautics and astronautics such as fairing separation and airdropping [1]. In the process of separating, the relative motion of different parts will significantly influence the flow structure and aerodynamic force, which will in turn influence the motion of the bodies [2]. The situation is worse in the condition of high Mach number because of the appearance of shock-shock interaction and shock-boundary-layer interaction, which poses a threat on the security of separation. If not designed properly, the separation could fail and different parts could even collide [3]. Therefore, it is necessary to carry out the numerical simulation on multi-body separation to strengthen the understanding of this question.

Overset mesh has the advantages of not having to regenerate grids during the calculation, and it has higher grid quality. Therefore, overset mesh is widely used for separation simulation.

In this paper, based on the overset grids, numerical simulations have been carried out to investigate the separating process of a blunt cone and a cylinder at Mach 24, focusing on the safety during separation. The influence of initial angles of attack has been considered. Meanwhile, the characteristics of the flow field during multi-body separation are discussed. The Navier-Stokes equations coupled with six-degree-of-freedom equations [4] are solved with FLUENT in current work.

2. Numerical methods

2.1. Flow solver
Solving Navier-Stokes with Finite Volume Method, the system of governing equations is cast in integral Cartesian form for an arbitrary control volume $V$ with differential surface area $dA$ as follows [5]:

$$\frac{\partial}{\partial t} \int_V W dV + \int_V (F - G) dA = \int_V H dV$$

(1)

And

$$W = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho E \end{bmatrix}, F = \begin{bmatrix} \rho v \\ \rho vu + p_i \\ \rho vv + p_j \end{bmatrix}, G = \begin{bmatrix} 0 \\ \tau_{xi} \\ \tau_{yi} \end{bmatrix}$$

(2)

$E$ stands for total energy per unit mass; $\rho$ represents density; $p$ represents pressure; $v$ represents velocity; $\tau$ stands for the viscous stress tensor; $q$ stands for the heat flux.

The viscous flux is discretized by central difference splitting and the inviscid flux is discretized with Roe scheme. The discrete flux at each face is as follow:

$$F = \frac{1}{2} (F_R + F_L) - \frac{1}{2} \Gamma \hat{A} \delta Q$$

(3)

The fluxes $F_R = F(Q_R)$ and $F_L = F(Q_L)$ are computed using the solution vectors $Q_R$ and $Q_L$ on the right and left side of the face. The matrix $|\hat{A}|$ is defined by:

$$|\hat{A}| = M |\Lambda| M^{-1}$$

(4)

Where $\Lambda$ represents the diagonal matrix of eigenvalues and $M$ is the model matrix that diagonalizes $\Gamma^{-1} A$, where $A$ stands for the inviscid flux Jacobian $\partial F / \partial Q$.

Dual time stepping method is used. The density-based solver is employed, which solves the governing equations of continuity, momentum, and energy simultaneously as a set. No turbulent model is added since the floe keeps in laminar state.

The relative motion of different components during separation is achieved by dynamic mesh. Overset mesh makes it easier to replace parts of a mesh without having to remesh large parts during the separation.

2.2. Six DOF Solver Theory

To calculate the translational and angular of the center of gravity of the multi-body, the six DOF solver is employed. The translational motion is solved in the inertial coordinate system:

$$\bar{v}_G = \frac{1}{m} \sum \bar{f}_G$$

(5)

The angular motion is computed using body coordinates:

$$\bar{\omega}_R = L^{-1} \left( \sum \bar{M}_B - \bar{\omega}_b \times L \bar{\omega}_b \right)$$

$$\bar{M}_R = R \bar{M}_G$$

(6)

(7)

$R$ is the transformation matrix from inertial to body coordinates.
\[
R = \begin{bmatrix}
\cos \theta \cos \psi & \cos \theta \sin \psi & -\sin \theta \\
\sin \phi \sin \theta \cos \psi - \cos \phi \sin \psi & \sin \phi \sin \theta \sin \psi + \cos \phi \cos \psi & \sin \phi \cos \theta \\
\cos \phi \sin \theta \cos \psi + \sin \phi \sin \psi & \cos \phi \sin \theta \sin \psi - \sin \phi \cos \psi & \cos \phi \cos \theta
\end{bmatrix}
\]

Equation (8)

ψ, θ and φ represent the attitude angles of the model in the body coordinates.

3. Calculation settings

3.1. Flow configuration

The separation of a blunt cone and a cylinder is simulated. The blunt cone is 2 meters long with 1 meter base diameter and 50 millimeters nose radius and weights 300 kilograms. The cylinder is 1 meter long with 1 meter base diameter and weights 120 kilograms. When separation starts, a 300 Newton force of axial direction is exerted on the cylinder for 1 second.

3.2. Boundary conditions

The condition of incoming flow is shown in Table 1. Initial angles of attack before separation are chosen to be 0 and 5 degree.

| Altitude(km) | Pressure(pascal) | Mach number | Temperature(K) |
|-------------|------------------|-------------|----------------|
| 60~70       | 5.2209           | 24          | 219.585        |

The wall is set to be no-slip with constant temperature of 300K.

3.3. Overset mesh

The grid of blunt cone is used as the background mesh, which covers the whole computational domain. The outer boundary of the cylinder grid is set to overset boundary condition, and the overlapping area shuns the boundary layer of the cylinder, making the calculation more accurate. In the process of hole-cutting, the background mesh and the component mesh have the same priority. In the process of overlap minimization, the cell donor priorities are based on distance to the nearest boundary.

Figure 1. Overset mesh at 0s.

Overset grid system before separation are shown in Figure 1. For different separation distance, the overlapping domain is slightly different but the topology is the same. The total number of computational
grids is approximately 4,270,000. The height of the first layer grid is $10^4$ m. Grids near the wall and at the shock wave location are refined.

4. Results and Discussions

In this paper, The OX axis points to the rear of the model. The OZ axis is perpendicular to the model reference plane and points to the left of the model. The OY axis is perpendicular to the XOZ plane in the reference plane and points above the model.

The studied objects can separate with each other successfully without collision under the situation of 0 degree and 5 degree angles of attack, and the separating process is shown in Figure 2 and Figure 3. The blunt cone and the cylinder continue to descend due to the gravity, and the cylinder is separated from the blunt cone by the reverse thrust.

![Figure 2](image1)

**Figure 2.** The separating process at 0 degree angle of attack.

![Figure 3](image2)

**Figure 3.** The separating process at 5 degree angle of attack.

Figure 4 shows the x and y coordinates of the centroid of the model during the separation for 0 and 5 degree angle of attack. At 0 degree angle of attack, the heights of the blunt cone and the cylinder are almost the same as time goes, and there is no relative motion between the multi-bodies in the y direction. At 5 degree angle of attack, the centroid coordinate of the cylinder is lower than that of the blunt cone after 0.2s, and the relative motion in the y direction is generated. At this time, shock wave begins to appear in front of the cylinder.

The axial separation distance at 5 degree angle of attack is larger than that of 0 degree, as shown in Figure 4(b).
Figure 4. Centroid position of the model during the separation for the two degrees angle of attack.

In conclusion, both 0 and 5 degree angle of attack can keep a safe distance. The angle of attack has little effect on the separation distance.

The initial attitude angles of the models are zero. During the separation, both the blunt cone and the cylinder start to rotate, and the pitch angle ‘psi’ is the most significant due to the action of moment.

At 0 degree angle of attack, when separation begins, negative pitch angle appears at both the blunt cone and the cylinder. As time goes, the cylinder's pitch angle becomes greater than the cone's. The yaw angle ‘theta’ and the roll angle ‘phi’ are all almost 0, which are basically negligible. See Figure 5 for details.

Figure 5. Attitude angles of the blunt cone and the cylinder at 0 degree angle of attack.

At 5 degree angle of attack, when separation begins, the blunt cone has a positive pitch angle ‘psi-cone’ that increases with time. The pitch angle of the cylinder ‘psi-cylinder’ is negative before 0.6s, and then becomes positive. After 0.8s, the value exceeds the blunt cone. The yaw angle and roll angle are basically negligible. See Figure 6 for details.
The yaw angle and the roll angle of the model.

(b) The pitch angle of the model.

Figure 6. Attitude angles of the blunt cone and the cylinder at 5 degree angle of attack.

In conclusion, at 5 degree angle of attack, the change of pitch angle is larger than that of 0 degree. As a consequence, 0 degree angle of attack is safer for the separation under the consideration of attitude angles.

Before the separation, the cone and the cylinder stick to each other and a detached shock wave appears in front of the blunt cone.

When the angle of attack is 0, the cylinder stays in the wake region of the cone during the whole separating process. The entire flow field is almost symmetry and two symmetric vortexes appear in the wake, as shown in Figure 7. Overall, the interaction between the cone and the cylinder is relatively small and the separating process is smooth.

Figure 7. The contour of static pressure at 0 degree angle of attack.

Different from the condition of 0 degree angle of attack, the flow field begins to show asymmetric characteristics while the angle of attack increased to 5 degree. The asymmetry of the cone’s wake makes the cylinder drifting away from the wake as physical time increases. Finally, the cylinder is exposed to the supersonic incoming flow, consequently generating a new detached shock wave. The two detached shock waves interact with each other at some points as shown in Figure 8, which changes the direction of the first detached shock wave, resulting in quite complex flow structure.
After analyzing the flow field structure, the aerodynamic of the blunt cone during the separation process are analyzed. Figure 9 shows the comparison of the drag of the whole blunt cone ‘Fx-cone’ and the part of bottom surface of it ‘Fx-bottom’ at 0 degree angle of attack, and Figure 10 shows that of 5 degree angle of attack.

The results illustrate that the drag of the whole blunt cone and the bottom of it have same changing trend and value, which confirms that the influence of separation on the blunt cone mainly results from the change of pressure at the bottom of the cone during the separating process. The pressure at the bottom surface of the blunt cone increases firstly and then decreases, which causes the drag of the cone to decrease firstly and then increase.

At the beginning, the drag of the blunt cone decreases as time goes because the interstage region is a low velocity recirculation zone and the outflow disturbance can only be transmitted through the shear layer. Therefore, the pressure in the interstage region is much higher than the normal bottom pressure. At the same time, the airflow is hindered due to the existence of the cylinder, and the obstruction effect is transmitted to the bottom of the blunt cone, so that the bottom pressure is raised to be slightly larger than the static pressure of the flow. This phenomenon is called “afterbody effect” in the experiment [6].
Figure 10. The comparison of the drag of the whole blunt cone ‘Fx-cone’ and the bottom surface of it ‘Fx-bottom’ at 5 degree angle of attack.

The appearance of shock/shock interaction results in unsteady oscillation of aerodynamic force of the model. Figure 11 shows the drag ‘Fx-cone’ and lift ‘Fy-cone’ of the blunt cone. It can be seen from the Figure 11 that the drag curves show unsteady feature after 0.4s at 5 degree angle of attack.

Figure 11. The aerodynamic force of the blunt cone.

The unsteady oscillation is more obvious through the comparison of the drag of the cylinder ‘Fx-cylinder’, as shown in Figure 12. A sudden jump of the drag of the cylinder appears at 0.5s, and the drag of the cylinder increases with the enhancement of the second shock wave after that moment.

In conclusion, considering the flow field structure and the stability of aerodynamic force, the 0 degree angle of attack is safer for the separation.
5. Conclusion
Based on overset grids and dynamic grids, three-dimensional Navier-Stokes equations coupled with six-degree-of-freedom equations are solved to simulate the separating process of a blunt cone and a cylinder at Mach 24 in this paper. The conclusions are listed as follows:

• The cone and the cylinder can separate with each other successfully without collision under the situation of both 0 and 5 degree angles of attack;

• In the condition of 5 degree initial angle of attack, the distance between the cone and the cylinder is larger and the change of pitch angle is larger than that of 0 degree during the separation.

• The flow field almost keeps symmetry during separation when the initial angle of attack is zero. Two separating vortexes are generated at the interval of two bodies and the cylinder behind always stays in the wake of the front cone. When the initial angle of attack increases to 5 degree, the cylinder behind starts to move out of the wake and to be exposed to the supersonic incoming flow. A new detached shock wave appears on the cylinder and interacts with the front shock wave and change the direction of it;

• The appearance of shock/shock interaction at the 5 degree angle of attack make the flow field and the aerodynamic force on aircraft unsteady, which will pose a threat on the security of separation;

• The influence of separation on the front blunt cone mainly results from the change of the flow field at the bottom of the cone during the separating process.

Acknowledgments
This research is supported by the National Natural Science Foundation of China (NSFC, Grant Nos. 91641123, 11872094) and the Foundation for Innovative Research Groups of the National Natural Science Foundation of China through grant 11721202.

References
[1] Newton J and Spaid F 1962 Interaction of secondary injectants and rocket exhaust for thrust vector control ARS Journal 32 p1203–11
[2] Mirzaei M, Najafie N and Shadaram A 2006 Numerical simulation of stage separation manoeuvre with jet interaction Aircraft Engineering and Aerospace Technology 78 p217–25
[3] Nichols R and Denny A 2013 Numerical simulation of a store in controlled separation 17th Applied Aerodynamics Conference p 3128
[4] Hall L and Parthasarathy V 1998 Validation of an automated Chimera/6-DOF methodology for multiple moving body problems 36th AIAA Aerospace Sciences Meeting and Exhibit p 753
[5] Pulliam T and Steger J 1980 Implicit Finite-Difference Simulations of Three-Dimensional Compressible Flow Aiaa Journal 18 p 159–67
[6] Zhou W, Bai P and Ma H 2000 The numerical study of flowfields around a separating missile. Chines Journal of Computation Physcis 17 p 532–6