Detached eddy simulation of the release progress of moving body separating from a cavity

Huan Li, Hongyin Jia, Pengcheng Cui, Jiangtao Chen, Guiyu Zhou and Yaobing Zhang *

Computational Aerodynamics Institute, China Aerodynamics Research and Development Center, Mianyang 621000, China

*Corresponding author e-mail: zhyb_super@cardegs.com

Abstract. Aiming at the complex unsteady flow during the release progress of moving body separating from a cavity, combined with the overlapping grid technique and Spalart Allmaras equation turbulence model, the three-dimensional unsteady flow solver is presented. On this basis, the validity of the simulation method is verified by M219 cavity example and WPFS example. Then the released moving body is conducted with SA-IDES. The research results show that upward force and pitch down moment of the moving body are affected by the fluctuating flow near the cavity and the SA–DES simulation is good in predicting high frequency flow. In conclusion, the outside shear layer and the inside fluctuating flow of the cavity has a significant impact on the aerodynamic loads and the trajectory of the moving body during the initial separation stage.

1. Introduction

The progress of the moving body separating from a cavity can be seen in modern aircrafts with its benefit of improvement in the aircrafts’ aerodynamic performance. However, when the moving body is released from the cavity, there are extremely complex flow physical phenomena. A large range of pressure fluctuation formed in the cavity may cause severe vibration and harsh noise [1]. Meanwhile, the trajectory of released moving body may be influenced by the pressure fluctuation by changing the forces and moments of the moving body. It’s obvious that investigation of the released progress is essential for the aerodynamic design of the high-performance aircraft in the future.

In order to study the complex flow field, a large number of experiments and numerical simulations have been carried out. The results show that the length depth ratio of the cavity [2-3], the shape of the cavity [4], incoming Mach number [5], and other factors will affect the flow characteristics in the cavity. At present, there are mainly two kinds of wind tunnel test technology to study the progress, including the controllable trajectory test (CTS) and the wind tunnel free flight test [6-8]. Compared with the experimental research, the numerical simulation research of the progress has the advantages without limitation of test equipment and test safety. Currently, in numerical simulation method, Reynolds-averaged Navier–Stokes (RANS) equations [9-10] are mostly used for the released progress of moving body, while large eddy simulations (LES) [11-12] or detached eddy simulations (DES) [13-14] are adopted for the cavity flow.
In this paper, the progress involves both moving body and cavity flow, SA–DES turbulence model is adopted to study the unsteady progress. The impact of cavity flow on the aerodynamic loads and the trajectory of the moving body during the initial separation stage is analyzed, which has certain guiding significance for the aerodynamic design of the high-performance aircraft in the future.

2. Numerical Method

The numerical simulations here are conducted by the China Aerodynamics Research & Development Center (CARDC) NNW-FlowStar software [15], an unstructured finite volume cell-center CFD solver. The flow solver behaved will in the several drag prediction workshops(DPW) and high-lift prediction workshops(HLWP) organized by AIAA and NASA, and the computational accuracy and robustness of the solver are as good as some other software such as Fluent, such as FUN3D, CFL3D, CFD++ [16].

2.1. Governing Equations

The unsteady Navier-Stokes equations are discretized by a cell centered finite-volume method. The integral form of unsteady Navier-Stokes equations for a bounded domain \( \Omega \) with a boundary \( \partial \Omega \) can be expressed as:

\[
\frac{\partial}{\partial t} \int_\Omega Q dV + \int_{\partial \Omega} (H(Q) \cdot \hat{n} - \overline{Q_v} \cdot \hat{n}) dS = \oint_{\partial \Omega} H(Q) \cdot \hat{n} dS
\]

Where, \( Q = \left[ \rho, \rho u, \rho v, \rho w, \rho E \right] \), \( \overline{v} \) is the wall velocity and \( H(Q) \) is the inviscid flux vector, \( H_v(Q) \) is the viscous flux vector. Here, \( \rho, (u, v, w) \) and \( E \) denote the density, velocity of three directions, and specific total energy of the fluid. \( \hat{n} \) is the outward pointing normal unit vector of boundary. The inviscid flux is replaced by a numerical Riemann flux function Roe schemes [17], and the viscous flux is discretized with central difference scheme.

2.2. SA turbulence model

SA turbulence model is not simplified directly by \( k-\varepsilon \) two equation model, but with the experience and dimensional analysis, is gradually developed to the one equation model suitable for solid turbulent flow. The integral form can be expressed as:

\[
\frac{\partial}{\partial t} \int_{\Omega} \bar{W}_r d\Omega + \oint_{\partial \Omega} (\bar{F}_{r,x} - \bar{F}_{r,y}) dS = \int_{\partial \Omega} \bar{Q}_r d\Omega
\]

Where:

\[
\bar{W}_r = \rho \bar{v}
\]

\[
\bar{F}_{r,x} = \bar{W}_r V = \rho \bar{v}V
\]

\[
\bar{F}_{r,y} = \frac{1}{\sigma} \left( v_L + \bar{v} \right) \frac{\partial \bar{W}_r}{\partial n} = \frac{1}{\sigma} \rho \left( v_L + \bar{v} \right) \frac{\partial \bar{v}}{\partial n}
\]

2.3. SA–DES method

There are many different forms of DES. The main method is to modify the length scale of turbulence model equation to mesh filter scale in LES model region. When the generating term and dissipation term of the turbulence equation reach equilibrium, vortex viscosity coefficient is proportional to the amplitude of the average deformation rate and the square of filter scale, similar to the viscosity coefficient of Smagorinsky sublattice vortex viscosity model. The nearest distance \( d \) from the wall surface in the right term of the SA one equation model is modified to:

\[
\bar{d} = \min(d, C_{u*,\Delta})
\]

After being modified, the DES method based on the SA turbulence model is obtained. \( \Delta \) is the maximum distance from the center of all coplanar control volumes surrounding the appointed control
volume to the center of the appointed control volume. The value of $C_{des}$ is related to the distribution of turbulent energy in the inertial sub-zone.

When $\bar{d} = d$, the DES equations are transformed into RANS equations. When $\bar{d} = C_{des}\Delta$, considering the local equilibrium, generating term and dissipation term of the turbulence equation reach equilibrium.

$$\bar{c}_b\rho\bar{\tilde{v}}\approx c_{w3}\rho\left[\frac{\bar{\tilde{v}}}{C_{des}\Delta}\right]^2$$

(7)

The vortex viscosity coefficient is similar to the viscosity coefficient of Smagorinsky sublattice vortex viscosity model.

$$\bar{\nu} \propto \bar{S}\Delta^2$$

(8)

2.4. Overlapping grid technique

Due to the unsteady flow phenomenon of the moving body separating from a cavity, the overlapping grid technique is adopted. The technique allows any grid zones to overlap each other. The wall surface intersection criteria is selected as the way to determine the hole boundary. After that, the alternative digital tree (ADT) method [18] is applied to solve the corresponding grid cell of the hole boundary in the sub-grid wrapping objects. Finally, the information of overlapping zones is transmitted by hole digging and interpolation. Obviously, the technology which is flexible and suitable for the simulation of the progress of moving body separating from a cavity.

3. Validation of Numerical Method

3.1. M219 cavity example

M219 cavity case is considered here to evaluate the accuracy of our SA-DES method. The wind tunnel experimental data published by QinetiQ is widely used to validate the numerical method. The sketch of the cavity can be seen in reference [19]. Mesh size inside the cavity is about 1mm. In order to assure $y+\sim1$, the thickness of the first layer mesh adjacent to the wall is $2 \times 10^3$ mm. The grid of the cavity can be seen in Fig. 1. The simulation was performed with free stream conditions of $Ma = 0.85$, $T = 266.53$ K, $P = 63000$ Pa. The total number of grid cells is 13 million and the simulation time step is $1 \times 10^{-5}$ s. Total 20000 time steps are computed, which equates to physical time 0.2 s.

Figure 2 and Figure 3 shows the complex cavity flow structure. Figure 4 shows the sound pressure level results at K20 and K29. Close inspection reveals good match among numerical simulation, experiment and the semi-empirical analysis, which demonstrates the reliability and the accuracy of the SA-DES in predicting the fluctuating cavity flow.
3.2. WPFS example
Wing/Pylon/Finned-Store (WPFS) model is the standard store separation experiment model. The wind tunnel experimental data published by AEDC is widely used to validate the overlapping grid technique. The detail diameters of the model can be seen in reference [20]. Figure 6 shows the grid of WPFS model, which contain two overlapping grid zones. The comparison of the experiment data and CFD data can be seen from figure 7, the result shows that calculation values are in good agreement with the experimental
values and the method in the paper has the ability to simulate the moving body flow field with the overlapping grid technique.

(a) overlapping grid

(b) slice of overlapping grid

Figure 6. Grid of WPFS model

(a) distance of the store versus time

(b) attitude angle of the store versus time

Figure 7. Comparison of the experiment data and CFD data

4. Simulation and analysis of the release progress

4.1. Moving body model and cavity model

To simulation the release progress of moving body separating from a cavity, the simulation models include two model: moving body model and cavity model. Moving body model here is the standard store of the WPFS model and cavity model is scaled M219 cavity with its length equals to 4.572 m while keeping the length-to-depth ratio same as original cavity.

Figure 8 shows the Slice mesh enlargement of the overlapping grid sections near the cavity. It can be seen that the mesh sizes in the interpolated boundary of the two zones are matched. Red mesh represents the subzone of the moving body and black mesh represents the major-zone of the cavity. The finest off-body spacing is 8mm and this grid surrounds the cavity and extends. It is manually specified in order to contain the entire release progress. The thickness of the first layer grid on the shell wall is kept at 2 × 10^-3 mm. The subzone containing moving body has about 22 million cells and the major-zone containing cavity has about 40 million cells. The simulation conditions are as follows: Mach number Ma = 0.95, attitude H=8km, angle of attack α = 0 °.
4.2. The Impact of the cavity flow field on the moving body

With the moving body released, the progress is affected strongly by cavity flow, especially the progress across the shear layer. Figure 9 shows the history of the moving body aerodynamic characteristics, it can be seen that fluctuations in the vertical force and pitch moment on the moving body are obvious. For example, there are large differences in pitch moment between \( t=100\)ms and \( t=108\)ms, or between \( t=157\)ms and \( t=165\)ms, which are targeted in figure 9(b). The reason is that the shedding vortexes from the cavity flow collide onto the moving body and impact on the surface with irregular distribution of high pressure and low pressure.

Figure 10 and figure 11 show that instantaneous iso-surface of Q-criterion (colored by pressure coefficient) and Mach number contours of cavity center (section colored by Mach number and body colored by pressure coefficient).

At \( t=100\)ms or \( t=108\)ms, the moving body just reaches the shear layer with its’ body immersed in cavity. From figure 10(a) and figure 10(b), it is clear the inside fluctuating vortex structures of the cavity impact on the moving body. Although the time difference between \( t=100\)ms and \( t=108\)ms is only 8ms with almost no change in position of the moving body, the pressure distribution of the body in these two
moment are discriminated obviously. The similar result can be related to another two moments: \( t=157 \text{ms} \) and \( t=165 \text{ms} \), at which the moving body is going through the shear layer. The fluctuating effect of the shear layer on aerodynamic loads of the moving body is stronger than that of the inside fluctuating flow of the cavity, which can be seen from figure 11(c) and figure 11(d).

**Figure 10.** Instantaneous iso-surface of Q-criterion \((Q = 3e5)\) for different time.
During 0~0.055 s, the moving body is controlled by the ejection force and the vertical speed increases to -3.7 m/s as is shown in Fig. 12(d). The moving body is loaded to a great upward pitch moment and the pitch angular velocity increase to 77.5°/s rapidly, and the pitch angle reach to 2.1 degrees. And then, the moving body keeps falling down with the gravity force and the aerodynamic force. From 0.055 s to 0.1 s, the vertical aerodynamic force on the moving body keeps stable at about 1000N and the falling speed increases to -4.1 m/s with a gentler accelerated speed. But after 0.1s, the moving body begins to be interfered by the shear layer as shown in figure 11. The falling speed increases further slowdown and the pitch angle accelerated speed is reverted at a later stage as shown in figure 12(b) and figure 12(d).

Because the vertical aerodynamic force and the pitch moment of the moving body inversely increases with the shear layer.

Figure 11. Mach number contours of cavity center cross section for different time
5. Conclusion
The released progress of the moving body separating from cavity was studied with SA-DES method based on overlapping grid technique. In the initial separation stage, the inside fluctuating vortex structures of the cavity impacted on the moving body. When the moving body reach at near the shear layer, the vertical aerodynamic force and the pitch moment of the moving body inversely increased and the trajectory of the body changed gradually. The outside shear layer and the inside fluctuating flow of the cavity has a significant impact on the aerodynamic loads and the trajectory of the moving body during the released progress.

Acknowledgments
This work is supported by National Numerical Windtunnel (NNW).

References
[1] S. J. Lawson 2009 Journal of aircraft 46(3) 1009.
[2] YANG Dang-guo 2010 Journal of Aerospace Power 25(7) 1567.
[3] XIE Lu AI Jun-jiang et al 2014 Advances in Aeronautical Science and Engineering 5(1) 18.
[4] XU Lu SANG Wei-min et al 2011 Chinese Journal of Applied Mechanics 28(1) 85.
[5] RizzettaDP VisbalMR 2003 AIAA Journal 41(8) 1452.
[6] TracyM, PlentovichE et al 1992 AIAA 1992-4363 Reston.
[7] RossJPetoJ 1992 Royal Aircraft Establishment 1992-2233.
[8] Dix RE,Bauer RCS 2000 AEDC 2000-99-4.
[9] Srinivasan, S. 1993 Journal of Fluids Engineering 113(8) 368.
[10] Tam, C. 1996 AIAA Journal 34(11) 2255.
[11] Larchevêque 2004 Journal of Fluid Mechanics 516(2) 265.
[12] Larchevêque 2003 Journal of Fluid Mechanics 15(1) 1993.
[13] Nayyar 2007 The Aeronautical Journal 111(7) 153.
[14] Viswanathan AK,Squires KD et al 2003 AIAA 2003-0265 Reston.
[15] Cui P C Deng Y Q Tang J et al 2016 Acta Aeronautica et Astronautica Sinica 37(10) 2992.
[16] Levy D Laflin K Tinoco E 2013 51st AIAA Aerospace Sciences Meeting DLR.
[17] P. L. Roe 1997 Journal of Computational Physics, 135(2) 250.
[18] BONET J 1991 International Journal for Numerical Methods in Engineering 31(1) 1.
[19] Yan Panpan Q. F. Zhang et al 2018 Journal of Mechanics 34(2) 103.
[20] ZHANG Qun-feng YAN Pan-pan et al 2016 Journal of Aerospace Power 31(3) 717.