Simulation of the Store Separation from Internal Weapons Bay Using M_SST DES Model

Junfeng Gong\(^1\), Fang Wang\(^2\)
\(^1\)UAV Research Institute, Northwestern Polytechnical University, Xi’an, China
\(^2\)Xi’an ASN Technology Group Co., Ltd., Xi’an, China
gong.junfeng@163.com

Abstract. The internal carriage of stores on an aircraft’s merit is especially importance for attack aircraft. For the safe carriage and separation of stores, it is important to know the flow field environment the store experiences during flight. A CFD investigation was conducted to determine cavity flow characteristics. M-SST DES Method are used as numerical simulate method. Compared with the wind tunnel test results, the open cavity flows are studied. The numerical simulation results are in good agreement with the wind tunnel test results, thus verifying the credibleness of the M-SST DES Method. As a typical research model, AEDC’s ‘Wing-Pylon-Finned Store’ wind tunnel test model is computed in this paper. Last, this paper investigates the store’s separation characteristic, which is internal carriage on and release from aircraft. Those results are also indicated that the DES method of numerical simulation flow-filed can embody the unsteady aerodynamic random factors of the store separation from internal weapons bay.

1. Introduction
For the modern and high performance aircraft, stealth has become an important concern and basic precondition. And other request includes the maneuverability and cruise performance. Internal carriage of stores on an aircraft’s merit is especially importance for high performance attack aircraft. Weapon bay flow is a typical cavity flow aerodynamics that is complicated and oscillations. Oscillations are driven by the instabilities in the shear layer and vortex shedding, with a strong coupling between the shear layer and flow inside the cavity. Most Computational Fluid Dynamics (CFD) research conducted on cavity flows indicated that the turbulence is key role to play in driving the flow mechanisms. DES (Detached-Eddy Simulation) approach is theoretically a good way of taking advantage of the merits of both RANS (Reynolds Averaged Navier-Stokes) and LES (Large Eddy Simulation) approaches. Simulate results, computer capability and time cost are integrate balance, DES is selected for its unique characteristic [1-3].
The main objective of this paper is to simulate of the store separation from an aircraft’s internal weapons bay using M_SST DES model. First open cavity flow simulation result are compare with wind tunnel test result. Then store separation simulation is validated. Last internal store separation from an aircraft simulation are research and analyzed.

2. M_SST DES Method
The notable modification to the k–w model came from Menter in 1994. For DES with the two-equation k–w model, the only modification is in the dissipation term

$$-\beta \rho \omega k$$  \hspace{1cm} (1)
The M_SST turbulent length scale is defined by
\[ l_{k-\omega} = k^{1/2} / \omega \]  
(2)

In M_SST DES Method, 1 is given by
\[ \tilde{l} = \min(l_{k-\omega}, C_{DES}\Delta) \]  
(3)

In equation (3), \( C_{\omega,\Delta} = 0.65 \). \( \Delta \) is the metric of the grid size, which essentially computes the size of the maximum cell length.
\[ \Delta = \max(\Delta_x, \Delta_y, \Delta_z) \]  
(4)

For unstructured grid, \( \Delta \) is the length that local cell’s center point to neighborhood cell’s center point.

When space between the cell (i.e. \( C_{\omega,\Delta} \)) is greater than the M_SST turbulent length scale \( (l_{k-\omega}) \), RANS is triggered. LES is activated when the converse occurs. This boundary between LES and RANS is therefore completely dependent on the geometry and on the density of the computational domain. For near the wall mesh, the \( \omega \) is larger and \( k \) is a limited value. So \( l_{k-\omega} \) is likely to be always smaller than the actual space between the cells near the wall so RANS will remain active there. When the cells are far to the wall the \( l_{k-\omega} \) is rapidly increase, so \( C_{\omega,\Delta} \) is used and LES is activated.

3. Test cases and result for cavity flow

Cavity flows can be described in many ways. Generally speaking, its can be divided by the free-stream Mach number. Now the typical way of categorising cavity flows is however directly based on the flow characteristics that occur in the cavity. Which flow is triggered in the cavity is strongly dependent on the cavity length-to-depth ratio (L/D) ratio. The categories include open cavity flow, closed cavity flow and transitional cavity flow[4,5].

The experimental data used to validate the numerical results in this paper were provided by Plentovich, Robert et al.[6]. A rectangular, three-dimensional cavity was mounted in a flat plate. The cavity length was 14.4 in. The cavity width was 9.6 in. The depth of the cavity was 2.4 in. The free-stream Mach number of the wind tunnel test was 0.6. The static-pressure was 144694Pa and the static-temperature was 340.15 K. The numerical model states are same to the states of the wind tunnel test.

![Figure 1. CFD grid of the open cavity model (L/D=6)](image)

Fig.1 gives the grid of the numerical model. The mesh is y-symmetry and “H” type. The pressure-far-field boundary conditions is apply to flow field. In the bottom solid surface, the no-slip condition is applied.400000 nodes are gridded within the cavity. In general, the first node close to the wall has y+ between 30—300. Initially, flow-field of the cavity is computed by RANS method, and given the result of the Cavity flows. At 0 sec, unsteady flow-field is initialized by this steady flow-field result. Unsteady flow-field is computed by M_SST DES Model method. Enough flow time passed, initial steady flow-field influence can be ignored.
The pulsatile pressure are looked like high frequency and random. It is foreseen that it is also related to the sampling frequency of the pressure value. However, on the whole, the pressure pulsation range is relatively stable, and the average value is stable.

Fig. 3 shows the symmetry plane calculation results for the open cavity (L/D=6). It can be seen that the calculation result based on the RANS method is stable. The results based on the DES method are variable and fluctuating, indicating the unsteady nature of the open cavity flow.

Fig. 4 shows the comparison of the CFD calculation results with the experimental results for open cavity bottom pressure. It can be seen that the calculation result based on the SST-based DES method is closer to the experimental value than the SST-based RANS method. Since the DES method is essentially an unsteady method, the amount of computation is much higher than that of the RANS method, so the time step of unsteady flow field calculation cannot be too small. Therefore, the time steps given in this paper are $\Delta t = 0.1\tau$, $\Delta t = 0.05\tau$, and $\Delta t = 0.01\tau$. Calculate 10 steps, 20 steps, and 100 steps respectively in the flow through the cavity.

The results show that as the calculated number of steps decreases, the calculation results are approximated to the RANS method. As the number of calculation steps increases, the calculation result approaches the experimental value. The related paper also confirmed that the time step has an effect on the simulation results of the DES method. Simone Crippa studied the Half-Span Delta Wing using the RANS method and the DES method. It was found that as the time step increases, the DES simulation result approaches the RANS method simulation result. In general, the reason for this phenomenon is that as the time step is reduced, the DES method can discriminate and capture smaller time scale vortices. Especially for the pressure pulsation, as the time step decreases, the pressure...
pulsation of the higher frequency flow field, i.e., noise, can be resolved. For practical engineering calculations, a balance needs to be struck between the accuracy of the calculations and the consumption. When the time step is small to a certain extent, the calculation accuracy of the flow field brought by the smaller time step is limited, and the computational resource consumption increases rapidly. Referring to the calculation result of Fig. 4, the time step of the cavity flow DES calculation in this paper is determined to be $\Delta t = 0.01\tau$. 

4. Test Cases and Result for store separation

4.1 Technology of Coupling solve

Flow chart of Coupling solve is laid out as follows:

![Flow chart of Coupling solve](image)

**Figure 5.** Flow chart of Coupling solve

Initially, the aircraft doesn’t separate from the parent aircraft, steady flow-field is computed. At 0 sec, unsteady flow-field is initialized by the steady flow-field result. At each time step, unsteady flow-field is integrated to give the aerodynamic forces and moments of the separating stores. External forces including gravity, ejector and other external forces, the aerodynamic forces and moments are passed to six-degree-of-freedom motion of separation stores model. Through this model, the new positions and attitudes are computed for the next time step. Relying on the information, the unstructured dynamic mesh model updates the grid of the unsteady flow-field. The next time step unsteady flow-field is solved. As this sequence, iterative time advancement is executed.

4.2 The Simulation Result Validated

![Geometry and surface grid of AEDC wind tunnel test model](image)

**Figure 6.** Geometry and surface grid of AEDC wind tunnel test model

AEDC [7] wind tunnel test model is showed in Fig.6. The detailed geometry definitions and experimental data for wind tunnel test can be found in Ref.(13). The state of test and CFD is $Ma=1.2$, $H=12000$ m. Angle of attack of the wing model is 0 deg. The quantity of the volume mesh is about 800000. Unsteady flow-field time step is taken as 0.0005 second.
Linear displacements graphs are shown in Fig. 7a. Angular displacements graph are given in Fig. 7b. CFD simulation linear and angular displacements are in very good agreement with experimental data in short time range. Along with time, the difference of CFD and experimental data became bigger. Especially, after t=0.3 second, the roll angular displacement error is greater than others. This difference mainly comes from that aerodynamic roll moment error is sensitive to roll inertia. Roll inertia is far smaller than pitch inertia and yaw inertia.

5. Results and Discussion
The cavity flow is complex and there is a constant unsteady oscillation. Therefore, when performing unsteady weapon separation simulations, whether it is the RANS calculation flow field or the DES calculation flow field, the separation of the flow field at the initial time is uncertain. This paper attempts to study the influence of the flow field oscillation with missile on the separation characteristics of the missile under different initial flow fields. As in the previous section, the RANS model was first used to simulate the flow field a cavity with weapons. After sufficient flow field time iterations, the effect of the initial flow field can be ignored. The calculation of cavity flow does not use the URANS method, and any flow field after full iteration using the RANS method is used as the initial flow field of the unsteady DES simulation flow. After the DES method simulates the flow field for an unfixed period of time, three different moments of flow are taken as the initial moments for the missile to separate from the bay. Thereafter, the initial flow field at this moment was used to perform missile separation simulation. In this paper, the three initial moments of the simulation of the flow field by the DES method are 0.2s, 0.22s, and 0.24s, respectively.
Fig. 8 shows the flow line of the symmetry plane of the weapon bay at the initial moment of separation. The flow at several initial moments is different. However, on the whole, there is a very strong flow shear layer in the upper part of the weapon bay. The shear layer interferes with the upper part of the weapon bay, causing the shear layer to fluctuate continuously. Under the action of the shear layer, the cavity forms a circular flow. The air flow above the missile flows backwards and the flow below the missile flows forward. Due to the complex flow of the cavity, there is a very strong three-dimensional effect.

Fig. 9 shows the flow field state of the cavity containing the missile. Although the inflow and boundary conditions are completely symmetric, it can be seen from Fig.9 that there is no symmetry in the flow, and the left and right flows also seem to fluctuate randomly. Fig.10 shows the pressure distribution on the symmetry line at the bottom of the cavity, showing that the surface pressure calculated by the DES-SST method is also pulsating. The following will study the changes in the field separation characteristics of a missile under the action of this pressure pulsation flow.

Fig. 11 shows the meshing of the moving mesh during weapon separation. It can be seen from the figure that the viscous mesh of the boundary layer of the missile adopts the subdomain control method.
that follows the movement of the missile body and does not deform as a rigid body. There is no decline in the quality of this part of the grid. This is very insignificant for viscous flow calculations.

Figure 12. Symmetrical surface flow lines at different moments in missile separation (Example 2)

Fig. 12 shows the symmetry plane streamline at different moments for missile separation in Example 2 (initial separation time T=0.22s). As the missile passes through the shear layer above the cavity, the complexity of the cavity flow greatly increases. At this point, the high-speed shear layer outside the cavity, under the blocking and guidance of the projectile, rushes into the lower part of the cavity, opposite to the local forward flow direction of the cavity. The impact of the two makes the flow of the cavity more complicated.

Since then, with the gradual upward movement of the projectile, the shear layer completely enters the lower part of the projectile, and relatively smoothly reaches the back of the cavity, and the flow of the cavity is gradually stabilized. Before this, the annular flow of the ballistic body was re-established and gradually increased, lengthened, and strengthened. The flow evolved into a relatively simple cavity flow.
Fig. 13 shows the weapon separation characteristics of the four different initial conditions in the upper section. It can be seen that the different separation initial states due to the complexity of the cavity flow affect the weapon separation characteristics of the buried weapon bay. First of all, calculating the symmetry of the model does not cause the missile's lateral motion to not appear. The calculation results are in accordance with actual experience. There is no complete symmetry in practice. From the above we can also see that even with the URANS method, this phenomenon can be foreseen. Because of the complexity of the cavity flow, the DES method inevitably has the difference in the initial state of the flow field, but from the calculation results, it can be seen that the separation characteristics are similar. With the increase of the dynamic pressure of the flow field, when the aerodynamic interference force and the gravity of the weapon reach the same magnitude, the separation characteristics of the weapon will be more sensitive to the initial state. Using this method, the dispersion of the separation characteristics can be given through multiple calculations.

6. Conclusion
Due to the broad application prospects of buried weapons in aircraft, this paper studies the flow of buried weapon cavities. Due to the complexity of the cavity flow, this paper decided to use the DES method based on the SST model. Firstly, the principle and characteristics of the DES method are analyzed in detail and the DES method based on the M-SST model is given. Then the DES method based on the SST model was used to simulate the cavity flow, the open cavity flow was calculated, and the flow characteristics of such cavity were analyzed. By comparing the numerical simulation of the cavity flow with the experimental results, a numerical simulation method for the flow of the bullet chamber in the DES method based on the SST model was established. At the end of this paper, we attempted to establish a non-structural dynamic grid, based on the DES method of the SST model, and used this method to simulate the separation of weapons from the buried weapon bay. Finally, a preliminary attempt was made to establish a DES method based on the SST model using an unstructured dynamic grid to simulate the separation characteristics of weapons from the buried weapon bay. This method was successfully used to simulate the process of weapon's separation from the aircraft’s buried weapon bay. This method was successfully used to simulate the process of weapon's separating from buried weapon bay of the aircraft. This paper considers the unsteady nature of the cavity flow itself, using the DES method to reflect the aerodynamic random factors that are not unique to the separation path of the actual weapon separation process.
7. References

[1] Werner Haase, Marianna Braza, Alistair Revell. DESider-A European Effort on Hybrid RANS-LES Modelling. Results of the European-Union Funded Project, 2004–2007

[2] Shia-Hui Peng, Werner Haase. Advances in Hybrid RANS-LES Modelling. Papers Contributed to the 2007 Symposium on Hybrid RANS-LES Methods, Corfu, Greece, 17–18 June 2007

[3] Shia-Hui Peng, Piotr Doerffer, Werner Haase. Progress in Hybrid RANS-LES Modelling. Papers Contributed to the 3rd Symposium on Hybrid RANS-LES Methods, Gdansk, Poland, June 2009

[4] Stallings, Robert L., Jr.; and Wilcox, Floyd J., Jr.; Experimental Cavity Pressure Distributions at Supersonic Speeds. NASA TP-2683, 1987.

[5] Tracy, M. B.; Plentovich, E. b.; and Chu, Julio; Measurements of Fluctuating Pressure in a Rectangular Cavity in Transonic Flow at High Reynolds Numbers, NASA TM-4363, 1992

[6] E. B. Plentovich, Robert L. Stallings, M. B. Tracy. Experimental Cavity Pressure Measurements at Subsonic and Transonic Speeds. NASA-TP-3358, 1993

[7] Fox, John H., “23. Generic Wing, Pylon, and Moving Finned Store” RTO-TR-26, Verification and Validation Data for Computational Unsteady Aerodynamics, October 2000