Published by :
https://www.ijert.org/
An International Peer-Reviewed Journal

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 06 , June - 2026

Coupled Multibody Dynamics-CFD and Two-Way Fluid-Structure Interaction Analysis of Aircraft Weapon Bay Doors using MSC ADAMS, Cradle scFLOW and MSC Nastran

Karthik A.V[1], Balaji S[1], Venkata Subramanian G[1], Gomathi Sankar[1], Chandan V[1]
[1]SOLIZE PARTNERS India Private Limited

Abstract - Aircraft weapon bay systems are subjected to highly unsteady aerodynamic environments during door deployment, resulting in complex interactions between airflow, structural deformation, and mechanical motion. Accurate prediction of hinge loads, pressure fluctuations, transient stresses, and door kinematics is essential for ensuring safe weapon release and structural reliability. Conventional uncoupled computational approaches often fail to capture the mutual interaction between aerodynamic loads and structural response.

This study presents a coupled simulation framework integrating MSC ADAMS, Cradle scFLOW, and MSC Nastran for the aerodynamic and structural analysis of a dual-door aircraft weapon bay. Two simulation methodologies are investigated. The first employs a multibody dynamic–computational fluid dynamics (MBD–CFD) co-simulation between MSC ADAMS and Cradle scFLOW to evaluate transient aerodynamic loads and hinge reaction forces during door deployment. The second utilizes a fully coupled two-way Fluid–Structure Interaction (FSI) methodology between scFLOW and MSC Nastran to capture the interaction between aerodynamic pressure loads and structural deformation.

The coupled simulations successfully predict transient pressure distributions, velocity fields, hinge reaction forces, structural stresses, and deformation behavior throughout the deployment sequence. Results indicate that aerodynamic loads significantly influence hinge forces and structural response. The proposed methodology provides a reliable virtual testing environment for aircraft weapon bay systems and reduces dependency on expensive experimental testing.

Keywords: Weapon Bay, Fluid Structure Interaction, MSC ADAMS, MSC Nastran, scFLOW, Co-Simulation, CFD, Aerospace Engineering.

1. INTRODUCTION

Modern combat aircraft employ internal weapon bays to reduce radar cross-section (RCS) and aerodynamic drag while improving stealth performance and mission survivability. Unlike externally mounted stores, internal bays require door systems that open into the freestream during weapon deployment, exposing them to complex and highly unsteady aerodynamic environments.

During deployment, bay doors interact with high-speed external airflow, leading to complex flow phenomena such as cavity oscillations, shear-layer instabilities, vortex shedding, pressure fluctuations, and recirculation zones inside the cavity. These flow features are inherently transient and strongly influenced by door motion, geometry, and operating conditions.

Such aerodynamic disturbances significantly affect multiple aspects of weapon bay performance, including:

- Door opening kinematics
- Hinge reaction forces and actuator loads
- Structural deformation and stress distribution
- Fatigue life of door mechanisms
- Weapon release trajectory and safety

In addition, the interaction between the shear layer spanning the cavity opening and the internal recirculating flow often produces large-amplitude pressure oscillations, which can induce vibration and structural loading. These effects become particularly critical during the initial phases of door opening, where rapid changes in flow topology occur.

Traditional simulation approaches often treat the problem in a decoupled manner. Computational fluid dynamics (CFD) analyses typically assume prescribed rigid-body motion of doors, neglecting structural flexibility and dynamic feedback from aerodynamic loads. Conversely, structural analyses often rely on simplified or static pressure distributions, ignoring the transient nature of aerodynamic forces. Such uncoupled methodologies fail to capture the strong interaction between fluid flow, structural deformation, and multibody motion, leading to inaccurate predictions of loads and structural response.

With advances in computational power and numerical algorithms, coupled simulation techniques have emerged as a powerful approach to address these limitations. Co-simulation frameworks enable simultaneous interaction between multiple physical domains, allowing realistic modeling of aero-mechanical systems. In particular, the integration of multibody dynamics, CFD, and finite element structural analysis provides a comprehensive representation of the system behavior.

In this context, the present study investigates two advanced coupled methodologies:

1. MSC ADAMS–scFLOW co-simulation, which captures the interaction between aerodynamic loads and multibody door kinematics
2. MSC Nastran–scFLOW two-way Fluid–Structure Interaction (FSI), which accounts for mutual coupling between aerodynamic pressure and structural deformation

The primary objective of this work is to evaluate the transient aerodynamic behavior, hinge loads, structural stresses, and deformation characteristics of a dual-door aircraft weapon bay under realistic operating conditions. The study aims to demonstrate the necessity and effectiveness of coupled simulation approaches in accurately predicting the complex physical interactions governing weapon bay systems.

LITERATURE REVIEW

Fluid–structure interaction (FSI) and multibody–aerodynamic coupling have become essential in analyzing complex aerospace systems such as aircraft weapon bays, where fluid flow, structural response, and kinematics are strongly interdependent. Early studies in computational fluid dynamics (CFD) established the fundamental governing equations and numerical approaches for simulating aerodynamic flows [1], [2]. However, these methods were primarily applied in uncoupled frameworks, limiting their ability to capture transient interactions between flow and structures.

Cavity flow behavior, which is highly relevant to weapon bay configurations, has been extensively studied due to its unsteady and oscillatory nature. Rossiter’s experiments [10] provided foundational insights into cavity flow oscillations, while later works highlighted the role of vortex shedding, recirculation, and shear-layer instabilities in generating pressure fluctuations [11], [19]. These aerodynamic phenomena significantly influence structural loading and vibration characteristics.

To address such challenges, aeroelasticity-based approaches have been developed to account for the interaction between aerodynamic forces and structural deformation. Dowell [3] emphasized the importance of coupling fluid dynamics with structural mechanics for predicting aeroelastic responses. Similarly, Blevins [4] investigated flow-induced vibration mechanisms, demonstrating how unsteady fluid loading can lead to structural fatigue and resonance.

Advancements in numerical methods have enabled the development of partitioned and coupled FSI frameworks. Tezduyar [16] and Bazilevs et al. [15] presented finite element-based approaches for solving FSI problems, highlighting the importance of accurate load transfer and mesh deformation techniques. Farhat et al. [17] and Piperno et al. [18] further improved partitioned coupling strategies by introducing efficient algorithms for data exchange between fluid and structural solvers.

In parallel, overset mesh and moving grid techniques have significantly enhanced the capability to simulate moving bodies in fluid domains. Henshaw and Schwendeman [13] demonstrated the effectiveness of overlapping grids in handling large motions without remeshing, which is particularly suitable for rotating door simulations in weapon bays.

Recent developments in industrial simulation tools have enabled high-fidelity co-simulation environments. MSC ADAMS provides robust multibody dynamic simulations, while MSC Nastran is widely used for structural analysis [5], [6]. Cradle scFLOW offers advanced CFD capabilities for complex turbulent flows [7], incorporating turbulence models such as the SST k–ω model proposed by Menter [8], which is well-suited for separation-dominated flows.

Despite these advancements, many studies still rely on either prescribed motion in CFD or static loading in structural analysis, which limits prediction accuracy. Fully coupled approaches combining multibody dynamics, CFD, and structural solvers are increasingly recognized as necessary for realistic simulation of aerospace systems. Such integrated methodologies enable accurate prediction of hinge loads, transient stresses, and deformation behavior under dynamic aerodynamic conditions.

2. WEAPON BAY CONFIGURATION

A. Geometric Model

The weapon bay configuration considered in the present study consists of a rectangular cavity integrated into the aircraft fuselage and two hinged doors, namely the **outboard door** and the **inboard door**. These doors are designed to open sequentially during weapon deployment, enabling smooth release while minimizing aerodynamic disturbance.

The cavity represents the internal storage region for weapons and is modeled with a length of **4.90 m** and a width of **1.00 m**, providing a slender geometry typical of modern fighter aircraft bays. The structural thickness of the cavity walls is assumed to be **10 mm**, ensuring adequate stiffness to withstand aerodynamic loading.

The **outboard door**, with a length of **0.70 m**, is directly attached to the cavity structure through a revolute hinge and serves as the primary interface with the external airflow.

The **inboard door**, having a slightly larger length of **0.75 m**, is connected to the outboard door, forming a multi-body hinged mechanism. Both doors have a uniform thickness of **5 mm**, representing lightweight structural panels commonly used in aerospace applications.

This dual-door arrangement allows for controlled opening kinematics and reduces aerodynamic interference during weapon release. The geometric configuration also enables the investigation of **aerodynamic interaction between the two doors**, as well as their combined influence on cavity flow behavior.

The complete geometric parameters used in the simulation are summarized in **Table I**, and the corresponding three-dimensional model is illustrated in **Figure 1**.

Table I. Weapon Bay Geometry

Parameter	Value
Cavity Length	4.90 m
Cavity Width	1.00 m
Outboard Door Length	0.70 m
Inboard Door Length	0.75 m
Door Thickness	5 mm
Cavity Thickness	10 mm

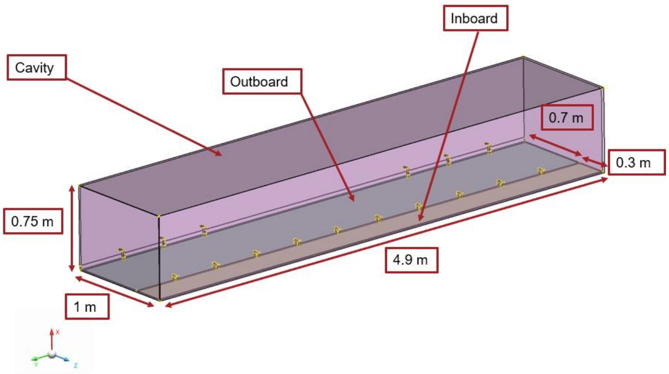

Figure 1. weapon bay geometry showing cavity, inboard door, and outboard door configuration

B. Operating Conditions

The simulations were performed at sea-level conditions with a freestream Mach number of 0.3.

Table II. Flow Conditions

Parameter	Value
Mach Number	0.30
Velocity	100 m/s
Pressure	101325 Pa
Temperature	25°C
Altitude	Sea Level

Door deployment consists of:

- Outboard door rotation = 90°
- Inboard door rotation = 150°

3. NUMERICAL METHODOLOGY

A. MSC ADAMS–scFLOW Co-Simulation

The multibody dynamic model consists of:

- Weapon bay cavity
- Outboard door
- Inboard door

Three joints were defined:

1. Fixed joint between cavity and ground
2. Revolute joint between cavity and outboard door
3. Revolute joint between outboard and inboard door

Door motion was prescribed using STEP functions:

$$\boldsymbol{\theta_2 = \mathrm{STEP}(t,0,0,1,90^\circ)}$$

$$\boldsymbol{\theta_3 = -\mathrm{STEP}(t,0,0,1,150^\circ)}$$

The MSC Co-Simulation Engine was used for bidirectional data exchange between ADAMS and scFLOW.

At each time step:

- Door positions were transferred from ADAMS to scFLOW.
- Aerodynamic loads were calculated in Cradle scFLOW.
- Forces and moments were returned to ADAMS.
- Door motion was updated.

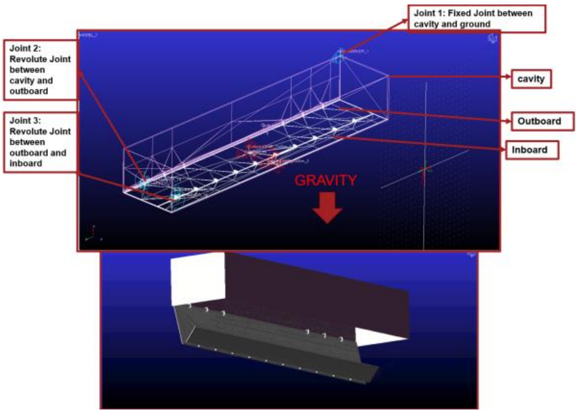

Figure 2. ADAMS multibody model showing cavity, inboard door, outboard door, and revolute joints

B. CFD Model

Aerodynamic simulations were conducted using Cradle scFLOW.

Governing Equations

Continuity Equation:

$$\partial\rho/\partial t + \nabla\cdot(\rho V) = 0$$

Momentum Equation:

$$\rho(\partial V/\partial t + V\cdot\nabla V) = -\nabla p + \nabla\cdot\tau + \rho g$$

Energy Equation:

$$\rho(\partial h/\partial t + V\cdot\nabla h) = \partial p/\partial t + \nabla\cdot(k\nabla T)$$

The SST k–ω turbulence model was selected due to its superior capability in predicting flow separation and adverse pressure-gradient effects commonly observed in cavity flows.

Table III. CFD Solver Settings

Parameter	Value
Solver Type	Density-Based
Turbulence Model	SST k–ω
Time Step	0.0002 s
Iterations	5000

4. MESH GENERATION

Overset mesh technology was employed to accommodate large rotational motion without remeshing.

Table IV. Mesh Statistics

Region	Cell Count
Background Domain	1,583,613
Outboard Overset	184,009
Inboard Overset	386,896
Total Mesh Cells	2,154,518

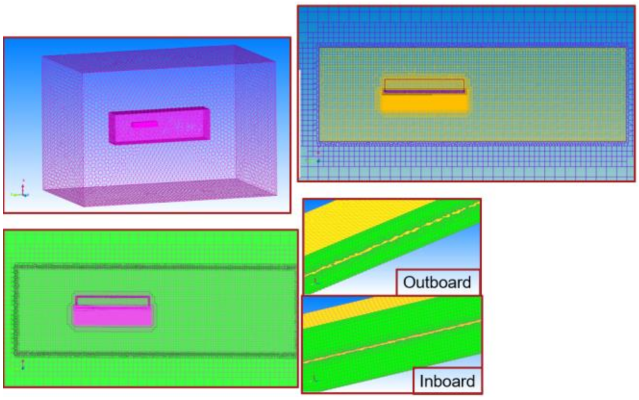

Figure 3. Computational mesh showing background mesh, overset domains, and cavity refinement regions

The finest mesh size of 0.005 m was applied at the overset interface to ensure accurate interpolation between moving and stationary grids.

5. RESULTS AND DISCUSSION

A. Steady-State Flow Analysis

The initial steady-state solution establishes the baseline aerodynamic behavior before door deployment.

Pressure contours indicate flow stagnation near the cavity leading edge, while low-pressure recirculation regions develop inside the cavity.

Velocity contours reveal peak flow velocities approaching the inlet velocity of approximately 100 m/s near the cavity opening.

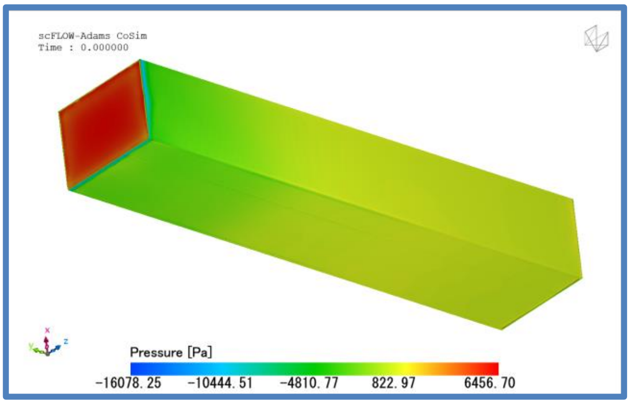

Figure 4. Steady-state pressure contour around the weapon bay cavity

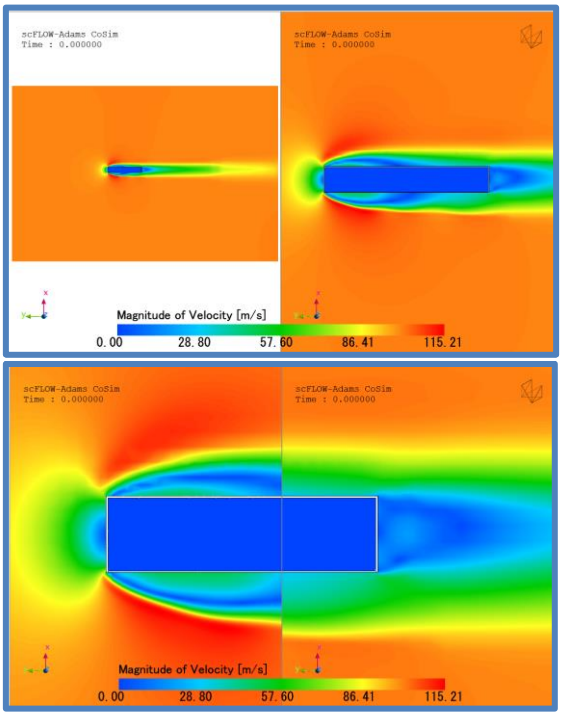

Figure 5. Velocity contour and velocity vectors showing cavity recirculation.

B. Transient Pressure Distribution

During door deployment, significant pressure fluctuations are observed inside the cavity.

Initially, localized pressure concentrations occur near hinge regions due to rapid flow impingement. As deployment progresses, large-scale vortical structures form within the cavity, resulting in strong pressure gradients.

Simulation results indicate pressure variations approaching ±15–20% of freestream pressure near door leading edges.

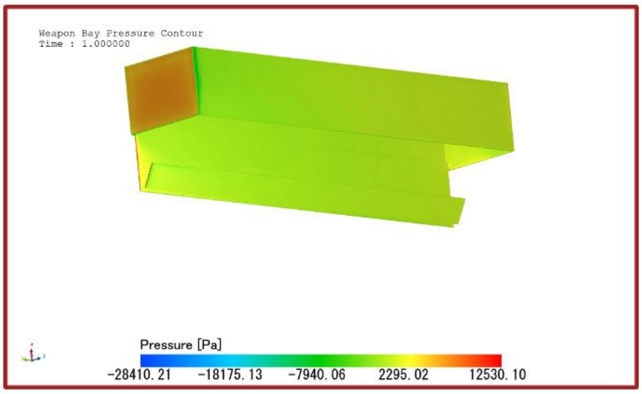

Figure 6. Transient pressure contour at intermediate door deployment

The highest-pressure loading occurs during the initial 20–40% of door opening when flow separation is strongest.

C. Velocity Field Evolution

Transient velocity contours demonstrate the development of:

- Shear-layer instabilities
- Flow separation
- Recirculation bubbles
- Vortex structures

Peak local velocities exceed 110 m/s near the cavity entrance due to flow acceleration around the door edges.

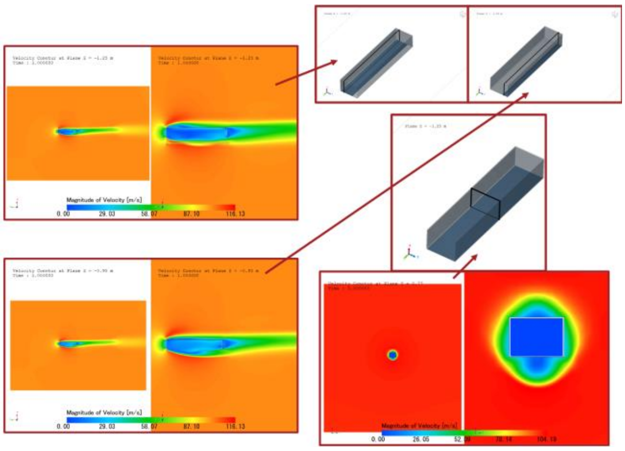

Figure 7. Velocity contour showing transient cavity flow structures during deployment

These high-gradient regions are responsible for significant aerodynamic loading on the door surfaces.

D. Joint Force Analysis

Joint reaction forces were extracted directly from the ADAMS–scFLOW co-simulation.

Joint 2 (Cavity Bracket)

Joint 2 experiences the largest aerodynamic loading.
Results indicate:

- Rapid force increase during initial deployment
- Maximum force occurs between 20–40% opening
- Peak loads exceed gravitational effects by several times

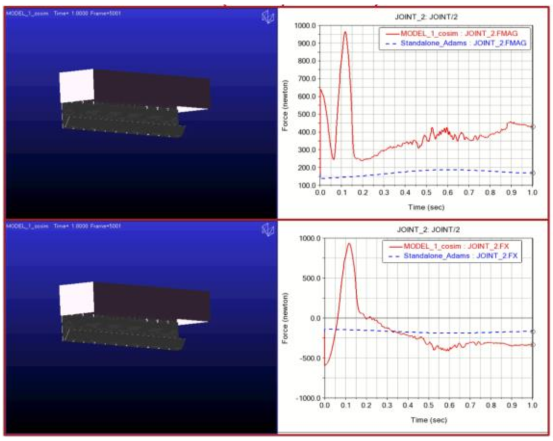

Figure 8. Joint reaction force history at Joint 2 (Cavity Bracket)

Joint 3 (Inboard Door)

Joint 3 experiences lower mean loads but larger oscillatory fluctuations caused by wake interaction from the outboard door.

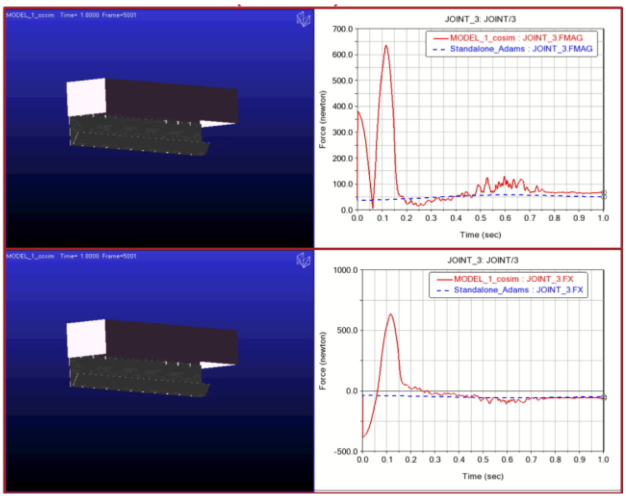

Figure 9. Joint reaction force history at Joint 3 (Inboard Door)

The results demonstrate the importance of aerodynamic coupling when designing hinge mechanisms and actuators.

6. TWO-WAY FLUID STRUCTURE INTERACTION

A. Coupling Methodology

The two-way FSI process consists of:

1. Pressure calculation in scFLOW
2. Load transfer to MSC Nastran
3. Structural deformation calculation
4. Geometry update
5. Flow field recalculation

This process continues iteratively until convergence.

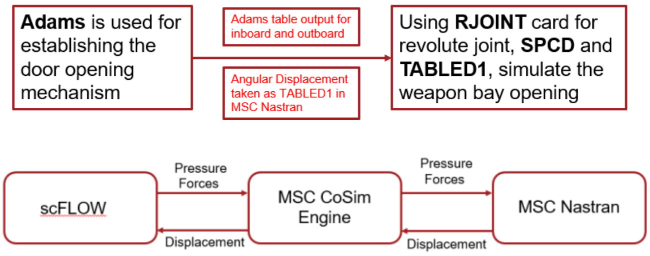

Figure 10. Two-way coupling architecture between scFLOW and MSC Nastran

B. Structural Response

Deformation Results

Maximum global displacement of the door is governed by its rigid-body rotation during deployment, resulting in a total positional change of approximately **0.94 m** from the initial closed configuration to the fully open position. This displacement represents the overall motion of the door rather than structural deformation.

In contrast, **local structural deformation** due to aerodynamic loading is relatively small and is captured separately in the deformation contour results. The door experiences **elastic bending**, with maximum deformation occurring at the **free edges**, where stiffness is lowest and aerodynamic pressure effects are most pronounced. The hinge regions remain comparatively rigid due to boundary constraints.

The localized deformation magnitude is observed to be approximately **14 mm**, indicating that although the door undergoes large rigid-body motion, the structural deflection remains within a small range. This confirms that the door behavior is dominated by controlled rotation, with limited elastic deformation under aerodynamic loads.

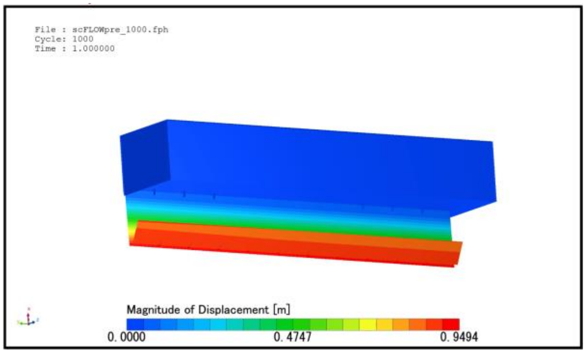

Figure 11. Total deformation contour obtained from two-way FSI simulation

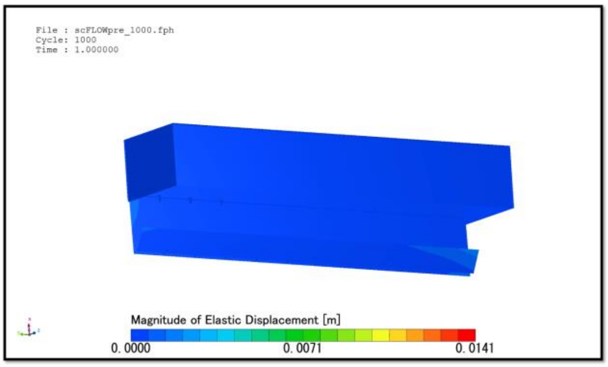

Figure 12. Elastic Deformation contour obtained from two-way FSI simulation

Maximum displacement is observed near the trailing edge and reaches approximately 14 mm depending on deployment angle.

Stress Distribution

Von Mises stress contours indicate stress concentration near hinge attachment locations.

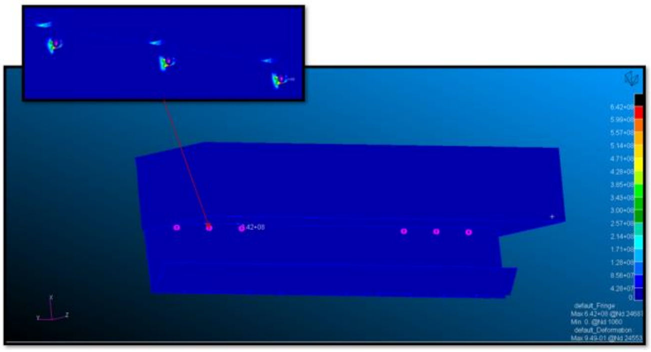

Figure 13. Von Mises stress distribution during peak aerodynamic loading

Maximum stresses are observed during the **rapid acceleration phases** of door deployment, where aerodynamic loads change abruptly. These stresses are predominantly **concentrated near the hinge (revolute joint) locations**, as these regions act as load transfer points between the door and the supporting structure.

Due to the boundary constraints at the hinges, these areas experience the highest load intensities, resulting in **peak stress concentrations**. In contrast, the inclusion of structural flexibility allows for redistribution of stresses across the door surface, reducing localized peaks compared to idealized rigid-body assumptions.

However, the **maximum Von Mises stress consistently occurs at the hinge interfaces**, indicating that these regions are the most critical from a structural integrity and fatigue perspective.

7. COMPUTATIONAL PERFORMANCE

Table V. Computational Resources

Parameter	ADAMS-scFLOW	Nastran-scFLOW
RAM	256 GB	256 GB
CPU	16 Cores	16 Cores
Mesh Cells	2.15 Million	3.08 Million
Iterations	5000	1000
Solution Time	21 Hours	41 Hours

The two-way FSI simulation required approximately twice the computational effort because of continuous geometry updates and structural calculations.

8. CONCLUSIONS

A comprehensive co-simulation framework integrating MSC ADAMS, scFLOW, and MSC Nastran was successfully developed for aircraft weapon bay analysis.
Major conclusions are summarized as follows:

1. ADAMS–scFLOW co-simulation accurately captured transient aerodynamic loads and hinge reaction forces during door deployment.
2. Overset mesh technology enabled robust simulation of large rotational motion without remeshing.
3. Peak aerodynamic loading occurred during the initial stages of door deployment due to flow separation and cavity interaction.
4. Two-way FSI analysis successfully captured the interaction between aerodynamic pressure and structural deformation.
5. Structural flexibility altered pressure distribution and stress response, demonstrating the necessity of coupled analysis for accurate prediction.
6. The proposed methodology provides a reliable virtual-testing platform for future weapon bay design and optimization.

Future work will focus on higher Mach-number conditions, aeroacoustics cavity resonance prediction, and weapon release simulations under coupled aerodynamic environments.

REFERENCES

[1] J. D. Anderson, *Computational Fluid Dynamics: The Basics with Applications*, McGraw-Hill, 1995.
[2] J. H. Ferziger and M. Perić, *Computational Methods for Fluid Dynamics*, Springer, 2020.
[3] E. H. Dowell, *A Modern Course in Aeroelasticity*, Springer, 2015.
[4] R. D. Blevins, *Flow-Induced Vibration*, Krieger Publishing, 2016.
[5] MSC Software, *MSC ADAMS User Guide*, Hexagon AB, 2025.
[6] MSC Software, *MSC Nastran Dynamic Analysis User Guide*, Hexagon AB, 2025.
[7] Software Cradle, *scFLOW Technical Documentation*, Hexagon AB, 2025.
[8] F. R. Menter, “Two-equation eddy-viscosity turbulence models for engineering applications,” *AIAA Journal*, vol. 32, no. 8, pp. 1598–1605, 1994.
[9] T. Colonius, “Modeling artificial boundary conditions for compressible flow,” *Annual Review of Fluid Mechanics*, vol. 36, pp. 315–345, 2004.
[10] J. Rossiter, “Wind tunnel experiments on cavity flows,” *Aeronautical Research Council Reports and Memoranda*, 1964.
[11] D. Rizzetta and M. Visbal, “Large-eddy simulation of supersonic cavity flows,” *AIAA Journal*, vol. 41, no. 8, pp. 1452–1462, 2003.
[12] R. Henderson, “Cavity aeroacoustics: The past, present and future,” *AIAA Paper 2000-1905*, 2000.
[13] M. Henshaw and D. Schwendeman, “Moving overlapping grids for high-speed flows,” *Journal of Computational Physics*, vol. 227, pp. 7469–7502, 2008.
[14] K. J. Bathe, *Finite Element Procedures*, Prentice Hall, 2014.
[15] Y. Bazilevs et al., “Computational fluid–structure interaction,” *Computational Mechanics*, vol. 43, pp. 3–37, 2008.
[16] T. Tezduyar, “Finite element methods for fluid–structure interaction,” *Archives of Computational Methods in Engineering*, vol. 13, pp. 83–130, 2006.
[17] C. Farhat, M. Lesoinne, and P. Le Tallec, “Load and motion transfer algorithms for FSI problems,” *Computer Methods in Applied Mechanics and Engineering*, vol. 157, pp. 95–114, 1998.
[18] S. Piperno, C. Farhat, and B. Larrouturou, “Partitioned procedures for transient aeroelastic simulations,” *Computer Methods in Applied Mechanics and Engineering*, vol. 190, pp. 3147–3170, 2001.
[19] A. Rona and K. Knowles, “Experimental investigation of cavity flow oscillations,” *Aeronautical Journal*, vol. 108, pp. 221–235, 2004.
[20] M. Visbal and D. Rizzetta, “Large-eddy simulation of cavity flowfields,” *Progress in Aerospace Sciences*, vol. 38, pp. 437–452, 2002.

We would like to extend our sincere gratitude **Cadence** for providing their **MSC ADAMS, CRADLE ScFLOW and MSC Nastran** software. Utilizing this advanced simulation tool has enabled us to achieve superior results compared to traditional numerical calculations. Your support has been invaluable in enhancing the accuracy and efficiency of our simulations.
Thank you for your continued partnership and contribution to our success.