https://www.hpc.mil/images/hpcdocs/newsroom/ugc_2011_proceedings_small.pdf

Computational Modeling of Geometrically Complex Modern Weapons Bays and Weapons Dispense at High Supersonic Speeds

Rudy Johnson
US Air Force Research Laboratory, Air Vehicles Directorate (AFRL/RB), Wright-Patterson AFB, OH
rudy.johnson@wpafb.af.mil

Chandrasekhar Kannepalli, Chris Chartrand, Roger Birkbeck, and Neeraj Sinha
Combustion Research and Flow Technology, Inc. (CRAFT Tech), Pipersville, PA
{kchandra, ccc, rbirkbeck, sinha} @craft-tech.com

Abstract

Computational modeling of the air flow in a geometrically-complex weapons bay, as well as the separation of weapons from a generic cavity model into a high-Mach (3.5–5.0) free stream, are summarized. Computational work on the geometrically-complex bay is focused on flow control strategies for the reduction of dynamic pressure loads. Results indicate significantly different acoustic characteristics for the bay when one versus two doors is open. When both bay doors are open, leading-edge mass blowing combined with aft-wall treatment is an effective approach to reduction of dynamic pressure loads. However, the effectiveness of this control strategy is reduced when only one bay door is open, due to the generation of strong resonant tones. Computational fluid dynamics (CFD) analysis has led to the development of novel, easily integratable "passive" control strategies involving "baffles" that have shown significant reduction in the tonal amplitude. The CFD simulations have enabled a good understanding of the complex flow field, led to the development of control strategies, and have guided the sub-scale wind-tunnel test program. Significance to the Department of Defense (DoD) is the integration of successful acoustic reduction concepts into realistic aircraft weapons bays.

CFD simulations of weapons separation from a generic bay at high-speed are being used to develop data acquisition techniques for sub-scale wind-tunnel testing in the high supersonic regime. The bays at these high-Mach numbers involve spatial and temporal flow field scales that are an order-of-magnitude smaller than those at Mach 1.5–2.0 regime, making the numerical simulations of such bays computationally more expensive. The availability of the high performance computing resources towards this end is gratefully acknowledged, while the significance to DoD is the improved understanding of the associated flow physics that will help in the better design of data acquisition techniques for store dispense at these high supersonic Mach numbers.

1. Introduction

This paper provides a sampling of the computational modeling accomplished during the third year of the high performance computing (HPC) Challenge Project "Weapons Bay Aeroacoustic Load Suppression and Separation Enhancement Simulations." Previous work included weapons bay acoustic modeling using the CRAFT CFD® software applied to a generic box cavity on a flat plate, and to weapons bay geometries representative of the F-22[1], and F-35[2] aircraft. The computational modeling has guided the design, development, and evaluation of weapons bay acoustic reduction techniques. Computational fluid dynamics (CFD) results were also used to support small-scale experiments conducted at the National Center for Physical Acoustics (NCPA) at the University of Mississippi, Oxford, Mississippi.

Since the previous User's Group Conference, the small-scale wind-tunnel model representing the F-35 weapons bay has been redesigned to allow flow through the aircraft inlet. In addition, the scale of the model was increased from 1/20th to 1/15th as a part of the redesign (Figure 1). CFD analysis was used to design the internal flow path to reduce inlet spillage that could contaminate the boundary-layer approaching the weapons bay. Two baseline aircraft configurations are examined. Selected flow control concepts are discussed and model results are presented with experimental data.

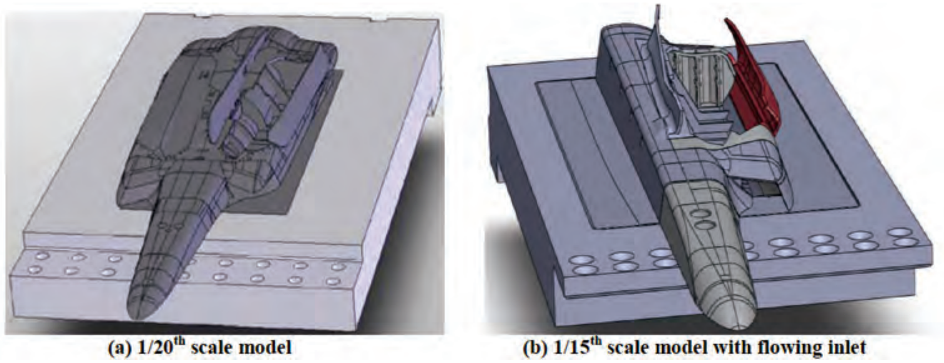


Figure 1. Small-scale weapons bay acoustics wind-tunnel model representative of the F-35

The unsteady flow character of the weapons bay can also have significant consequences for internally carried stores, either through damage to sensitive electronics or structural components. In addition, the unsteady store loads encountered can influence store separation trajectories[3]. Small-scale drop testing is a technique used to study store separation in a laboratory environment. The current research in this area attempts to synchronize the data collection to develop an understanding of the effect that the unsteady flow has on the store separation trajectory. Photogrammetric techniques are currently used to track the store motion, but a current project to develop small-scale telemetry packages for data acquisition during drop testing is underway. Computational modeling of a generic box cavity in a flat plate is accomplished with a store positioned in the opening of the cavity. The resulting store loads are used to establish an estimate of the design specifications for this telemetry package.

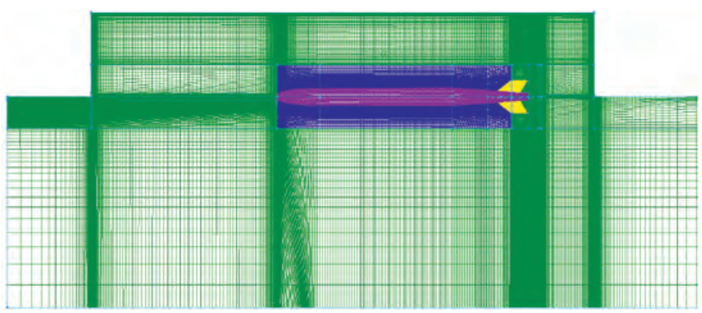


Figure 2. Computational mesh for a store located in the opening of THE L/D=6 generic bay

The remainder of the paper will first discuss the use of CFD and small-scale wind-tunnel testing to study the unsteady flow field in a weapons bay that is representative of the Joint Strike Fighter (JSF) aircraft at supersonic Mach number of 1.5. A description of the computational approach to collecting unsteady store loads in a generic cavity is given. Details of the computational methodology will be presented, followed by a discussion of the weapons bay behavior for two baseline configurations involving both bay doors open and one door closed. An analysis of one of the acoustic control strategies will be provided then a look into the store loads experienced in a Mach 3.5 cavity are presented. Finally, the conclusions and recommendations of these studies will be presented.

2. Modeling Approach

The process used for the acoustic modeling and the store loads are described. A short description of the CFD software completes this section.

2.1 Weapons Bay Acoustics

As indicated in Figure 1b, an approximate model of the F-35 bay has been designed that is floor-mounted. The original CAD model of the full aircraft was cropped along a plane parallel to the aircraft axis[2]. Care was taken to position the model relative to the wind-tunnel boundary-layer and to incorporate an inlet geometry with flow-through ducting to ingest the approaching flow; thus preventing it from spilling over the model and into the bay (an improvement over the previous 1/20th scale model, Figure 1a). CFD was used to carefully design the duct to insure venting at the back of the model, downstream of the bay. The details of the upstream fore-body, the nose and the engine inlet are included to provide as realistic an inflow to the weapons bay as possible. Since the size of the model has been increased, the left-hand portion of the aircraft was removed to minimize blockage effects during tunnel operation. Geometric details of the complex internal structures of the bay, as well as the bay doors in two different configurations, are modeled (Figure 1b).

Since, modeling the entire flow field around the model in a single simulation is prohibitively expensive, a zonal approach involving three different domains is used. An initial Reynolds-averaged Navier-Stokes (RANS) simulation of the wind-tunnel nozzle and empty test section provides the inflow conditions to a second RANS simulation involving the fore-body model mounted on the test section floor (Figure 3). The latter fore-body simulation provides the inflow conditions to the Large-eddy simulations (LES) of the complex weapons bay. Figure 3a shows the computational extents of the domains for the three zonal simulations, while Figure 3b shows the Mach number contours on the mid-span plane passing through the center of the bay, demonstrating the continuity of the solutions across the different zonal simulations. The wind-tunnel simulation will not be discussed, but the fore-body simulation that provides the inflow boundary-conditions to the weapons bay LES on a plane upstream of the bay will briefly be discussed below.

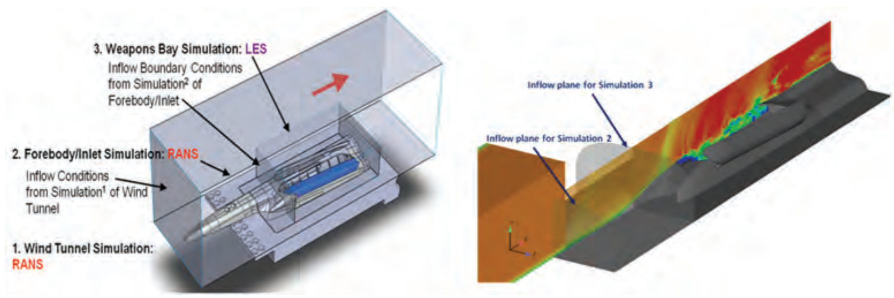
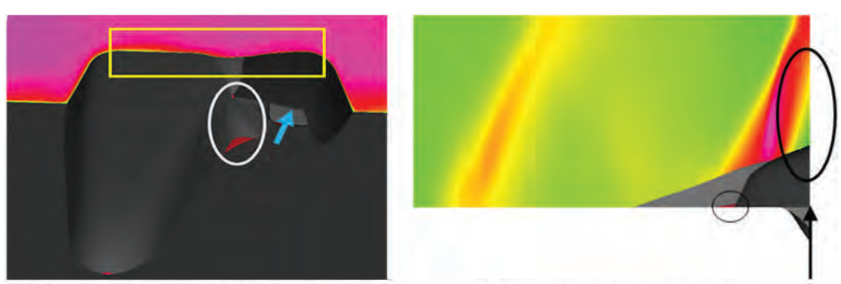


Figure 3. Zonal approach used for the weapons bay simulations

Figure 4a shows the fore-body geometry and the inlet along with iso-surfaces of separated flow regions (in red). Also shown on the far background are the mean U-velocity contours on the inflow/approach plane (that is just upstream of the weapons bay leading-edge) to the weapons bay simulation. Figure 4b shows the mean pressure contours and the iso-surfaces of separated flow zones in the mid-span plane of the bay. Clearly, the flow-through inlet allows the impinging flow to pass with minimal separated flow zones (that are very small); and hence does not result in any flow spillage into the bay. As a result, a nice, "clean" boundary-layer-type of flow approaches the bay, as can be seen from the U-velocity contours on the far background plane in Figure 4a. Further, the inlet only causes a weak shock structure with "quicker" recovery of the pressure to the nominal Q_∞ conditions upstream of the bay leading-edge. In an earlier paper, Johnson, et al.[2] showed that the modeling of the flow-through inlet is very important to assure a clean and uncorrupted flow entering the weapons bay, as would be the case for the prototype aircraft.



(a) Separated flow regions & u-velocity on approach plane of bay

(b) Pressure on the bay mid-span plane

Figure 4. Fore-body/inlet RANS simulation

2.2 Store Loads

The intention of this work is to develop an understanding of the magnitude and frequency content of store loads for store released from a weapons bay into a Mach 3.5 free stream. This data will be used to guide the selection of sensors (accelerometers & telemetry hardware) to capture the store loads during small-scale wind-tunnel drop testing. Since Mach 3.5 is beyond the typical flow regime where the CRAFT CFD® solver is typically run, an empty bay solution is generated to work out the grid resolution and time-step requirements. The basic approach is the same as describe above; a RANS model of the blow down tunnel nozzle and empty test section is accomplished to provide the boundary-conditions for an LES simulation of the generic L/D=6 weapons bay, Figure 5. A vertical slice of the computational mesh along the centerline is shown in Figure 2, with the store model positioned in the opening of the cavity. The cavity mesh alone contains ~7.5 million points, while the total from the cavity and store is ~14.7 million.

Figure 5. NCPA wind-tunnel nozzle & L/D=6 cavity mounted on the tunnel ceiling

The RANS simulation indicates the tunnel boundary-layer is ~1.45 in. thick as it approaches the bay. This will need to be controlled to be more representative of expected flight conditions, but for the current modeling effort this profile is used. The time-step required to model the unsteady flow is 2.0e-8 seconds, which is 5 to 6 times smaller than what was previously required for Mach 1.5 & 2.0 acoustic modeling for the same bay. The empty bay calculation shows that boundary-layer; and hence the shear-layer over the cavity are more stable than in lower Mach number flow fields due to the smaller, less coherent shear-layer structures. Similar observations were reported by Murray & Elloit[4].

2.3 CFD Software

All the numerical simulations in the present effort have been carried out using CRAFT Tech's CRAFT CFD® software[5]. CRAFT CFD® is a multi-block, structured flow solver. For unsteady LES, the spatial numerical scheme implemented uses a higher-order (fifth-order) upwind-biased reconstruction procedure[6,7] in conjunction with Roe's approximate Riemann solver. The fifth-order reconstruction scheme permits the resolution of waves using a minimal number of points, while also minimizing the numerical dispersion and dissipation errors. For problems dominated by the interaction of physical structures at scales only just larger than the grid scale, such a scheme is mandatory. The RANS models in CRAFT CFD® are based on a formulation of the k-ε turbulence model that is capable of modeling free-shear flows and near-wall flows within a unified framework[8]; while two sub-grid scale models, the Smagorinsky model, and a one-equation sub-grid scale kinetic energy model, are available for LES applications. In addition, a hybrid RANS-LES model[12] coupling the RANS k-ε turbulence models with the one-equation sub-grid scale kinetic energy model of LES is also available. The CRAFT CFD® code has been validated for various LES applications, including cavity flows[9–12]. In particular, it was rigorously validated (as part of a previously-funded US Air Force Research Laboratory (AFRL) program involving Separation Enhancement and Acoustic Reduction (SEAR)[15,16,12,13,14]) with wind-tunnel test data for the prediction of aero-acoustic loads in cavity flows at both subsonic and supersonic flow conditions. The time integration schemes in CRAFT CFD® include a four-stage, low-storage, second-order Runge-Kutta scheme, a directional ADI solver that is both computationally efficient and inexpensive in memory usage, and a fully-implicit, symmetric, Gauss-Seidel scheme. Sub-iterations are used to achieve second-order temporal accuracy. For efficient computation of large three-dimensional (3D) problems, a parallel framework based on distributed memory architectures has been implemented; wherein the computational domain is decomposed into sub-domains of equal size, and each of these sub-domains is solved for on different processors. Exchange of information across processors is implemented via the Message Passing Interface (MPI) libraries. The parallel efficiency of the code scales very well, and it runs successfully on several operating system platforms[1].

3. Results

3.1 Weapons Bay Acoustics

The simulations in this study were performed for a free stream Mach number, $M\infty=1.5$, and at a dynamic pressure, $Q_\infty=12.6$ psi. Two baseline configurations are considered, both doors open (Figure 6a) and outboard door closed (Figure 6b). In addition, flow control strategies designed and tested for each configuration will be discussed.

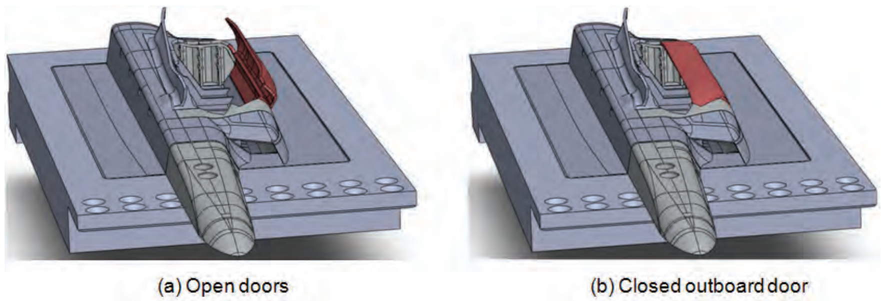


Figure 6. Bay door configurations analyzed

A very high-resolution, structured grid was developed for the wind-tunnel models that incorporated the external features (fore-body, engine inlet, fuselage undercarriage, weapons bay doors, etc.), as well as the dominant internal features, (the engine duct and a few bulkheads). Much of the grid generation effort was spent developing the computational mesh for the complex bay (approximately 10 million grid-points). Although the cavity was simplified significantly for the sub-scale testing, the geometry inside the cavity still remains complex from the perspective of making a structured grid that is sufficient to run a fifth-order spatial accurate solution via LES. The simulations presented here did not use any sub-grid scale model, and hence used a MILES approach. The simulations used the implicit ADI time integration scheme with three sub-iterations to minimize the linearization error on the implicit side. A constant physical time-step equivalent to 0.1 microseconds is used for advancing the flow solution in a time-accurate sense.

The flow field in the simulation is initialized with no-flow in the bay and the free stream conditions outside of the bay. The unsteady weapons bay flow field naturally evolves as the solution is advanced in time. At some point during the evolution, the turbulent unsteady flow will reach a statistical steady-state. Subsequently, snapshots or ensembles of the flow field solution are saved over a time period equivalent to 0.03s for statistical averaging. This corresponds to about 25 flow-through time periods, t_c (where t_c=L/U_c, L is the length of the cavity and U_c~0.57U_∞). Note that 0.57 is the κ value typically used in the Rossiter[17] or Heller-Bliss[18] empirical formula for evaluating the Rossiter or resonant modes/frequencies associated with the flow over a cavity. Here, U_c (where U_c=κU_∞) is the mean convection speed of vortical structures/disturbances travelling downstream along the shear-layer above a cavity. Each simulation required about 200 CPU-hours with 128 processors on an IBM P5 machine.

3.1.1 Analysis of Baseline Configurations

The non-uniform "tooth-like" leading-edge for the weapons bay (Figure 7a) results in differential flow expansion into the bay. This sets up a strong stream wise vortex that entrains high-speed free stream fluid into the bay. The plots in Figures 7b and c of the iso-surface of the mean U-velocity (equivalent to 325m/s and corresponding to a local Mach number ~1.0), colored by the depth (of high-speed flow penetration) into the bay relative to the tip/edge of the bay show a "groove-like" structure that is an imprint of the above-discussed vortex. It is also representative of the mean shear-layer shape over the bay. Since the iso-surface level closely corresponds to the sonic Mach number, the flow above this surface can be said to be nominally supersonic, and that below it to be subsonic. For the open door configuration in Figure 7b, the groove-like structure is deeper and extends to a greater distance downstream in the bay; hence the vortical flow induced is stronger. For the closed door configuration (Figure 7c), the tooth-like structure on the leading-edge of the bay is partially closed by the door; hence the differential flow expansion into the bay is reduced, resulting in a weaker vortex. Consequently, the iso-surface "groove" is shallower, implying that the high-speed fluid entering the bay is reduced; hence the span wise curvature of the shear-layer is smaller. Note that in both of the baseline configurations; the flow predominantly heads straight down the bay with higher speed fluid along the inboard side.

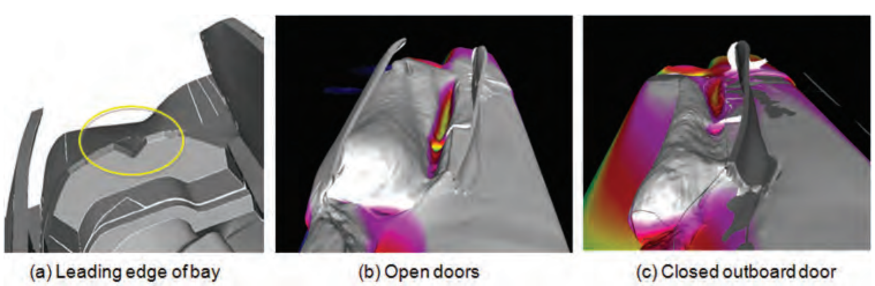


Figure 7. Iso-surface of U=325m/s colored with distance (depth) from bay edge

The mean-flow streamlines colored by the local speed in Figure 8 show that for the open door configuration, the main (high-speed) flow accelerates as it dips down into the bay. It forms a few re-circulation bubbles along the outboard side of the bay, but no distinct re-circulation pattern sets up that extends the entire length of the bay. The differential flow pattern due to the non-straight leading-edge is highlighted, and can be seen to be higher for the open doors configuration. This involves a larger turning of the flow from the entire leading-edge region to its straightening along the inboard side of the bay. For the closed door configuration, the main high-speed flow heads straight along the inboard side of the bay impinging on the inboard corner of the aft-wall. It then re-circulates along the outboard side, forming one big re-circulation zone communicating fluid from the aft-end to the front-end of the bay. The predominantly lighter colors on the inboard side and darker (blue) colors of the streamlines along the outboard side imply high-speed flow (in the downstream direction) and low-speed flow (in the upstream direction), respectively.

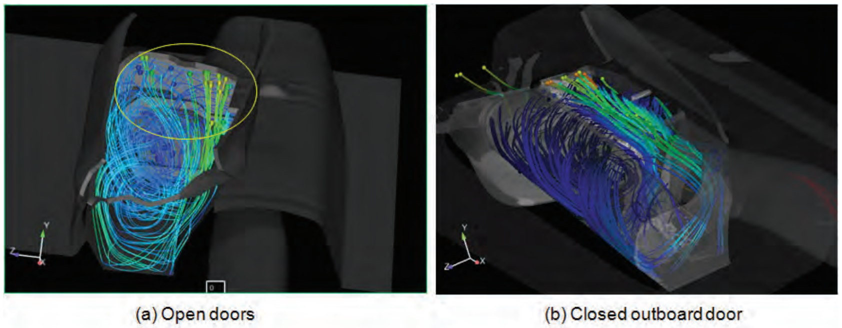


Figure 8. Mean-flow streamlines colored by the local speed

The OAPSL distribution is shown in Figure 9 for the two configurations. In general, the levels are higher for the closed door configuration over the entire bay (especially along the inboard side and on the aft-wall). This is rather a surprising result when one considers the fact that the surface area of flow affecting the bay has decreased (i.e., one-half of the bay is closed to the free stream flow), but the “aero-acoustic” imprint on the bay surfaces has increased. This is due to the classical Rossiter-like or longitudinal resonance mode facilitated by a large re-circulation region which extends the length of the bay with the outboard door closed. More details of the analysis is reported by Kannepalli, et al.[19].

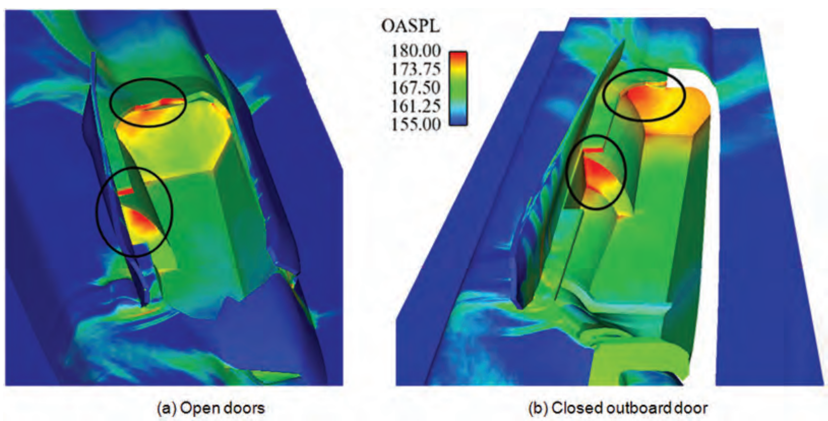


Figure 9. OASPL distribution on bay surfaces

Figure 10 shows a line plot of the OASPL distribution along the mid-span of the ceiling. The surprising result elaborated earlier, that the closed door configuration OASPLs are higher (~3–6 dBs) than the open doors configuration, is again evident in this figure. Also shown in the figure is the comparison with the data from the experiment[20], which is in excellent agreement (within 1–2 dBs). The closed door configuration displays a distinct double-inflexion distribution of the OASPL that is associated with the presence of resonant tones. The minimum at an x/L~0.7 has been seen for rectangular cavities too (with an L/D>4 and Mach 1.5) by Dix, et al.[21] and Arunajatesan, et al.[16]

Figure 11 shows the SPL spectra for the baseline outboard door closed configuration on the ceiling at x/L~0.95. From the CFD, there is a dominant tone of ~975 Hz and secondary tones at 480, 1,500, and 1,933 Hz. These frequencies compare to Rossiter modes R2 ~ 895 Hz, R1 ~384 Hz, R3 ~1,350, and R4 ~1,840 Hz, computed using the empirical formula with constants (σ, κ) = (0.25, 0.57) and an L/D~6 (Heller-Bliss[18]). The dominant 2nd mode has a wavelength that is approximately equal to the length of the bay. Similar results of the 2nd mode dominance for supersonic Mach 1.5 rectangular bays have been reported by Dix, et al.[21] and Arunajatesan, et al.[16]. The entire weapons bay can be considered to be resonating with a dominant "Rossiter-like or longitudinal mode" of ~975 Hz with amplitude of ~25 dBs. ***This frequency corresponds to ~60 Hz for the full-scale bay, and with amplitude of ~25 dBs—it is of a serious concern from a structural standpoint.*** There is good agreement with the experiment with all the tones being captured very well. However, the tones reported in the experiment are at slightly lower frequencies. The experimental model has a few more internal details of the geometry inside the bay compared to the computational model.

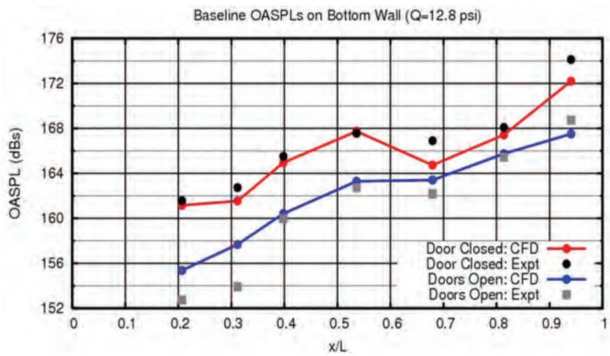


Figure 10. Bay ceiling OASPL

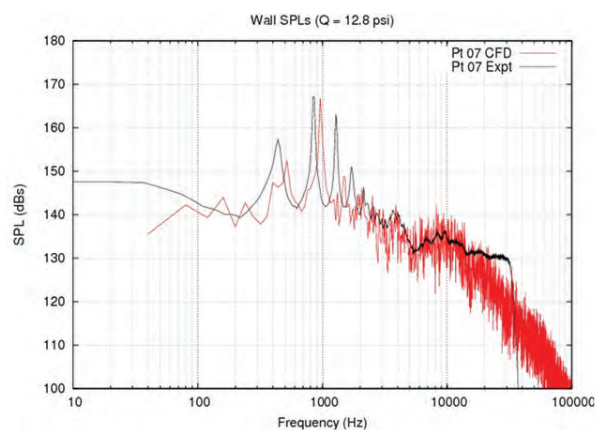


Figure 11. Computational and experimental spectra outboard door closed

3.1.2 Analysis of Control Strategies

Leading-Edge Mass Blowing (LEMB)

The motivation behind the use of LEMB is to help reduce the flow expansion into the bay and decrease the high-speed fluid entrainment into the bay. In addition, the slot jets would act like a "fluidic-spoiler" and also help stabilize and/or loft the shear-layer above the cavity. LEMB affords an adjustable concept that can be tuned (by increasing or decreasing the mass flow rate) for a wide range of operational flight envelope when compared to spoilers that are designed and work most efficiently for a given configuration. It was also earlier used in the SEAR program[13,15,14] with subsonic cavities, and Arunajatesan, et al.[16] demonstrated the effectiveness of this concept for a rectangular L/D=6 cavity at a free stream Mach 1.5.

LEMB involved blowing of sonic jets of air through the leading-edge slots, Figure 12a. The mass flow rates for the open doors and closed outboard door configurations are 3.6 lbs/s and 2.2 lbs/s, respectively, for the full-scale aircraft. Figure 12b summarizes the effect of LEMB on the OASPL distribution on the ceiling of the bay for both the configurations. The broadband levels in the open doors case decreased by 1–2 dBs in the aft-end of the bay, but the flow rate used exceeds 3 lbs/s. The LEMB was not effective at the mass flow rate studied for the closed outboard door configuration.

Other blowing rates with LEMB were examined at NCPA in the experiment in tandem with other control strategies involving pressure-relief-system (PRS) and aft-wall treatments such as porous liners that showed good reduction in the broadband noise levels[20,17].

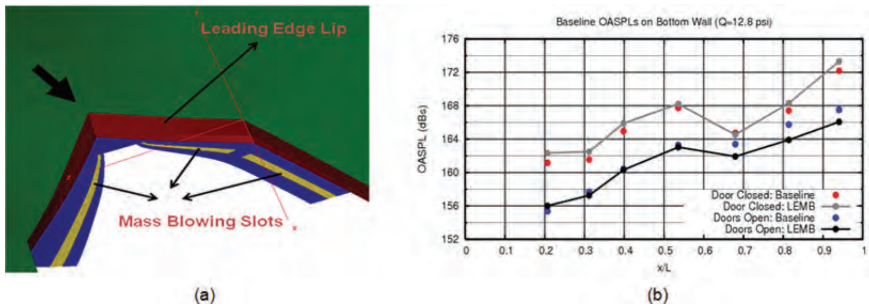


Figure 12. Illustration of LEMB slots, and (b) effect of LEMB on the OASPLs on the ceiling of the bay

Baffles

Baffles are physical partitions or walls positioned in the bay just like bulkheads, but are not required to provide structural integrity. These concepts were explored for the closed (outboard) door configuration motivated by the fact that cutting off the quiescent bay volume underneath the outboard door may be a good control strategy. These baffles could either be longitudinal, the green surface along the stream wise direction in Figure 13a that would physically partition the bay and block out the volume underneath the outboard door. They could also possibly be along the transverse/span wise direction (Figure 13b) and provide impedance to the acoustic disturbance/wave from freely moving upstream; i.e., destroying the acoustic feedback loop.

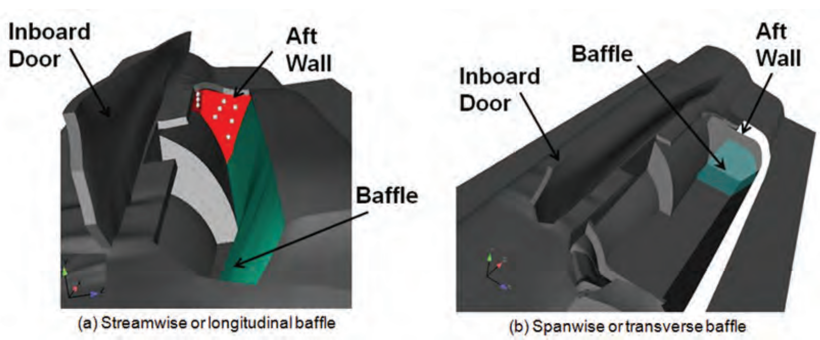


Figure 13. Illustration of (a) the stream wise baffle, and (b) the span wise baffle used as control concepts for the closed outboard door configuration

For brevity, only the span wise baffle results will be discussed here, see Kannepalli, et al.[19] for more details. This baffle was located at an x/L=0.87. It was designed to cut-off the re-circulating flow under the outboard door with a Mach <0.3 (as in Figure 14) from the baseline case involving the closed outboard door. The presence of this baffle lofts or deflects the flow (not shown) near the aft-wall region; hence the OASPLs on the aft-wall in Figure 15 are reduced by ~5 dBs over the entire wall (a 43% reduction in the fluctuating pressure component). In effect, it behaves like a "spoiler" for the aft-wall. It is very interesting to note that this effect is not only local to the aft-wall region, but also extends upstream, as can be seen in Figure 16, for the OASPL distribution on the ceiling. Except very close to the location of the baffle, the OASPLs over the entire ceiling of the bay are also lowered. This would imply that it has also hindered with the acoustic feedback loop responsible for resonance in the bay. The resulting SPL spectra indicate that the dominant tones shift to higher frequencies of 1,320 Hz and 1,754 Hz, respectively (that correspond to the 3rd and 4th modes seen in the baseline closed door configuration of the experiment). The overall noise levels; however, decrease as the amplitudes of these tones is also lower (only ~10–15 dBs).

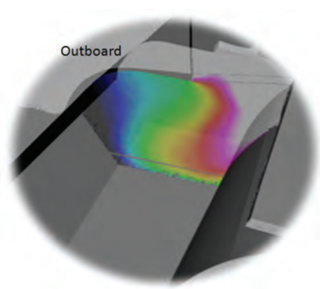


Figure 14. Mach number contours on a plane at x/L=0.87 in the outboard door closed configuration used to design the span wise baffle

It is important to note that there is some optimization necessary for this baffle based on its stream wise (x) location that shifts the dominant modes to higher frequencies and lowers the OASPLs. Also, conceivably more than one such baffle could be used.

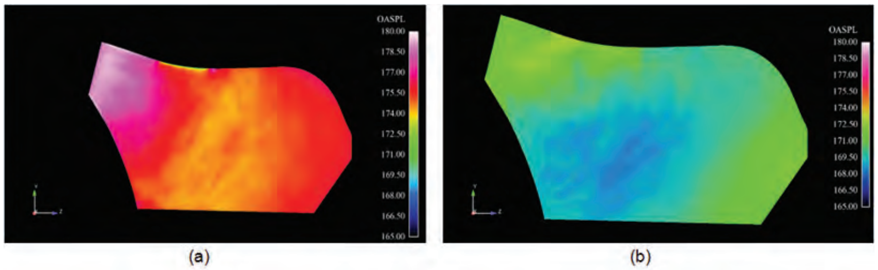


Figure 15. Aft-wall OASPL distribution; a) baseline door closed, and b) with the span wise baffle

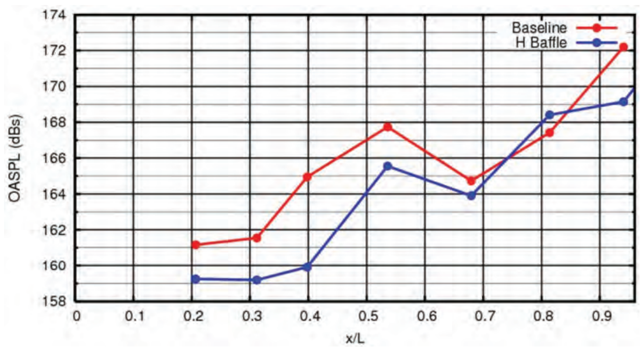


Figure 16. Comparison of the ceiling OASPLs

3.2 Store Loads

As mentioned earlier, an empty bay simulation and the case with the store submerged in the shear-layer in the opening of the cavity are modeled to generate unsteady time histories of the store aerodynamic loads. Once the simulations reached an approximate statistical steady-state data, was collected for 6 ms, equivalent to 6 cavity cycles based on the second Rossiter mode. This took approximately 250 CPU-hours on an IBM P5 using 108 processors for the case with the store in the shear-layer.

Looking at the mean stream wise (U) velocity contours in both the mid-span and mid-stream wise location indicates that the presence of the store acts to loft the high-speed flow away from the cavity, while increasing the growth of the shear-layer into the cavity, Figure 17. The store in this location also produces asymmetric flow within the cavity.

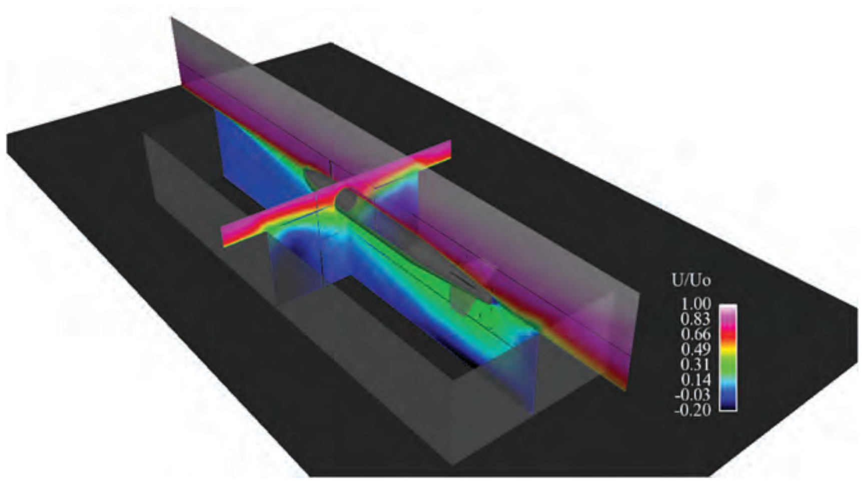


Figure 17. Mean U velocity contours with the store in the cavity opening

A quick comparison to the empty cavity indicates the change in the OASPL due to the presence of the store, Figure 18. Once again, the asymmetry in the solution is seen on the aft-wall for both cases. The free stream side of the store nose clearly takes a beating from the shear-layer as it diverts the high-speed flow away from the cavity. In general, the store causes slight reductions in the OASPL due to the blockage effects, which are consistent with what is reported in the literature[22].

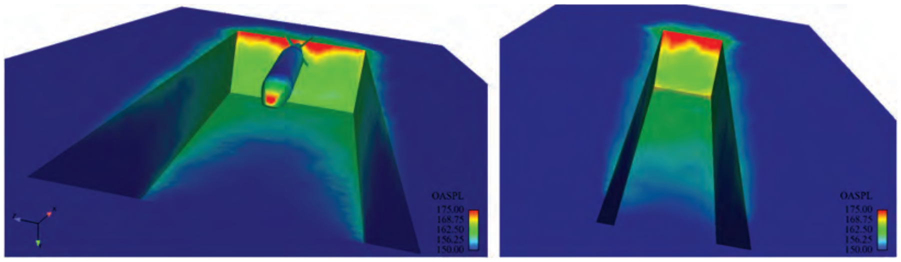


Figure 18. Surface contours of OASPL for the cavity with store and clean cavity

With a Mach 3.5 free stream velocity and the relatively thick shear-layer, the spectral character in the cavity is broadband. This is also reflected in the frequency spectra of the unsteady store loads, Figure 19. This indicates that the stream wise force component and rolling moment are relatively smaller in magnitude than the side and normal forces and the pitch and yawing moments; reflecting what is typically of concern during store separation analysis. The drop-off in the unsteady components beyond 10 KHz indicates limits on the data sampling and bandwidth requirements for the telemetry package. Further analysis to look at dynamic pressure scaling effects on the fluctuating loads, both frequency content and magnitude, is planned.

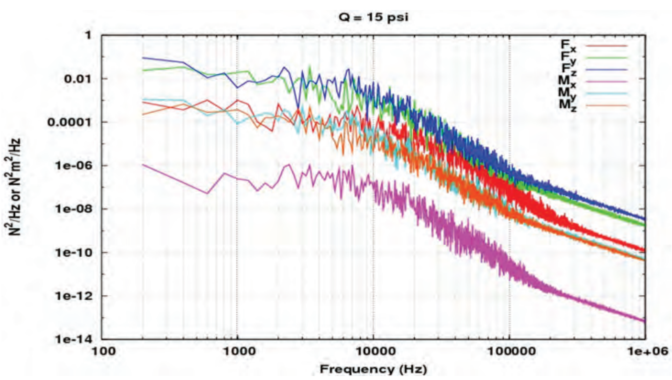


Figure 19. Frequency spectra of the unsteady forces/moments on the store

4. Conclusion

Computational analysis of a complex weapons bay geometry representative of that on the F-35 aircraft was done for a free stream Mach of 1.5. Baseline configurations with both bay doors open and with the outboard door closed were carried out. The dynamic pressure loads on the bay for the baseline open doors configuration showed broadband spectral behavior when compared to traditional rectangular cavities at the same Mach number. However, the closed door configuration shows a dramatic change in behavior with the generation of tones, as well as an increase in the OASPLs. The dominant tone present has a very high amplitude of ~20 dBsm and corresponds to a frequency ~60Hz, which is of serious concern from a structural stand point for the full-scale aircraft.

For the blowing rates studied, LEMB was not effective in suppressing the OASPLs in the bay for the outboard door closed configuration. However, a simple, passive control concept involving “baffles” has been shown to result in significant reduction in the tonal amplitude. Their principle mode of suppression is based on “physically” impeding or cutting-off of the feedback loop mechanism responsible for the setting up of resonant tones. The baffles hence satisfy both an easily “integratable” and working control concept over a wide operational range of the aircraft.

Modeling of a store placed in the opening of a generic L/D=6 cavity submerged in a Mach 3.5 free stream flow field was accomplished to help establish criteria for the development of a telemetry system suitable for small-scale store drop testing. Results indicate a frequency range of interest, and also provide information on the magnitude of the fluctuating components of the store loads. Further modeling to look at dynamic pressure effects on the fluctuating store loads components will be accomplished in the near future. Fully time-accurate simulations of representative drop tests are also planned.

Acknowledgments

The support of Al Piranian (JSF Program Office) and Jim Grove (AFRL/RBAI) and the computing time provided by the HPCMP Office for this effort are gratefully acknowledged.

References

1. Arunajatesan, S., Sinha, N., Stanek, M.J., Grove, J.E., and Johnson, R.A., "High Performance Computational Modeling of Unsteady Surface Loads in Complex Weapons Bays", *Proceedings of the DoD HPCMP Users Group Conference 2009*, San Diego, CA, June 15–18, 2009.

2. Johnson, R.A., Kannepalli, C., Arunajatesan, S., and Sinha, N., "Computational Modeling of Geometrically-Complex Weapons Bays", *Proceedings of the DoD HPCMP Users Group Conference 2010*, Schaumburg, IL, June 14–17, 2010.

3. Johnson, Rudy, Stanek, Michael, and Grove, James, "Store Separation Trajectory Deviations due to Unsteady Weapons Bay Aerodynamics", *AIAA-2008-188*, 46th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, January 2008.

4. Murray, R.C. and Elliot, G.S., "Characteristics of the Compressible Shear-Layer over a Cavity", *AIAA Journal*, Vol. 39, No. 5, pp. 846–856, May 2001.

5. Sinha, N., Hosangadi, A., and Dash, S.M., "The CRAFT NS code and preliminary applications to steady/unsteady reacting, multi-phase jet/plume flow field problems", *CPIA Pub. 568*, May 1991.

6. Rai, M.M. and Moin, P.," Direct numerical simulation of transition and turbulence in spatially-evolving boundary-layer", *Journal of Computational Physics*, Vol. 109, pp.162–192, 1993.

7. Kannepalli, C., Arunajatesan, S., and Dash, S.M., "RANS/LES Methodology for Supersonic Transverse Jet Interactions with Approach", *AIAA-2002-1139*.

8. Papp, J. L. and Dash, S. M., "Turbulence Model Unification and Assessment for High-Speed Aero-propulsive Flows", *AIAA Paper 2001-0880*, Jan. 2001.

9. Arunajatesan, S. and Sinha, N., "Large-Eddy Simulations of Supersonic Impinging Jet Flow Fields", *Paper No. AIAA-2002-4287*, 38th AIAA/ASME/SAE/ASEE Propulsion Conference & Exhibit, Indianapolis, IN, July 7–10, 2002.

10. Kannepalli, C., Arunajatesan, S., Calhoon, W.H., Jr., and Dash, S.M., "Large-Eddy Simulations of High-Speed Flows", *Paper No. HT-FED2004-56162*, 2004 Joint ASME-JSME Fluids Engineering Summer Conference, Charlotte, NC, July 11–15, 2004.

11. Calhoon, W.H., Jr., Kannepalli, C., and Dash, S.M., "LES Studies Of Scalar Fluctuations at High Convective Mach Numbers", Third AFOSR International Conference on Direct Numerical Simulation and Large Eddy Simulation (TAICDL), University of Texas at Arlington, Arlington, TX, USA, August 5–9, 2001.

12. Arunajatesan, S. and Sinha, N., "Hybrid RANS-LES Modeling for Cavity Aero-acoustics Predictions", *International Journal of Aeroacoustics*, Vol. 2, No. 1, pp. 65–91, 2003.

13. Arunajatesan, S., Shipman, J., Cavallo, P.A., Birkbeck, R, Sinha, N., Ukeiley, L.S. and Alvi, F.A., "Flow Control for Enhanced Store Separation", *AIAA Paper 2007-1239*.

14. Shipman, J.D., Arunajatesan, S., Cavallo, P.A., Sinha, N., Ukeiley, L. and Alvi, F., "Flow Control for Enhanced Store Separation", *Paper No. AIAA-2007-1236*, 45th Aerospace Sciences Meeting and Exhibit, Reno, NV, January 8–11, 2007.

15. Ukeiley, L.S., Sheehan, M. Coiffet, F., Alvi, F.S., Arunajatesan, S., and Jansen, B., "Control of Pressure Loads in Complex Cavity Configurations", *Paper No. AIAA-2007-1238*, 45th Aerospace Sciences Meeting and Exhibit, Reno, NV, January 8–11, 2007.

16. Arunajatesan, S., Kannepalli, C., Sinha, N., Sheehan, M., Shumway, G., "Suppression of Cavity Loads Using Leading-Edge Blowing Concepts", *AIAA Journal*, Vol. 47, No. 5, 2009.

17. Rossiter, J.E., "Wind-Tunnel Experiments on the Flow over Rectangular Cavities at Subsonic and Transonic Speeds", *Aeronautical Research Council, Reports & Memoranda No. 3438*, London, England, October 1964.

18. Heller, H.H. and Bliss, D.B., "The Physical Mechanism of Flow-Induced Pressure Fluctuations in Cavities and Concepts for Their Suppression", *AIAA Paper 75-491*, March 1975.

19. Kannepalli, C., Chartrand, C., Birkbeck, R., Sinha, N., and Murray, N.", Computational Modeling of Geometrically-Complex Weapons Bays", *AIAA Paper 2011-2774*, 17th AIAA/CEAS Aeroacoustics Conference, Portland, OR, 6–8 June 2011.

20. Murray, N. "Enhanced Acoustical Environment for Modern Weapons Bays", *Interim Report, AFRL SBIR Phase II Subcontract No. 09-C-3906/C390*, National Center for Physcial Acoustics, University of Mississippi.

21. Dix, R.E. and Dobson, T.W., Jr., "Database for Internal Store. Carriage and Jettison, Vol. 1", *AEDC-TR-90-23*, Arnold Eng. Develop. Center, November 1990.

22. Heller, H.H., Holmes, G., and Covert, E.E., "Flow-Induced Pressure Oscillations in Shallow Cavities", *AFFDL-TR-70-104*, DTIC Accession Number: AD880496, Air Force Flight Dynamics Laboratory, Wright-Patterson AFB, OH, December 1970.