

# Validation of Immersed Boundary Wall-Modelled Large Eddy Simulations with F-16XL Flight Data

Jordan B. Angel\*, Aditya S. Ghatge, Gaetan K. W. Kenway, Man Long Wong, and Cetin C. Kiris  
*Volcano Platforms Incorporated, Palo Alto, CA 94304, USA*

**The purpose of the work is to assess the present capability of Wall-Modelled Large Eddy Simulations (WMLES) for simulating military aircraft at high-alpha conditions that may challenge traditional steady-state methods due to massive flow separation and unsteady interactions between off-body vortices. Specifically, this paper presents WMLES simulations of the F-16XL aircraft at a specific high alpha, low Mach flight condition (FC-25) from the Cranked-Arrow Wing Aerodynamics Project (CAWAP) flight test that has been repeatedly identified as a difficult case for making accurate predictions on the outboard section of the wing where the flow is largely separated. This paper is deliberately focused in scope to give a straightforward assessment of WMLES for this case. The computational framework used, Volcano ScaLES, is a new Cartesian octree based Navier–Stokes solver with an immersed boundary approach written from the ground up in C++/CUDA to maximize computing throughput on modern graphics processing units (GPUs). Excellent agreement with flight data is achieved, notably at the highly unsteady outboard stations. Simulations are run using a 7× NVIDIA GeForce RTX 4090 GPU workstation and a 8× A6000 workstation.**

## I. Introduction

Computational fluid dynamics (CFD) may be able to reduce wind tunnel and flight testing by providing flow field solutions for realistic aircraft designs only if the results can be trusted and if it is indeed more practical when a database of flight conditions is needed. Presently, Reynolds-averaged Navier-Stokes (RANS) based approaches are mature and provide trusted results over a wide range of conditions, but may fail to give acceptable results for flight conditions that violate the boundary layer assumptions of RANS modeling. Maneuverable fighter aircraft fly a much wider envelope of flight conditions compared to transport aircraft and typical delta wing designs create large vortices over the wings during high angle of attack flight to reduce stall [1]. Critical flight conditions (e.g. takeoff, landing, and high-speed maneuvers) likely require high fidelity simulations to resolve complex unsteady flow phenomena [2, 3].

In the mid-1990s, the Cranked-Arrow Wing Aerodynamics Project (CAWAP) was initiated by Lamar [4] which was an impressive effort bringing together flight, wind tunnel, and simulation data to understand the performance of the cranked-arrow wing design using the F-16XL aircraft. A photograph\* of the F-16XL is shown in Figure ?? . An overview of the development of the aircraft and its performance compared to the F-16 was given by Hillaker [5], the designer of the F-16 and F-16XL. Originally conceived as the Supersonic Cruise and Maneuver Program (SCAMP), General Dynamics developed the F-16XL in collaboration with NASA Langley Research Center to enhance the capabilities of the already successful F-16. Two F-16XLs were built as flight demonstrators with the objective of showing configuration changes to the F-16 could provide major improvements including significantly enhanced payload capacity, improved fuel capacity without the need for external fuel tanks, and dramatically improved maneuverability. Several flight tests were performed for a range of conditions and these experiments have been the focus of numerous computational studies and follow-on workshop efforts [4, 6]. The follow-on effort of CAWAPI (CAWAP International) expanded to include NATO allied countries and a comprehensive comparison of CFD results was presented in [7]. The effort was successful at predicting many of the flight conditions but a particular case highlighted for being particularly challenging was designated FC-25 and corresponds to a low Mach, high alpha condition. This case became a focus of many numerical studies and in later workshops unsteady methods such as URANS and delayed detached eddy simulations (DDES) were performed [8–10]. These simulations improved on past work and presently there is very limited room for accuracy improvements except for the outboard stations where the flow is highly unsteady and discrepancies are still common in the literature.

Our aim in this paper is to assess the practicality of a Cartesian WMLES Navier-Stokes solver for predicting the massively separated and complex vortical flow seen in the FC-25 test. Section II describes the computational approach

---

\*Corresponding author.

\*Photo accessed from <https://www.nasa.gov/gallery/f-16xl-ship-no-1/>

adopted in this work and provides some details for the solver design. Next, Section III details the comparisons to flight data in the form of pressure cuts at several stations along the wing and fuselage. Details of the hardware and computational cost are also provided.

## II. Numerical Method

### A. Numerical formulation

Volcano ScaLES solves the compressible Navier–Stokes equations using a nominally 4th order spatially accurate finite difference discretization with favorable Kinetic Energy and Entropy Consistency properties making it suitable for Large Eddy Simulations (LES) over a vast range of flow regimes. The viscous flux discretization utilizes a mix of 4th order and 2nd order discretizations with high spectral bandwidth thereby allowing further robustness for high Reynolds number flows without additional numerical dissipation via the inviscid flux numerical operator in turbulence resolving regions. Physics-based numerical sensors that are functions of local velocity gradients as well as pressure and density fluctuations allow for localized use of limiters needed to capture flows with discontinuities such as shocks and steep density gradients. A key convenience afforded by the use of Cartesian grids with unit aspect ratio is the use of explicit time-stepping scheme even in flows involving highly complex geometries; the perfectly isotropic grid cells near the geometry do not introduce any numerical stiffness associated with complex grid topologies. There are other significant advantages of Cartesian meshes including significantly reduced memory requirements for storing the grid as well as efficient and high accuracy in the underlying numerical formulation. The popular Strong Stability Preserving (SSP) variant of the classical 3rd order Runge–Kutta scheme developed by Gottlieb and Shu [11] is utilized for all the work presented in this paper.

Geometries are represented in the numerical formulation using an immersed boundary algorithm capable of enforcing inviscid no-penetration boundary conditions as well as viscous skin friction with appropriate Reynolds number asymptotic properties using an equilibrium wall-model. The wall-model that provides a shear-stress constraint at the wall uses solution information interpolated via probing at a fixed distance of  $3\Delta/2$  into the fluid away from walls. Additional methods to damp generation of spurious numerical noise at walls as well as at coarse-fine interfaces are also utilized in the present work. Simulations presented in this work utilize either the constant coefficient Vreman subgrid scale closure [12] or the dynamic Smagorinsky model [13, 14] to model the subgrid-scale stresses.

The CFD solver has been validated using a sequence of WMLES including: a) aerodynamics of iced straight wings [15], b) automotive aerodynamics [16], c) subsonic jets [17], d) airframe noise [18], and e) vibro-acoustics of a launch vehicle [19].

### B. Octree Cartesian Grid System

The Volcano ScaLES solver uses Cartesian grids generated using a recursive octree approach. Grids around engineering-quality configurations consist of an unstructured tree-of-cubes with millions of leaf cubes each containing a structured grid of  $4^3$  or  $8^3$  cells. This method gives nearly all the computational advantages of structured grids while retaining the ability to closely follow surfaces and minimize the number of “dead” cells inside the geometry. Typically, 90-95% of the total cells in a grid are compute cells, while the remaining ones are considered “dead” or “blanked” cells.

### C. Computational Architecture

Motivated by the need to fully exploit all available resources on modern GPUs, Volcano ScaLES is written exclusively in C++17 and CUDA. On simulations involving non-canonical geometries such as those considered in this paper, we achieve approximately 30% of peak FLOPs on compute-oriented devices along with nearly perfect (>95%) weak scaling efficiency from 1 to 8 GPUs. Unlike most CFD codes, the majority of the computations are compute bound and we achieve high performance even in graphics-oriented, bandwidth-restricted devices, such as the Nvidia GeForce RTX 4090. The combination of Cartesian methods and an explicit time stepping scheme yields extremely low memory requirements: ScaLES can simulate between 8.5 and 10 million cells per gigabyte of GPU memory depending on the complexity of the configuration. Simulations containing between 400 and 500 million cells are routinely performed on desktop computers containing just two RTX 4090 GPUs with 24 Gb of memory each. The ability to fully exploit GPU hardware such as the Nvidia A10G accelerators on AWS that offer better price-performance than more expensive Nvidia A100/H100-based hardware, significantly lower the cost of large eddy simulations to a level that is practical for use in an industrial setting.

### III. WMLES of F-16XL at Flight Condition 25

This section provides a concise description of the flight case and the computational set up for the simulations. Details of the flight test that are not immediately relevant are omitted. For a comprehensive overview of this flight condition we recommend the CAWAP and CAWAPI review papers, [4, 6, 7].

#### A. Case Description

The nominal conditions used to set up the problem are provided in Table 1 for reference. The top rows indicate the quantities used to define the flow conditions with their symbols used in this paper and their units. All simulations are initialized from freestream conditions derived from Table 1. The altitude is used to estimate freestream density and temperature from the US 1976 Standard Atmosphere Model. These two thermodynamic variables along with the standard gas properties of air define a freestream sound speed which, in turn, defines the velocity field using the Mach number, angle of attack, and side slip angle. The mean aerodynamic chord (MAC) is taken to be the reference length scale and implies a Reynolds number  $Re \approx 32 \times 10^6$ . We take the typical aerospace coordinate system definition so that the flow is in the positive  $x$  direction and the positive  $y$  direction points outboard on the wing.

**Table 1** Nominal freestream and reference conditions

| Altitude | Angle of attack | Side slip     | Mach number | Ref. Length           |
|----------|-----------------|---------------|-------------|-----------------------|
| [ft]     | $\alpha$ [deg]  | $\beta$ [deg] | $M$ [-]     | $c_{\text{ref}}$ [ft] |
| 10 000   | 19.84           | 0             | 0.242       | 24.7                  |

#### B. Boundary Conditions

The simulations are run in free-air since the experimental data was collected in flight. A wall model is used to compute a wall shear stress at the no-slip boundaries. As with previous studies, we consider the symmetric case so that our computational domain includes half of the aircraft with a symmetry boundary condition at  $y = 0$ ft. The F-16XL has a single engine with a serpentine duct intake which is shown in Figure 2. The inlet of the fan is given a static pressure condition and the exhaust is given total pressure and temperature conditions. Table 2 gives the pressure and temperature values applied at the fan inlet and exit planes. Note that a careful matching of the implied derived quantities such as inlet Mach number are not pursued in this paper since the propulsion system is not expected to significantly influence the flow above the wing which is our focus.

**Table 2** Propulsion boundary conditions

| Inlet static pressure | Exit total pressure | Exit total temperature |
|-----------------------|---------------------|------------------------|
| [psia]                | [psia]              | [R]                    |
| 8.72                  | 26.3                | 1209                   |

#### C. Volcano ScaLES

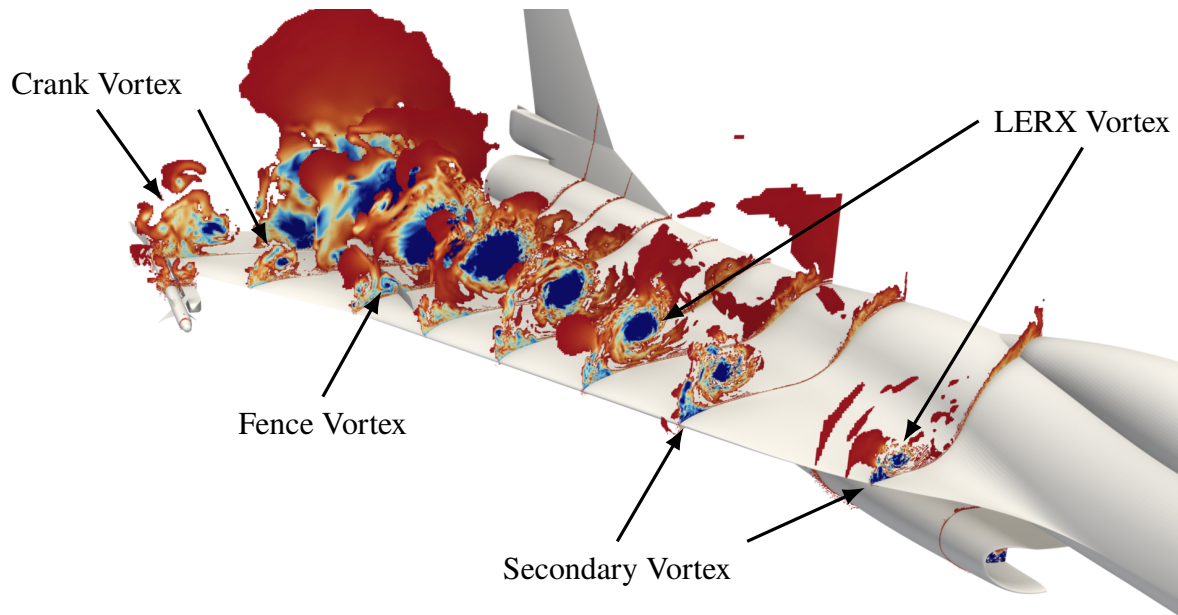
ScaLES solves the compressible Navier-Stokes equations using a nominally 4th order spatially accurate finite difference discretization with favorable Kinetic Energy and Entropy Consistency properties making it suitable for Large Eddy Simulations (LES) over a vast range of flow regimes. Geometries are represented in the numerical formulation using an immersed boundary algorithm with appropriate Reynolds number asymptotic properties using an equilibrium wall-model. There are other significant advantages of Cartesian octree meshes including significantly reduced memory requirements for storing the grid as well as efficient and high accuracy in the underlying numerical formulation. Grids around engineering-quality configurations consist of an unstructured tree-of-cubes with  $O(10^6)$  leaf cubes each containing a structured grid of  $8^3$  or  $4^3$  cells.

On simulations involving non-canonical geometries such as those considered in this paper, we achieve approximately 30% of peak FLOPs on compute-oriented devices along with nearly perfect (>95%) weak scaling efficiency from 1 to 8

GPUs. The combination of Cartesian methods and an explicit time stepping scheme yields extremely low memory requirements: ScaLES can simulate between 8.5 and 10 million cells per gigabyte of GPU memory depending on the complexity of the configuration. Simulations containing up to 400-500 million cells are routinely performed on desktop computers containing just two RTX-4090 GPUs with 24 GB of memory each. The ability to fully exploit lower-specification hardware significantly lower the cost of LES simulations to a practical level for use in an industrial setting.

#### D. Mesh Considerations

At such a high angle of attack, the Leading Edge Root Extension (LERX) creates a strong vortex over the wing which promotes flow attachment. Furthermore, the fence at the wing crank creates a vortex over the outboard portion of the wing. This outboard portion is highly unsteady with complex interactions between the crank vortex and the fence vortex. To account for these flow features, volumetric refinement regions are added over the wing manually. The over-wing vortices are easily captured using simple tapered cylinders with the axial vector aligned with the vortex core. An inexpensive coarse grid simulation can be run quickly which provides enough information to select regions of refinement. Figure 1 shows several slices of the coefficient of total pressure on the wing for such a simulation. From this depiction it is relatively easy to judge the spatial extent and relative strength of the vortices. All cells within these regions are refined to an input resolution.

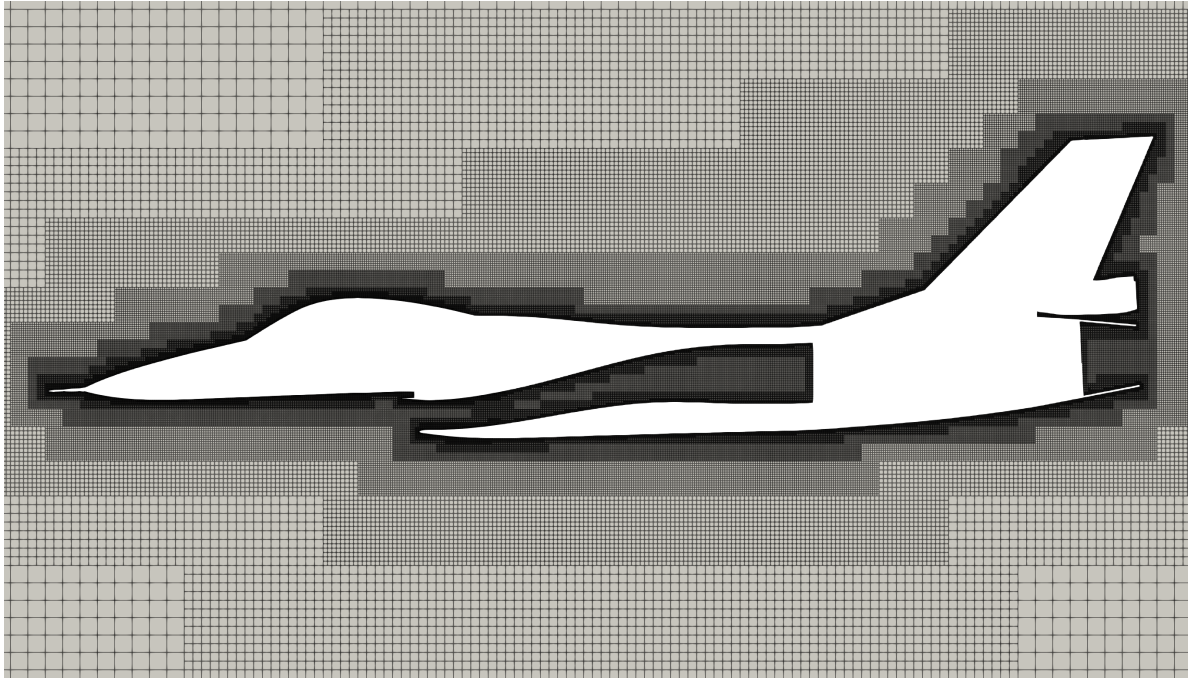


**Fig. 1 Slices of total pressure coefficient showing the vortex structures over the wing. Several vortices and their origins are indicated in the figure with arrows.**

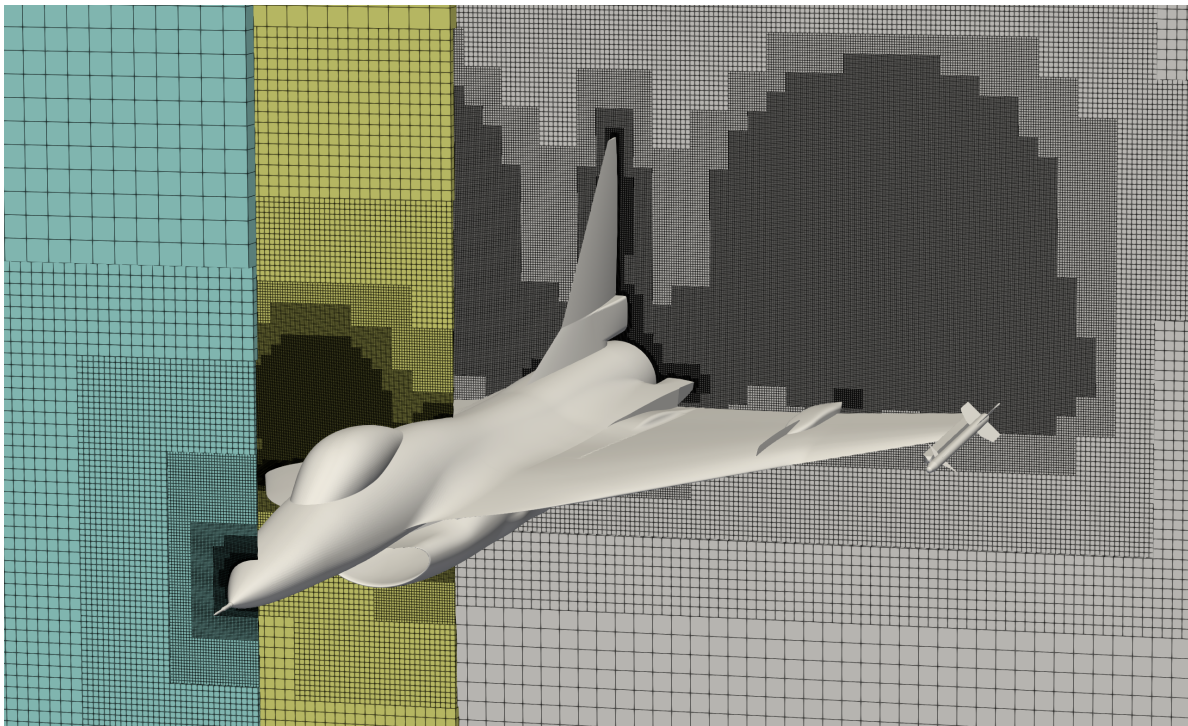
The high curvature of the leading edge of the wing and the high angle of attack causes the flow to separate almost immediately. Consequently, resolving the leading edge curvature is the primary concern for the near-wall mesh. To account for this, portions of the surface are tagged using input regions. Blocks containing tagged surface triangles are refined to the requested mesh spacing. The geometry definition was obtained as an STL file. The discretization of the surface is coarse relative to the volume spacing near walls and there is also a defect along the upper surface of the wing shown in Figure 4. The defect appears as many small bumps along a line of constant  $y$  on the inboard section of the upper surface of the wing. The bumps are relatively small and due to the flow being massively separated they are not expected to influence the solution in an appreciable way. Unfortunately, a butt line used for validation lies along this defect and its presence is observed as small fluctuations in the averaged solution.

Computational resources used are provided in Table 3. The first column indicates a label to refer to the two meshes. The second and third column give the finest cell resolution and total number of grid cells for G1 and G2. The third and





**Fig. 2** Cartesian octree mesh at the plane  $y = 0\text{ft}$  plane showing the inlet duct.

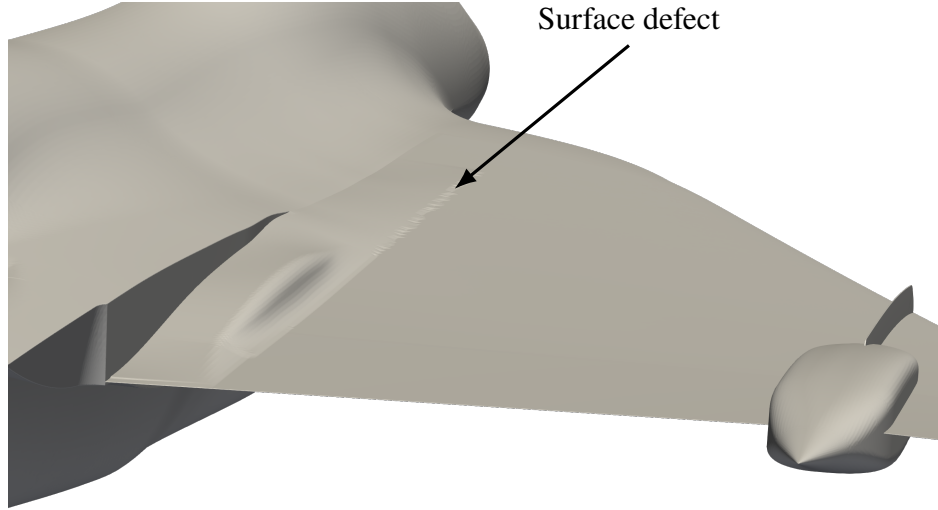


**Fig. 3** Surface geometry (shown mirrored) and Cartesian octree mesh at the constant coordinate planes  $x = 0\text{ft}$ ,  $23\text{ft}$ , and  $48\text{ft}$ .

**Table 3 Computational cost for the simulations presented in this section.**

|                          | G1          | G2       |
|--------------------------|-------------|----------|
| Min. $\Delta x$ [mm]     | 1.00        | 0.50     |
| No. of Cells [ $10^6$ ]  | 414         | 1,010    |
| Turnaround Time [days]   | 1           | 3        |
| MUPS [ $10^6$ cells/sec] | 5400        | 5200     |
| Hardware                 | 7× RTX 4090 | 8× A6000 |

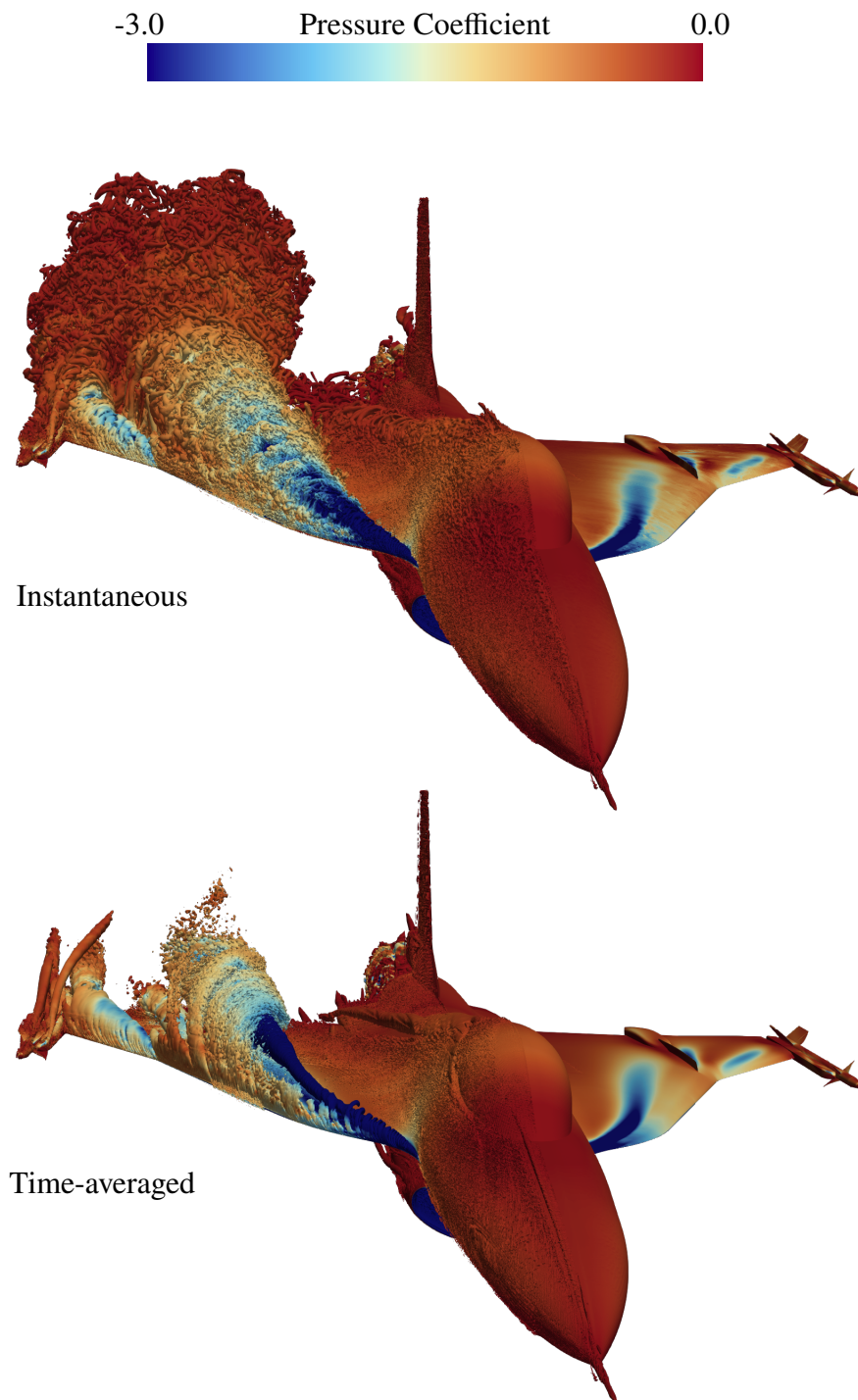
fourth columns provide some performance metrics in terms of the wallclock time needed to integrate a single convective time unit and the number of millions of grid cells updated per second (MUPS) i.e. 5400 MUPS means that 5.4 billion cells are updated per second.



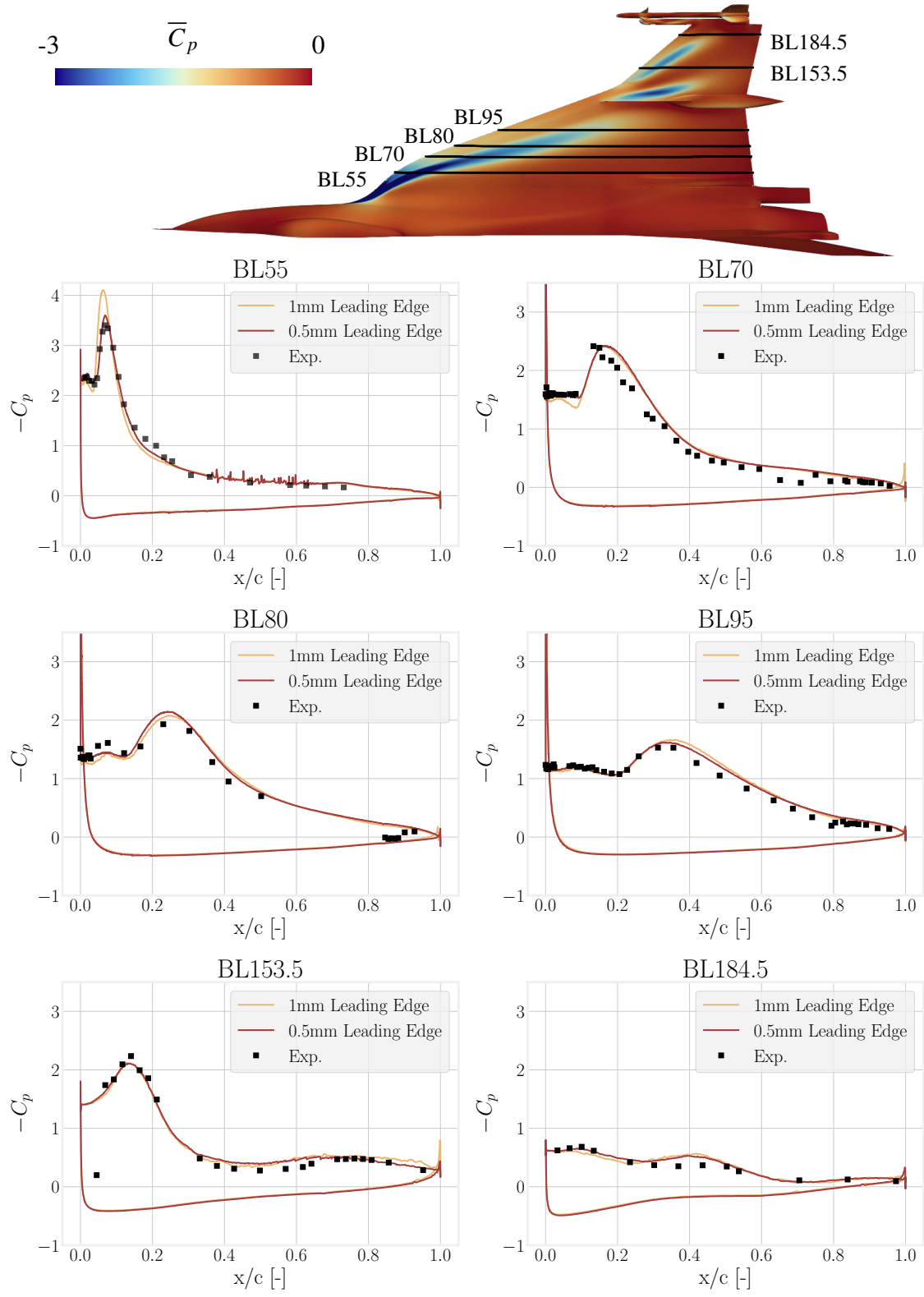
**Fig. 4** A zoomed view looking upstream shows the defect in the surface STL. A CAD file was not readily available to generate an improved surface triangulation.

### E. Comparison to Flight Data

During the flight tests, pressure measurements were taken at several stations shown in Figures 6 and 7. The butt lines (BL) have constant  $y$  (spanwise) coordinates and the fuse stations (FS) have constant  $x$  (streamwise) coordinates. At the top of both figures is a subfigure indicating where each line plot is extracted from the surface. The line plots have an abscissa of nondimensional length, either  $x/c$  or  $y/(b/2)$  and the range for  $C_p$  is chosen to match previous CAWAP summary papers. The titles of each subplot indicates the cut by its type (BL or FS) and its distance in inches from  $y = 0$  (BL) or  $x = 0$  (FS). The numerical average  $C_p$  is plotted with solid lines and the experimental data is plotted as black squares. Flow over the inboard section of the wing is quasi-steady and dominated by the LERX vortex which keeps the inboard flow mostly attached. A detailed discussion of the inboard cuts would be redundant to many previous publications since these locations have been well-predicted for many years and the present simulations tend to match other numerical results here. The only remark warranted here is to point out the noise in the solution at BL55 due to the surface triangulation issues pointed out previously. The primary interest is in what improvements, if any, can be realized using scale-resolving simulations to study the highly unsteady outboard portion of the wing, in particular BL153.5 and BL184.5. The unsteadiness can be seen the  $C_p$  average here which still shows some nonsmooth behavior compared to the other stations. We note the suction peak due to the crank vortex and the trailing portion of the wing are predicted with excellent agreement.

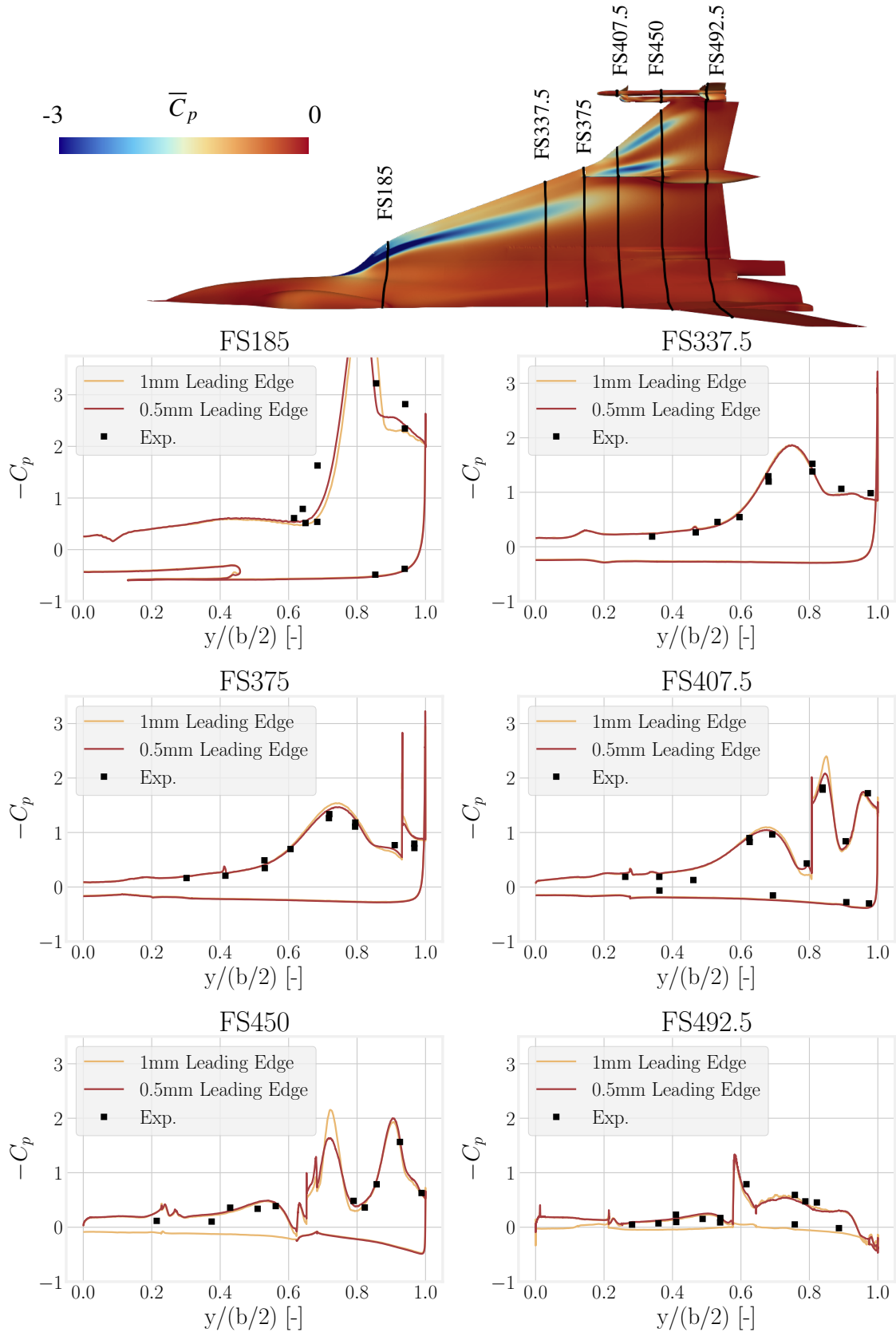


**Fig. 5** Front view of the F-16XL showing the surface (mirrored) and  $Q$ -criterion colored by pressure coefficient. Top: instantaneous solutions. Bottom: time-averaged solution.



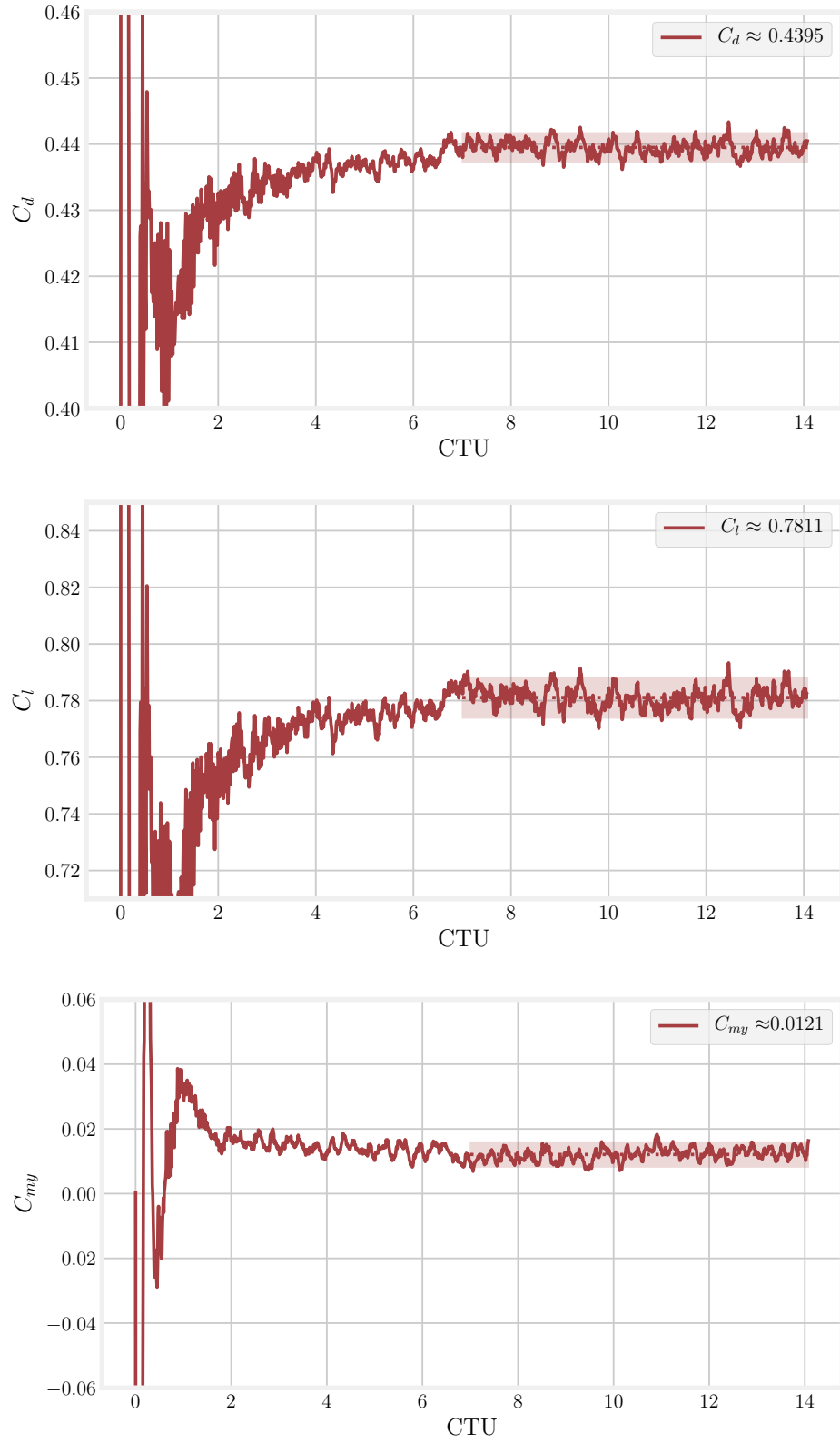
**Fig. 6** Top: average surface pressure coefficient ( $\bar{C}_p$ ) showing the butt lines. Below:  $\bar{C}_p$  compared to flight data (black squares) along the butt lines.





**Fig. 7** Top: Average surface pressure coefficient ( $\overline{C_p}$ ) showing the fuse stations in black. Below:  $\overline{C_p}$  compared to flight data (black squares) along the fuse stations.

## F. Forces and Moments



**Fig. 8** Time history of drag ( $C_d$ ), lift ( $C_l$ ), and moment coefficients ( $C_{my}$ ).  $CTU = U_\infty / c_{ref}$ .

No experimental data is available for forces at this condition but previous numerical studies have provided these values so we include them here for comparison. Reported numbers are obtained by averaging after a transient start up period that was estimated to be about 5.5 CTUs and the averaging window is about 8.5 CTUs wide. A time history of the force coefficients are depicted in Figure 8 and the averaged values are given in legend. The predicted values are in agreement with other simulations.

## IV. Conclusions

This work gives a perspective of the current readiness for WMLES applied to aerodynamic problems relevant to military aircraft design. Simulations results obtained were compared against pressure measurements. The outboard portion of the wing, which has been shown in numerous studies to be a difficult region to predict, showed excellent agreement with flight data and it was achieved using very modest computational resources. Performance metrics were also provided. A notable advantage of the approach presented here is the low memory footprint that enables simulations of 300-400 million grid points (for this particular case) using only two inexpensive consumer grade GPUs. It is worth emphasizing this point in the context of industrial research and design where large supercomputing clusters are likely not available but most CFD engineers are likely to have a workstation. Current computing hardware trends are being driven by recent breakthroughs in machine learning which has increased the demand for GPUs but also made access to GPUs more readily available. Furthermore, there are numerous companies that specialize in building GPU workstations making them quite affordable relative to traditional engineering workstations which typically are built around expensive multi-core CPUs. Alternatively, if a GPU cluster is available, many cases could be run simultaneously to dramatically increase the overall throughput since each case would require only a single node. Finally, we note that explicit time-stepping with structured grid blocks is highly amenable to GPU parallelism where fine granularity of parallelism (down to the cell) is often key to achieve optimal performance.

## References

- [1] Rizzi, A., and Luckring, J., “Historical development and use of CFD for separated flow simulations relevant to military aircraft,” *Aerospace Science and Technology*, Vol. 117, 2021, p. 106940. <https://doi.org/10.1016/j.ast.2021.106940>.
- [2] Smith, B. R., “Challenges to the use of CFD in the Military Aircraft Industry,” *AIAA SciTech*, 2015, pp. 1–22.
- [3] Smith, B. R., and McWaters, M. A., “Aerodynamic Database Requirements for the Detailed Design of Tactical Aircraft: Implications for the Expanded Application of CFD,” *Use of Computational Fluid Dynamics for Design and Analysis: Bridging the Gap Between Industry and Developers*, NATO AVT-366, 2022, pp. MP–AVT–366–02.
- [4] Lamar, J., Obara, C., Fisher, B., and Fisher, D., “Flight, Wind-Tunnel, and Computational Fluid Dynamics Comparison for Cranked Arrow Wing (F-16XL-1) at Subsonic and Transonic Speeds,” 2001.
- [5] Hillaker, H., *F-16XL flight test program overview*, ??? <https://doi.org/10.2514/6.1983-2730>, URL <https://arc.aiaa.org/doi/abs/10.2514/6.1983-2730>.
- [6] Lamar, J., and Obara, C., “Review of Cranked-Arrow Wing Aerodynamics Project: Its International Aeronautical Community Role,” *45th AIAA Aerospace Sciences Meeting and Exhibit*, 2007, p. 1. <https://doi.org/10.2514/6.2007-487>, URL <https://arc.aiaa.org/doi/abs/10.2514/6.2007-487>.
- [7] Rizzi, A., Jirásek, A., Lamar, J. E., Crippa, S., Badcock, K. J., and Boelens, O. J., “Lessons Learned from Numerical Simulations of the F-16XL Aircraft at Flight Conditions,” *Journal of Aircraft*, Vol. 46, No. 2, 2009, pp. 423–441. <https://doi.org/10.2514/1.35698>, URL <https://doi.org/10.2514/1.35698>.
- [8] Rizzi, A., and Luckring, J. M., “What Was Learned in Predicting Slender Airframe Aerodynamics with the F-16XL Aircraft,” *Journal of Aircraft*, Vol. 54, No. 2, 2017, pp. 444–455. <https://doi.org/10.2514/1.C033569>, URL <https://doi.org/10.2514/1.C033569>.
- [9] Lofthouse, A. J., and Cummings, R. M., “Numerical Simulations of the F-16XL at Flight-Test Conditions Using Delayed Detached-Eddy Simulation,” *Journal of Aircraft*, Vol. 54, No. 6, 2017, pp. 2077–2099. <https://doi.org/10.2514/1.C034045>, URL <https://doi.org/10.2514/1.C034045>.
- [10] Morton, S. A., and McDaniel, D. R., “F-16XL Simulations at Flight Conditions Using Hybrid Near-Body/Offbody Computational Fluid Dynamics,” *Journal of Aircraft*, Vol. 54, No. 6, 2017, pp. 2050–2069. <https://doi.org/10.2514/1.C034095>, URL <https://doi.org/10.2514/1.C034095>.

- [11] Gottlieb, S., and Shu, C.-W., “Total variation diminishing Runge-Kutta schemes,” *Mathematics of computation*, Vol. 67, No. 221, 1998, pp. 73–85.
- [12] Vreman, A., “An eddy-viscosity subgrid-scale model for turbulent shear flow: Algebraic theory and applications,” *Physics of fluids*, Vol. 16, No. 10, 2004, pp. 3670–3681.
- [13] Moin, P., Squires, K., Cabot, W., and Lee, S., “A dynamic subgrid-scale model for compressible turbulence and scalar transport,” *Physics of Fluids A: Fluid Dynamics*, Vol. 3, No. 11, 1991, pp. 2746–2757.
- [14] Lilly, D., “A proposed modification of the germano sugrid-scale closure method,” *Phys Fluids A*, Vol. 4, 1992, pp. 633–635.
- [15] Wong, M. L., Ghate, A. S., Kenway, G. K. W., Angel, J. B., and Kiris, C. C., “Immersed Boundary Wall-Modelled Large Eddy Simulations of Straight Wings with Three-Dimensional Ice Accretion,” *AIAA Aviation 2024 Forum*, 2024.
- [16] Angel, J. B., Ghate, A. S., Kenway, G. K. W., Wong, M. L., and Kiris, C. C., “Immersed Boundary Wall-Modelled Large Eddy Simulations for Automotive Aerodynamics,” *AIAA Aviation 2024 Forum*, 2024.
- [17] Wong, M. L., Ghate, A. S., Kenway, G. K. W., Angel, J. B., and Kiris, C. C., “Immersed Boundary Wall-Modelled Large Eddy Simulations of Subsonic Jet Flows and Acoustics,” *AIAA Aviation 2024 Forum*, 2024.
- [18] Wong, M. L., Ghate, A. S., Kenway, G. K. W., Angel, J. B., and Kiris, C. C., “Immersed Boundary Wall-Modelled Large Eddy Simulations of a Nose Landing Gear for Aeroacoustics,” *AIAA Aviation 2024 Forum*, 2024.
- [19] Kenway, G. K. W., Ghate, A. S., Browne, O. M. F., Wong, M. L., Angel, J. B., and Kiris, C. C., “Immersed Boundary Wall-Modelled Large Eddy Simulations of a Generic Hammerhead Payload Fairing,” *AIAA Aviation 2024 Forum*, 2024.