

DLR LK6E2 Missile Airframe Validation Study Using Wall-Modeled Large Eddy Simulations

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

This paper presents numerical simulation results for a realistic generic missile in transonic flow conditions using a recently developed Cartesian octree immersed boundary Wall-Modelled Large Eddy Simulation (WMLES) solver, Volcano ScaLES. The missile used in this paper is the subject of NATO Science & Technology Organization (STO) efforts AVT-316 (“Vortex Interaction Effects Relevant to Military Air Vehicle Performance”, previously) and AVT-390 (“Vortex Flow Predictions for Stability and Control of Missile Airframes”, currently) which study the prediction of an experimentally-demonstrated break in rolling moment for total angles of incidence σ beyond 16° . Predicting the wind tunnel results from an angle of attack sweep is considered as a validation case for the solver and the study is taken on in two parts. First, results of a consistent grid refinement study are presented for three angles of attack. Satisfactory accuracy is achieved with a medium resolution “best practice” grid that is suitable for engineering analysis, and subsequently this grid is used to simulate three additional angles of attack to complete an angle sweep. The LES solutions accurately capture the break in rolling moment which eluded many Reynolds-Averaged Navier–Stokes (RANS) solvers in the results from AVT-316. Finally, as a measure WMLES readiness for routine industrial design use, computational resources and timings are also detailed e.g., the solver is capable of completing an aerodynamics simulation using the 435M cell grid in two days using a commodity workstation with only 2× NVIDIA RTX 4090s while simulations using the best practice mesh at 761M cells are completed in roughly twenty-four hours using a single server with 8× NVIDIA L40S cards.

I. Introduction

Computational fluid dynamics (CFD), particularly steady-state Reynolds-Averaged Navier–Stokes (RANS) methods, have been used widely in industrial aircraft development programs to reduce design costs or to accelerate testing cycles; while new development programs are likely to expand their reliance on CFD, as solvers and hardware capabilities continue to improve [1]. The applicability of numerical simulation to the design and testing of aircraft is limited to flow conditions for which the solver has been validated, typically through a representative case study where data is available to confirm the numerical predictions. The maneuverability of agile tactical aircraft and missiles requires testing aerodynamic performance at flight conditions that are sometimes beyond the capability of most current methods of analysis. Aerodynamic flows that occur during critical flight conditions (e.g. take-off, landing, and high-speed maneuvers) are often highly unsteady and complex nonlinear dynamics such as shock buffeting, massive flow separation, and complex vortex interactions are typical [2–4]. An insightful perspective on these limitations is given in a paper by Smith [1] which details the requirements and the opportunities for expanded use of CFD in a development program of a new fighter aircraft. In that work, a classification is defined where test conditions fit into either an ‘A’, ‘B’, or ‘C’ category with ‘A’ being the most routine and least demanding conditions (cruise, low alpha, no vortex separation) and ‘C’ representing the conditions which require scale-resolving simulations or wind tunnel testing owing to the several complex nonlinear flow phenomena mentioned already. With some estimates from the F-35 development program, estimates demonstrate that, for a fully digital development campaign, the conditions in group ‘C’ would dominate the other two categories, consuming more than 80% of the total required core hours. The greatest reduction in program cost through CFD would come from cost-effective methods for solving problems in the ‘C’ class. Two capabilities are highlighted that could significantly reduce the computational costs of development: mature and well-validated Wall-Modelled Large Eddy Simulation (WMLES) and GPU accelerated solvers. This paper may be seen as an effort towards demonstrating these capabilities.

There are no universal criteria such that one can claim that a numerical solver has been “validated”; validation can only be established for particular problems where reliable data already exists [5]. As an example, the NATO Science &

*Corresponding author.

Technology Organization (STO) efforts AVT-316 and AVT-390 [6–9] were organized to assess the capability of CFD solvers to predict an experimentally-demonstrated break in rolling moment for total angles of incidence σ beyond 16° . The transonic regime and high-angle of incidence create conditions for several complex and unsteady phenomena: shock-induced separation at the nose, shock boundary layer interactions over the wings, shock-vortex interactions from the leading-edge vortices, and largely separated regions over the wings at higher angles. The geometry of the missile is depicted in a rolled configuration in Figure 1 along with a visualization of Q-criterion that shows some of the flow features described.

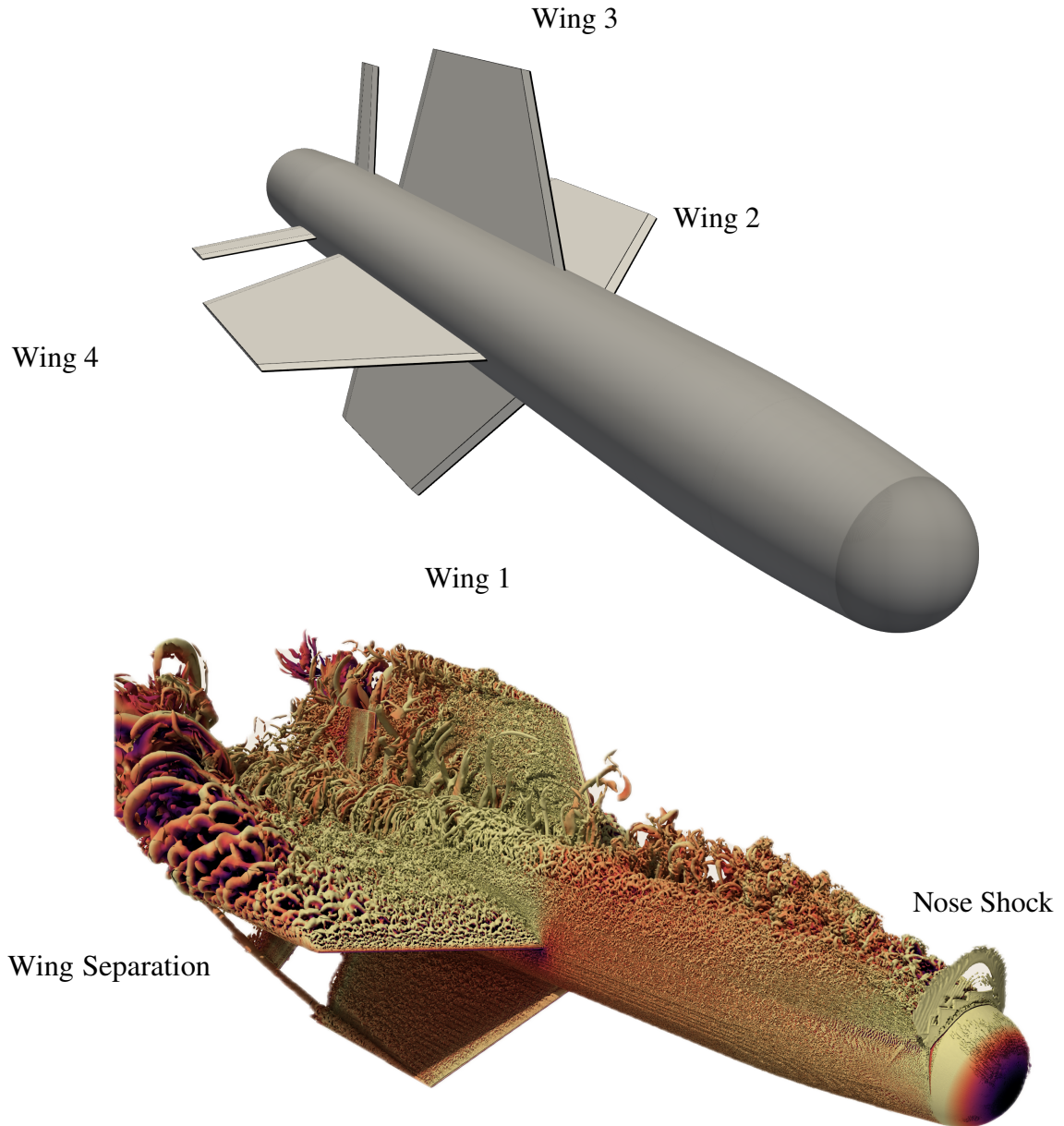


Fig. 1 Left: The missile frame considered in this paper shown with a perspective view. Right: Iso-surface of Q-criterion colored by the streamwise velocity component for a typical solution.

It has been noted that the break in the rolling moment is due to an asymmetric breakdown of the leading edge vortices causing a sudden change in the lift behavior for increasing angle [9]. This problem is viewed to be a representative case

that may arise during the industrial design of a missile. Predicting these kinds of critical phenomena is essential for enabling the practical usage of LES for industrial design and so, the problem is taken up here as a meaningful validation case for the GPU native Volcano ScaLES solver. Other validation studies using Volcano ScaLES include: aerodynamics of the F-16XL at high angles of attack [10], aerodynamics of iced straight wings [11], automotive aerodynamics [12], subsonic jets [13], vibroacoustics of a launch vehicle [14], and commercial aircraft high-lift aerodynamics [15].

The next section describes the solver with some discussion of designing simulation software for modern hardware. The simulation results of Section III are divided into two parts. The first half presents results from a systematic grid refinement study performed using three mesh resolutions at three different angles of incidence. For these three angles, results are compared that show sharply diminishing differences as the mesh spacing is refined. Based on the results of this study, a satisfactory grid resolution is chosen and used to perform three additional simulations for a total angle sweep using six angles. The results of the sweep are presented in the second half of Section III where details of the flow are examined. Section IV will provide an overview of the results and some discussion of future work planned.

II. Volcano ScaLES

Use of general-purpose programmable GPUs has been mainstream in the scientific supercomputing community for more than a decade at this time. Recent breakthroughs in generative artificial intelligence and deep learning technologies that rely heavily on this hardware are driving a surge in GPU production and it appears likely they will continue. The increased production and proliferation of GPUs have made them easily accessible outside of dedicated computing centers. Engineering companies that use computationally aided design are now building on-premises GPU clusters. These devices potentially offer an enormous boost in computational throughput, but the degree to which this potential is realized depends entirely on the design of the simulation software which will look very different compared to CPU based predecessors. The GPU hardware is of a fundamentally different design than the CPUs which developers have programmed for decades and effectively using the hardware requires different design patterns and algorithms. In this section we briefly describe the computational software used for the simulations.

Volcano ScaLES is a compressible Navier–Stokes solver utilizing a nominally fourth-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. Motivated by the need to fully exploit all available resources on modern GPUs, Volcano ScaLES is written exclusively in C++17 and CUDA. The solver is designed for Cartesian octree meshes which are well-suited for large-scale simulations on GPUs. The octree meshes are generated very rapidly which significantly lowers the barrier for rapid experimentation by the user. Grids with over a billion cells are generated in a couple of minutes. Changing geometries is often as simple as a one-line change in the input file. The majority of computations are compute bound, and high performance is achieved even on graphics-oriented devices, bandwidth-restricted devices which are far less expensive compared to higher memory bandwidth hardware like the H100. In simulations involving non-canonical geometries, such as those considered in this paper, we achieve approximately 30% of the 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 400 to 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 lowers the cost of LES simulations to a practical level for use in an industrial setting.

Geometries are represented using an immersed boundary method with appropriate asymptotic Reynolds number properties using an equilibrium wall-model. The wall model that provides a shear stress constraint on the wall uses solution information interpolated through probing at a fixed distance of $3\Delta/2$ in the fluid away from the walls, where Δ is the local grid spacing. The viscous flux discretization utilizes a mix of fourth-order and second-order operators 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 an 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. The simulations presented in this work use the dynamic procedure Smagorinsky model [16, 17] to model the subgrid-scale stresses. The Strong Stability Preserving (SSP)

variant of the classical third order Runge–Kutta scheme developed by Gottlieb and Shu [18] is utilized for all the work presented in this paper.

III. Grid Refinement Study and Experimental Comparisons

This section provides a concise description of the test case and the computational setup for the grid refinement study. Details of the test that are not immediately relevant are omitted. For a comprehensive overview, we recommend the following papers, [6–9].

A. Initial and Boundary Conditions

The nominal conditions used to set up the problem are provided in Table 1 as a reference. The upper rows indicate the quantities used to define the flow conditions with the symbols used in this paper and their units. All simulations are initialized from the freestream conditions derived from Table 1. The reference length d_{ref} is the diameter of the missile. Simulations are run in free-air with a domain $200 \times L$ where $L = 10d_{\text{ref}}$ is the length of the body. The freestream Mach number, speed of sound, and total angle of incidence, σ , define the initial velocity vector field.

Table 1 Nominal freestream and reference conditions.

Static pressure	Static temperature	Roll angle	Mach number	Ref. length
p_{∞} [Pa]	T_{∞} [°K]	λ [deg]	M [-]	d_{ref} [m]
47 075	270	45	0.85	0.05

B. Mesh Considerations

Here we will describe the Cartesian octree mesh* used for the grid refinement study. As already mentioned, the domain is taken to be $200 \times L$ where L is the length of the missile. The mesh is constructed using a bottom-up procedure. A single spacing parameter h is specified as input which represents the baseline spacing for all blocks intersecting geometry. Other levels of the octree are filled on the basis of constraints given as input. Constraints can be volume or surface refinement regions tagged so that their local spacing matches some refinement level relative to h . That is, blocks tagged with a local spacing corresponding to a relative level ℓ will have a spacing $h/2^{\ell}$. For example, a base spacing $h = 1$ mm with a volume region specified to have a relative level $\ell = -1$ would refine all blocks in the region to a local spacing of $h = 2$ mm. Surface regions can also specify the distance that the spacing should extend into the volume.

The mesh is depicted in Figures 2 and 3. Figure 2 shows a center line slice of each mesh. This view is used to show two volume refinement regions that are used in anticipation of the separated vortical flow. Since the same mesh is used for all angles, the slope of the conical volume regions are chosen based on observations of coarse grid solutions at the most separated angle. These are the only volume refinement regions we specify. Next, Figure 3 shows a constant x cut of the meshes near the leading edge of Wing 2. A surface refinement region can be seen on the suction side of Wing 2 where the refined region extends a fixed distance into the volume. As the mesh is refined, the interface will remain roughly fixed in this location. Wing 4 has a similar refinement. These regions are tagged for refinement based on the expectation that the flow will be largely separated will lift into a spiral vortex. A “well-balanced” property is maintained such that adjacent blocks never differ in their relative levels by more than one. The geometry and refinement regions define the topology for the mesh that will remain fixed and global refinement is carried out by reducing the baseline spacing h . A summary of three meshes with their spacing h and total number of grid points is given in Table 2.

*We will use ‘mesh’ or ‘grid’ interchangeably.

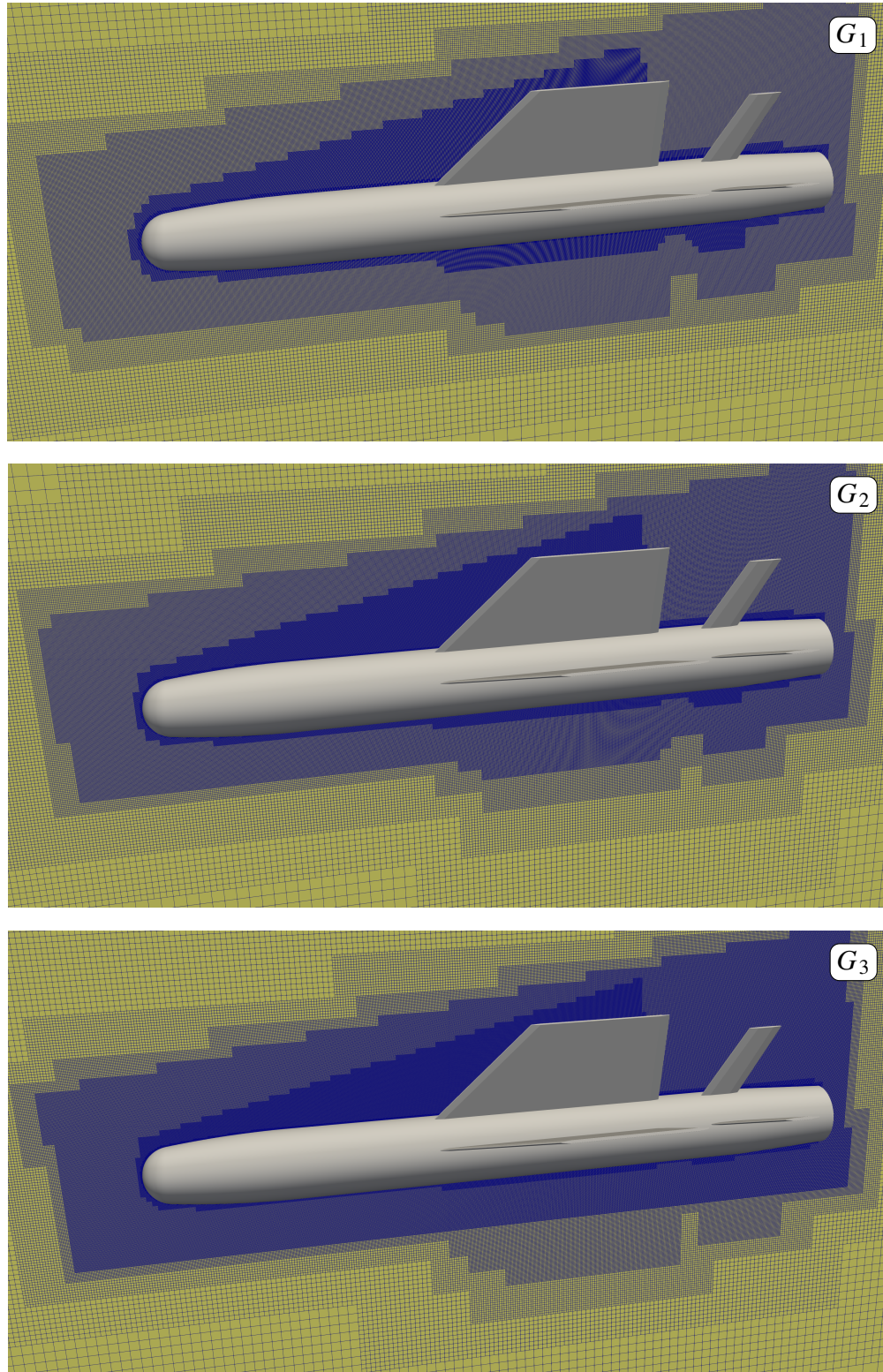


Fig. 2 Constant $y = 0$ m slice of the computational mesh through the missile center line.

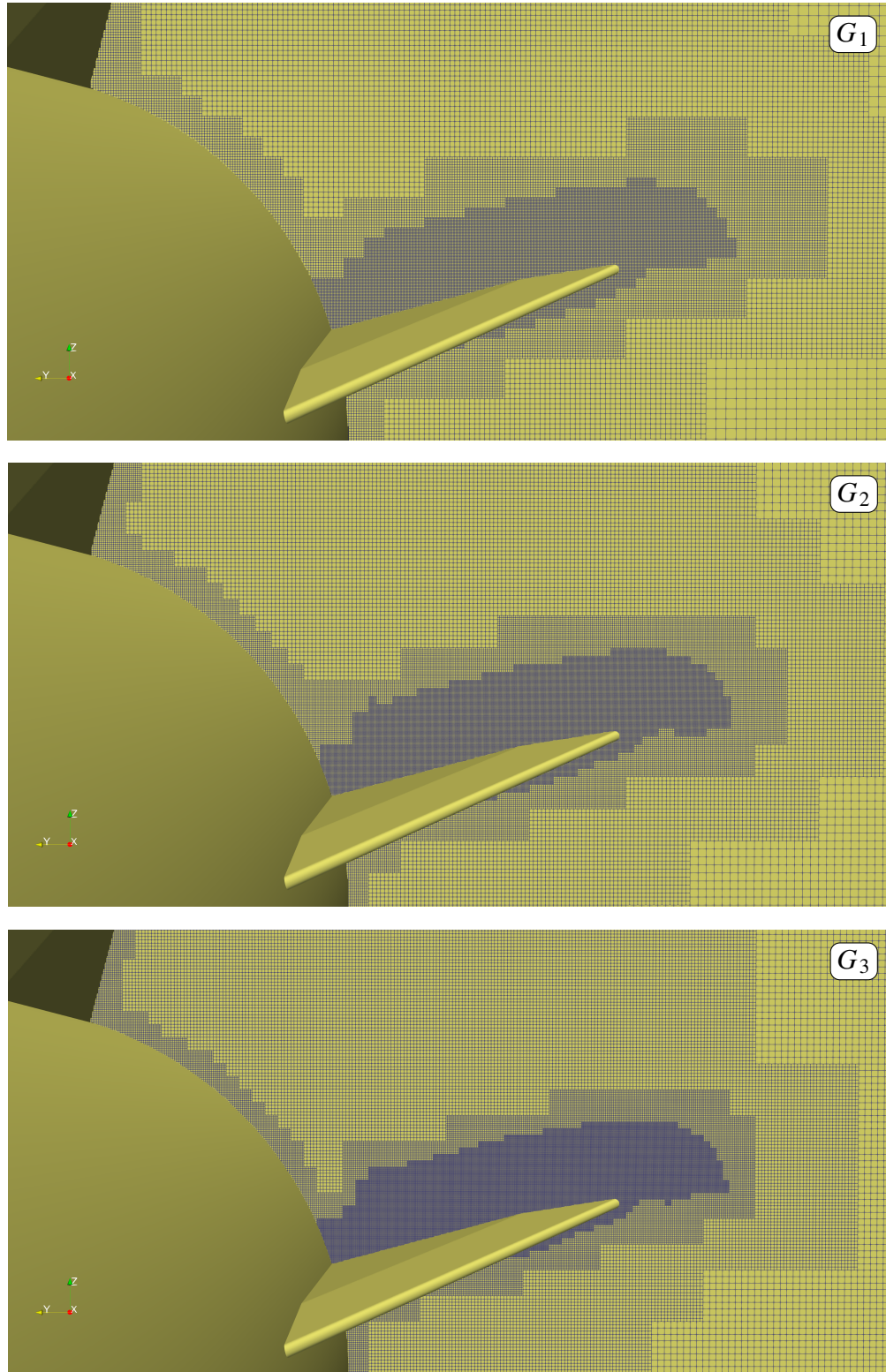


Fig. 3 Constant $x = -0.22$ m slice of the computational mesh looking downstream over the leading-edge of Wing 2.

C. Grid Refinement Study

Table 2 provides details of the meshes used in the refinement study. We label the grids G_1 , G_2 , and G_3 , numbering the grids from coarsest to finest. The second and third columns give the baseline spacing h and the total number of cells in the grid. The mesh G_1 has 435 million grid cells, but can be run on two NVIDIA RTX 4090 cards with 24 Gb of memory each. Meshes G_2 and G_3 require more memory and are run on a server using 8× NVIDIA L40S cards. The third and fourth columns provide some performance metrics in terms of the wall clock time needed to integrate 50 ms of physical time and the number of millions of grid cells updated per second (MUPS), i.e. 7100 MUPS means that 7.1 billion cells are updated per second.

Table 2 Computational cost for the simulations presented in this section.

Label	h [mm]	No. of cells [10^6]	Wall time / 50 ms [hr]	MUPS [10^6 cells/sec]	Hardware
G_1	8.75×10^{-2}	435	42	1950	2× RTX 4090
G_2	7×10^{-2}	761	26	6800	8× L40S
G_3	5.5×10^{-2}	1360	60	7100	8× L40S

Three angles of incidence are chosen, two from the pre-break regime, and one post-break angle: 14° , 16° , and 17.5° . Each angle is simulated using all three grids and their roll moment time histories are plotted in Figure 5. The time histories are plotted with solid lines and colored by their grids. Dashed lines are drawn to indicate the computed mean, and a shaded band is also drawn to indicate a standard deviation about the mean.

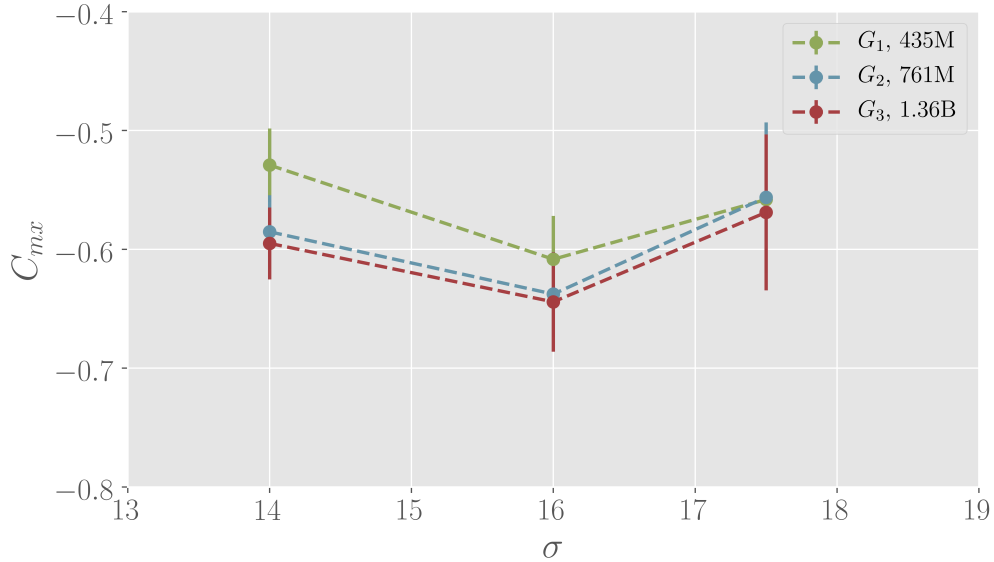


Fig. 4 Rolling moment versus angle of incidence for the grid refinement study.

At 14° and 16° , refining the coarse grid significantly changes the solution, but 17.5° shows almost no difference. It will be shown in the next section that at this angle, much of the coherent vortical flow has broken down into a large separation bubble. With this in mind, it is not expected that grid refinement will change the rolling moment since additional resolved content would not contribute to the moment anyway. The negligible differences in the rolling

moments are most likely due to the expected uncertainty inherent in any estimation of a mean and, with respect to rolling moment, the solution is converged. At all angles, the solutions from the medium resolution G_2 grid are essentially the same as the fine grid. The surface pressure averaged over time is plotted in Figure 6. The differences between grid resolutions are minor on this scale. Even the G_1 solutions show an asymmetric vortex break down, although less pronounced than the G_2 and G_3 solutions. Because of the roll angle, Wing 4 sees a slightly larger effective angle of incidence compared to Wing 2 and as the angle of incidence increases, surface pressure at the trailing edge of Wing 4 decreases more than Wing 2, primarily in the outboard region. This difference in pressure gradients influences the locations of the shocks and later it will be shown that the normal shock on Wing 4 vanishes at a slightly smaller σ than the normal shock on Wing 2. See [8] for comparisons to the wind tunnel surface pressure.

Table 3 Rolling moment coefficients by angle of incidence.

σ	G_1	G_2	G_3
Total angle	(435 M)	(761 M)	(1.36 B)
14.0°	-0.529	-0.585	-0.595
16.0°	-0.609	-0.638	-0.644
17.5°	-0.558	-0.556	-0.569

Figure 4 and Table 3 show the mean and standard deviation respectively for all simulations. The results convincingly show grid convergence for 14° and 16°. Based on the Grid Convergence Index (GCI) [5], the expected change from further grid refinement is about 2%. As already discussed, due to the change in flow physics, increasing grid resolution for $\sigma = 17.5^\circ$ is not expected to have a strong influence on the rolling moment so the answer is essentially converged for our purpose. We conclude that the G_2 grid is sufficiently accurate and provides the best trade-off between accuracy and computation time.

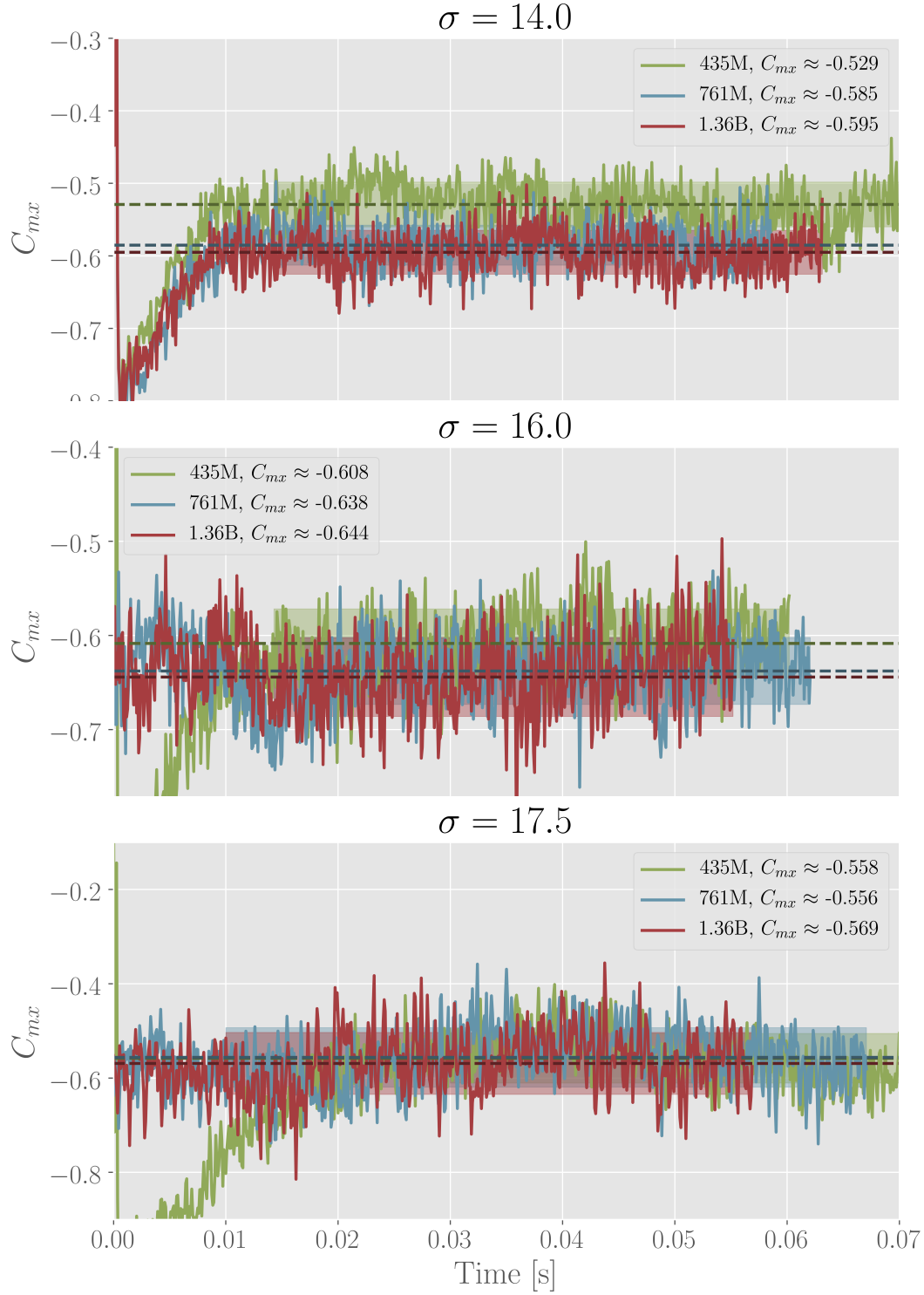


Fig. 5 Time histories for rolling coefficient from the grid refinement study. Means are shown as dashed lines and ± 1 standard deviation is shown by a shaded region. The start of the shaded region indicates the start of time-averaging.

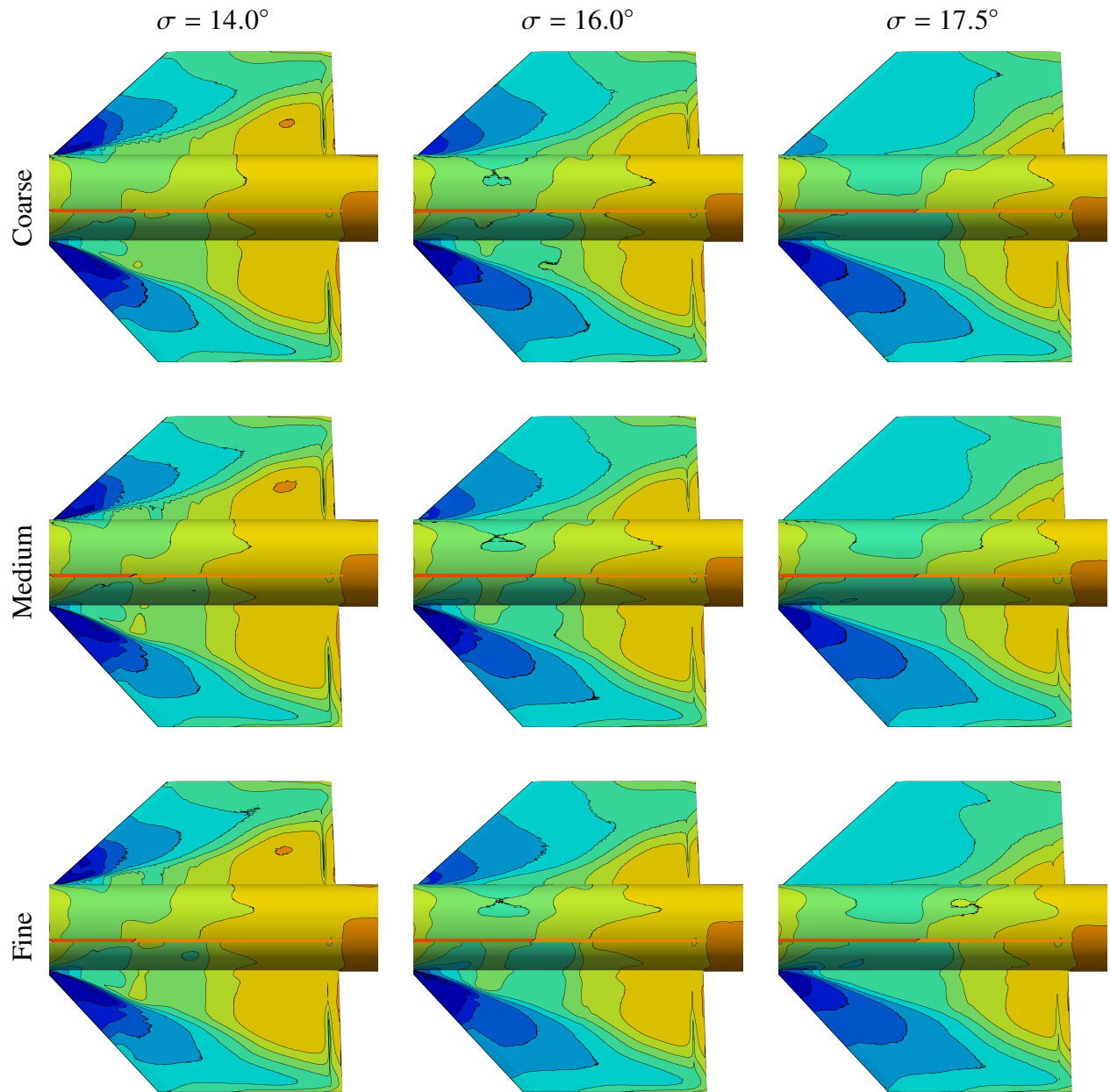


Fig. 6 Surface C_p distributions for angles $\sigma = 14.0^\circ$, 16.0° , and 17.5° .

D. Angle Sweep Results and Comparisons to Wind Tunnel Data

To fill out the sweep we add the three angles $\sigma = 13^\circ$, 15° , and 19° . The rolling moment curve for the entire sweep is shown in Figure 7 along with wind tunnel data collected from two different tunnels plotted in black with the data spread shaded gray. The time histories for each of these simulations are plotted in Figure 8 with the time-average value depicted as a dashed blue line. Figure 8 also shows the experimental data spread that is estimated for each angle by using the shaded region in Figure 7. The simulations reproduce the wind tunnel results remarkably well and the moment break beyond 16° is seen clearly. An individual simulation takes roughly twenty-four hours to complete using $8 \times$ NVIDIA L40S GPUs. Two nodes with this configuration were used to run simulations simultaneously, completing all six simulations in just over three days. This is a compelling indication that LES can be used, today, as a practical design tool with demonstrated improvements at challenging flight conditions. Before examining the flow in detail we remark that the angles chosen here are not clustered to sample the weaker break near 14° ; future simulations are planned for this regime.

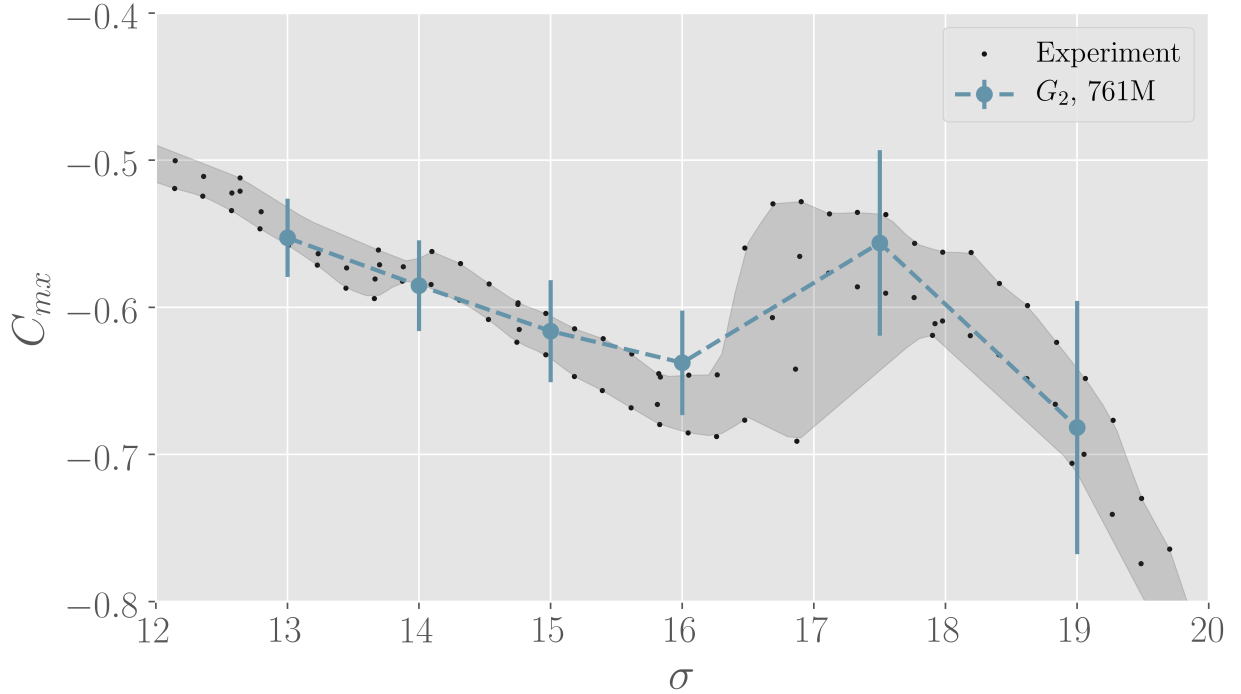


Fig. 7 Complete angle sweep using the medium mesh. Mean values are shown with circles and connected by a dashed line. Experimental values are shown in black with the spread shaded in gray [8].

It is important to note that wind tunnel experiments carried out at over a range of Mach numbers $0.3 < M < 0.85$ did not show a roll moment break at $\sigma = 16^\circ$ for $M < 0.6$ and the break near 14° was only seen at $M = 0.85$ [9], so it is reasoned that the shocks play an essential role in causing this behavior. Figure 9 shows density plotted on a constant z slice above Wing 4 for $\sigma = 13^\circ$ as well as the core of the missile colored by the gradient of the average pressure coefficient, $|\nabla \bar{C}_p|$. This figure highlights some of the well-known characteristics of shock-vortex interactions that are also seen in the simulations. Near the leading-edge root of Wing 4, the vortex roll-up begins and a narrow vortex core forms which can be traced downstream to the point where it passes through a normal shock which is evident in the surface pressure. The adverse pressure gradient from the shock reduces the axial velocity of the vortex, changing the balance between the centrifugal force and the core pressure gradient. The vortex begins to expand at an increased rate compared to its pre-shock state which can be observed in the density. Eventually, all coherence is lost and the vortex is completely broken down.

An explanation for the break in rolling moment beyond $\sigma = 16^\circ$ given in [9] states that around $\sigma = 15^\circ$, the leading normal shock on Wing 4 disappears. This changes the flow separation topology on Wing 4, leading to a large region of separated flow on the outboard section of the leading edge. This is consistent with the present WMLES. Figure 10 plots

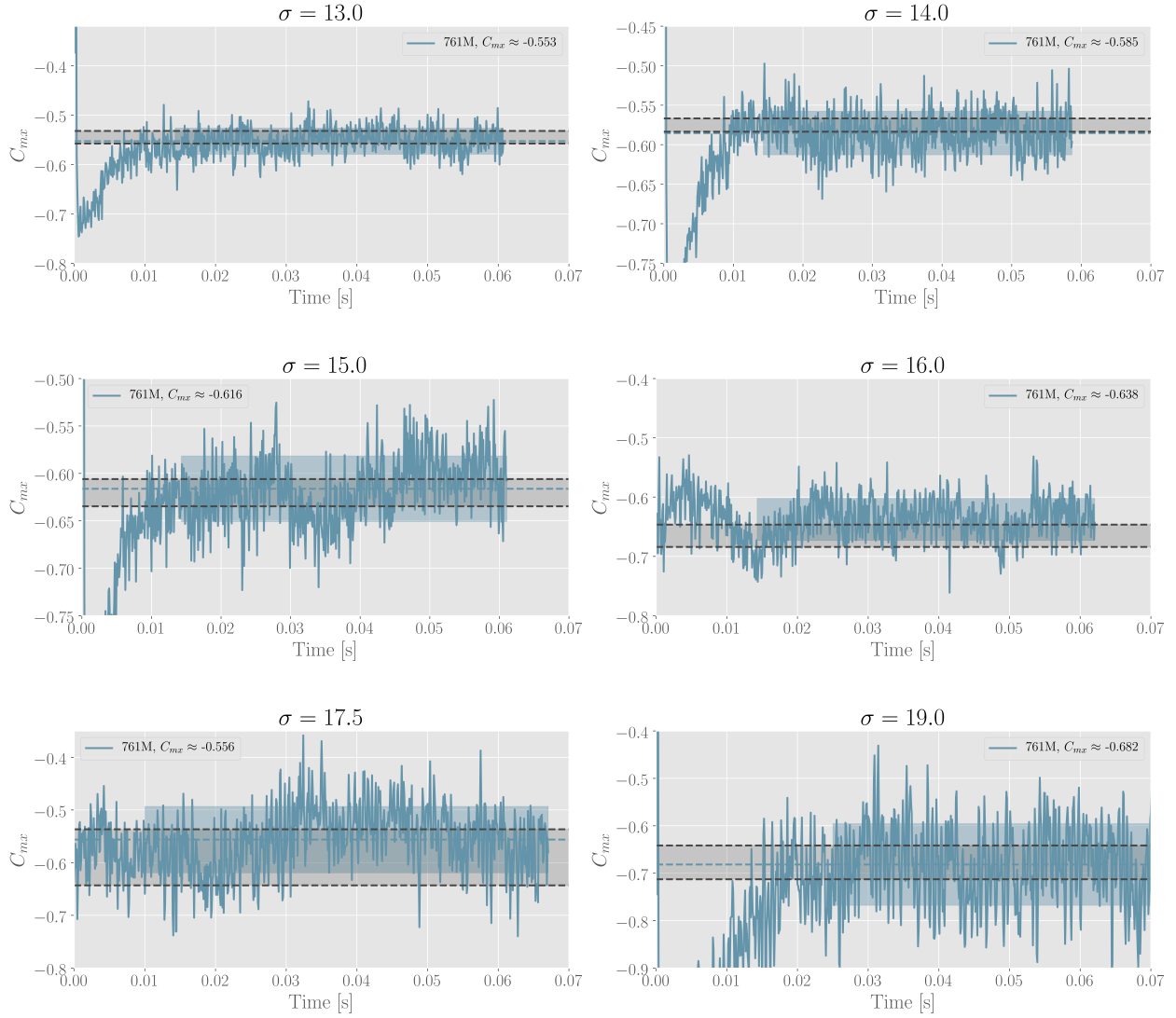


Fig. 8 Load histories for all six angles of the sweep. Experimental data band shown in gray and black.

constant y slices of density for both Wing 2 and Wing 4 at $\sigma = 13^\circ$, 14° , and 15° . Two normal shocks are seen, and at $\sigma = 13$ the adverse pressure gradient due to the shock diverts the separated flow from the leading-edge outboard. Looking at the shocks on Wing 4 for $\sigma = 14^\circ$, the leading shock appears weaker and fluctuations behind the shock suggest some rotational flow is starting to pass across the shock, but largely the separated flow is confined to the leading edge. Increasing to $\sigma = 15^\circ$ the leading normal shock is almost diminished and the outboard separated flow begins to cover more area of the wing. All the while, the shock structure on Wing 2 is relatively stable. The leading shock also weakens on the Wing 2 side but not as severely. Importantly, due to the roll angle, the incoming flow on the Wing 2 side tends to be glancing while Wing 4 tilts more towards the incoming flow which sees higher incoming momentum. The same plots for the higher angles are shown in Figure 11. Once $\sigma = 16^\circ$, the simulation results show the adverse pressure gradient near the leading edge of Wing 4 has reduced to a smoother gradient although the presence of the shock has not completely gone yet. Finally, beyond 16° , the pressure gradients over Wing 4 collapse, massive flow separation occurs, and a weaker normal shock settles connecting Wing 4 and Wing 3. Figure 12 confirms this in the surface streamlines

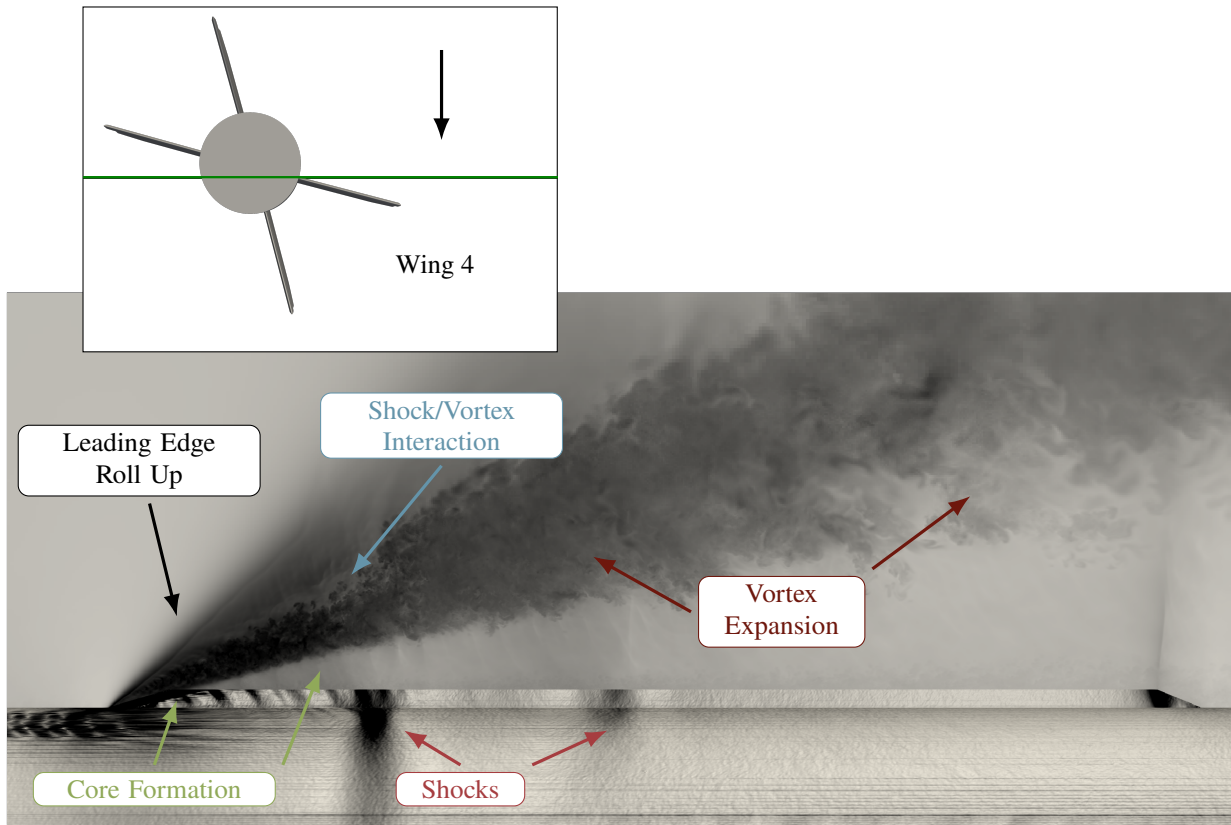


Fig. 9 Density is shown on a constant z slice just above Wing 4. The fuselage is seen colored by $|\nabla C_p|$. The features of the shock-vortex interaction at $\sigma = 13^\circ$ are indicated with arrows. A perspective view of the slice is shown for context.

computed from the instantaneous skin friction for all six incidence angles. The angles are plotted in increasing order from left to right and from top to bottom. Starting from the top, the 13° and 14° simulations show a typical vortex influence with some separation near the leading edges, more noticeably on Wing 4. As the angle increases to 15° and 16° , the separation region on Wing 4 moves more inboard and creates an identifiable separation zone. Beyond 16° , the separation bubble has moved completely over Wing 4 and the leading edge vortex has broken down, while the leading edge vortex from Wing 2 is largely intact. The Wing 2 leading-edge vortex remains mostly stable with some evidence of vortex bursting seen in the streamlines towards the trailing edge.

Another way of demonstrating the loss of coherence for the Wing 4 vortex is shown in Figure 13 using density on a plane at a constant z coordinate that cuts the vortex core near the leading edge. The surface is also colored as in previous figures to indicate the location of shocks. Shock-vortex interactions are seen clearly for the lower angles. At $\sigma = 16^\circ$, the last evidence of coherent vorticity can be seen in the density fluctuations. Increasing the angle further leads to a total loss of the vortex core and the flow over Wing 4 changes character to a large separated bubble with a free shear layer originating at the wing edge.

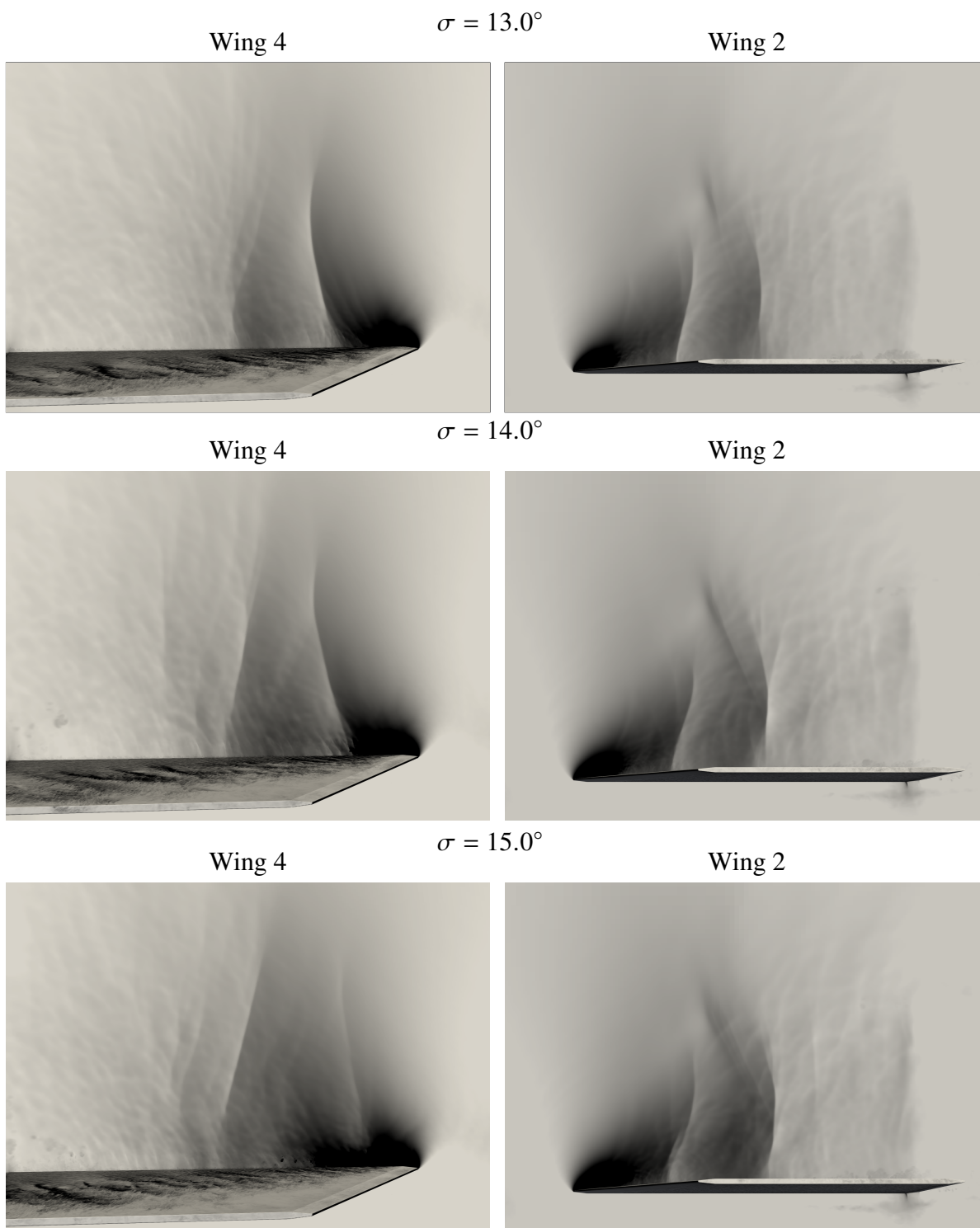


Fig. 10 Density viewed over Wing 4 and Wing 2 showing the normal shocks and regions of separated flow for the lower angles. The surfaces are colored by skin friction with lighter regions indicating flow separation. Both views are looking inboard so that the flow appears reversed.

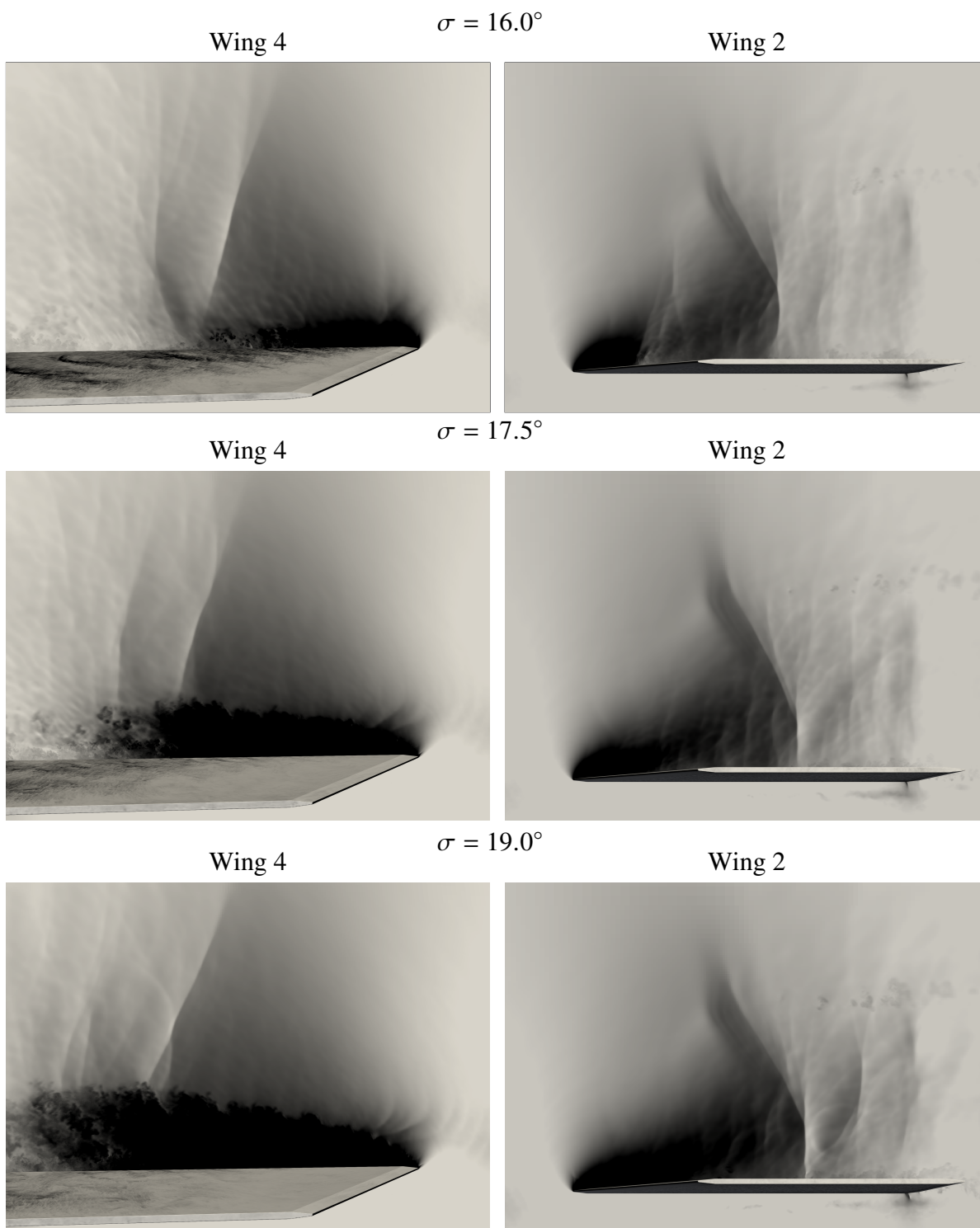


Fig. 11 Density viewed over Wing 4 and Wing 2 for the higher angles showing the normal shocks and regions of separated flow. The surfaces are colored by skin friction with lighter regions indicating flow separation. Both views are looking inboard so that the flow appears reversed.

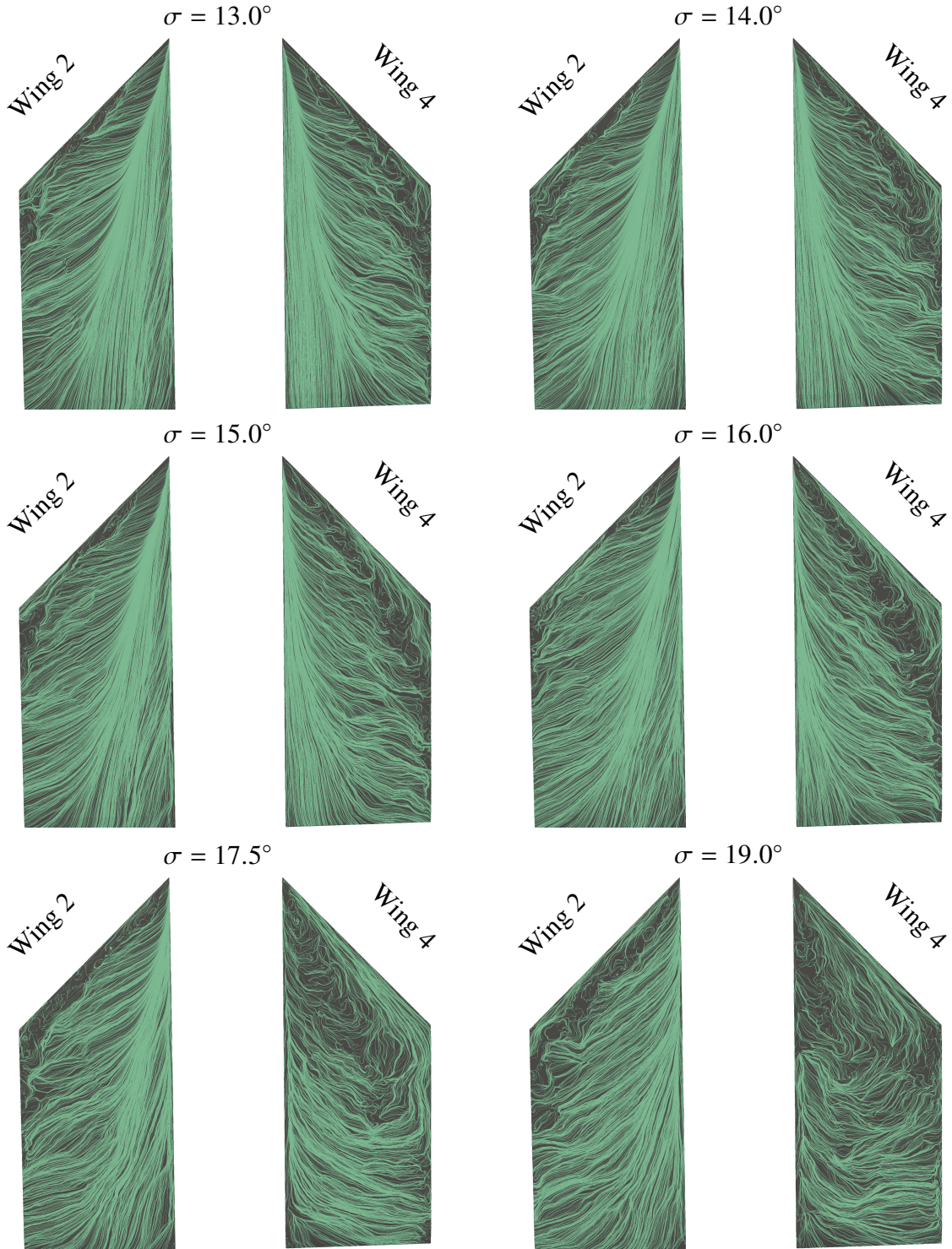


Fig. 12 Instantaneous skin friction streamlines showing the asymmetric vortex breakdown. The separation on Wing 4 moves inboard with increasing angle. Beyond some critical angle between 16.0° and 17.5° the leading-edge vortex breaks down and the flow on Wing 4 shows a large separation bubble.

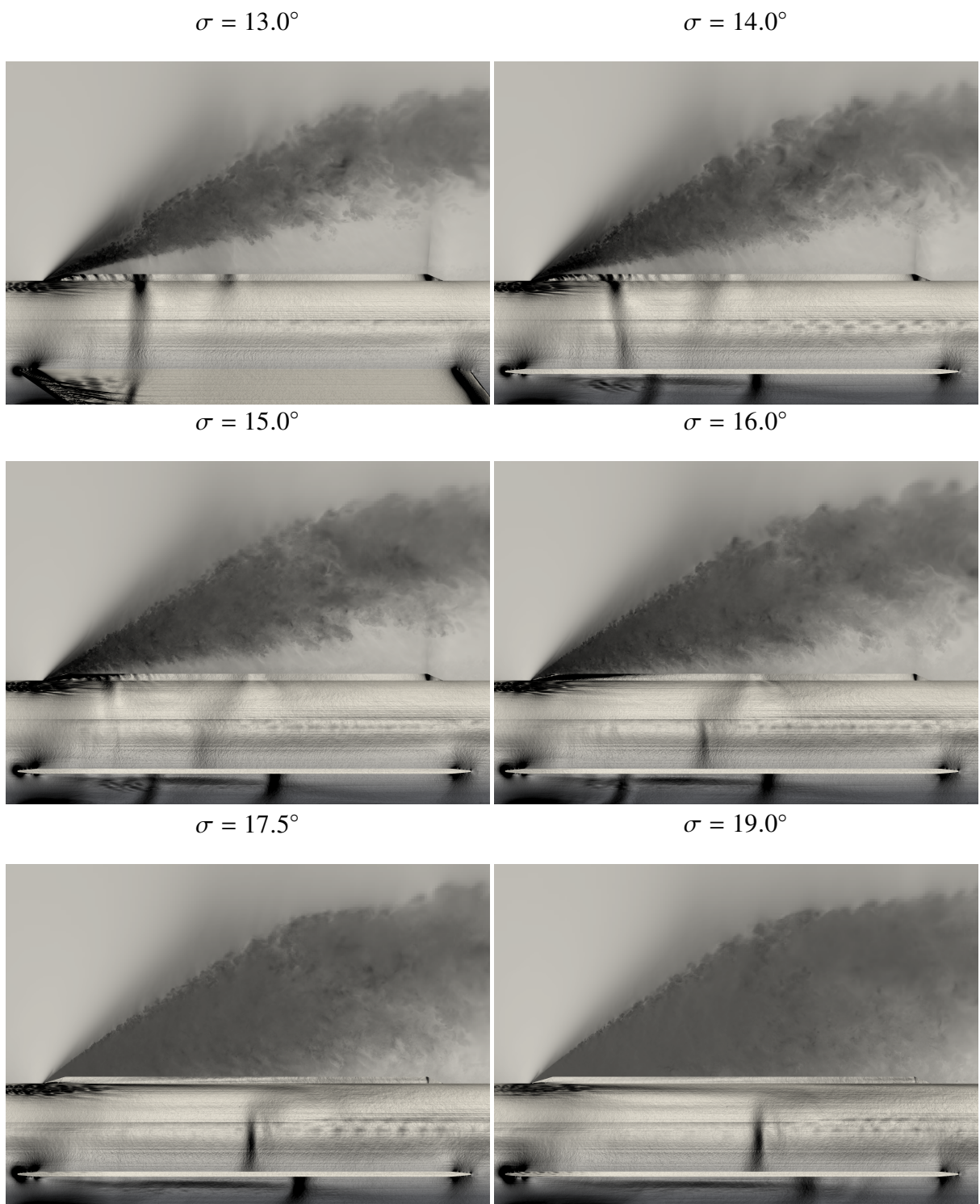


Fig. 13 Density viewed over Wing 4 on a constant z plane that cuts the vortex core near the leading edge. Loss of coherent vorticity is evident with total break down beyond $\sigma = 16^\circ$.

IV. Conclusions

This work presented results of a validation study of an immersed boundary WMLES solver applied to a generic missile in transonic flow. A consistent grid refinement study at three angles of incidence showed that a medium resolution 756 million cell mesh provided essentially converged solutions. Additional simulations at other angles were carried out to perform an angle sweep and results accurately predict a known rolling moment break that has challenged state-of-the-art CFD codes. The sweep simulations were computed using only 8× NVIDIA L40S cards taking about twenty-four hours to complete per simulation. Besides the rapid turnaround time, a major enabler for industrial use is the rapid and iterative grid generation procedure that is used by ScaLES. The 1.36 billion point grids used in this work took fewer than 5 minutes to generate. This rapid workflow allows for systematic grid convergence studies even in non-canonical real-world problems where flight or experimental data is not available. 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.

Acknowledgments

The experimental data used in this paper was provided by the German Aerospace Center (DLR). The authors would like to thank Kai Richter (DLR) and Christian Schnepf (DLR) for making the wind tunnel data available, and the NATO STO AVT-390 project team for providing the geometry and conditions. We would like to acknowledge and thank Nigel Taylor (MBDA UK), James DeSpirito (Army Research Laboratory), and Magnus Tormalm (Swedish Defense Research Agency) and the rest of the AVT-390 team members for the generous invitation that initiated this work and for discussions and suggestions that we found tremendously helpful when preparing this paper.

References

- [1] 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.
- [2] Smith, B. R., “Challenges to the use of CFD in the Military Aircraft Industry,” *AIAA SciTech*, 2015, pp. 1–22.
- [3] 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>.
- [4] 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>.
- [5] Oberkampf, W. L., and Trucano, T. G., “Verification and validation in computational fluid dynamics,” *Progress in Aerospace Sciences*, Vol. 38, No. 3, 2002, pp. 209–272. [https://doi.org/https://doi.org/10.1016/S0376-0421\(02\)00005-2](https://doi.org/https://doi.org/10.1016/S0376-0421(02)00005-2), URL <https://www.sciencedirect.com/science/article/pii/S0376042102000052>.
- [6] Taylor, N. J., “AVT-390: Overview of the Principal Outcomes for the DLR LK6E2 Airframe,” *AIAA AVIATION FORUM AND ASCEND*, 2024. <https://doi.org/10.2514/6.2024-4242>.
- [7] Tormalm, M. H., DeSpirito, J., Schnepf, C., Dikbas, E., and Anderson, M., “Further Comparisons of Predicted and Measured Aerodynamic Characteristics of the DLR LK6E2 Missile Airframe (Scale Resolving),” *AIAA AVIATION FORUM AND ASCEND*, 2024. <https://doi.org/10.2514/6.2024-4240>.
- [8] Schnepf, C., Schüle, E., Weiss, S., and Henne, U., “Additional Wind Tunnel Data for the Transonic LK6E2 Missile Airframe,” *AIAA AVIATION FORUM AND ASCEND*, 2024. <https://doi.org/10.2514/6.2024-4238>.
- [9] Anderson, M., Schnepf, C., Dikbas, E., Tormalm, M. H., DeSpirito, J., Loupy, G., and Rausa, A., “Further Comparisons of Predicted and Measured Aerodynamic Characteristics of the DLR LK6E2 Missile Airframe (RANS),” *AIAA AVIATION FORUM AND ASCEND*, 2024. <https://doi.org/10.2514/6.2024-4239>.
- [10] Angel, J. B., Wong, M. L., Kenway, G., Ghate, A., and Kiris, C. C., “Validation of Immersed Boundary Wall-Modelled Large Eddy Simulations with F-16XL Flight Data,” *AIAA AVIATION FORUM AND ASCEND 2024*, 2024, p. 4374.
- [11] 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.

- [12] 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.
- [13] 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.
- [14] 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.
- [15] Ashton, N., Ghate, A. S., Kenway, G. K. W., Angel, J. B., Wong, M. L., and Kiris, C. C., “Immersed Boundary Wall-Modeled Large Eddy Simulations for the 5th High-Lift Prediction Workshop,” *AIAA Scitech Forum, 2025, AIAA Paper to appear*, Orlando, FL, Jan 6-10, 2025.
- [16] 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.
- [17] Lilly, D., “A proposed modification of the Germano sugrid-scale closure method,” *Phys Fluids A*, Vol. 4, 1992, pp. 633–635.
- [18] Gottlieb, S., and Shu, C.-W., “Total variation diminishing Runge-Kutta schemes,” *Mathematics of computation*, Vol. 67, No. 221, 1998, pp. 73–85.

Appendix: Surface Pressure Cuts

The diagram in Figure 14 shows the spatial location of several cuts where C_p is compared to experimental data for the simulations in Section III. Figures 15, 16, 17, 18, 19, and 20 plot the time-averaged surface data in solid blue along with the wing tunnel data as black square markers. Some discrepancies can be seen but overall the agreement is excellent.

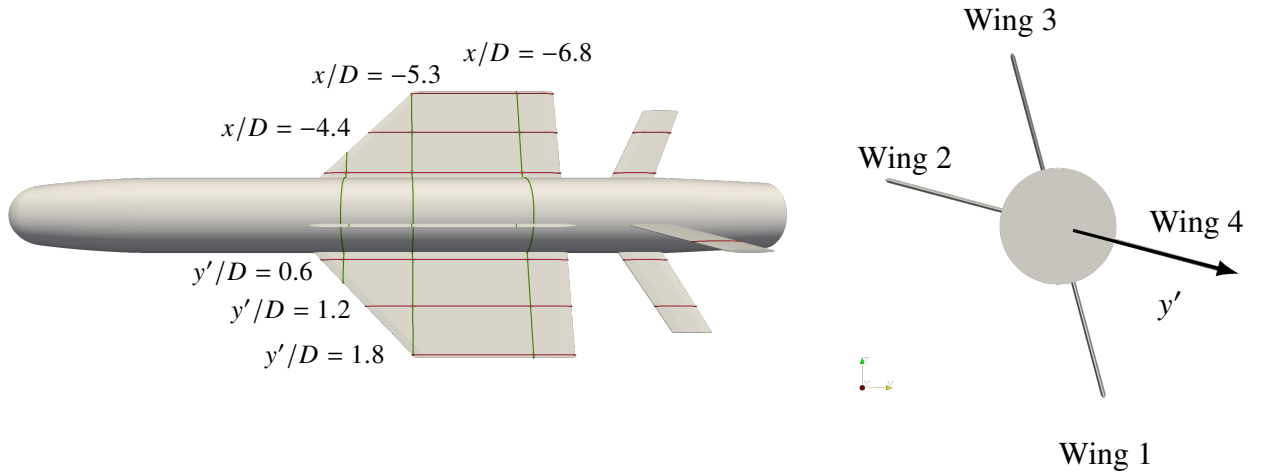


Fig. 14 Missile geometry at roll $\lambda = 45^\circ$. The coordinate y' extends along the span of Wing 4.

$$\alpha = 14.0^\circ, \lambda = 45^\circ, M = 0.85$$

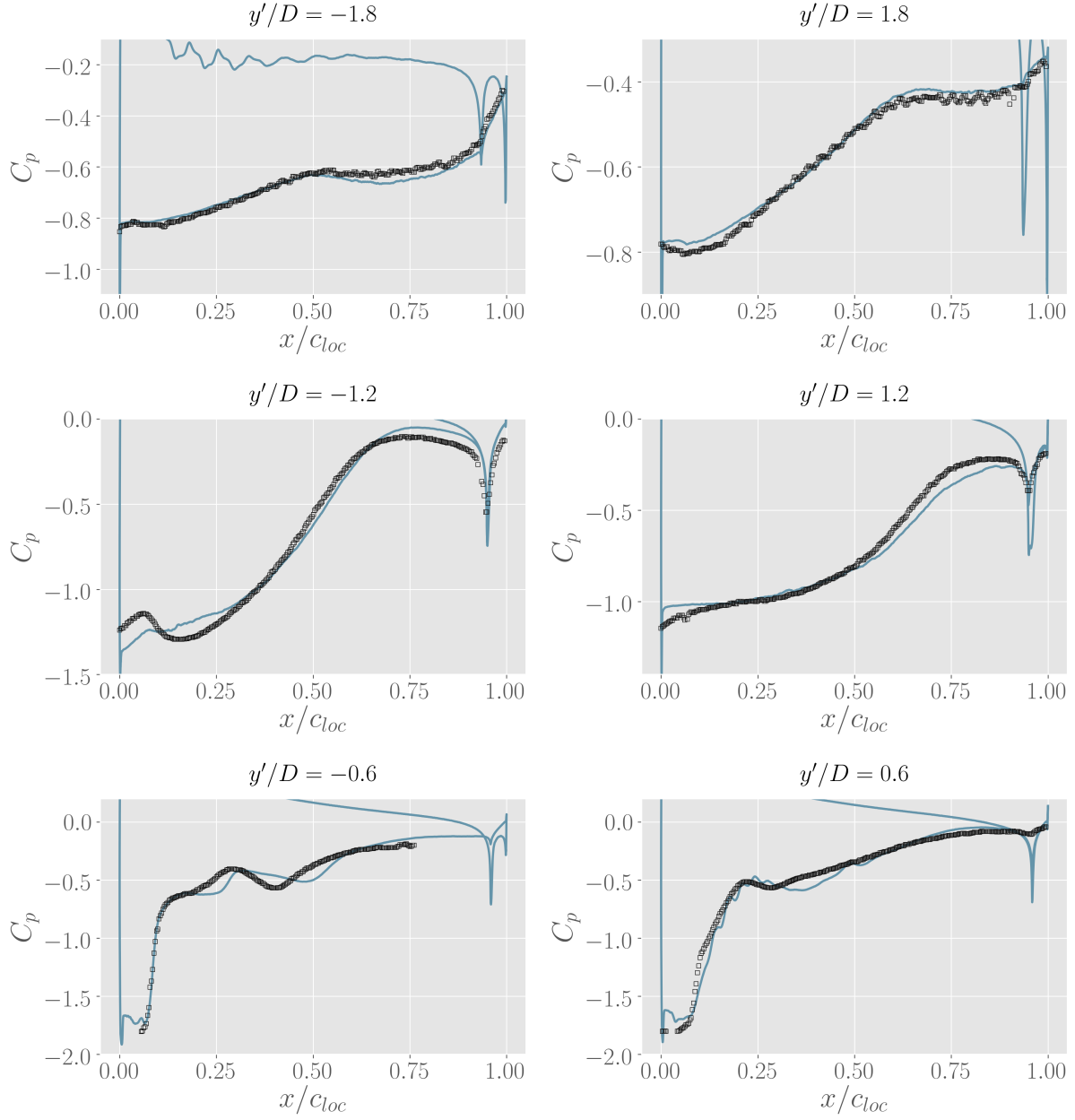


Fig. 15 Cuts for constant y stations at $\sigma = 14^\circ$.

$$\alpha = 14.0^\circ, \lambda = 45^\circ M = 0.85$$

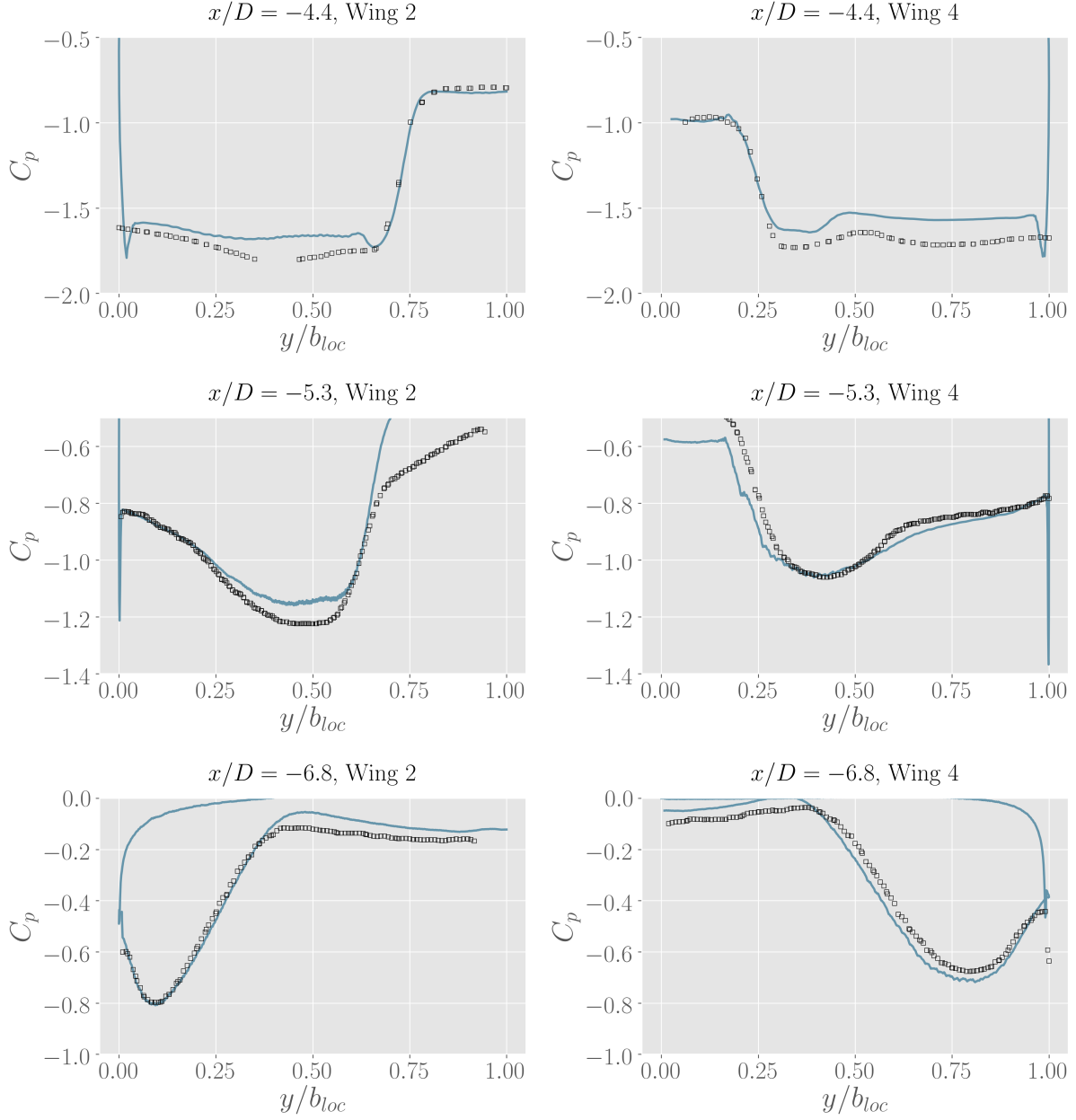


Fig. 16 Cuts for constant x stations at $\sigma = 14^\circ$.

$$\alpha = 16.0^\circ, \lambda = 45^\circ \quad M = 0.85$$

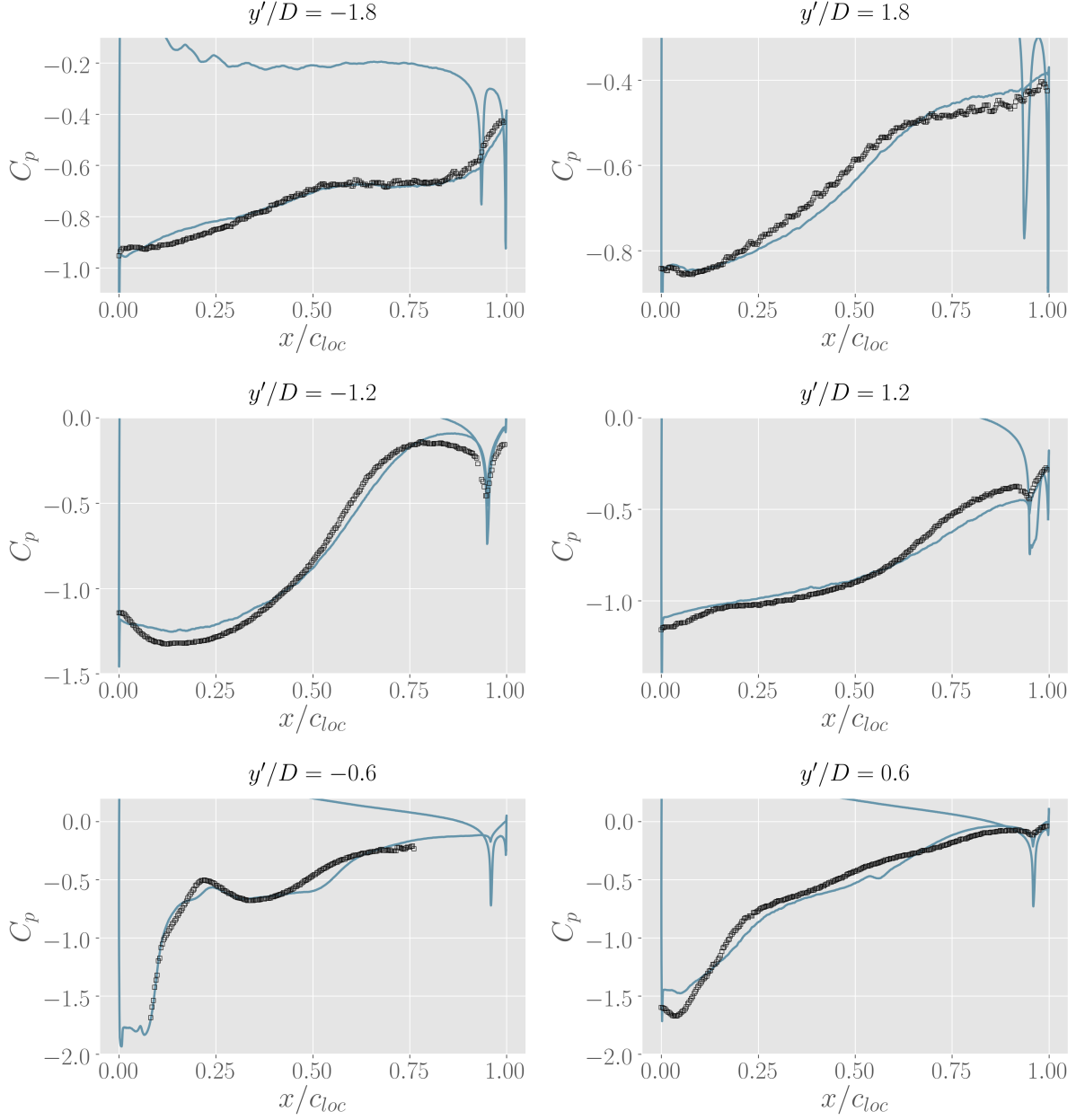


Fig. 17 Cuts for constant y stations at $\sigma = 16^\circ$.

$$\alpha = 16.0^\circ, \lambda = 45^\circ M = 0.85$$

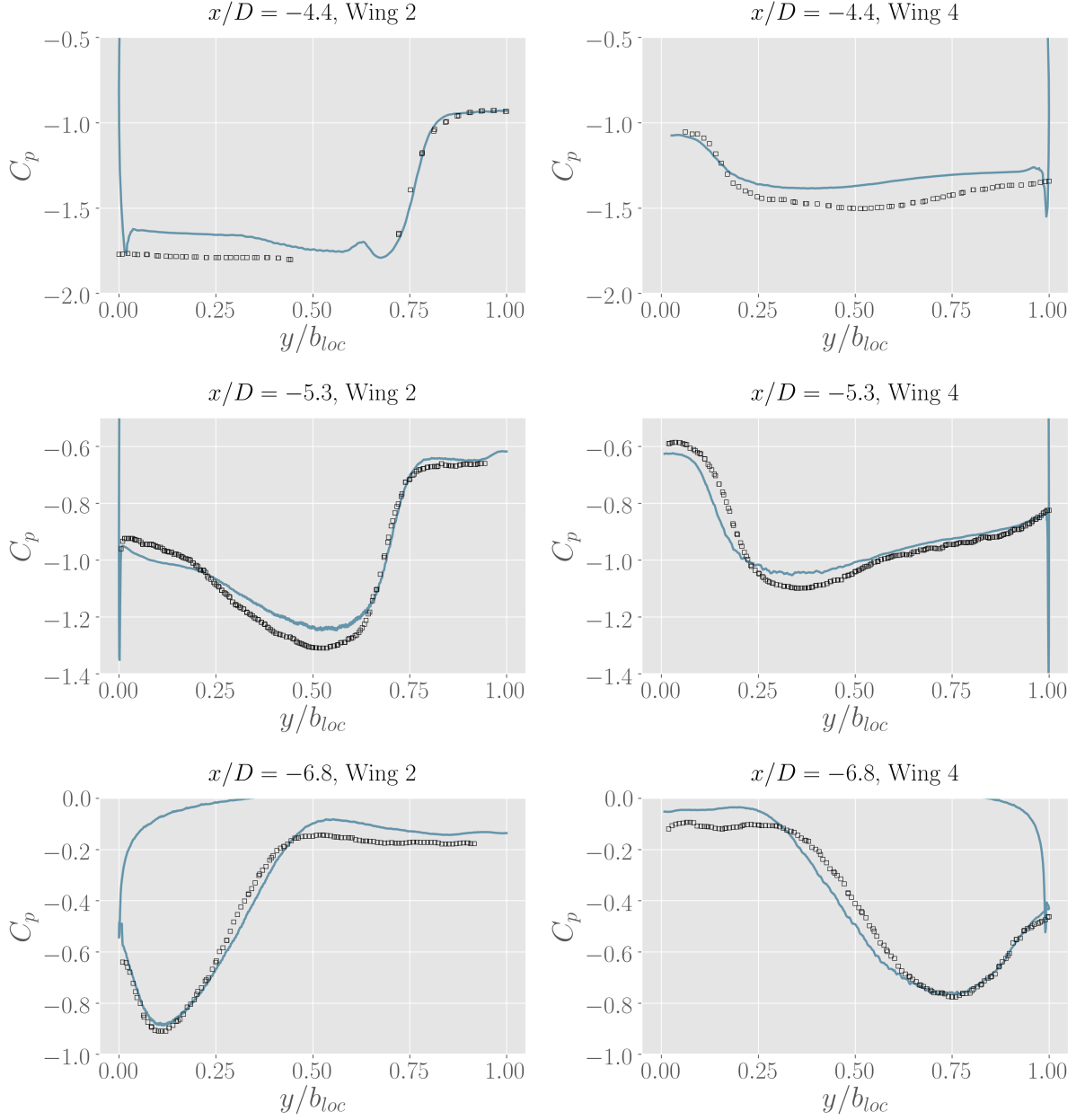


Fig. 18 Cuts for constant x stations at $\sigma = 16^\circ$.

$$\alpha = 17.5^\circ, \lambda = 45^\circ \quad M = 0.85$$

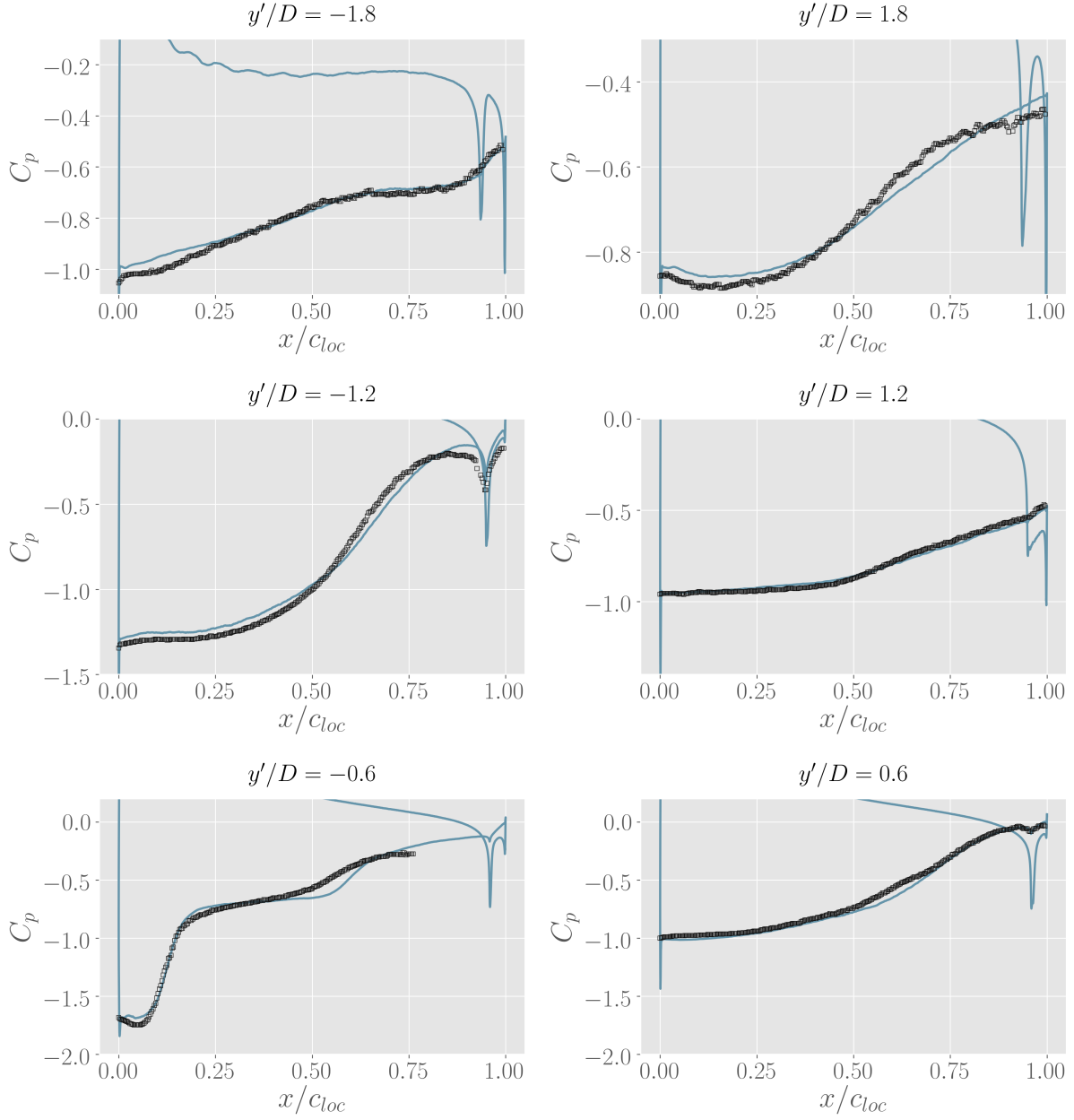


Fig. 19 Cuts for constant y stations at $\sigma = 17.5^\circ$.

$$\alpha = 17.5^\circ, \lambda = 45^\circ M = 0.85$$

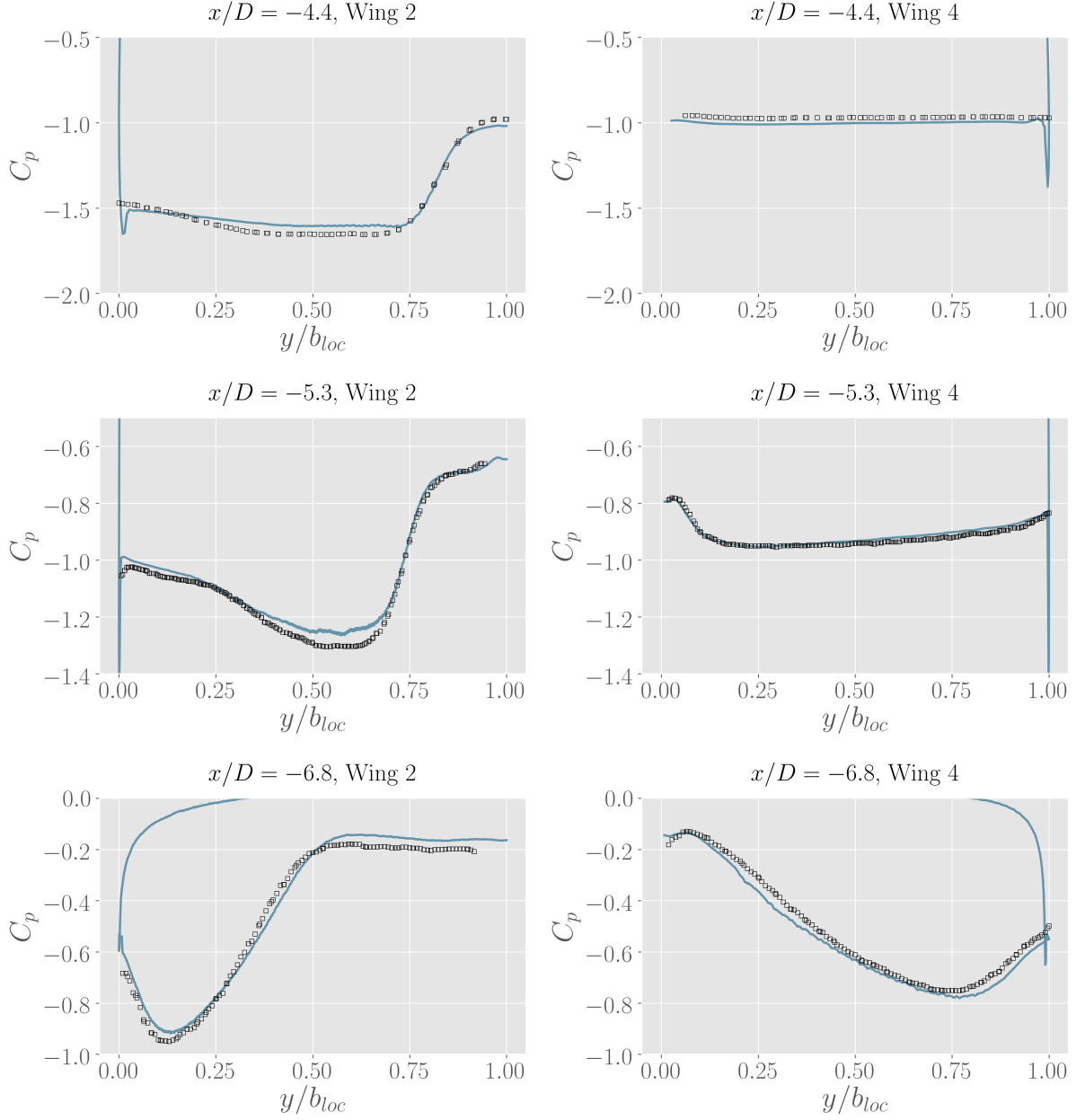


Fig. 20 Cuts for constant x stations at $\sigma = 17.5^\circ$.