

Computational Analysis of a Boundary-Layer Ingesting Tailcone Thruster Configuration Within the National Transonic Facility Using the LAVA Solver

Luis S. Fernandes*, Leonardo G. Machado†, Jeffrey A. Housman‡ and Jared C. Duensing§
NASA Ames Research Center, M/S 258-2, Moffett Field, California, 94035

Boundary-layer ingesting (BLI) propulsion systems are one of the many technologies currently under investigation within the aerospace community to meet NASA's Advanced Air Transport Technology project goal of a sustainable future in aviation. To that end, an experimental campaign was conducted in the National Transonic Facility (NTF) to study the propulsion-airframe integration effects of the Boundary-Layer Ingesting Tailcone System installed in the aft portion of a 2.7% scale design of the NASA Common Research Model (CRM). This test was accompanied by a numerical simulation effort using the Launch, Ascent, and Vehicle Aerodynamics (LAVA) solver, to validate the applicability of current best-practice Reynolds-averaged Navier Stokes (RANS) models in accurately predicting the relevant flow-physics in free-air. A total of 205 steady RANS simulations were performed using LAVA, covering the range of conditions present in the NTF test matrix. Flow conditions ranged from 5- to 15-million Reynolds number, Mach numbers of 0.75, 0.80 and 0.85, and angles-of-attack between -3° and 4° . Four different mass-flow plugs were also tested to assess propulsor operating condition effects. Sensitivity to these flow conditions are analyzed in detail, especially in terms of the nacelle flow distortion which was the main subject of the study. Aerodynamic load coefficients, aftbody boundary-layer measurements and fuselage pressure tap results are also discussed, providing a wide range of validation results from this experimental campaign. This validation effort shows that current best-practices using RANS models within the LAVA solver flow solver are well suited for predicting the inlet flow distortion characteristics of novel aircraft configurations employing BLI at the aft end of the fuselage.

I. Nomenclature

AATT = Advanced Air Transport Technology	α = Angle of attack
BLI = Boundary-layer ingesting	C_L = Lift coefficient
BL = Boundary-layer	C_D = Drag coefficient
CFD = Computational fluid dynamics	$C_{M,y}$ = Pitching moment coefficient
CRM = Common Research Model	Re = Reynolds number
DPW = Drag Prediction Workshop	Ma = Mach number
HL = Highlight	P_t = Total-pressure
LAVA = Launch, Ascent and Vehicle Aerodynamics	$P_{t,\infty}$ = Freestream total-pressure
MAC = Mean aerodynamic chord	s = Length along boundary-layer rake
MFP = Mass-flow plug	D_{nac} = Nacelle inlet diameter
NTF = National Transonic Facility	α_{flow} = Inlet flow angle-of-attack
PAI = Propulsion-airframe integration	β_{flow} = Inlet sideslip flow angle
PR = Pressure recovery	V_x, V_y, V_z = Cartesian velocity components
RANS = Reynolds-averaged Navier Stokes	θ = Inlet azimuthal angle
TCT = Tailcone thruster	ϕ = Fuselage azimuthal angle

*Research Scientist, Science & Technology Corporation, Computational Aerosciences Branch, AIAA Member, luis.fernandes@nasa.gov

†Research Scientist, Science & Technology Corporation, Computational Aerosciences Branch, AIAA Member, leonardo.machado@nasa.gov

‡Research Scientist, Computational Aerosciences Branch, AIAA Senior Member, jeffrey.a.housman@nasa.gov

§Computational Aerosciences Branch Chief, Computational Aerosciences Branch, NAS Division, AIAA Member, jared.c.duensing@nasa.gov

II. Introduction

THE goal of the Advanced Air Transport Technology (AATT) project is to bring forth NASA’s vision for a sustainable future in aviation. To that end, the project supports the development of revolutionary concepts and technologies for fixed wing subsonic transports that will enable cleaner, quieter and more efficient aircraft. Given the 2035–2045 time-frame over which these key enabling technologies are supposed to reach mature readiness levels, specific system-level success metrics that separately target aircraft noise, nitrogen oxide (NO_x) emissions, and fuel consumption have been defined and partitioned into near term, mid term, and far term goals (Celestina and Long-Davis, 2019). One such next-generation technology currently being explored in the context of AATT’s mission to reduce fuel burn is boundary-layer ingesting (BLI) propulsion systems. A number of vehicle configurations employing this innovative concept have been proposed and are currently under investigation (Kawai et al., 2006; Drela, 2011; Welstead and Felder, 2016; Jansen et al., 2022).

At a system level, the benefits of BLI stem mainly from two contributions. Firstly, the lower engine inlet and jet-exhaust velocities resulting from the ingested low-speed boundary-layer flow contribute to a higher propulsive efficiency. As a result, reduced fuel burn is able to sustain the same drag force experienced by the aircraft. In addition, the level of propulsion-airframe integration (PAI) offered by BLI technology results in lower weight and drag, with potential to benefit noise shielding as well. Hardin et al. (2012) report a 3–5 % reduced fuel burn for a compact-inlet, aft-mounted propulsion system when compared to a baseline pylon-mounted ultra-high-bypass (UHB) propulsion system, following a trade-factor-based analysis using a one-dimensional engine cycle model. A more recent study by Ochs et al. (2017) incorporating three-dimensional computational fluid dynamics (CFD) to more accurately account for the coupling between inlet, fan, fan exit-guide-vanes, and nozzle, shows a 4.2–4.5 % propulsive efficiency gain for a propulsive system employing BLI over a conventional one. These values are supported by a new study from Machado et al. (2024) analyzing BLI benefits on variants of the NASA Common Research Model (CRM) — identical to the ones analyzed in this work — and Transonic Truss-Braced Wing geometries. Hall et al. (2017) use control volume analyses to present a detailed breakdown of the dissipation mechanisms that need to be balanced by power added by the propulsor, and report a 9 % reduction in required propulsor mechanical power for a BLI configuration compared with a non-BLI one using the D8 concept (Drela, 2011). These studies offer a promising assessment of the potential savings in incorporating BLI into present and future airframe designs.

Two types of BLI systems can be distinguished based on how the PAI has been employed. Type-I BLI, also known as 180° distortion, is characterized by a low total-pressure region with large variation in swirl (flow angle in the circumferential direction) over the bottom portion of the engine inlet, corresponding to air ingested from the boundary-layer developing on the upper part of the aircraft’s body, typically the fuselage. The upper-half portion of the engine ingests freestream air with little to no swirl, similar to a conventional pylon-mounted propulsor. Both the D8 (Drela, 2011) and blended wing body (Kawai et al., 2006) concepts employ this PAI configuration. The other variation, called Type-II BLI, or 360° distortion, typically consists of an engine installed at the closeout of the fuselage, ingesting a ring of boundary-layer flow that develops over the fuselage of the aircraft. The inlet flow-field signature for this configuration is characterized by a low total-pressure region around the hub, with circumferentially localized low total-pressure and high-swirl regions that result from the combined effect of wing down-wash, fuselage up-sweep and vertical tail wake. Type-II BLI is employed in the Single-aisle Turboelectric AiRCraft with Aft Boundary-Layer propulsion (STARC-ABL) (Welstead and Felder, 2016) and SUBsonic Single Aft eNginE (SUSAN) (Jansen et al., 2022) concepts.

The goal of this work is to benchmark a modern CFD solver against wind tunnel data from a conventional commercial transport aircraft model employing Type-II BLI in the near-transonic regime, and develop best-practices for modeling such configurations. System-level studies for advanced concepts employing BLI will greatly benefit from an improved understanding of the coupled physics of the problem, particularly the PAI coupling. Further insight on the sensitivity of key flow features at the engine inlet, such as total-pressure and swirl, with changes in angle-of-attack and propulsor operating-conditions will also help inform future concept studies employing BLI. With this vision in mind, the aftbody section of a 2.7 % scale of the NASA CRM configuration was modified to accommodate a retrofitted tailcone thruster (TCT), hereafter designated as the CRM-TCT configuration. This model was subject to a comprehensive testing campaign (Chan et al., 2024) conducted at the National Transonic Facility (NTF) at NASA Langley Research Center in 2023. The full span model was installed in the NTF and tested at flight Reynolds numbers, based on the mean aerodynamic chord (MAC), ranging from 5- to 15-million, and Mach numbers of 0.75, 0.80 and 0.85. The data collected as part of this effort is used in the present work for comparison against results from the Launch, Ascent, and Vehicle Aerodynamics (LAVA) CFD code.

This paper is organized as follows. A brief description of the NTF test setup is first provided in Sec. III. A more

in-depth description of the wind tunnel test is provided in the companion paper by [Chan et al. \(2024\)](#). This is followed by a summary of the numerics and RANS models employed to simulate the wind tunnel test in Sec. IV. Preliminary studies performed to establish best-practices for running the entire database of conditions are described in Sec. V. This includes a grid refinement study and a turbulence model sensitivity study to identify and improve areas where the numerical results show discrepancies with the experimental data. Test matrix results covering a wide range of sensitivities to Mach number, Reynolds number, angle-of-attack and propulsor operating conditions are then reported and discussed in Sec. VI. Final conclusions are drawn in Sec. VII. Another companion paper describing the work done in parallel using the USM3D-ME flow solver can be found in [Bozeman \(2024\)](#).

III. Description of the National Transonic Facility Test Setup

Since being declared operational in late 1984, the NTF has been used as a research tool in a wide variety of aerodynamic investigations, ranging from fundamental research to studies of advanced components and concepts ([Wahls, 2021](#)). As the world’s largest pressurized cryogenic wind tunnel, it enables testing of scaled models at flight Reynolds numbers using supercold nitrogen (N_2).

The Boundary-Layer Ingesting Tailcone System (BLITS) test aims at understanding and quantifying the PAI effects of a tailcone system in the aft portion of a CRM-type airplane model. A 76 inch scale model of the CRM, consisting of a modified fuselage incorporating a tailcone thruster designed for BLI and PAI technology assessment, wings, vertical tail, and horizontal tails (with a -2° incidence angle) was manufactured for testing in the NTF ([Chan et al., 2024](#)). An overview of the geometry with the TCT aftbody installed is shown in Fig. 1(a), together with a zoomed in view of the TCT aftbody in Fig. 1(b). To investigate the integration effects of the aftbody TCT, a conventional aftbody was also designed and manufactured, herein labeled the *clean* configuration. The TCT aftbody is composed of the fuselage extension, engine nacelle, mass-flow plug (MFP) and the struts to support the testing hardware installed at the nacelle highlight. A set of four mass-flow plugs were designed to assess the PAI effects under a range of engine operating-conditions. Details regarding the sizing of these mass-flow plugs for the NTF CRM-TCT model, as well as preparation of the facility for PAI testing, can be found in [Jones et al. \(2021\)](#). A side view of the different mass-flow plug profiles is provided in Fig. 1(c). Additional dimensional details regarding the model are provided in Table 1.

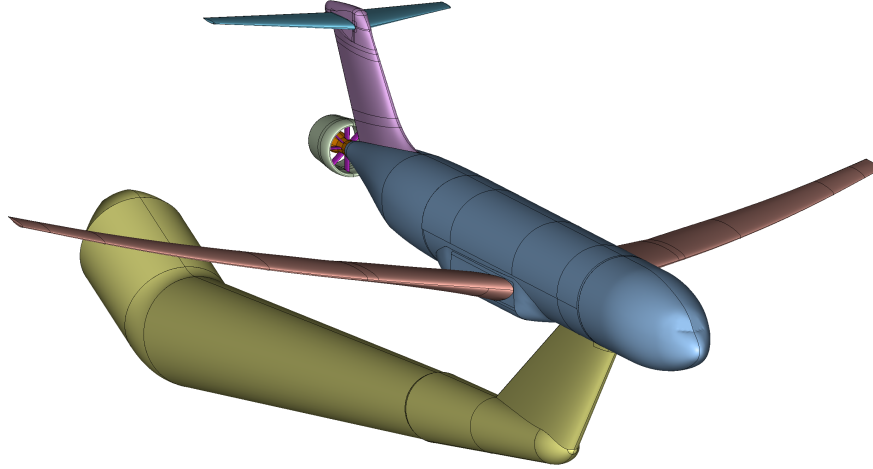
Table 1 Dimensional details of the 2.7 %-scale CRM-TCT model tested in the NTF.

Model Scale [%]	Model Span [in]	Wing Aspect Ratio ($\mathcal{A}$) [-]	Reference Length (c_{ref}) [in]	Reference Area (S_{ref}) [ft ²]	Reference Moment Center ($\vec{x}_{ref}$) [in]
2.7	62.4672	9.0	7.4466	3.0110	[-3.617, 0.0, -1.718]

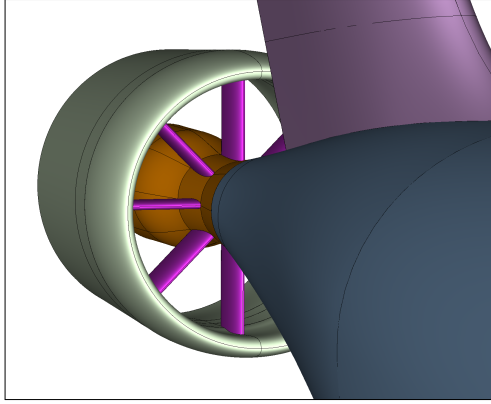
The modularity of the configuration allows for a seamless exchange between the two aftbody designs and the four mass-flow plugs available for the TCT aftbody, as well as the removal of the horizontal tail for the negative angles-of-attack[†]. Finally, the test article is mounted to the wind tunnel model interface using the 20° bent stub sting (NTF stub sting #5), lower swept strut, and 0° yaw strut block. To assess the effects of the sting mounting hardware, both free-air (*w/o sting*) and mounted (*w/ sting*) configurations are simulated. Finally, all simulations performed consider the geometry in free-air.

A summary of the configurations and freestream conditions considered in the present work is presented in Table 2. The complete set of Reynolds numbers and Mach numbers is simulated only for the *clean* and *flow-through TCT* configurations. Note that throughout this work the *flow-through TCT* label refers to the geometry with the *Cruise MFP* installed, and the Reynolds numbers mentioned are based on the mean aerodynamic chord of the model (c_{ref} in Table 1). Out of these, a single condition — 10-million Reynolds number, 0.80 Mach number — was selected to assess sting effects. The full angle-of-attack sweep ranges from -3° to 4° . Data was collected at 0.25° increments in angle-of-attack for the experimental alpha sweeps, while a 1° increment is used in the numerical simulations to capture the load curves. Another shorter angle-of-attack sweep spanning only the positive range (0° to 4°) was defined for a subset of cases. The reason for this, as noted above, is due to the fact that the horizontal tail was removed prior to running the negative angles-of-attack during the test. The gas model properties for the simulations at a Reynolds number of 5-million emulate those of standard atmosphere air, while those for the Reynolds numbers of 10- and 15-million use molecular nitrogen

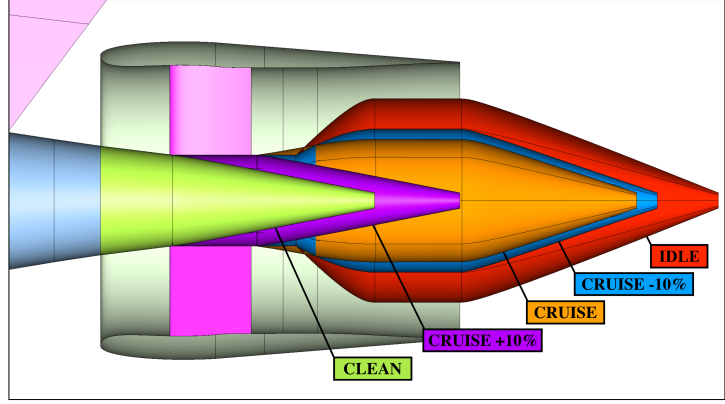
[†]The decision to remove the horizontal tail at negative angles-of-attack was made during the experimental campaign, driven by concerns that the high moment load carried by the horizontal tail under these conditions would damage the force balance instrumentation.



(a) Geometry of the CRM-TCT with the wind tunnel mounting hardware attached to the fuselage.



(b) Detailed view of the aftbody TCT.



(c) Side view of the different mass-flow plug profiles.

Fig. 1 Geometry of the CRM-TCT model analyzed in the present work.

(N_2) as required in the experiment to achieve these flight Reynolds number conditions. In both cases, the perfect gas model is utilized. Finally, the flow-through configurations employing different mass-flow plugs are simulated at a freestream Reynolds number of 5-million only, and are meant to assess engine operating-condition effects.

A. Wind Tunnel Instrumentation

A vast amount of data was extracted during the experimental campaign, much of which is used throughout this work for CFD code validation purposes. Firstly, forces and moments were measured using a six-component force balance mounted inside the test article, and were then converted into meaningful aerodynamic load coefficients such as lift, drag and pitching moment. These were further corrected in an attempt to account for wall and blockage effects. All experimental aerodynamic load coefficients used for comparison in this work use the corrected dataset. For the clean aftbody configuration, two sets of boundary-layer rakes were distributed around the end of the fuselage to characterize the near-wall flow at different azimuthal stations. A set of five rakes were placed at the *forward* axial station ($X = 47.719$ inches), at azimuthal angles of 0° , 45° , 90° , 135° and 180° . The *aft* axial station ($X = 58.228$ inches) boundary-layer was characterized using a set of just three rakes, placed at circumferential angles of 60° , 120° and 180° . The location of these boundary-layer rakes is illustrated in Fig. 2(a). Note that the geometry of these rakes was not part of the computational model; instead, the solutions were sampled at the locations where the measurement is made.

In addition, three sets of nacelle rakes were designed for flow-distortion characterization at the nacelle highlight. The location of the probes on each of these rakes follows the SAE Recommended Practice ARP1420 document (S-16 Turbine Engine Inlet Flow Distortion Committee, 1978), which defines several metrics used to analyze total-pressure distortion

Table 2 Summary of test conditions and configurations simulated.

Configuration	T_0 [°F]	Reynolds Number [$\times 10^6$]	Mach Number [-]	Sting Included	Angle-of-attack Sweep [°]	Gas Model
<i>Clean</i>	120	5	0.75, 0.80, 0.85	✓	−3 – 4	Air
	−50	10	0.80	✗	0 – 4	N ₂
	−50	10	0.75, 0.80, 0.85	✓	−3 – 4	N ₂
<i>Flow-through TCT (w/ Cruise MFP)</i>	−120	15	0.75, 0.80, 0.85	✓	−3 – 4	N ₂
	120	5	0.75, 0.80, 0.85	✓	−3 – 4	Air
	−50	10	0.80	✗	0 – 4	N ₂
<i>Flow-through TCT w/ Cruise −10 % MFP</i>	−50	10	0.75, 0.80, 0.85	✓	−3 – 4	N ₂
	−120	15	0.75, 0.80, 0.85	✓	−3 – 4	N ₂
	120	5	0.75, 0.80, 0.85	✓	0 – 4	Air
<i>Flow-through TCT w/ Cruise +10 % MFP</i>	120	5	0.75, 0.80, 0.85	✓	0 – 4	Air
<i>Flow-through TCT w/ Idle MFP</i>	120	5	0.75, 0.80, 0.85	✓	0 – 4	Air

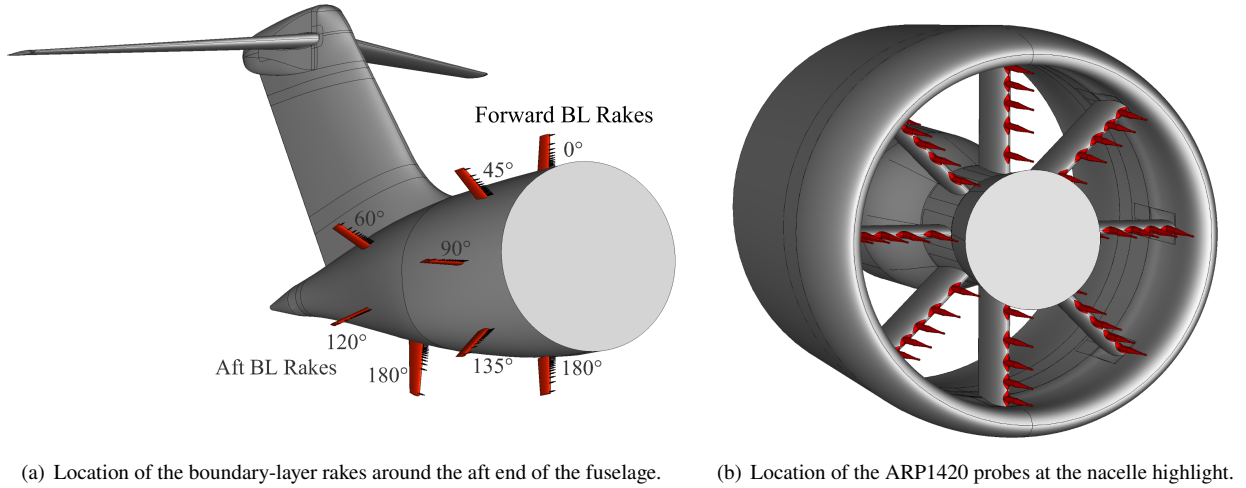


Fig. 2 Wind tunnel instrumentation locations. Note that the red-accented features are not modeled in the CFD simulations performed.

patterns typical of gas turbine inlets in an experimental setting. Firstly, a fixed rake composed of eight struts with five pressure probes each gathers data for most representative flow conditions. The location of the 40 fixed ARP1420 probes resulting from this configuration is shown in Fig. 2(b). The two remaining sets of rotary rakes collect detailed flow-field data with a 5° angular resolution at select conditions of interest. These rotary rakes have six struts total, but only three of them are fitted with instrumentation. For total-pressure distortion, each strut is fitted with nine probes each at ARP1420 locations and an additional four intermediate radial stations. Finally, flow angularity distortion is measured using three

five-hole probes installed on each of the three instrumented rakes. Only the eight-strut variation of the TCT nacelle was modeled throughout this work, since variations in the flow-field upstream of the measurement station caused by a difference in the number of struts were assumed to be minimal. The eight-strut geometry is shown in grey in Fig. 2(b).

IV. Computational Methodology

In this section, the computational framework utilized for the numerical simulations is described. The discretization schemes utilized in the curvilinear solver of the Launch, Ascent, and Vehicle Aerodynamics (LAVA) solver framework are first presented, followed by a description of the grid topologies utilized to mesh the geometry.

A. Launch, Ascent, and Vehicle Aerodynamics (LAVA) CFD Framework

The CFD solver used in the present work is contained within the LAVA solver framework (Kiris et al., 2016). This flexible computational framework was developed with highly complex geometries in mind. As such, it supports a variety of mesh paradigms, such as the structured Cartesian, curvilinear overset and unstructured arbitrary polyhedral. The structured overset approach is used in the present work.

A finite difference formulation is applied to the compressible curvilinear Reynolds-averaged Navier-Stokes (RANS) equations in strong conservation law form. For the studies discussed here, the RANS system of equations is closed using the negative variant of the Spalart-Allmaras (SA) turbulence model (Spalart and Allmaras, 1992). Several modifications to this baseline SA model are also investigated, all using the negative variant: the SA model with Rotation/Curvature (RC) corrections (Shur et al., 2000) (SA-RC), the SA model with the nonlinear Quadratic Constitutive Relation (QCR) approximation, version 2000 (Spalart, 2000), applied to the turbulent stresses (SA-QCR2000), and the SA model with both corrections applied (SA-RC-QCR2000). Results using the two-equation $k - \omega$ baseline ($k - \omega$ BSL) turbulence model (Menter, 1992) are also obtained.

For the convective flux, a second-order discretization consisting of a modified Roe scheme with third-order left/right state reconstruction (Housman et al., 2009a,b) and a Koren limiter (Koren, 1993) was used throughout this work. The discretized set of nonlinear equations is marched in pseudo-time to steady-state. At each nonlinear iteration, the solution of the linearized system of equations is approximated using preconditioned Generalized Minimal Residual (GMRES) (Saad and Schultz, 1986). In addition, the implicit nature of the scheme allows for a CFL ramping procedure to be applied during runtime to further accelerate convergence to steady-state. Finally, both Message Passing Interface (MPI) and OpenMP protocols are leveraged to achieve a highly scalable, parallel CFD code.

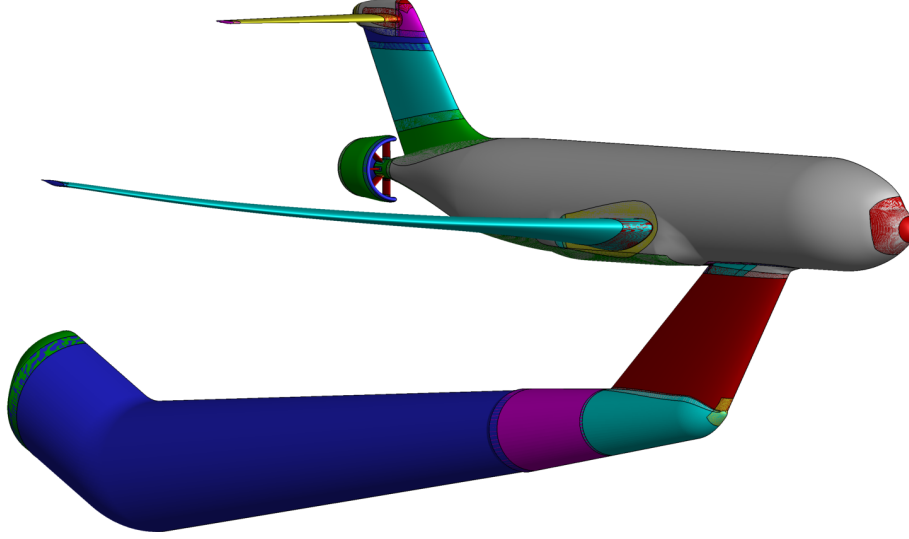
B. Grid Topology and Boundary Conditions

The gridding strategy employed in the present work follows best-practice guidelines from the Sixth Drag Prediction Workshop (DPW-VI) (Tinoco et al., 2018). Grid topologies were designed with previous best-practices established for the LAVA solver in mind, and build upon the knowledge acquired during the pre-test phase of this work. A summary of the simulations and sensitivity studies performed before the NTF test took place is available in Fernandes et al. (2022), including a validation study with the DPW-VI test data using the original CRM geometry.

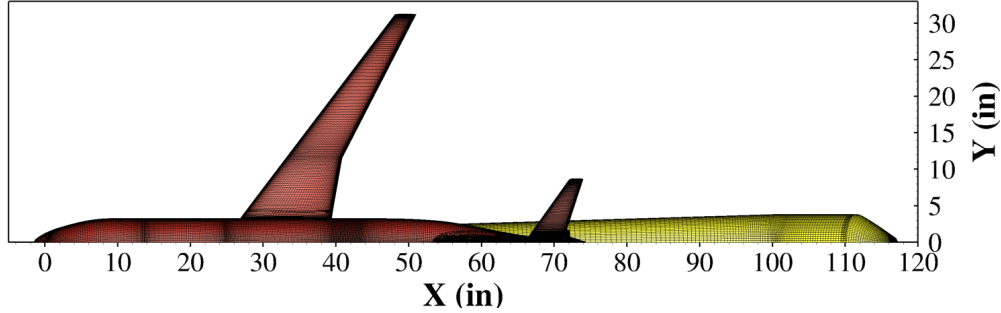
The overset grid paradigm is utilized in the present work. The surface of the CRM-TCT model is discretized with a set of overlapping surface-grid patches, which are then extruded hyperbolically to form the near-body volume grid. A view of the computational-model surfaces is shown in Fig. 3(a), where distinct colors were applied to different grid patches to emphasize the overlapping grid regions. A set of axis-aligned Cartesian boxes with grid stretching applied in all three coordinate directions slowly coarsens the grid away from the near-body regions. Figure 4 presents several slices of the grid system, highlighting the transition from the near-body grids extruded from the surface to the Cartesian boxes that coarsen onto the far-field. Given the symmetry of the model about the $Y = 0$ meters plane, formed by the roll and yaw axes, the grid systems generated only covered half of the physical domain, and symmetry boundary conditions were applied at the symmetry boundary unless otherwise noted (Fig. 3(b)).

V. Preliminary CFD Sensitivity Studies for Determination of Best-Practices

Prior to simulating the entire NTF database using LAVA, a series of preliminary sensitivity studies were performed to establish best-practices. These included a grid convergence study, checking for turbulence modeling sensitivities, comparing full- and half-span model solutions, and assessing solution sensitivity to geometry simplifications that were made in the sting-fuselage intersection area. In this section, two of these studies are described, namely the grid and



(a) Representation of the patches used to discretize the surface of the *flow-through TCT* configuration.



(b) Top-view of the near-body computational domain surfaces with the *Coarse* grid level overlaid.

Fig. 3 Close-up of the computational domain wall surfaces.

turbulence modeling sensitivity studies.

A. Grid Refinement Study

To assess the spatial discretization error, and ensure that grid convergence follows the expected 2nd order trend for the numerical scheme utilized in the present work, a grid family consisting of four grid levels was generated. These grid levels are labeled *Coarse*, *Medium*, *Fine* and *Extra Fine*, and the refinement ratio between successive grid levels is roughly 1.35 in each curvilinear direction. Total grid point counts and other grid-related metrics are shown in Table 3 for each refinement level. This grid refinement study was conducted for the two main configurations, namely the *clean* and *flow-through TCT* configurations, both incorporating the wind tunnel mounting hardware. The baseline cruise condition was chosen to conduct the grid convergence study, at a 10-million Reynolds number, Mach number of 0.80 and 2° angle-of-attack. Moreover, Fig. 5 compares slices of the *Coarse* and *Extra-Fine* refinement level grids.

Results for the load coefficients of the two configurations simulated are plotted in Figs. 6 and 7. The aerodynamic load coefficients are plotted against $N^{-2/3}$, where N is the number of grid points for each grid level, which reflects both the expected spatial 2nd order of accuracy of the numerical scheme used, and the three-dimensional problem size. In the asymptotic regime, the loads should follow a linear variation with this grid factor, which is an idealized scenario since flow features such as shocks and grid singularities can locally change the order of accuracy of the scheme. This is indeed the case for the drag coefficient, which shows an almost perfect linear trend with grid refinement for both configurations. The results show a difference of around 5 drag counts between the *Medium* and *Extra Fine* grid levels. Finally, the three coefficients also display monotonic convergence as the grid is refined.

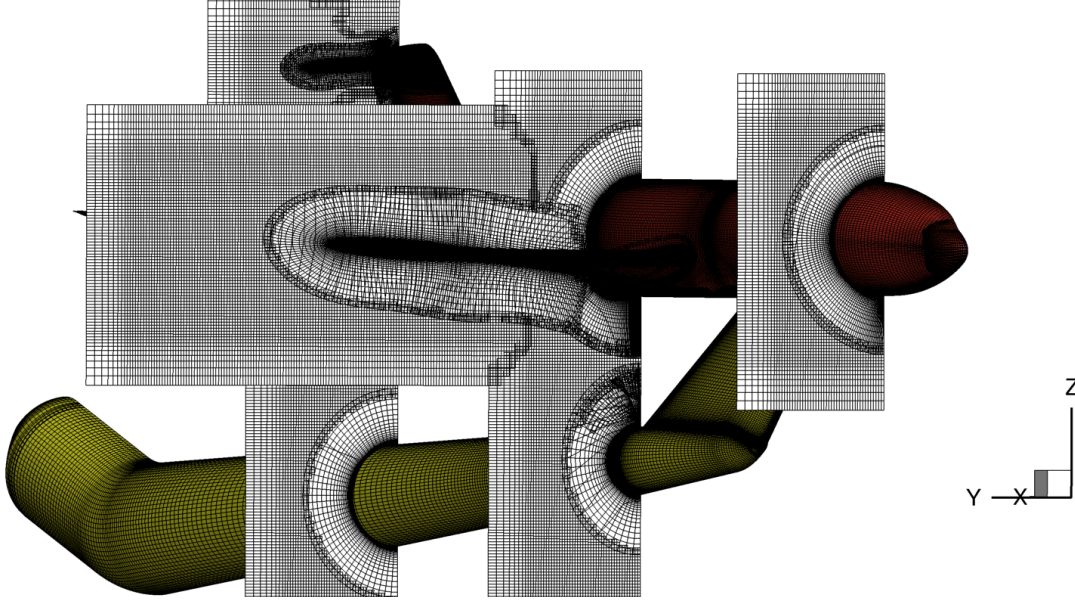
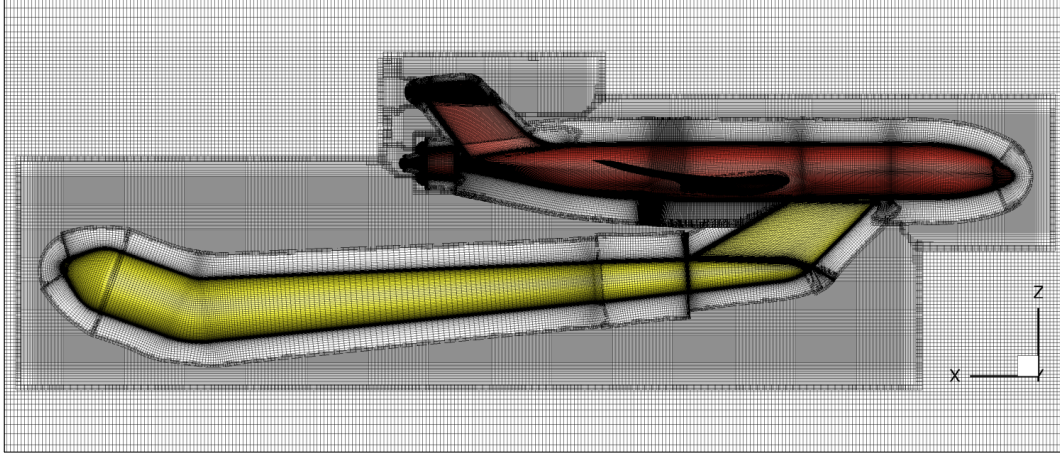


Fig. 4 Slices of the near-body volume grids on the *Coarse* grid level.

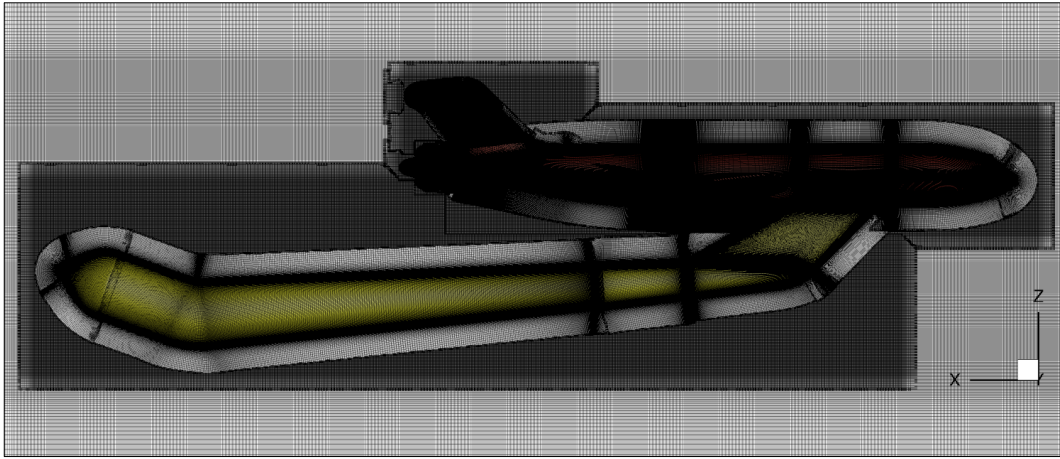
Table 3 Grid-related metrics for each refinement level studied. Values of average y^+ are shown for the cruise condition ($Re = 10 \times 10^6$, $Ma = 0.80$, $\alpha = 2^\circ$).

Configuration	Grid Metric	Coarse	Medium	Fine	Extra Fine
<i>Clean</i> (w/ <i>sting</i>)	# Grid Points [$\times 10^6$]	38.5	91.6	216.9	538.9
	# Solve Points [$\times 10^6$]	29.4	69.6	165.0	405.6
	Average y^+	0.726	0.537	0.400	0.297
<i>Flow-through TCT</i> (w/ <i>sting</i>)	# Grid Points [$\times 10^6$]	49.6	116.9	276.4	685.3
	# Solve Points [$\times 10^6$]	37.4	88.0	208.4	512.8
	Average y^+	0.727	0.538	0.400	0.296
Refinement Ratio		1.0	1.350	1.823	2.460
Wall Spacing [$\times 10^{-7}$ m]		4.93	3.65	2.71	2.00
Volume Stretching Ratio		1.18	1.13	1.09	1.07

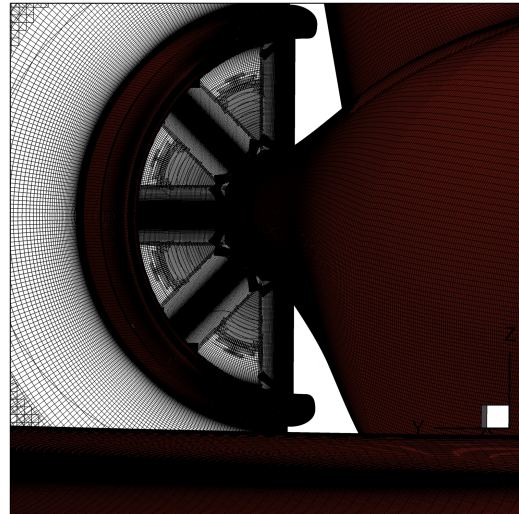
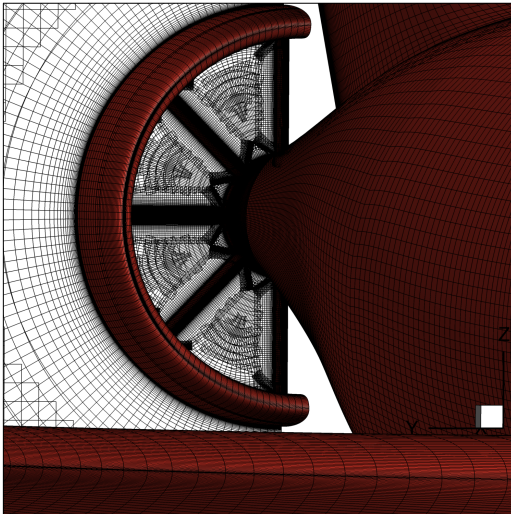
Figures 8 and 9 present the alpha-sweep performed for the positive angles-of-attack on the four grid levels for the *clean* and *flow-through TCT* configurations. Included in each plot is also the result of running the *Medium* grid level without the wind tunnel supporting structure (labeled *w/o sting*). This result was included to understand the effects of running the model in free-air, as opposed to including the wind tunnel mounting hardware. The results show that in all but the 4° angle-of-attack case, the lift and pitching moment coefficients do not change significantly with grid refinement. The same does not apply for the drag coefficient, which shows clear sensitivities to grid refinement level, especially between the *Coarse* and *Medium* grid levels. The *sting* supporting element has a visible effect on the load curves, especially the drag coefficient, even though the *sting* surfaces are not taken into account in the load calculations. This effect appears to decrease as the angle-of-attack increases. Overall, the trends captured follow the experimental data when it comes to the slope of the curves with angle-of-attack. However, the numerical results show an angle-of-attack shift relative to the wind tunnel data, even when comparing to corrected wind tunnel data. This is a common occurrence



(a) Side view of the near-body volume grids for the *Coarse* grid level.



(b) Side view of the near-body volume grids for the *Extra-Fine* grid level.



(c) Slice through the TCT grids for the *Coarse* grid level. (d) Slice through the TCT grids for the *Extra-Fine* grid level.

Fig. 5 Comparison between the *Coarse* and *Extra-Fine* grid levels.

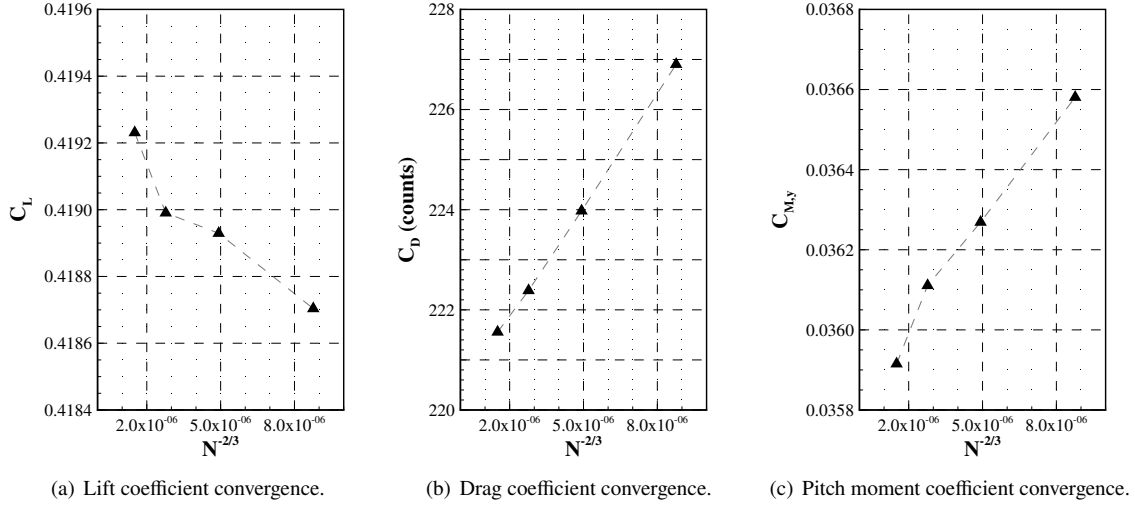


Fig. 6 Convergence of the load coefficients with grid refinement for the *clean* configuration.

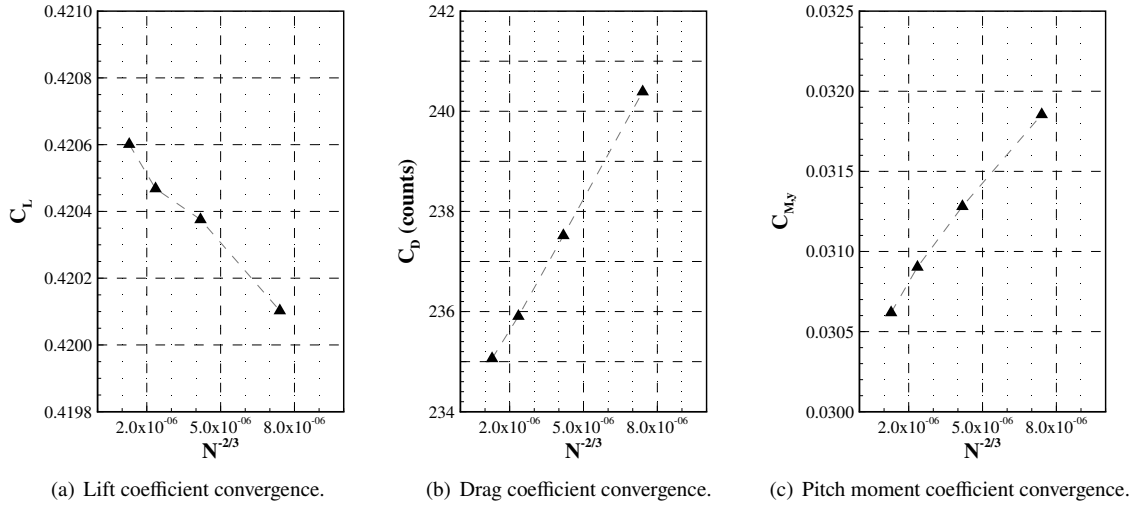


Fig. 7 Convergence of the load coefficients with grid refinement for the *flow-through TCT* configuration.

when comparing wind tunnel data to free-air simulations (Tinoco, 2023) due to the difficulty in correcting for wind tunnel wall and blockage effects, which are not modeled. One way to circumvent this limitation is to plot the loads against each other, which removes the angle-of-attack from the interpretation of the results. Therefore, load plots shown hereafter will not use angle-of-attack as an independent variable.

When the data is plotted in this manner, as shown in Figs. 10 and 11, the agreement with the corrected wind tunnel data is significantly improved. The drag polar curves now collapse on top of the experimental data for both configurations. The pitching moment is under-predicted in both configurations, and a slight difference in the slope of the C_L vs $C_{M,y}$ curve is also observed between the grid refinement results and the experimental data. It should be noted that these two discrepancies in the pitching moment prediction do not improve with grid refinement.

Since integrated aerodynamic loads can only provide a global view of grid convergence, sensitivity of local quantities of interest to grid refinement was also assessed. One such quantity for which wind tunnel data is available is the boundary-layer total-pressure profile data acquired at the boundary-layer rakes shown in Fig. 2(a). This data was collected only for the *clean* configuration, as there was a concern that the boundary-layer probes could break and

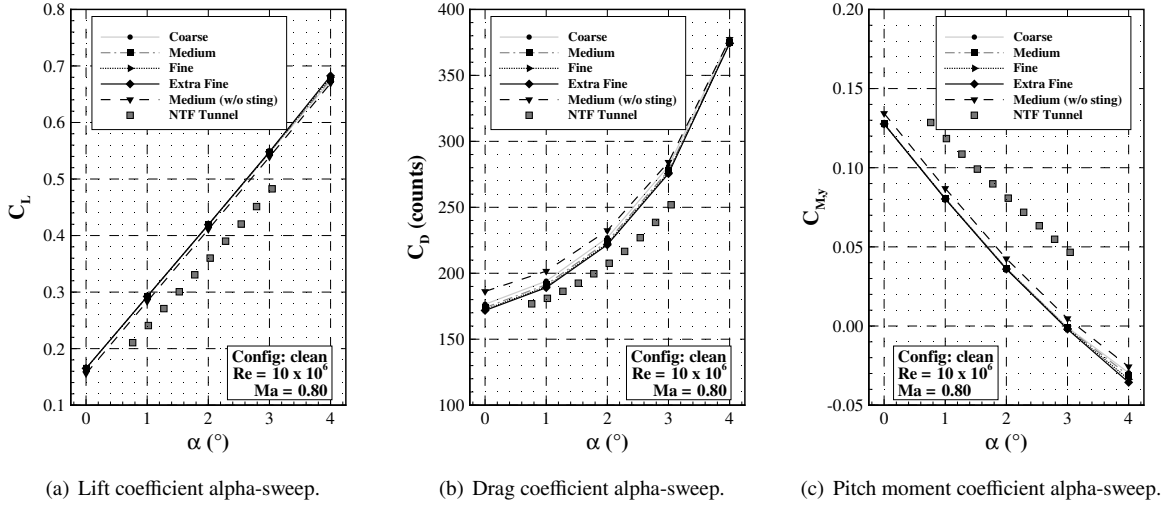


Fig. 8 Load coefficient alpha-sweep with different grid refinement levels for the *clean* configuration.

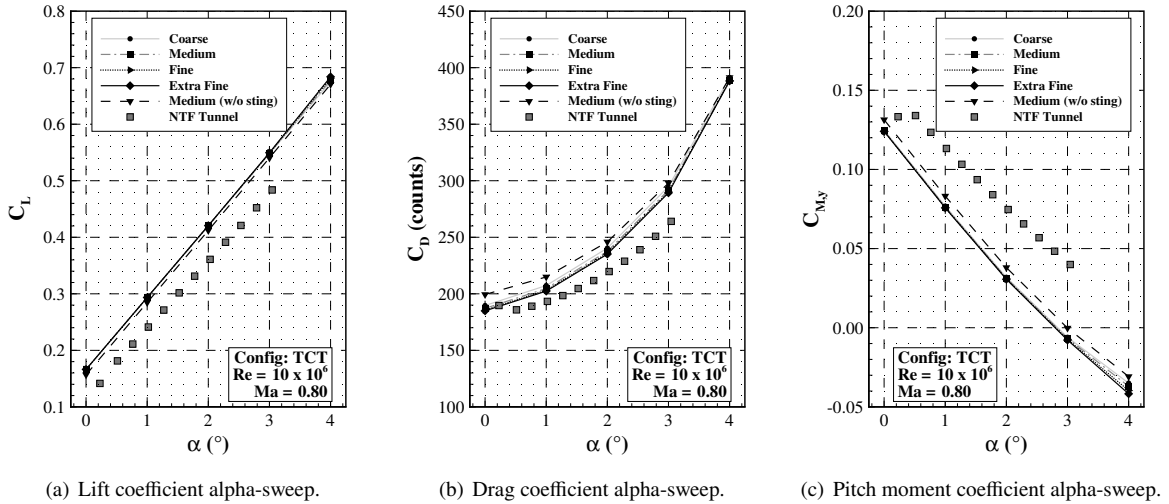


Fig. 9 Load coefficient alpha-sweep with different grid refinement levels for the *flow-through TCT* configuration.

damage the aftbody TCT located downstream. Furthermore, the flow condition for which the complete (*forward* and *aft*) boundary-layer rake data is available ($Re = 5 \times 10^6$, $Ma = 0.80$) differs from the condition reported in the grid refinement study thus far ($Re = 10 \times 10^6$, $Ma = 0.80$). For the purpose of these comparisons, therefore, the 5-million Reynolds number condition was also simulated for all grid levels at an angle-of-attack of 2° only. A subset of these boundary-layer profiles is plotted in Fig. 12. Sensitivity to the grid refinement level is mostly contained to the outer part of the boundary-layer, where the total-pressure recovers to slightly different values at the *forward* 135° and 180° locations. These values converge towards the wind tunnel total-pressure outside of the boundary-layer as the grid is refined. Furthermore, the circumferential locations that are not subject to the wake or influence of upstream elements such as the *sting* or the wing (such as the *forward* 0° , 45° , and 90° rakes, and the *aft* 60° rake) show excellent agreement with the experimental measurements. An interesting profile is picked up by the rake located at the *forward* 135° location. Here, the *S-shape* is well captured along the outer edge of the rake. However, below 0.6 inches away from the wall there is an inflection of the measured boundary-layer profile that is not captured, regardless of grid refinement level. This profile will be further discussed in Sec. V.B. Finally, the computed boundary-layer profile at the *forward* 180°

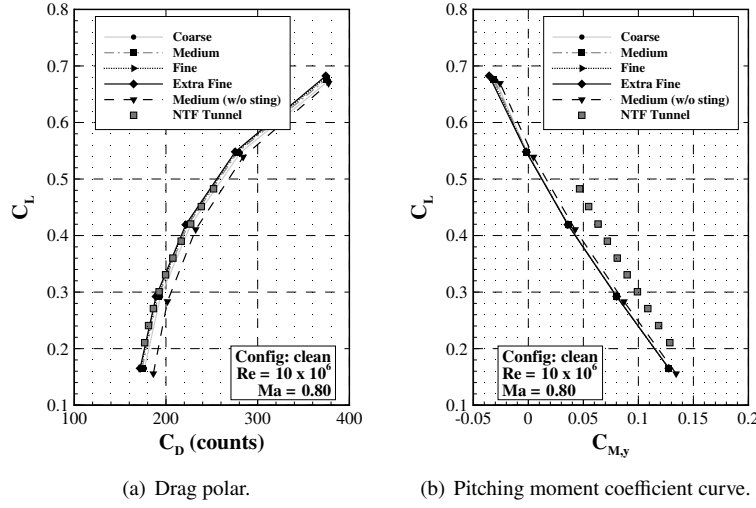


Fig. 10 Load coefficients for the *clean* configuration plotted against lift coefficient.

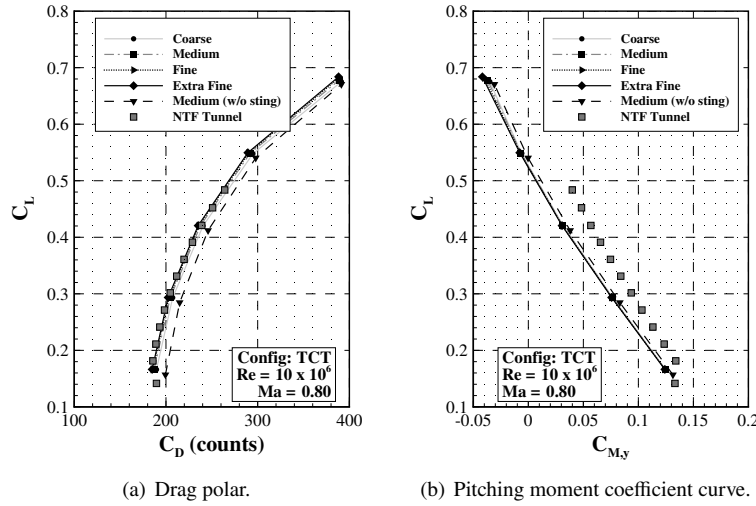


Fig. 11 Load coefficients for the *flow-through TCT* configuration plotted against lift coefficient.

location shows a more pronounced gradient relative to the wind tunnel data. This discrepancy does not improve with grid refinement.

A final metric used in the evaluation of grid convergence is the quality of the predicted flow-field at the nacelle highlight in the *flow-through TCT* configuration, as this is the main flow region of interest to this study. Grid sensitivities in this region are visualized in Fig. 14, which shows the pressure recovery (ratio of local to freestream total-pressure) profiles along the six rings defined by the forty ARP1420 probes illustrated in Fig. 2(b). Overall, the profiles predicted by the simulations show very good agreement with the experimental data. The solutions converge as the grid is refined, and little grid sensitivity is observed in most flow regions. One exception to this is the small region on the outermost ring (Ring 6) around $\theta = 0^\circ$, downstream of the vertical tail, where visible differences are observed between the *Coarse*, *Medium* and *Fine* grid levels. Past this grid level, the differences are minimal. This particular region coincides with the symmetry plane at $Y = 0$ meters, where a first-order symmetry boundary condition is applied. A full-span model of the *flow-through TCT* configuration was run and compared with the half-span model, showing that the experimental pressure recovery drop at $\theta = 0^\circ$ can be perfectly recovered using the *Medium* grid level when the full-span model is

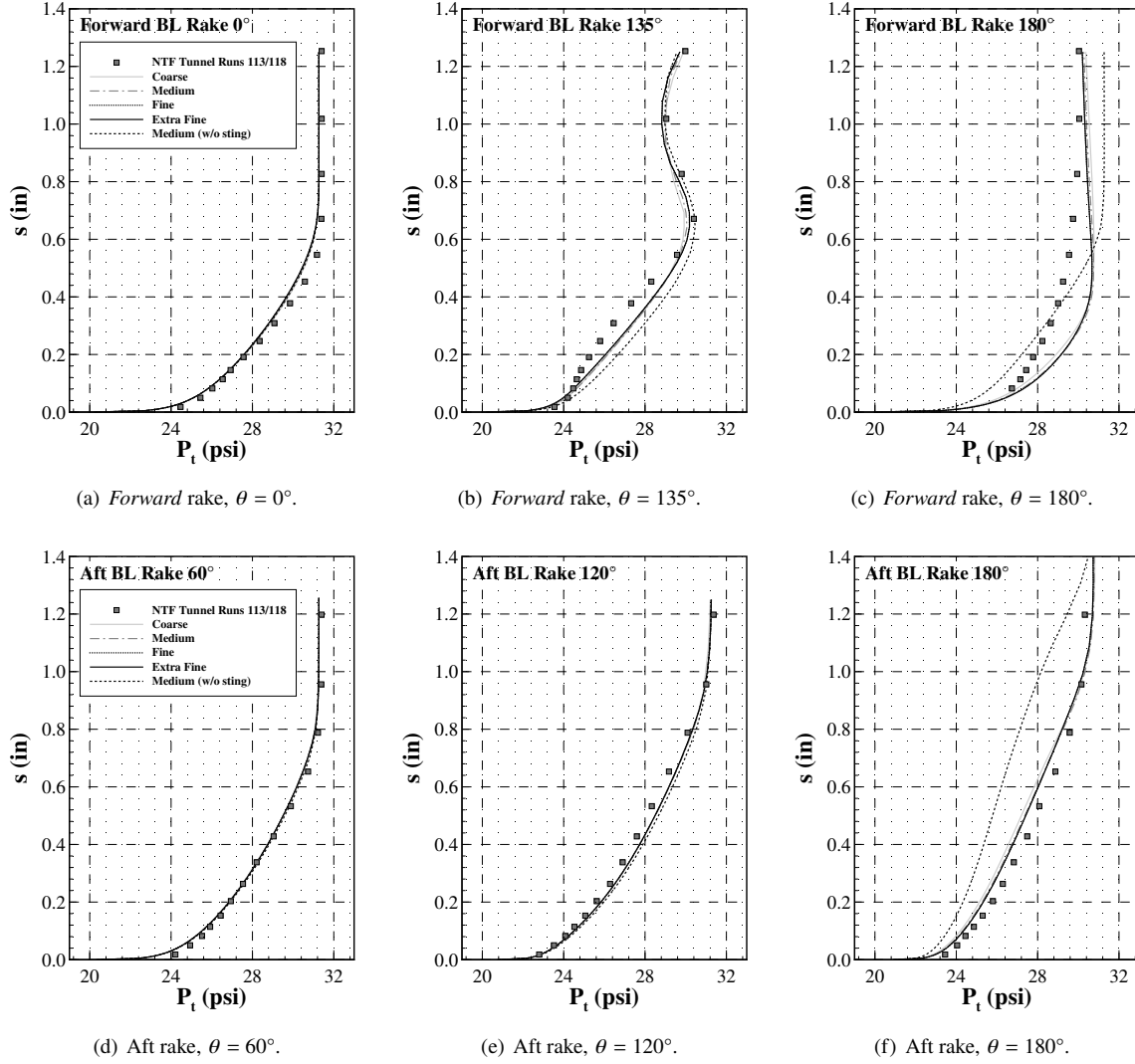


Fig. 12 Grid sensitivity of boundary-layer rake total-pressure (P_t) profiles for the *clean* configuration at the $Re = 5 \times 10^6$, $Ma = 0.80$, $\alpha = 2^\circ$ condition.

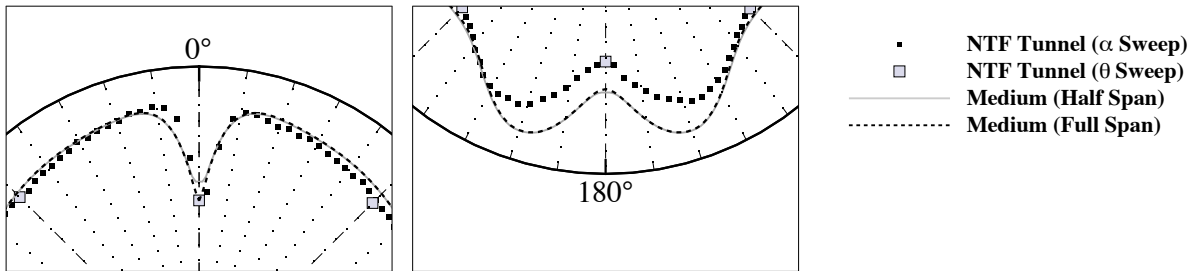


Fig. 13 Comparison between the half- and full-span models of the *flow-through TCT* configuration running on the *Medium* grid level at the symmetry plane.

utilized. This comparison is shown in Fig. 13. At the bottom 180° position the symmetry boundary condition also appears to have a minimal effect, limited to a region around 2° to 3° away from 180° . Elsewhere, the two curves collapse

to the same value. Given the small extent over which the half- and full-span solutions differed, the computational savings of using the half-span model for simulation of the final database is preferred. Another region where small grid sensitivity is visible is the region spanning the rake angles between 150° to 210° , where the fuselage upsweep vortices and the wake generated by the sting-fuselage intersection together create a tooth-like feature in the profile. Past the *Medium* grid level the differences in the solution are hardly visible. Another important observation regarding these profiles is the level of asymmetry displayed by the experimental profiles (which can be assessed by noting that the simulation profiles are symmetric with respect to the 0° to 180° plane). Although not substantial, it provides a data point when it comes to the level of uncertainty in the measurements.

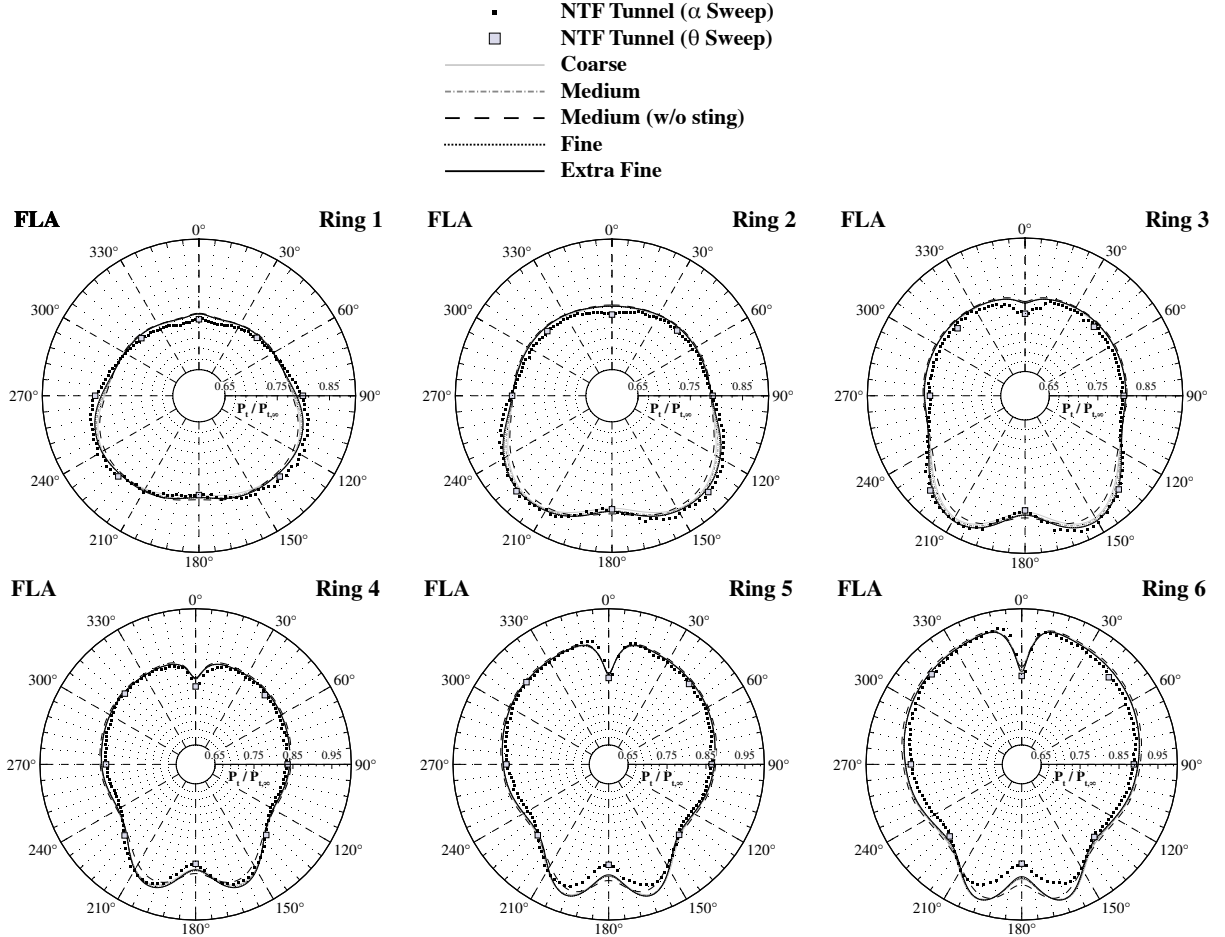


Fig. 14 Pressure recovery profiles extracted from the ARP1420 probe locations at the nacelle highlight for all grid levels simulated. Results are shown for the *flow-through TCT* configuration at the $Re = 10 \times 10^6$, $Ma = 0.80$, $\alpha = 2^\circ$ condition. Two sets of experimental data taken with the eight-strut fixed rake are shown. The symbol $\square$ corresponds to data taken during an angle-of-attack sweep, while the symbol $\bullet$ displays data gathered as the nacelle's roll angle θ is swept between 0° and 45° in 2.5° increments at a fixed angle-of-attack of 2° , covering the entire nacelle highlight area. Results without the sting for the *Medium* grid level also shown.

The discrepancies between the numerical and experimental results observed thus far in regions under the fuselage, namely the boundary-layer profile at the *forward* 180° rake (Fig. 12(c)) and the pressure recovery profiles in rings 5 and 6 at the ARP1420 locations in the nacelle highlight (Fig. 14), suggest that something is missing in the computational modeling of this region. The initial assumption was it had something to do with the sting-fuselage attachment, which happens upstream relative to where the discrepancies are being observed, and could thus impact the solution further downstream. This possibility was extensively investigated during pre-database sensitivity studies. One of the possibilities investigated was a more accurate representation of a small slot around the sting-fuselage intersection region which is

filled in the experiment by a somewhat permeable fabric that allows some amount of incoming air to flow into the model (Chan et al., 2024). Although this gap was never modeled as a permeable boundary, the geometry of the gap was incorporated in a variation of the baseline *flow-through TCT* configuration to assess the influence of this geometric feature in the flow solution predicted downstream, since it had been simplified in the baseline geometry. However, the results showed that incorporation of this gap did not improve the flow solution downstream, indicating that the source for the discrepancies observed potentially lies somewhere else. An interesting observation that further discredits the sting-fuselage intersection as the main culprit for the inaccuracies observed under the fuselage comes from comparing the pressure recovery profiles of the *Medium* grid level solution with and without the sting (long-dash line in Fig. 14). When inspecting the region spanning the rake angles between 150° to 210° in the outermost rings 5 and 6, where the discrepancies between the numerical and experimental data are largest, the differences between the two solutions (with and without sting) are smaller than the difference between the numerical and experimental data. This indicates that the sting itself is not responsible for the relatively large differences observed in this region of the ARP1420 pressure recovery profiles.

This suggests that the inaccuracy is most likely linked to the modeling of the fuselage upsweep vortices, rather than geometric differences between the test and computational models in the sting-fuselage intersection. This possibility is further elevated by observing how varying the turbulence model has a stronger impact in this region compared to varying the geometry. Turbulence model sensitivities are discussed in the following section.

B. Turbulence Model Sensitivity Study

In an attempt to resolve the discrepancies observed in regions under the fuselage, a turbulence model sensitivity study was conducted. The negative variant of the Spalart-Allmaras (SA) turbulence model and modifications thereof (SA-RC, SA-QCR2000 and SA-RC-QCR2000), in addition to the two-equation $k - \omega$ BSL turbulence model, were tested to assess the potential improvements or shortcomings of each closure when compared to the experimental data. These sensitivity studies were performed at a Reynolds number of 5-million, Mach number of 0.80 and angle-of-attack of 2° on the *clean* configuration, for comparison with boundary-layer rake measurements at this freestream condition. For the *flow-through TCT* configuration, nacelle highlight flow measurements taken at a Reynolds number of 10-million, Mach number of 0.80 and angle-of-attack of 2° were utilized for these studies.

Figure 15 shows six representative boundary-layer profiles at the *forward* and *aft* axial stations for the five turbulence models tested. The rake placed at $\theta = 0^\circ$ (above the fuselage) shows very little sensitivity to the turbulence model utilized. The same is true of the rakes placed at $\theta = 45^\circ$ and $\theta = 90^\circ$ (not shown). However, the boundary-layer profile located at $\theta = 135^\circ$ shows a significant sensitivity to the turbulence model closure. In contrast to the SA-RC-QCR2000 model, the baseline SA model is able to capture the inflection of the boundary-layer profile between 0.1 inches to 0.5 inches away from the body. Observing that the SA-RC and SA-QCR2000 profiles lie in between the baseline SA and SA-RC-QCR2000 solutions further indicates that the inflection of the profile is promoted by both corrections, and is not solely caused by one of them. The baseline SA profile also shows a closer agreement with the experimental data further away from the body. In this region of the profile, however, the SA-RC model more closely matches the baseline SA solution (and therefore the experimental data), while the SA-QCR2000 variant follows the SA-RC-QCR2000 profile. The $k - \omega$ BSL turbulence model is also unable to capture the inflection in the profile. Less sensitivity to the turbulence model utilized is observed for the boundary-layer rake placed at $\theta = 180^\circ$. Here, the discrepancy with the experimental data is only somewhat lower for the baseline SA model (and lower still for $k - \omega$ BSL), but the steepness of the profile still shows a mismatch between 0.1 inches to 1.0 inches away from the body. In the *aft* axial station, small sensitivity is seen for the profiles at $\theta = 60^\circ$ and $\theta = 120^\circ$, with the only outlier being the $k - \omega$ BSL model. Under the fuselage, at $\theta = 180^\circ$, $k - \omega$ BSL appears to do a better job at capturing the curvature of the boundary-layer profile as the total-pressure recovers to freestream values, while the baseline SA also shows good agreement with the experimental data.

Figure 16 shows contours of the eddy viscosity ratio with overlaid streamlines at two different axial slices: one located at the *forward* boundary-layer rake probe station, and another located just upstream of the nacelle inlet. These are shown for the *flow-through TCT w/ Cruise MFP* configuration for three different turbulence model solutions — baseline SA, SA-RC-QCR2000 and $k - \omega$ BSL. Streamline differences in the underside of the fuselage indicate a stronger sensitivity to the turbulence model in this region, likely related to the development of the fuselage upsweep vortex. The SA turbulence model variants predict a larger vortex with a wider radius of influence when compared to the $k - \omega$ BSL model. Differences in the development of the upsweep vortices are also already visible at the upstream (top row) slice. The streamlines appear to *wrap* around the vortex more intensely in the SA-RC-QCR2000 solution compared to

the other two models, suggesting a delay in the vortex development in the latter turbulence model closures. It is also interesting to note that while at the upstream station the model showing the highest eddy viscosity ratio values is the $k - \omega$ BSL model, close to the nacelle it is the SA-RC-QCR2000 that shows the largest amount of turbulence. This is especially true under the fuselage.

While the boundary-layer profiles for the *clean* configuration indicate good experimental agreement using the baseline SA model, the same cannot be said for the total-pressure profiles at the nacelle highlight for the *flow-through* TCT configuration. Figure 17 shows the comparison for the five turbulence models tested. It is clear that the baseline SA model shows an evolution between Rings 1 and 3 that does not match that of the experimental data in the range 100° to 260° . In the top half of the hemisphere all turbulence models behave similarly, with the exception of Ring 2, where the baseline SA model underpredicts the total-pressure ratio relative to the remaining models and the experiment. Overall, the SA-RC-QCR2000 closure shows the best agreement with the experimental data. Rings 1 to 3 follow the experiment closely. Ring 4 shows a slight overprediction appearing near 180° , while Rings 5 and 6 extend this overprediction to the lobes situated around 160° and 180° . The exact value predicted at 180° is well-captured by both the baseline SA and the SA-RC models, but the solution degrades around the lobes. The SA-QCR solution indicates that the overprediction at this location is mainly influenced by the QCR correction, rather than the RC one. With respect to the $k - \omega$ BSL

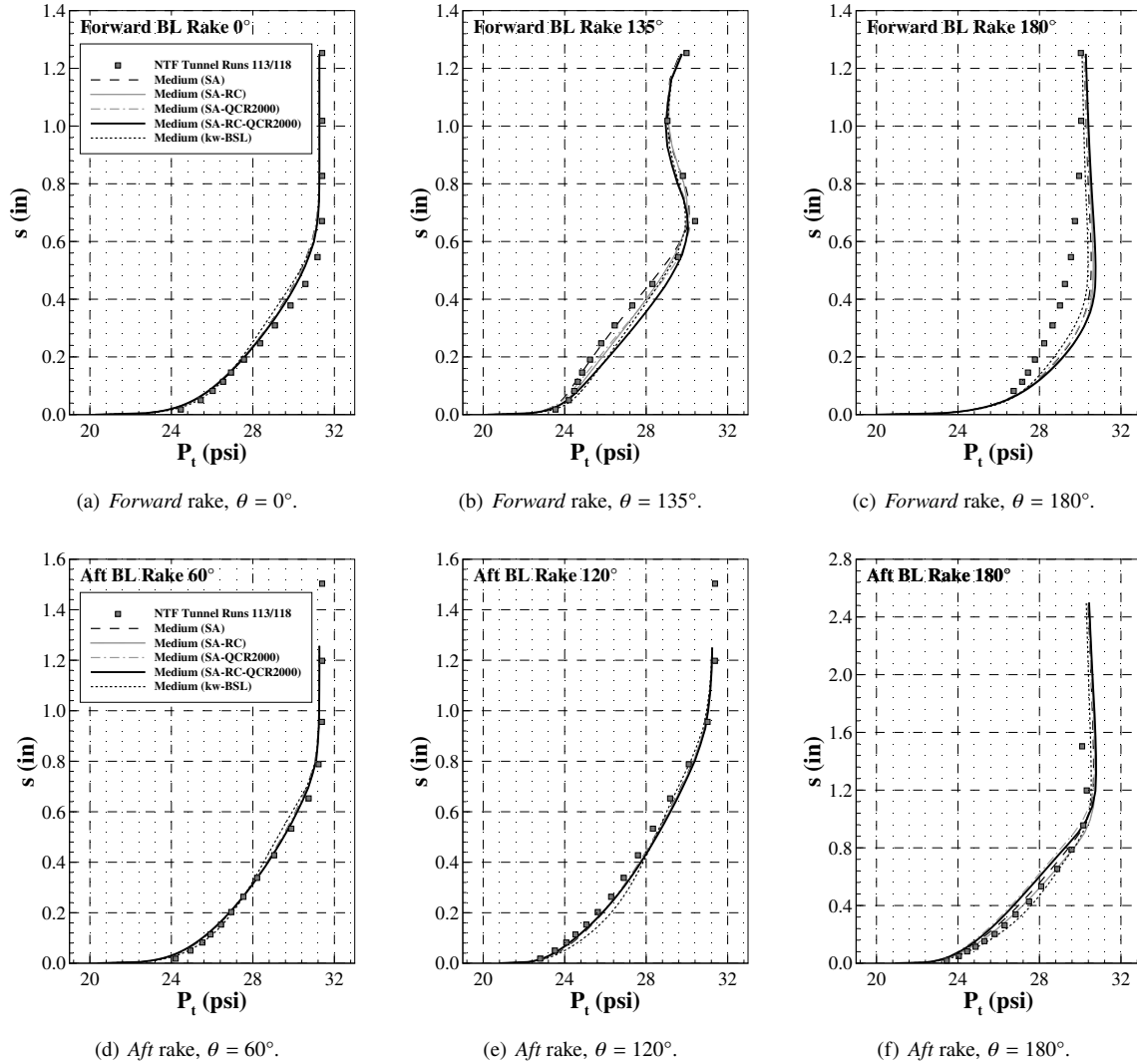


Fig. 15 Turbulence model sensitivity of boundary-layer rake total-pressure (P_t) profiles for the *clean* configuration at the $Re = 5 \times 10^6$, $Ma = 0.80$, $\alpha = 2^\circ$ condition.

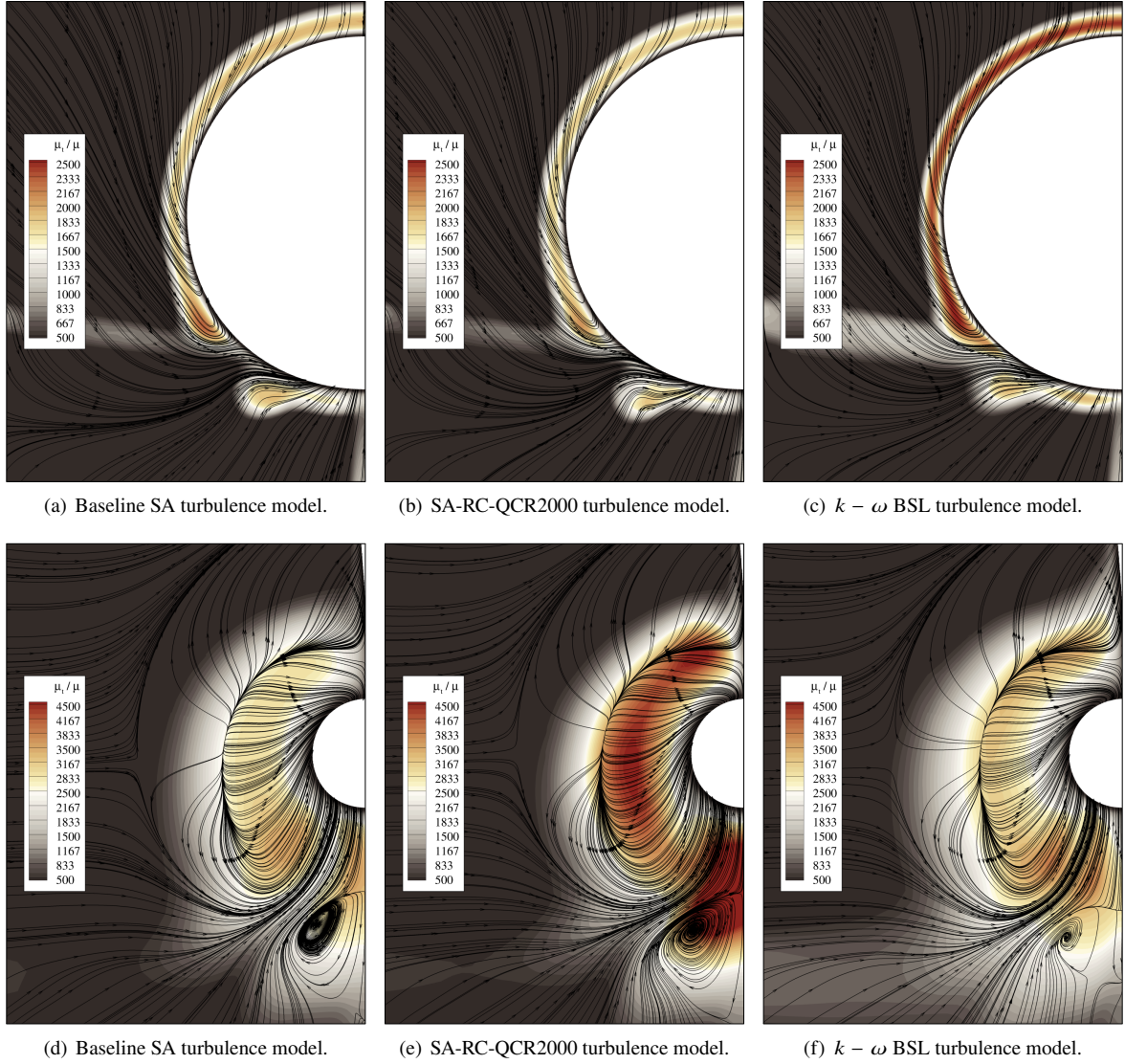


Fig. 16 Eddy viscosity ratio contours with overlaid streamlines at axial stations for three turbulence models. Results shown for the *flow-through TCT w/ Cruise MFP* configuration at the $Re = 10 \times 10^6$, $Ma = 0.80$, $\alpha = 2^\circ$ condition. Two axial stations are shown, one corresponding to the location of the *forward* boundary-layer rakes (top row) and another just upstream of the nacelle inlet (bottom row).

two-equation model results, Rings 1 to 3 show an overprediction of the total-pressure loss over the bottom of part of the fuselage. This extends to Rings 4 and 5, and in Ring 6 it matches well with the experimental profile.

It is important to note that the $k - \omega$ BSL results took significantly longer to converge to a steady-state solution when compared to any of the SA model variants tested. And for the SA model variants tested, the SA-RC-QCR2000 closure provided the most satisfactory results when compared to the experimental measurements. As a result, in big part due to its demonstrated accuracy in the sensitivity studies discussed here, the SA-RC-QCR2000 turbulence model closure is utilized in the remaining database solutions reported in the sections that follow.

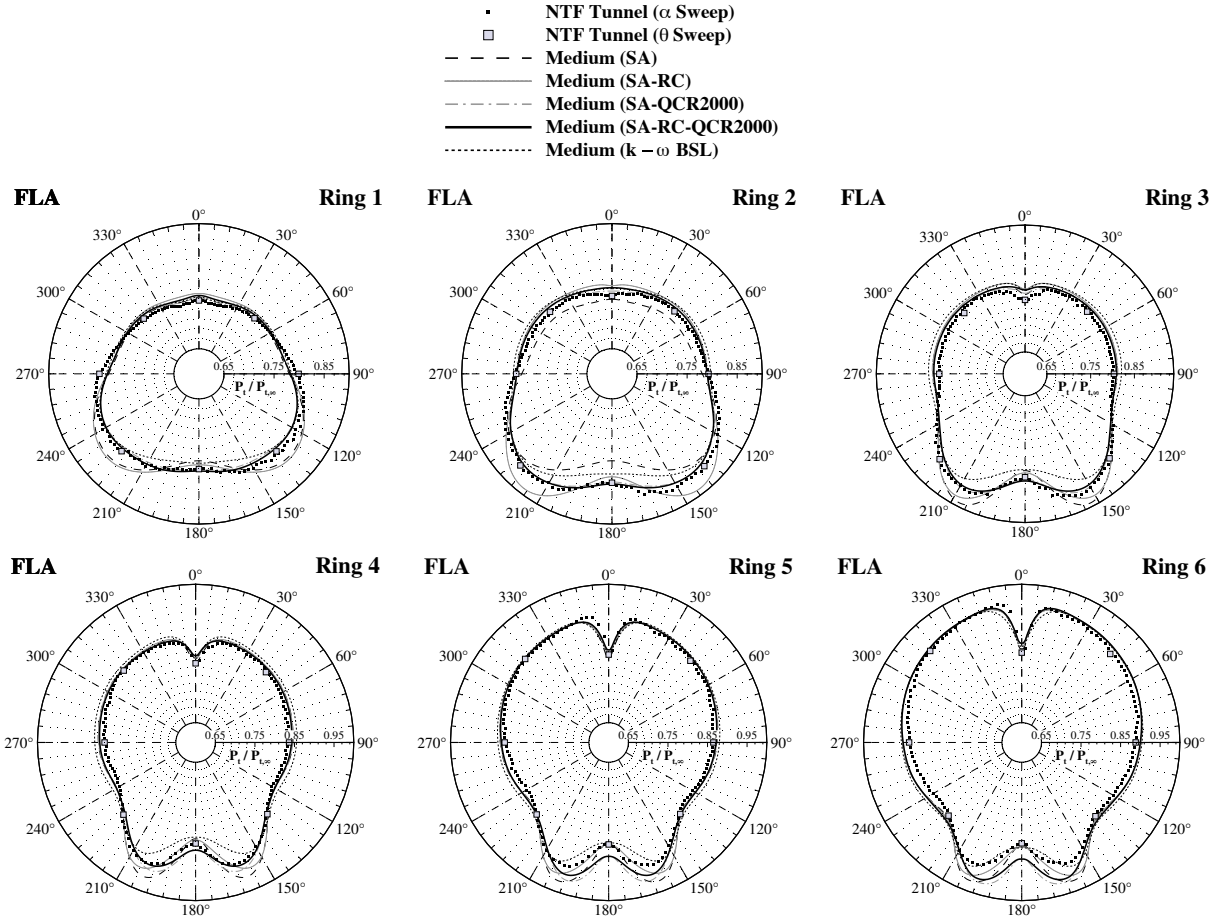


Fig. 17 Pressure recovery profiles at the ARP1420 probe locations for the turbulence models studied. Results shown for the *flow-through TCT w/ Cruise MFP* configuration at the $Re = 10 \times 10^6$, $Ma = 0.80$, $\alpha = 2^\circ$ condition.

VI. Discussion of Results from the NTF CRM-TCT Test Database

This section describes and compares the results obtained in the LAVA database across a wide variety of conditions and configurations. An effort was made to report several conditions rather than focus on a single one, so as to show the capability of the RANS model in capturing the wind tunnel flow in a wide range of flow conditions. The section is divided into five main categories corresponding to different types of data shows in the comparisons, namely, aerodynamic load coefficients, nacelle mass-flow rate, nacelle inlet flow distortion, aftbody boundary-layer rakes and aftbody static pressure taps.

A. Aerodynamic Load Coefficients

This section focuses on the aerodynamic coefficients obtained under several conditions run. The effect of adding the TCT to the clean aftbody is analyzed throughout the section by comparing *clean* and *flow-through TCT* configuration results. Unless otherwise noted, the *Cruise* mass-flow plug is utilized in the *flow-through TCT* configuration discussed. Loads obtained with different mass-flow plugs installed in the TCT nacelle are also analyzed. The load coefficient results are reported in terms of C_L vs. C_D and C_L vs. $C_{M,y}$, following the reasoning described in Sec. V.A.

Figure 18 shows the load coefficient results for the 5-million Reynolds number cases at different Mach numbers for the *clean* and *flow-through TCT* configurations. This was the only Reynolds number condition for which results at negative angles-of-attack were run in the wind tunnel with the horizontal stabilizer removed, and these are shown in blue on the plots. The drag polar results show excellent agreement between the simulations and the wind tunnel data. Not only are the absolute drag coefficient values well predicted, but also the 10 – 20 drag count difference between

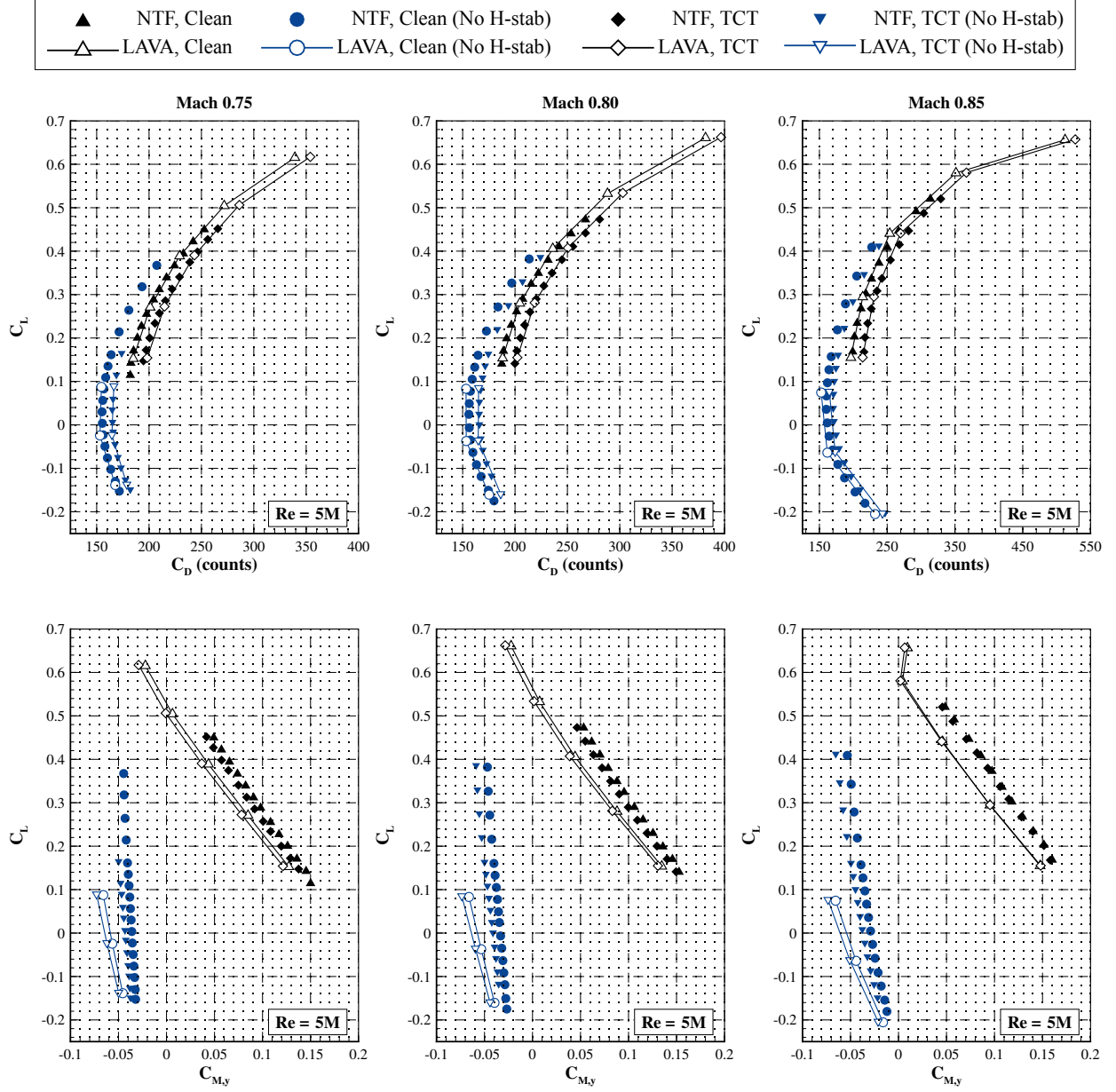


Fig. 18 Load coefficients at the 5-million Reynolds number condition under different Mach numbers for the *clean* and *flow-through TCT w/ Cruise MFP* configuration. Negative (blue color, horizontal tail uninstalled) and positive (black color, horizontal tail installed) angles-of-attack are shown. Filled symbols represent the NTF wind tunnel data, while open symbols correspond to the simulation results.

the *clean* and *flow-through TCT* configurations matches the delta observed in the test data. An increasing drag trend is also observed with increasing Mach number, especially at high-lift conditions. The pitching moment trends show reasonable agreement with the experimental data. A difference in the slope of the tunnel and numerical C_L vs. $C_{M,y}$ curves is observed at the negative angles-of-attack, where the horizontal stabilizer is uninstalled. The slopes tend to match better for the positive angles-of-attack with the horizontal tail installed. Again, the differences between the *clean* and *flow-through TCT* configurations are well predicted. The effect of adding the horizontal tail for the positive angles-of-attack (shown in black) is a distinct increase in the pitching moment coefficient, together with the change in the curve slope. Finally, this delta remains somewhat constant with increasing Mach number for the configurations without the horizontal stabilizer (shown in black), but is observed to shrink with increasing Mach number when the

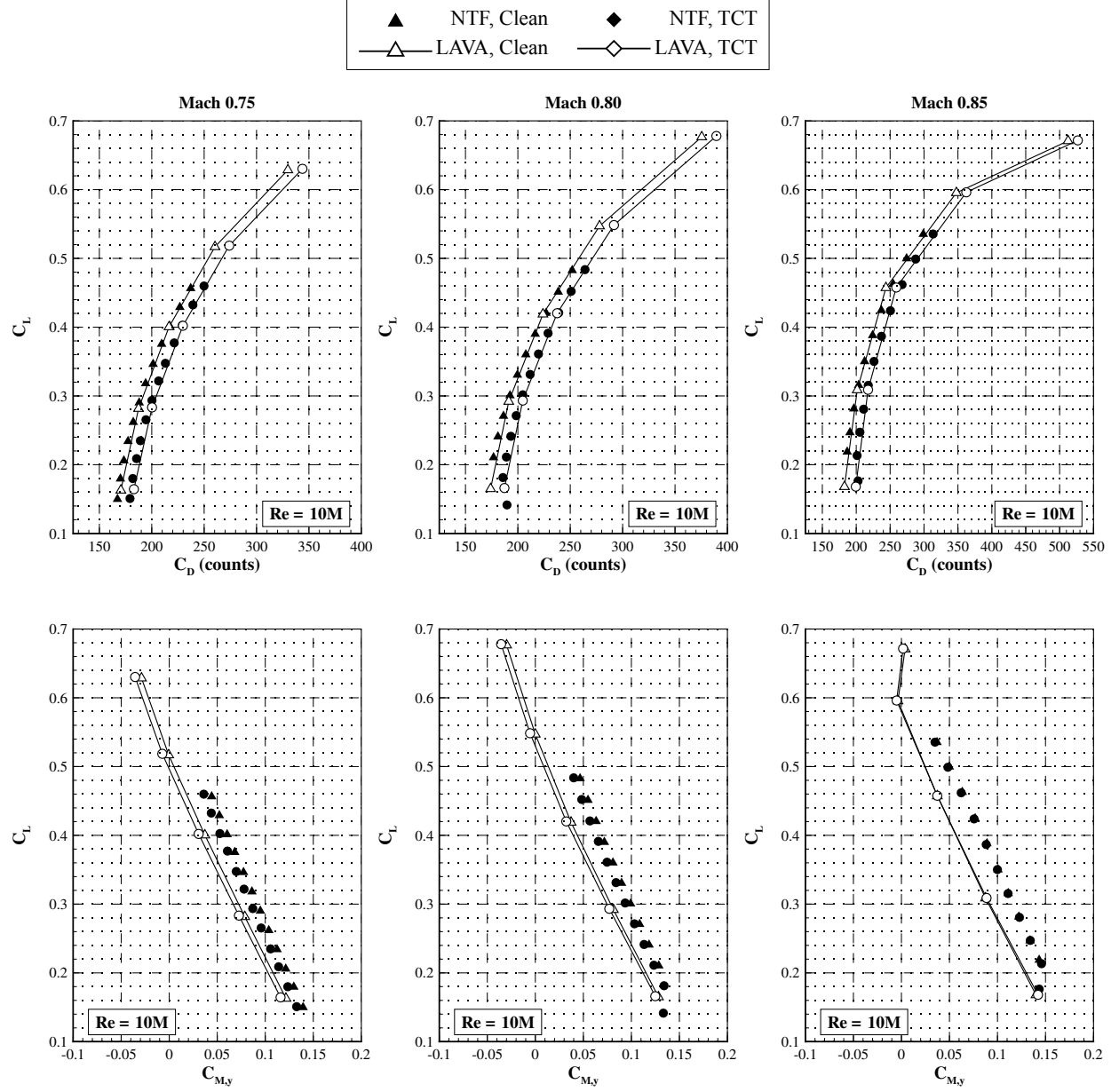


Fig. 19 Load coefficients at the 10-million Reynolds number condition under different Mach numbers for the *clean* and *flow-through TCT w/ Cruise MFP* configuration. Filled symbols represent the NTF wind tunnel data, while open symbols correspond to the simulation results.

horizontal tail is installed.

Similar considerations apply for the 10- and 15-million Reynolds number conditions shown in Figs. 19 and 20, for which only the positive angles-of-attack were run. Again, excellent agreement is observed in the predicted drag coefficient when compared with the NTF tunnel data, while the pitching moment is slightly underpredicted. The effect of increasing the Reynolds number is a slight reduction in both the drag and pitching moment coefficients at the equivalent Mach number condition. These effects are observed both in the wind tunnel data and in the numerical predictions.

Tailcone thruster operating condition effects are analyzed in Fig. 21. Only the conditions that were commonly run across the different mass-flow plug types are presented, i.e., the *flow-through TCT* configurations with different MFPs and the horizontal stabilizer installed at positive angles-of-attack. Results show only slight changes in both lift and pitching moment coefficient across the different MFPs. The trends indicate an increase in drag coefficient as

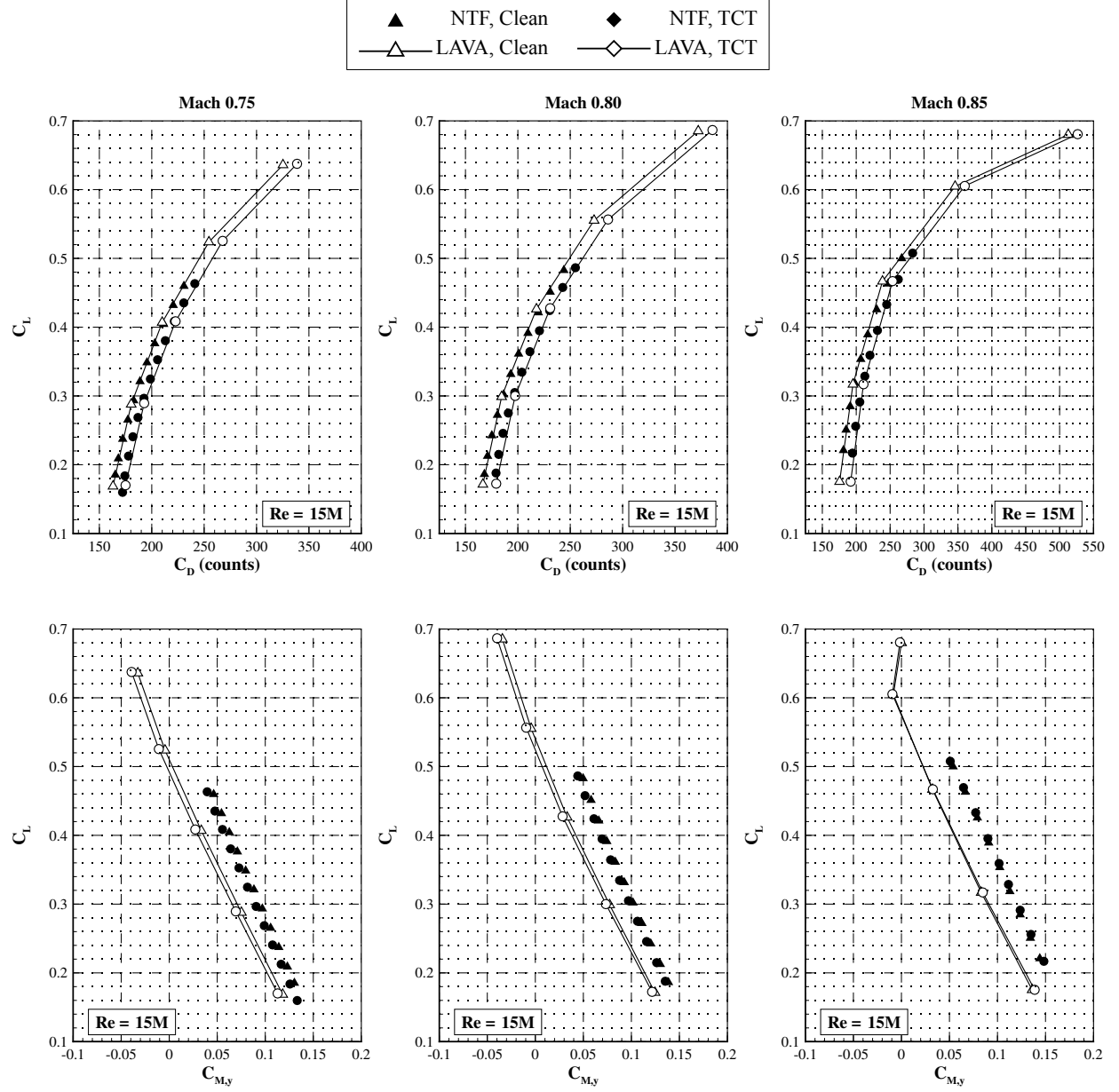


Fig. 20 Load coefficients at the 15-million Reynolds number condition under different Mach numbers for the *clean* and *flow-through TCT w/ Cruise MFP* configuration. Filled symbols represent the NTF wind tunnel data, while open symbols correspond to the simulation results.

the mass-flow rate across the nacelle increases from the *Idle* MFP to the *Cruise +10 %* MFP. This trend is inversed for the pitching moment coefficient, which lowers at higher mass-flow rate conditions. Despite the slight deviation in the predicted pitching moment coefficient when compared with the tunnel data, the deltas between different MFP configurations are well predicted in the RANS-modeled database.

B. Nacelle Mass-Flow Rate

The process for calculating the nacelle flow rate in the experiment — described in detail in [Chan et al. \(2024\)](#) — was replicated in the simulations, taking into account the total-pressures predicted at the ARP1420 measurement locations and the isentropic flow equations to calculate an average Mach number at the nacelle highlight. This process results in a

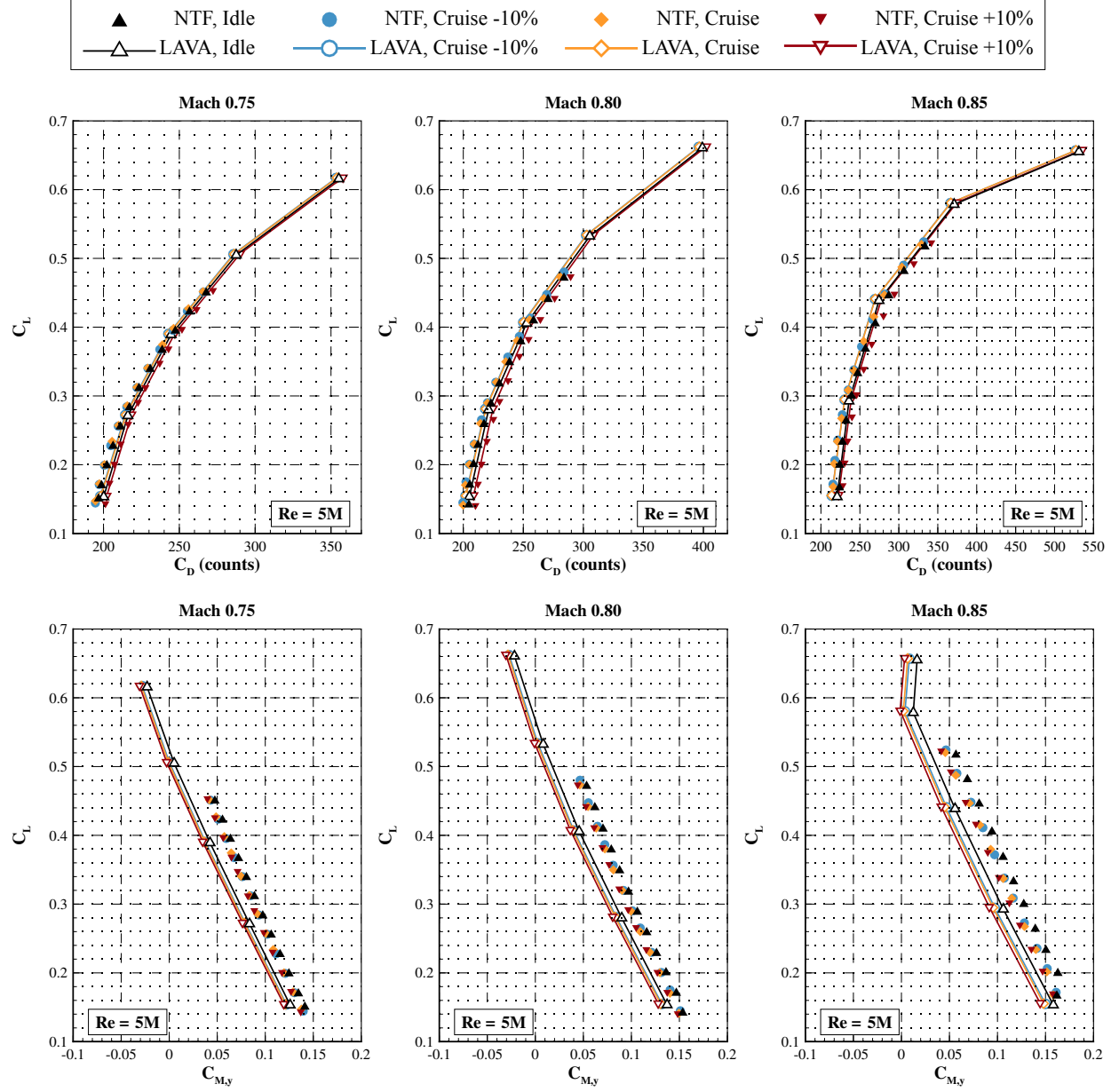


Fig. 21 Load coefficients at the 5-million Reynolds number condition under different Mach numbers, across the different mass-flow plugs installed on the *flow-through TCT* configuration. Filled symbols represent the NTF wind tunnel data, while open symbols correspond to the simulation results.

slightly different mass-flow rate compared to integrating the mass flux across the nacelle area, but allows for a more accurate comparison with the wind tunnel data.

Figure 22 analyzes the effect of increasing Mach number at fixed Reynolds number and MFP on the nacelle mass-flow rate. Overall, the mass-flow rate across the nacelle decreases as the Mach number increases. Additionally, the effect of increasing Reynolds number can be analyzed by comparing the same Mach number curve across the two figures. An increase of roughly 27 % in mass-flow rate is observed in going from 10- to 15-million Reynolds number conditions for any given Mach number. The simulations underpredict the experimental mass-flow rate by 1 % to 2 % at 10-million Reynolds number, and by less than 1 % for the 15-million Reynolds number cases. The deltas across the different Mach number alpha-sweeps also appear to match well between the simulations and the tunnel data.

The effect of having different MFPs installed on the aftbody assembly on the *flow-through TCT* configuration is

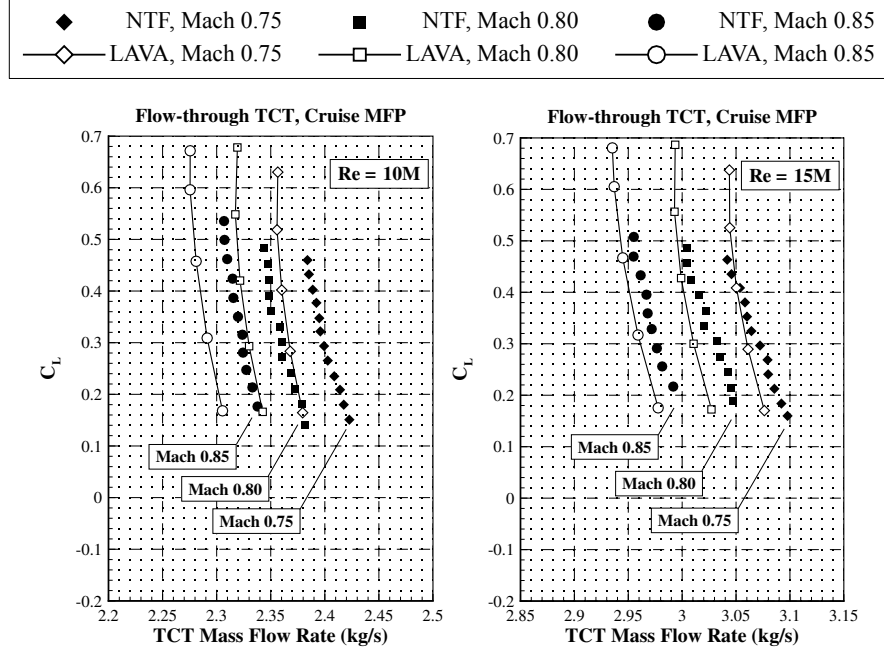


Fig. 22 Nacelle mass-flow rate at 10- and 15-million Reynolds numbers, on the *flow-through TCT w/ Cruise MFP* configuration. The effect of increasing Mach number is emphasized in each figure.

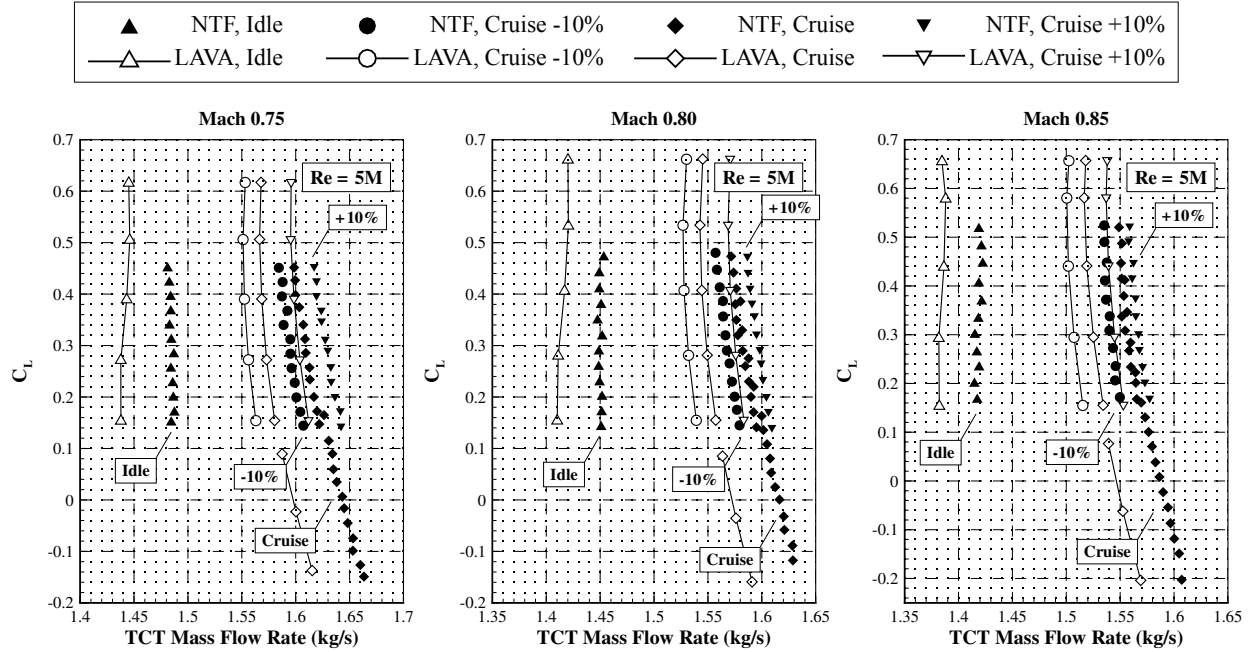


Fig. 23 Nacelle mass-flow rate at a Reynolds number of 5-million on the different *flow-through TCT* configurations. The effect of different MFPs installed on the TCT assembly is emphasized in each figure. A different Mach number is shown in each column (from left to right, the Mach number ranges from 0.75 to 0.85).

analyzed in Fig. 23. Specifically, each plot shows the mass-flow rates at a Reynolds number of 5-million for the four MFPs tested at a fixed Mach number condition. Note that only the *Cruise* MFP was run at negative angles-of-attack, which is why this is the only curve extending into the negative lift coefficient range of the plot. Again, the numerical

predictions slightly underpredict the wind tunnel data across all cases shown, but not by more than about 3.6 %. A slight improvement is observed at the higher Mach number condition, but not by a substantial amount. As designed, the mass-flow rate is minimum with the *Idle* MFP and maximum with the *Cruise +10 %* MFP. Importantly, the deltas in mass-flow rate across the different mass-flow plugs match remarkably well between the wind tunnel data and the RANS-database results. Finally, note that the *Cruise -10 %* and *Cruise +10 %* MFPs do not show a delta of $\pm 10\%$ relative to the *Cruise* MFP since they were designed at a Reynolds number of 10-million. Due to cost and schedule constraints during the experiment, these MFPs were not run at that Reynolds number, and the $\pm 10\%$ figures could not be verified (Chan et al., 2024).

C. Nacelle Inlet Flow Distortion

One of the main objectives of this experimental study was to characterize the inlet flow distortion that the nacelle is subject to when ingesting the boundary-layer flow developing over the model fuselage. In this section, the inflow distortion prediction is compared with the experimental data using two metrics. First, the total-pressure profiles collected by the ARP1420 probes are compared with the wind tunnel data. Lastly, the flow angles estimated with the five-hole probes are plotted against the predicted flow angularity. This provides a complete assessment of the effectiveness of the RANS model in predicting the coupled pressure and velocity fields at the nacelle highlight.

1. Total-Pressure Profiles from ARP1420 Probes

The pressure recovery (PR), defined as the ratio of local total-pressure to the freestream total-pressure at each ARP1420 probe is analyzed in this section. The discussions are centered around the sensitivities in the nacelle inlet flow distortion caused by changes in angle-of-attack, Reynolds number, and nacelle mass-flow rate. Qualitative comparisons between the wind tunnel data and numerical predictions are first provided in the form of contour plots of the pressure recovery at the nacelle highlight, followed by quantitative comparisons of the pressure ratio curves at the six circumferential rings defined by the ARP1420 probes.

The contour plots presented for the experimental data are reconstructed from the ARP1420 probe data collected during nacelle roll angle sweeps. These sweeps collected data at fixed angle-of-attack, and rotated the nacelle over 45° at 2.5° increments, providing coverage of the full 360° inlet flow since there was a strut at every 45° . Figures 24 and 25 illustrate the effects of increasing angle-of-attack on the inlet pressure recovery at Reynolds number of 5-million and 10-million, respectively, at the highest Mach number condition and for the *Cruise MFP* configuration. Qualitatively, the comparison between the NTF tunnel data and the LAVA simulation results is very good. Among the main features that can be identified in each plot, the following ones stand out: low-momentum flow characterized by a low-PR close to the hub walls, especially in the top half of the model; the low-PR footprint left by the passage of the flow over the vertical tail surface at 0° ; two symmetric lobes of high-PR on the underside of the fuselage, caused by the entrainment of near-freestream flow into the nacelle capture area due to the combined effect of the nacelle upsweep vortices and downwash from the wing/body fairing. Out of all these features, one that appears to show differences between the wind tunnel data and the simulation results is a slight overprediction of the PR at the center of the two symmetric lobes on the underside of the fuselage. The remaining character of the distortion pattern at the nacelle highlight is well captured by the RANS model.

The effect of increasing angle-of-attack seen in these figures is two-fold. Firstly, there is a decrease in the near-freestream (high-PR) flow that is captured by the nacelle in the upper region of the inlet. This is accompanied by a wider low-PR region around the hub in the top half of the inlet area. Conversely, the extent of the two high-PR lobes located in the underside of the fuselage widens as the angle-of-attack increases, and the PR at the center of each lobe increases accordingly. These effects make intuitive sense with the direction of increasing angle-of-attack. As the angle-of-attack increases, more of the bottom half of the inlet is exposed to flow unaffected by the model. While the opposite is true on the top half, there is also a compounding effect related to the ingestion of a larger amount of boundary-layer (low-PR) flow by the inlet as the angle-of-attack increases. This is evident at both 5-million and 10-million Reynolds number conditions.

Similar qualitative comparisons are shown in Fig. 26, this time focusing on the effects of increasing Reynolds number while fixing the Mach number and angle-of-attack at 0.85 and 2° , respectively. The effects of increasing the Reynolds number from 5-million to 15-million are more subtle compared to the sensitivity to angle-of-attack. Nonetheless, there appears to be an increase in overall PR in the inlet with increasing Reynolds number, as suggested by the lighter colors of the high-Reynolds contours. The extent of the features described previous looks qualitatively similar. Once again, good agreement is seen between the wind tunnel data and the LAVA database results.

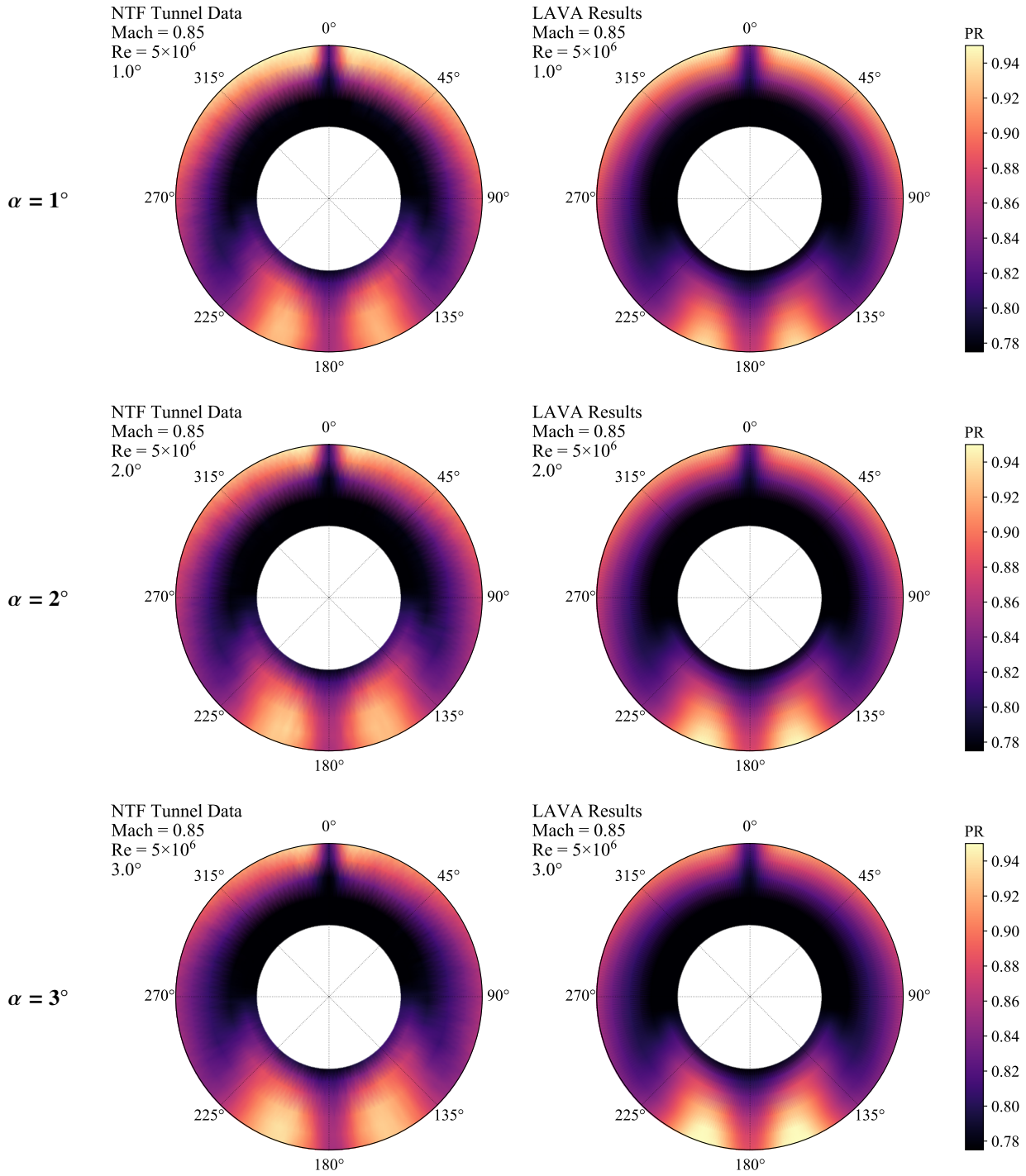


Fig. 24 Contour plots of the total-pressure recovery (PR) reconstructed from the ARP1420 data taken at different nacelle roll angles. The effect of increasing angle-of-attack is visualized for the *flow-through TCT w/ Cruise MFP* configuration running at the $Re = 5 \times 10^6$, $Ma = 0.85$ condition.

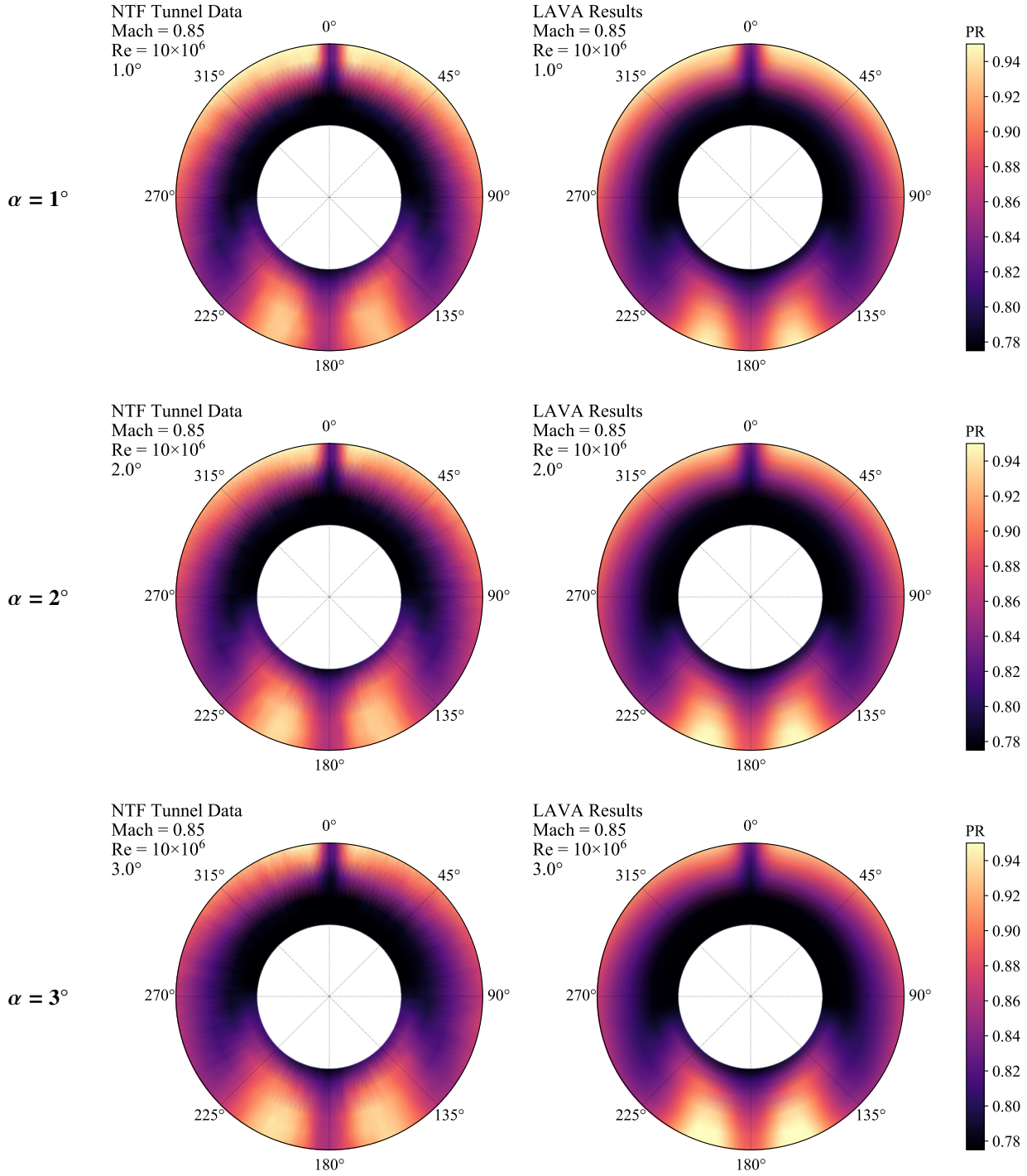


Fig. 25 Contour plots of the total-pressure recovery (PR) reconstructed from the ARP1420 data taken at different nacelle roll angles. The effect of increasing angle-of-attack is visualized for the *flow-through TCT w/ Cruise MFP* configuration running at the $Re = 10 \times 10^6$, $Ma = 0.85$ condition.

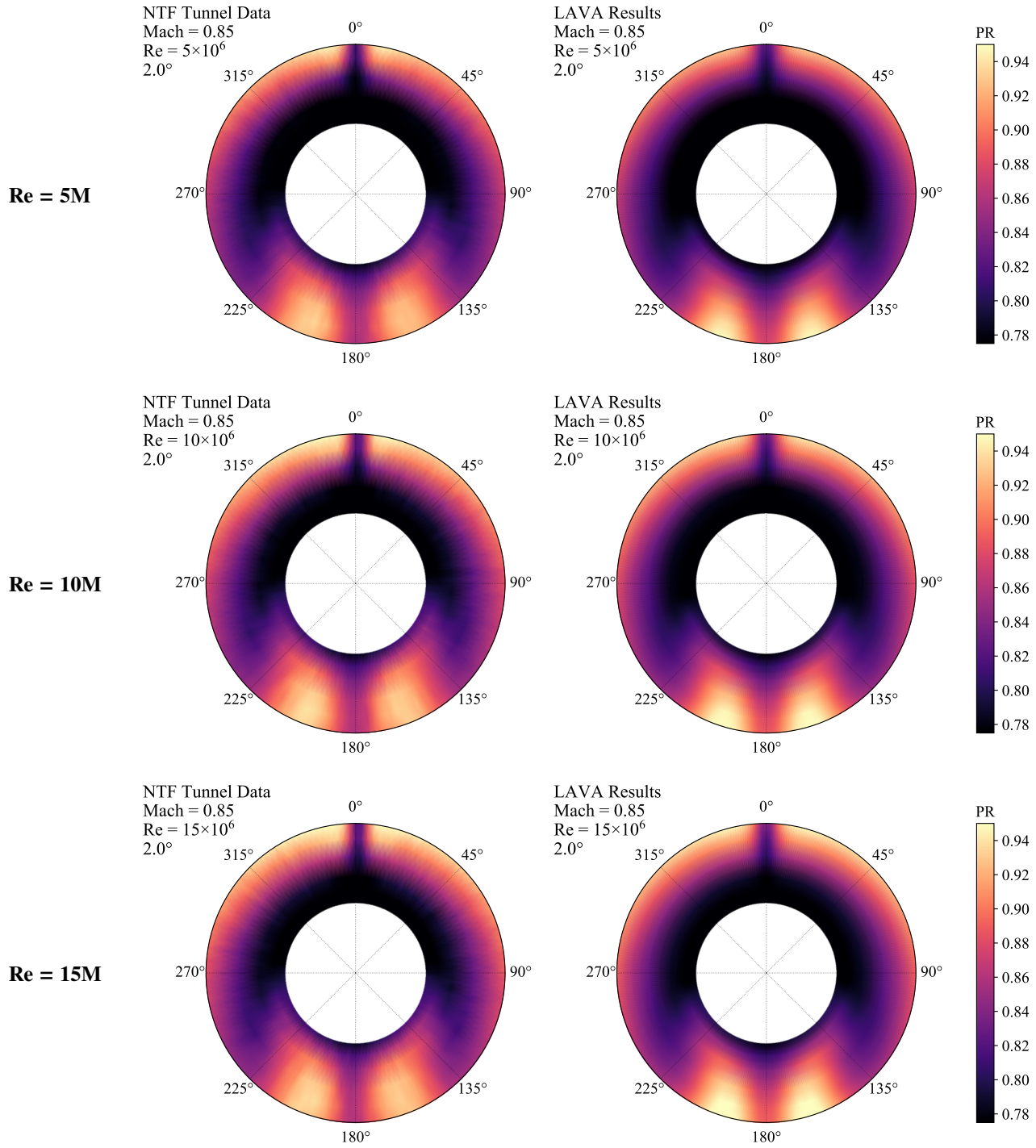


Fig. 26 Contour plots of the total-pressure recovery (PR) reconstructed from the ARP1420 data taken at different nacelle roll angles. The effect of increasing Reynolds number is visualized for the *flow-through TCT w/ Cruise MFP* configuration running at the $Ma = 0.85$, $\alpha = 2^\circ$ condition.

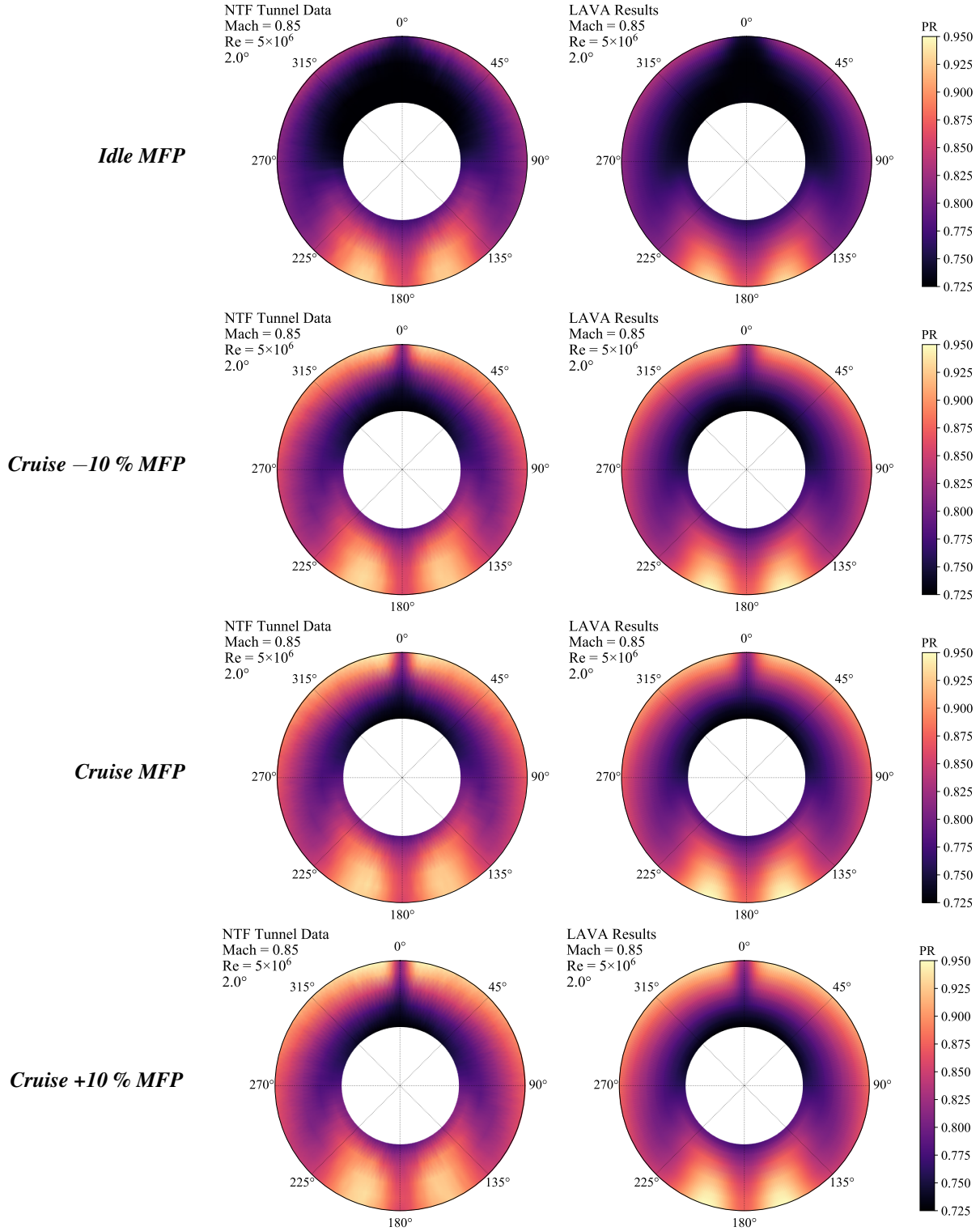


Fig. 27 Contour plots of the total-pressure recovery (PR) reconstructed from the ARP1420 data taken at different nacelle roll angles. The effect of increasing the nacelle mass-flow rate is visualized for the *flow-through TCT* configurations with different MFPs running at the $Re = 5 \times 10^6$, $Ma = 0.85$, $\alpha = 2^\circ$ condition.

Finally, Fig. 27 emphasizes the effects of increasing the nacelle mass-flow rate by comparing the inlet flow distortion across the four different MFPs tested in the *flow-through TCT* configuration. Results are shown at the 5-million Reynolds number, Mach 0.85 condition at an angle-of-attack of 2° . The differences here are striking. At the lowest flow rate setting with the *Idle MFP*, a large amount of boundary-layer (low-PR) flow is captured by the inlet, especially on the top half of the inlet. The larger blockage introduced by the wider *Idle MFP* causes a PR drop of up to 20 % in this region. Conversely, as the mass-flow rate increases, more near-freestream (high-PR) flow is pulled into the nacelle inlet capture area and ends up being ingested by the TCT. These trends are well-captured by the RANS model, and the comparisons between the experiment and numerical data show good agreement.

A quantitative comparison of the experimental and numerical total-pressure recovery profiles at the nacelle highlight is provided in Fig. 28 for the 5-million and 15-million Reynolds number condition at 2° angle-of-attack. Results for the three Mach numbers tested are provided for each. The results support the trends described thus far, and confirm the good agreement suspected from the qualitative plots. There are, however, some regions more susceptible to discrepancies between the two sets of data. These are the regions between 45° and 315° for Ring 1 (closest to the hub), and the regions containing the two high-PR lobes described earlier for the remaining rings (a wider region between 120° and 240° for

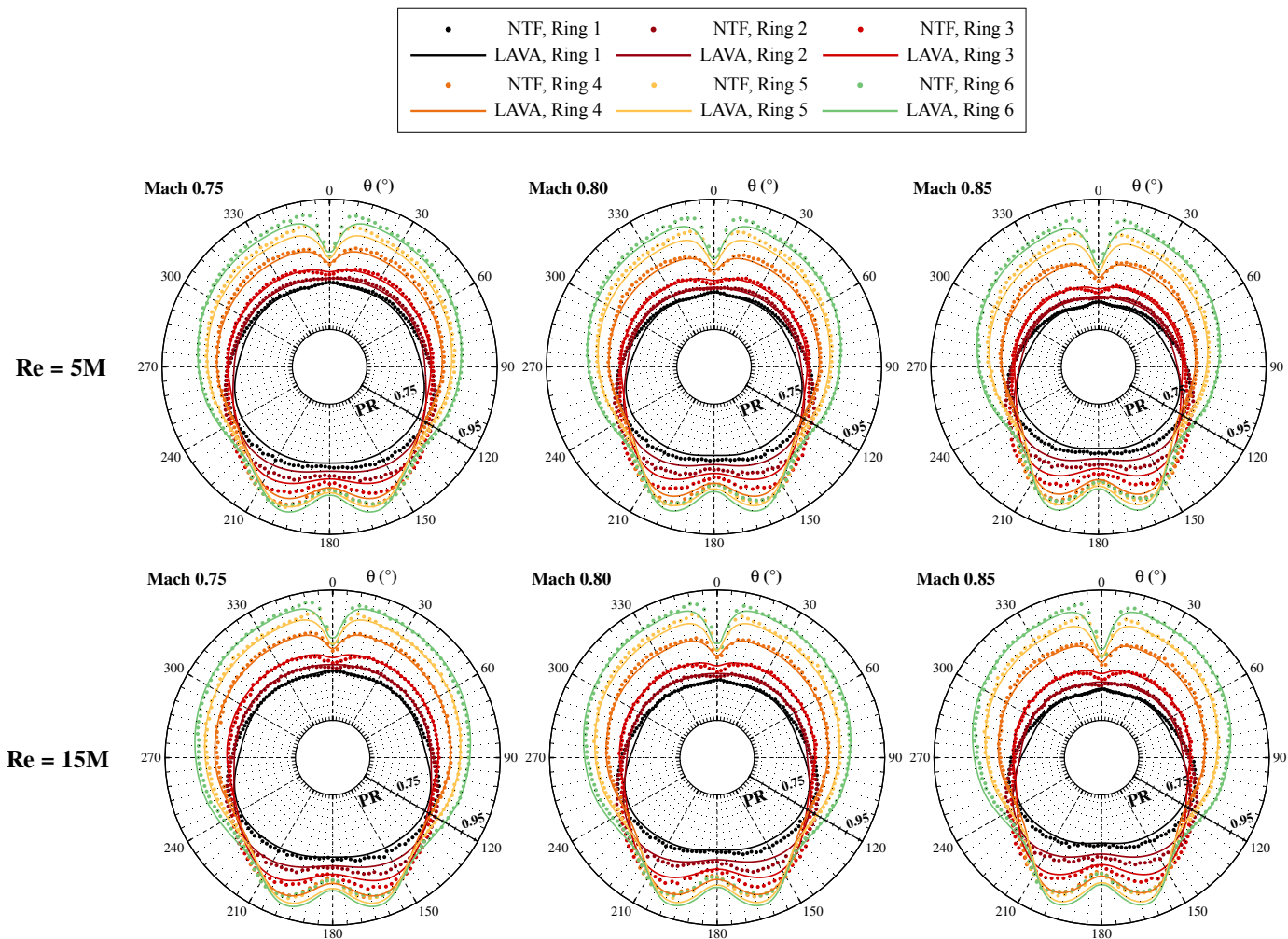


Fig. 28 Comparison of predicted and measured total-pressure recovery (PR) values along the six rings defined by the ARP1420 probes. Results shown for the *flow-through TCT w/ Cruise MFP* configuration at an angle-of-attack of 2° , Reynolds numbers of 5- and 15-million (top and bottom rows, respectively), and the three Mach numbers considered.

mid-radii rings 2 and 3, and a shorter region between 150° and 210° for rings 4, 5 and 6 closest to the casing).

2. Flow Angularity from 5-Hole Probes

Another measure of flow distortion at the nacelle inlet is the flow angularity in the velocity field. In the wind tunnel test, this data was collected with the 5-hole probe nacelle, containing a total of 9 5-hole probes distributed along three rakes (3 probes on each rake at different radii). This time, nacelle roll angle traverses were accomplished by rotating the nacelle 120° at 5° increments, allowing for the three rakes to cover the entire 360° profile. This data was collected only at a Reynolds number of 5-million, and for the *Cruise MFP* and *Cruise +10 % MFP flow-through TCT* configurations.

Flow angularity can be measured in several different ways. In this paper the results will be presented in terms of a flow angle-of-attack α_{flow} and a flow sideslip angle β_{flow} , defined in terms of the velocity components at the nacelle highlight as $\alpha_{\text{flow}} = \text{atan2}(V_z, V_x)$ and $\beta_{\text{flow}} = \text{atan2}(V_y, V_x)$, where V_x , V_y and V_z are aligned with the aircraft roll, pitch and yaw axis, respectively. Results are presented in the form of circumferential profiles at constant radii along three different rings, with radii of 0.746 inches (Ring 1), 1.023 inches (Ring 2) and 1.424 inches (Ring 3).

Starting with a qualitative look at the flow angularity predictions at the nacelle highlight, Figs. 29 and 30 show the effect of increasing angle-of-attack on the flow angle-of-attack α_{flow} and sideslip angle β_{flow} , respectively. The results reconstructed from the wind tunnel data look qualitatively similar to the LAVA results. Local minima and maxima are predicted at the correct locations and with the correct intensity. One exception to this observation occurs for the flow angle-of-attack at the 0° position close to the hub, where the local minima is slightly overpredicted. Close to the casing the wind tunnel plots also display a darker red tone on the top half of the inlet, indicating more pronounced flow angles compared to the predicted values. With respect to the flow sideslip angle, results also show good qualitative agreement. Wind tunnel data around the 0° position does not smoothly transition from positive to negative values, which is either a result of the relatively low 5° resolution with which the data was collected or a symptom of local flow asymmetries.

Apart from this, however, the contours show symmetry about the vertical axis as expected. Two lobes located between 135° – 180° and 180° – 225° indicate an interesting local reversal in flow direction close to the hub, likely an effect of the fuselage upsweep vortices. These features are more evident in the numerical results, but are also present in the wind tunnel data if one looks closely. Finally, the overall effect of increasing angle-of-attack is very subtle for both the flow angle-of-attack and sideslip angle at the nacelle highlight, suggesting that the flow angularity is more directly impacted by the local geometry.

Figure 31 summarizes the results for the two flow angles at the nacelle highlight at an angle-of-attack of 2° for the two mass-flow plugs tested with the *flow-through TCT* configuration. Good agreement is still observed with the high mass-flow rate plug installed. The effect of increasing the propulsor mass-flow rate is an overall increase in the flow angularity distortion observed, with local minima becoming lower and local maxima becoming higher when compared to the *Cruise MFP* case. This trend is well captured qualitatively by the LAVA simulations.

Figure 32 shows a quantitative comparison between the measured and predicted flow angles at the three Mach numbers tested for the 5-million Reynolds number condition at an angle-of-attack of 2°. The results shown pertain to the *flow-through TCT w/ Cruise MFP* configuration. Three vertical dashed lines placed at azimuthal angles of 60°, 180° and 300° indicate the locations where data collection carries over to the next A, B or C rake, as each covers a range of 120°. The results show very good quantitative agreement with the experimental data for both the flow angle-of-attack and sideslip angle. A few regions where some small discrepancies exist are around 180° for the flow angle-of-attack along Ring 3 (under the fuselage, closest to the casing) and between 135° and 225° for the sideslip angle along Rings 1 and 2 (under the fuselage, around the center and closer to the hub). These regions align with the regions of largest discrepancy also observed for the pressure recovery at the nacelle highlight, and are likely linked to the accurate modeling of the fuselage upsweep vortices. Nonetheless, the differences between the measured and predicted values are within 4° at most, which when one considers the quoted manufacturer uncertainty of $\pm 1^\circ$ for these probes (Chan et al., 2024) is reasonable. Additionally, a trend reversal occurs at the outermost ring (Ring 3) for both flow angles around 0°. This reversal is correctly captured in the flow angle-of-attack. However, for the sideslip angle the experimental data trend moves away from $\beta_{\text{flow}} = 0^\circ$. Given that the flow should be symmetric about the $Y = 0$ meters plane, the wind tunnel data for this ring around the 0° position is questionable. Another region where the validity of the experimental sideslip angles may be put into question occurs for Ring 1 at the transition from rakes C to A (at the 300° position). The discontinuity in the tunnel data incorrectly suggests a worse agreement between the measured and predicted values than is likely the case, especially when the symmetric transition point between rakes A and B (at the 60° position) is analyzed.

Figures 33 and 34 show the effect of propulsor operating condition on the flow angle-of-attack and sideslip angles, respectively, at different Mach number conditions. Results were separated for each ring for clarity. Overall, the trends

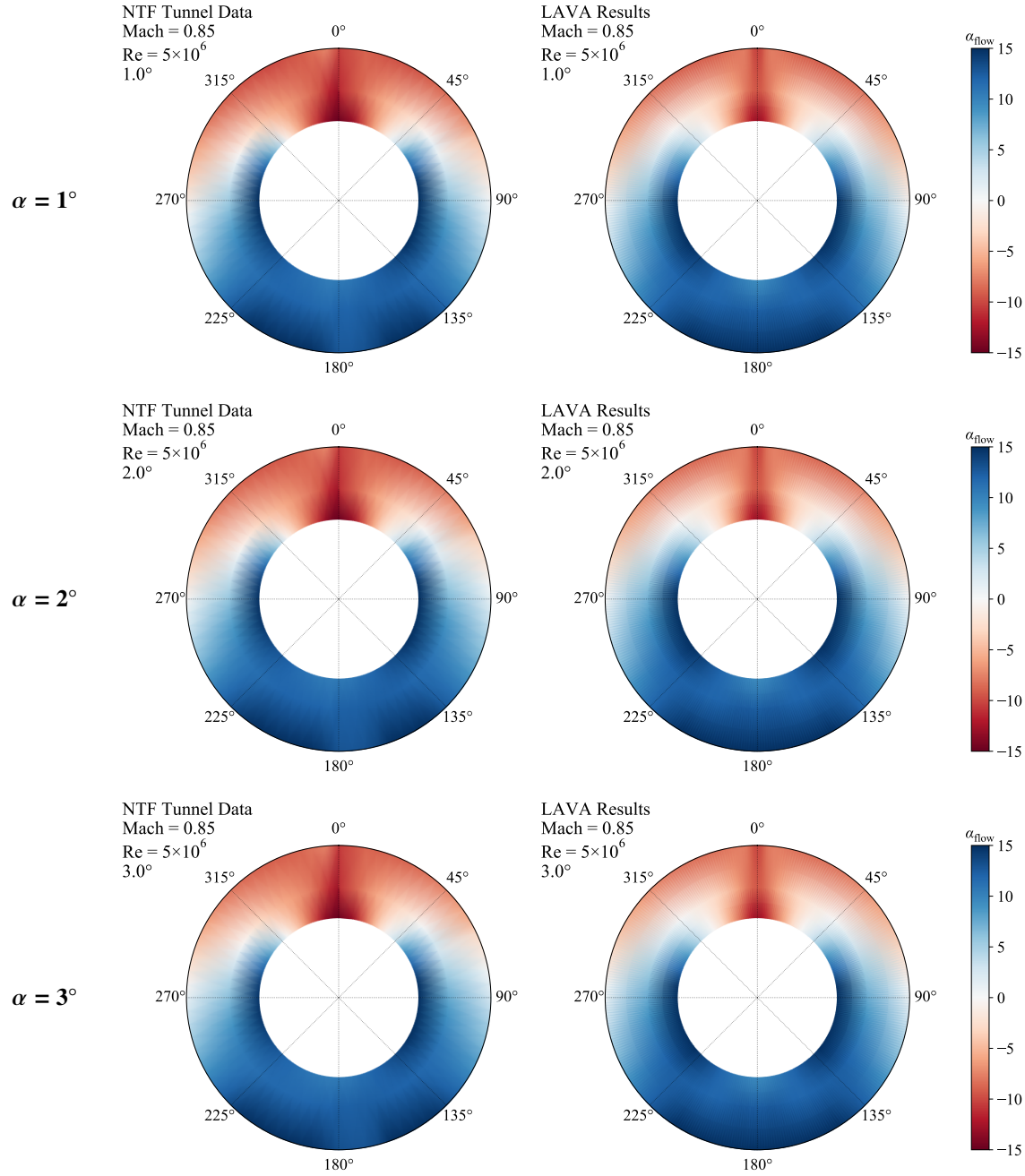


Fig. 29 Contour plots of the flow angle-of-attack reconstructed from the 5-hole probe data taken at different nacelle roll angles. The effect of increasing angle-of-attack is visualized for the *flow-through TCT w/ Cruise MFP* configuration running at the $\text{Re} = 5 \times 10^6$, $\text{Ma} = 0.85$ condition.

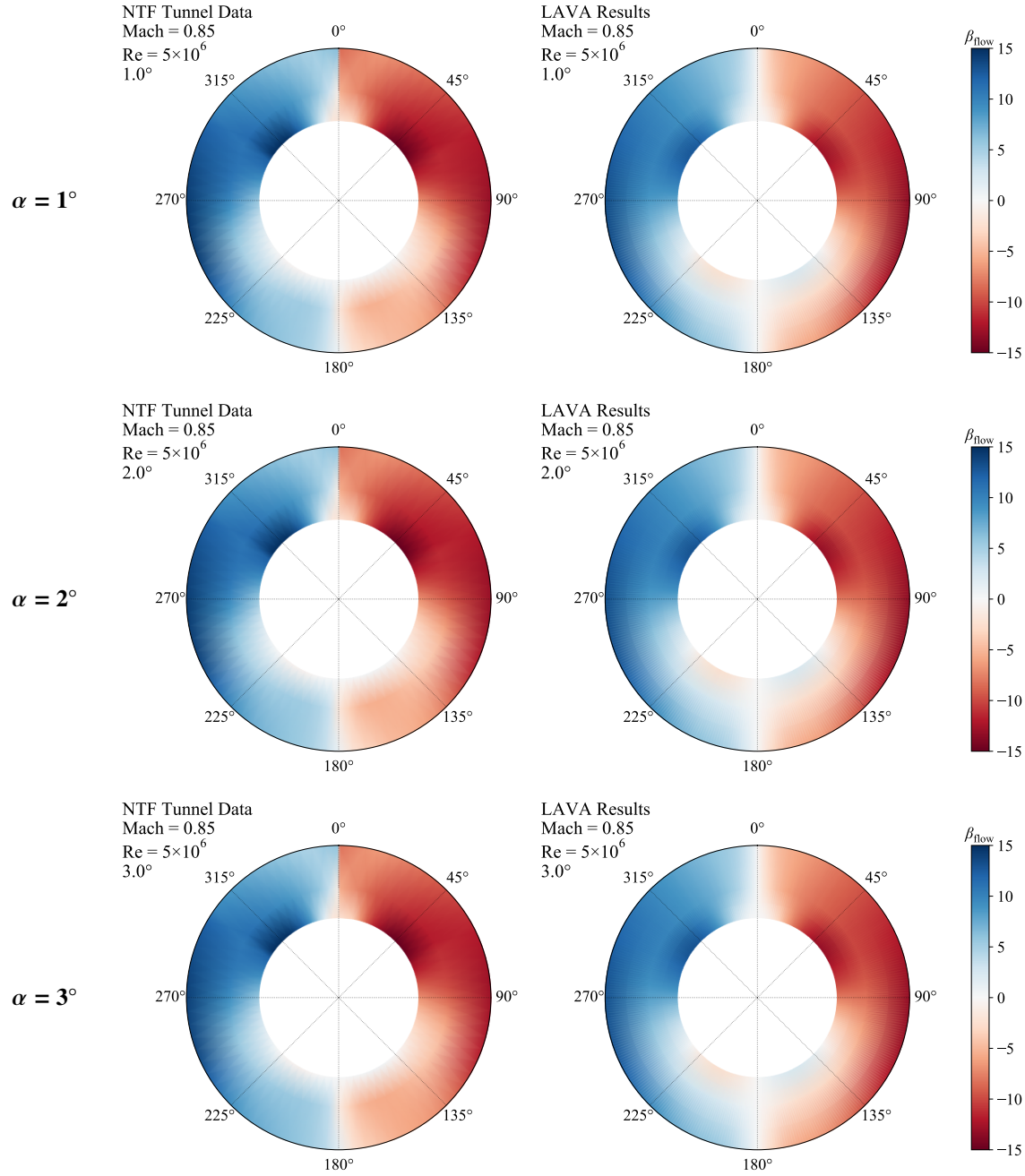


Fig. 30 Contour plots of the flow sideslip angle reconstructed from the 5-hole probe data taken at different nacelle roll angles. The effect of increasing angle-of-attack is visualized for the *flow-through TCT w/ Cruise MFP* configuration running at the $Re = 5 \times 10^6$, $Ma = 0.85$ condition.

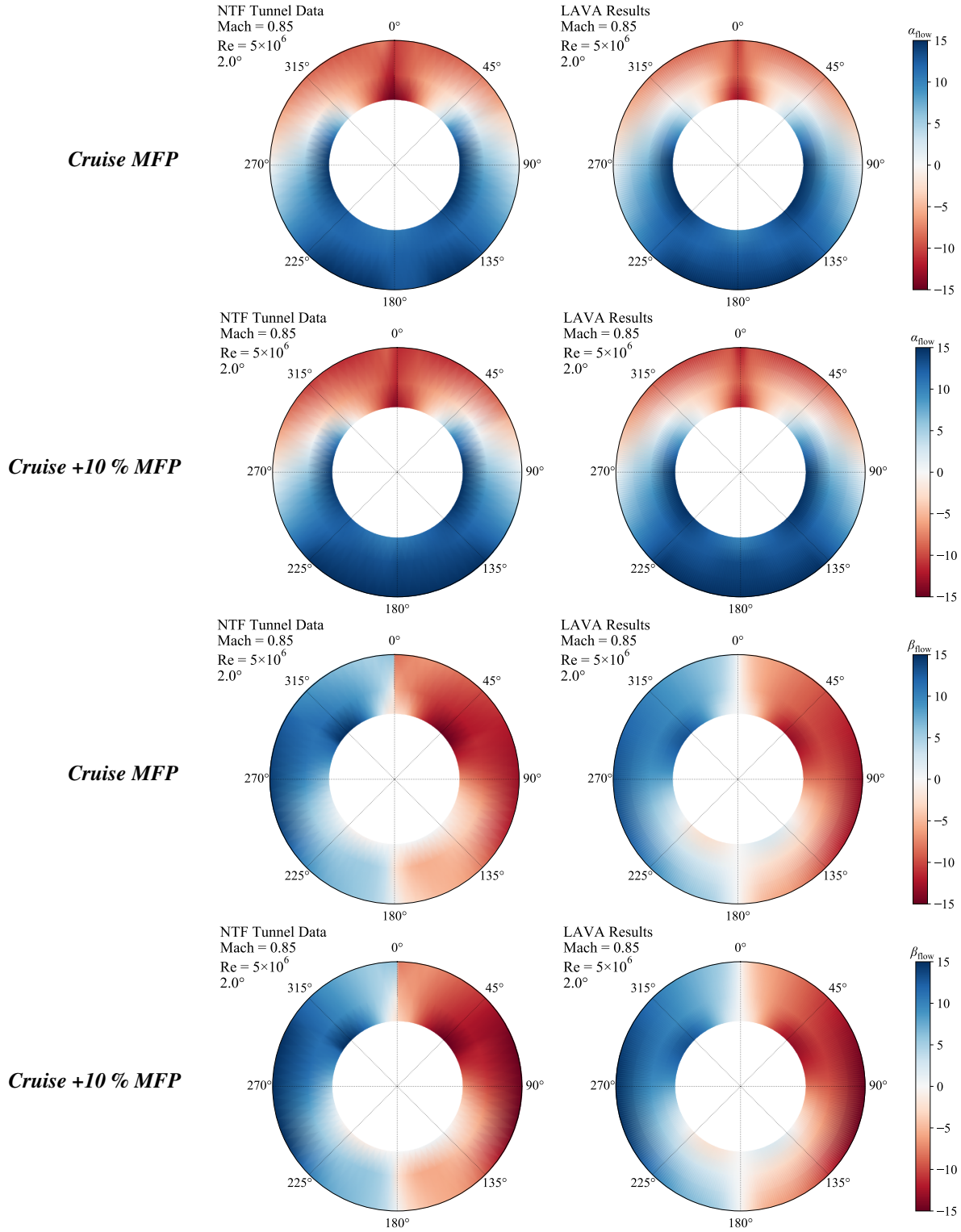


Fig. 31 Contour plots of the flow angle-of-attack (top two rows) and sideslip angle (bottom two rows) reconstructed from the 5-hole probe data taken at different nacelle roll angles. The effect of increasing the nacelle mass-flow rate is visualized for the *flow-through TCT* configurations with different MFPs running at the $Re = 5 \times 10^6$, $Ma = 0.85$, $\alpha = 2^\circ$ condition.

are well-captured by the RANS model. With respect to the flow angle-of-attack, discrepancies of up to 2° are observed along the range covered by rake A. Nonetheless, the deltas between the two mass-flow plugs appear to match well with experiment. The effect of increasing the propulsor mass-flow rate is more subtle near the hub (Ring 1) and more pronounced closer to the casing (Ring 3). A wider range of α_{flow} is predicted for the higher mass-flow rate configuration, increasing from $[-8^\circ, 15^\circ]$ to $[-11^\circ, 17^\circ]$. A similar widening of the range covered by the flow sideslip angle is observed along Ring 3 in Fig. 34. Conversely, the range is lowered along Ring 1 when the mass-flow rate increases, which is especially evident at the lower Mach number condition. For this flow angle, the main regions of discrepancy between the predicted and measured values are within the 135° to 225° range.

D. Boundary-Layer Measurements

Boundary-layer measurements at forward and aft axial stations of the aftbody (Fig. 2(a)) were collected for the *clean* configuration at the 5-million Reynolds number condition. Results are presented in Fig. 35 for the 0.85 Mach number condition only, as they are representative of the remaining Mach numbers simulated as well. Three angles-of-attack are shown, namely 1° , 2° and 3° . Note that for the latter two the profiles were shifted by +2 psi and +4 psi, respectively, to allow for proper visualization of the profiles. Numerical results shown for each profile were obtained by targetting the corresponding lift coefficient at each angle-of-attack for a fair comparison.

Results at the forward 0° , 45° and 90° locations show excellent agreement with the wind tunnel data. These profiles show a traditional boundary-layer profile characterized by the recovery of freestream pressure away from the walls. The profile located at 135° is affected by the downwash from the wing/body junction, and shows good agreement in the outer half of the boundary-layer profile. The shape of the profile close to the wall is not well captured by the SA-RC-QCR2000 turbulence model used for the database, as discussed in Sec. V.B. Finally, the boundary-layer profiles at the 180° station show a thicker boundary-layer when compared with the experimental measurements. The comparison does not improve at higher angles-of-attack.

The bottom row in Fig. 35 shows the results for the three boundary-layer rakes at the aft station in the aftbody. The

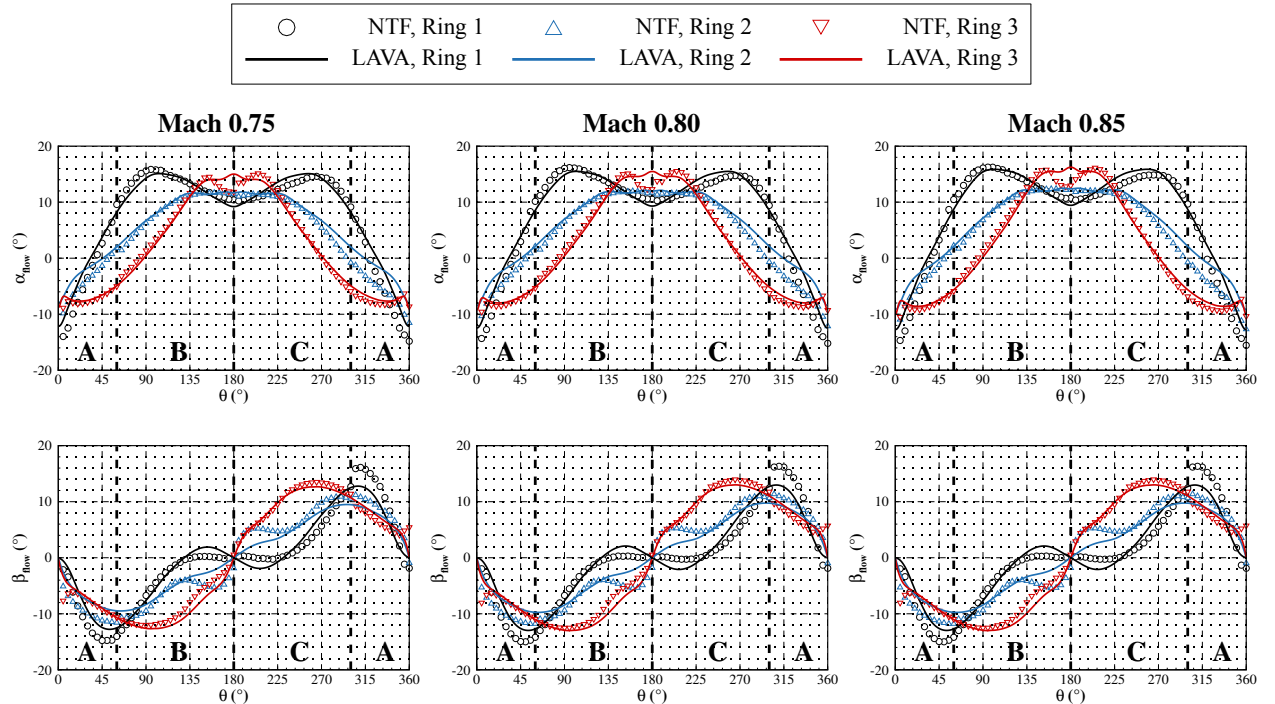


Fig. 32 Flow angularity profiles measured with the 5-hole probes at the nacelle highlight, in terms of flow angle-of-attack (top row) and flow sideslip angle (bottom row). Results shown for the *flow-through TCT w/ Cruise MFP* configuration at a Reynolds number of 5-million and an angle-of-attack of 2° . Each column corresponds to a different Mach number.

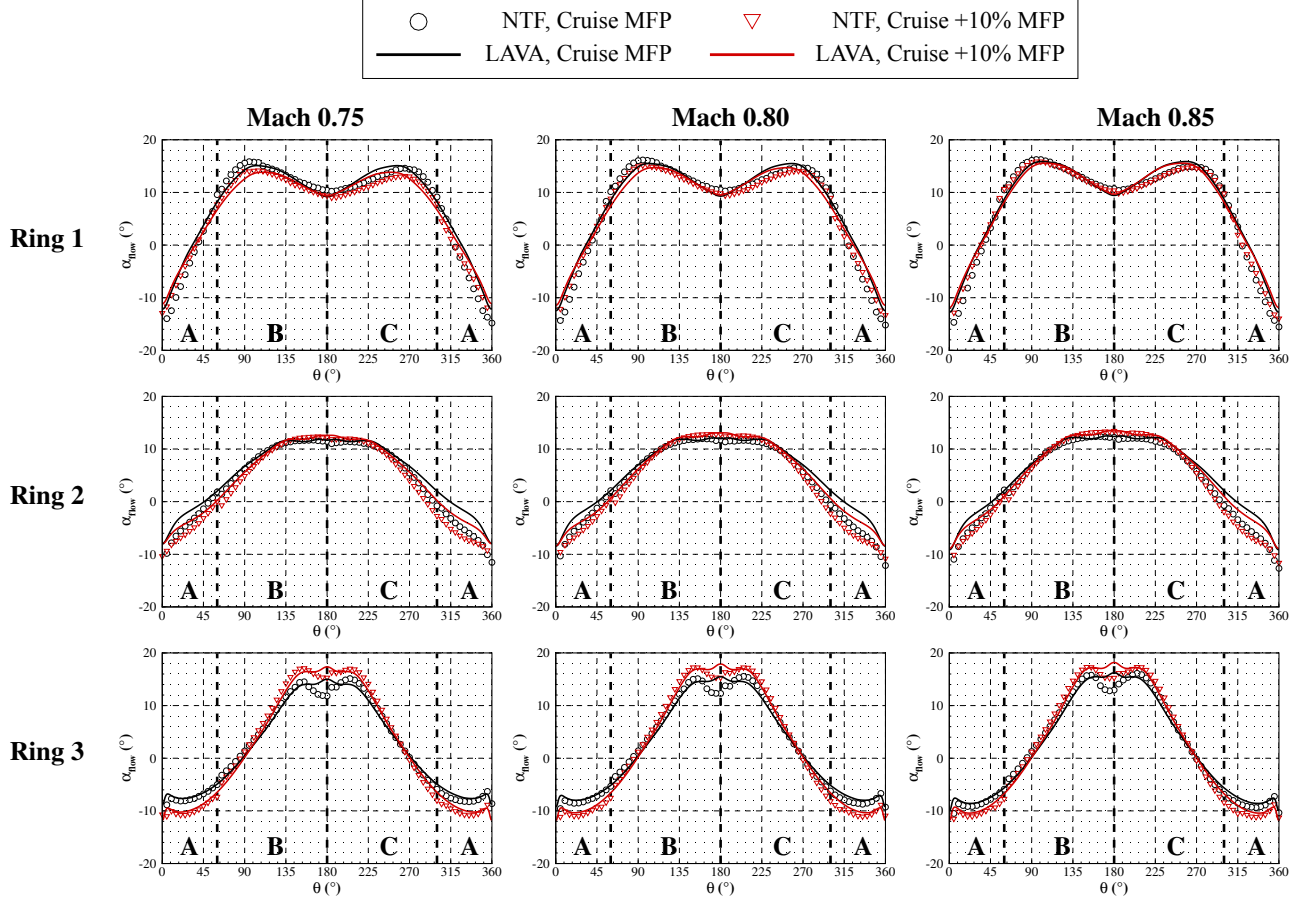


Fig. 33 Flow angle-of-attack profiles measured with the 5-hole probes at the nacelle highlight. Results shown for the *Cruise MFP* and *Cruise +10 % MFP* installed in the *flow-through TCT* configuration at a Reynolds number of 5-million and an angle-of-attack of 2° , emphasizing the sensitivity to the propulsor operating condition.

60° station results show good agreement with the tunnel data. At 120° , the predicted total-pressures slightly overpredict the experimental data. Conversely, at 180° the profiles underpredict the wind tunnel data up to 1 inches away from the wall, and then seem to overshoot the experimental values. Note, however, that the numerical values recover the freestream total-pressure at the outermost probes, whereas the wind tunnel profiles do not.

E. Aftbody Static Pressure Taps

The static pressure coefficient along the fuselage aftbody for different configurations tested is analyzed in this section. These static pressure ports were located in the aftbody portion of the fuselage and distributed axially along lines of constant azimuthal angle ϕ , namely $\phi = 45^\circ, 90^\circ, 135^\circ$ and 180° , following the same convention as the boundary-layer rake angles illustrated in Fig. 2(a). The streamwise coordinate is normalized by the nacelle inlet diameter, which provides a direct indication of the aftbody extent upstream of the nacelle highlight that is influenced by different nacelle configurations in terms of nacelle diameters. Results are separated for each row of pressure plugs at fixed azimuthal angle ϕ . The wind tunnel data for the plugs located at azimuthal angles of $45^\circ, 90^\circ$ and 135° show two data points for each axial station, corresponding to the port and starboard sides of the model mirrored about the symmetry plane. This provides an indication of the uncertainty in the experimental measurements, since both data points should return the same value given the symmetry of the configurations tested. This uncertainty appears to be very low for most data points, although there are some isolated instances where differences between the port and starboard side pressure coefficients are visible.

Figure 36 shows a comparison of the aftbody pressure coefficients between the *clean* and *flow-through TCT w/*

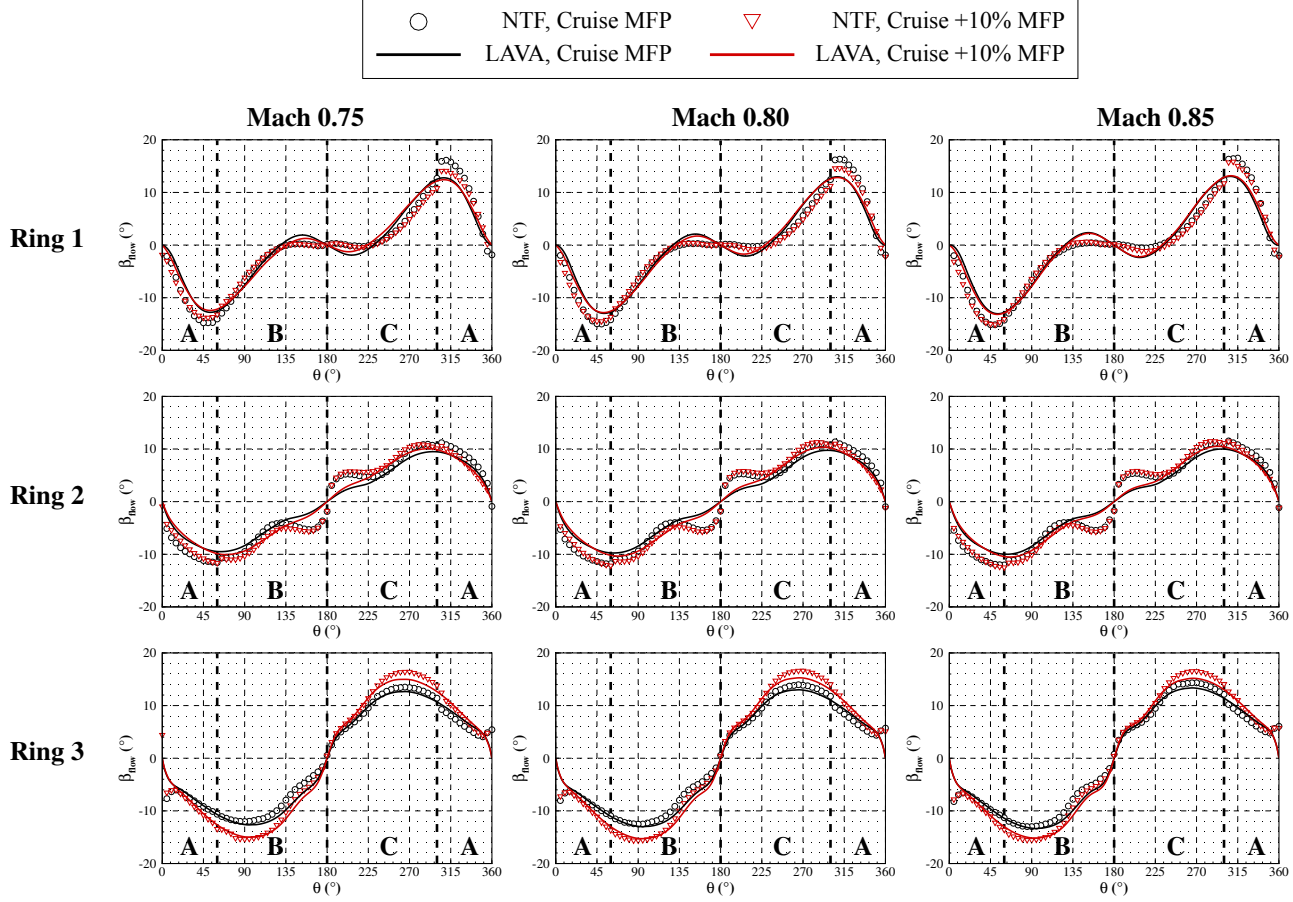


Fig. 34 Flow sideslip angle profiles measured with the 5-hole probes at the nacelle highlight. Results shown for the *Cruise MFP* and *Cruise +10 % MFP* installed in the *flow-through TCT* configuration at a Reynolds number of 5-million and an angle-of-attack of 2° , emphasizing the sensitivity to the propulsor operating condition.

Cruise MFP configurations at the 5-million Reynolds number condition and an angle-of-attack of 2° . Results for the three Mach number conditions simulated are reported in each column. The axial station corresponding to the nacelle highlight is indicated in each figure. A larger region of influence due to the presence of the nacelle is observed at the 45° azimuthal angle, spanning about 3 nacelle diameters upstream of the nacelle highlight. At the lower azimuthal angles this appears to drop to about 2 nacelle diameters upstream of the nacelle highlight. The presence of the nacelle in the *flow-through TCT* configuration is felt as an increase in the wall pressures relative to the *clean* aftbody. Overall, the agreement between the predicted and measured aftbody pressure coefficients is excellent. Only a small portion close to the nacelle highlight in the *flow-through TCT* configuration is observed to overshoot the pressure peak at the 45° and 90° azimuthal angles.

The effect of different mass-flow plugs on the aftbody static pressures upstream of the nacelle highlight is analyzed in Fig. 37 for the same Reynolds number and angle-of-attack. Results for the four mass-flow plugs tested in the *flow-through TCT* configuration are compared. In general, the larger the blockage introduced by the mass-flow plug in the nacelle internal flow, the larger the pressure peak that is felt upstream of the nacelle highlight. This trend is clear in all figures as the maximum pressure peak is observed for the *Idle* MFP, while conversely the minimum pressure peak is seen in the *Cruise -10 % MFP* configuration. The difference between the three *Cruise* MFPs is mostly felt around 1 nacelle diameter upstream of the nacelle highlight, while for the *Idle* MFP the upstream influence is extended to around 2 nacelle diameters. Again, the predictions show very good agreement with the wind tunnel data across all MFPs and Mach numbers, suggesting the RANS model is doing an excellent job at predicting the aftbody flow near the surface.

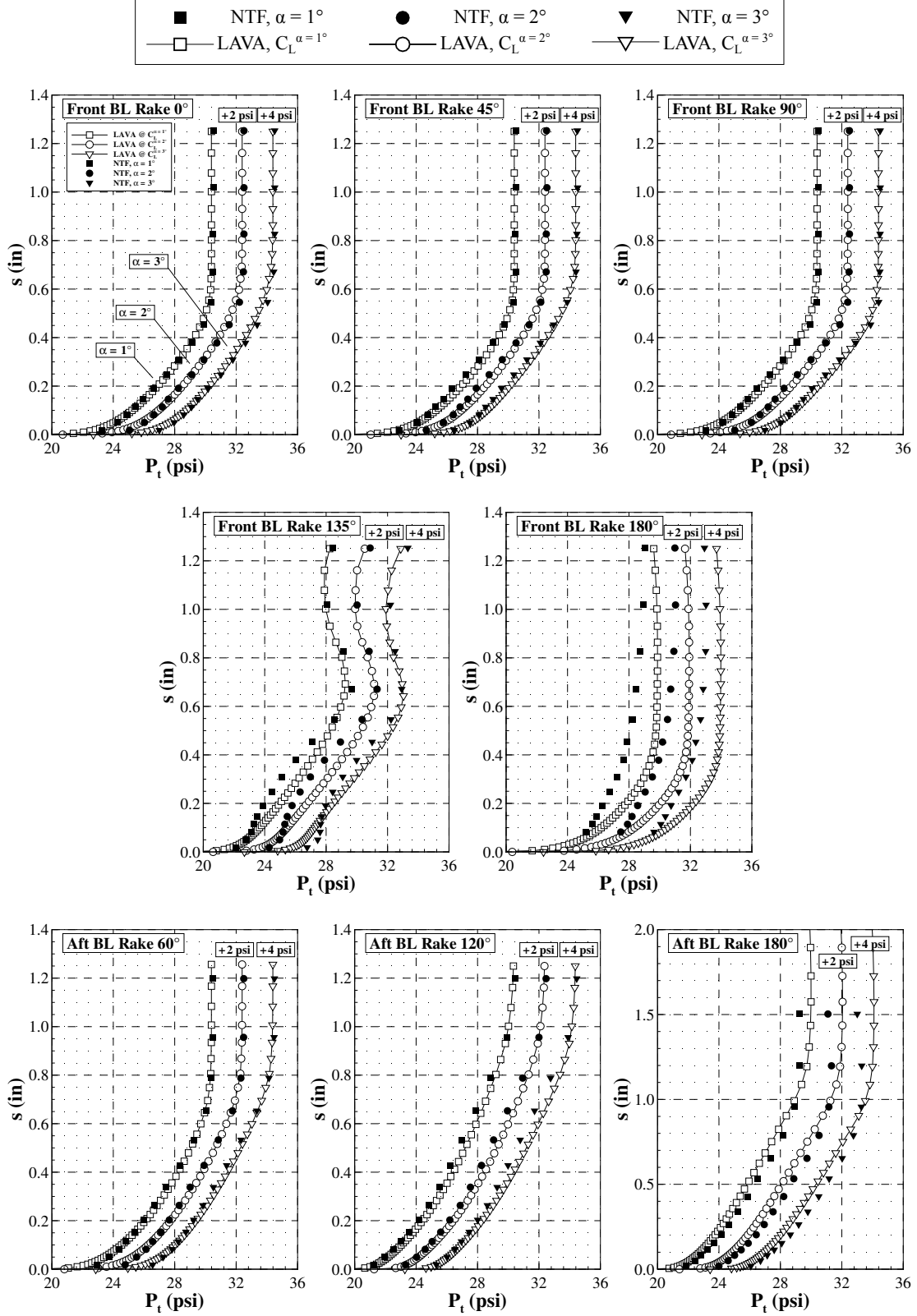


Fig. 35 Boundary-layer profiles in the forward and aft sections of the aftbody for the *clean* configuration, at the $Re = 5 \times 10^6$, $Ma = 0.85$ condition. Results for angles-of-attack of 1° , 2° and 3° are shown, where the latter two were shifted by +2 psi and +4 psi, respectively, for visualization purposes.

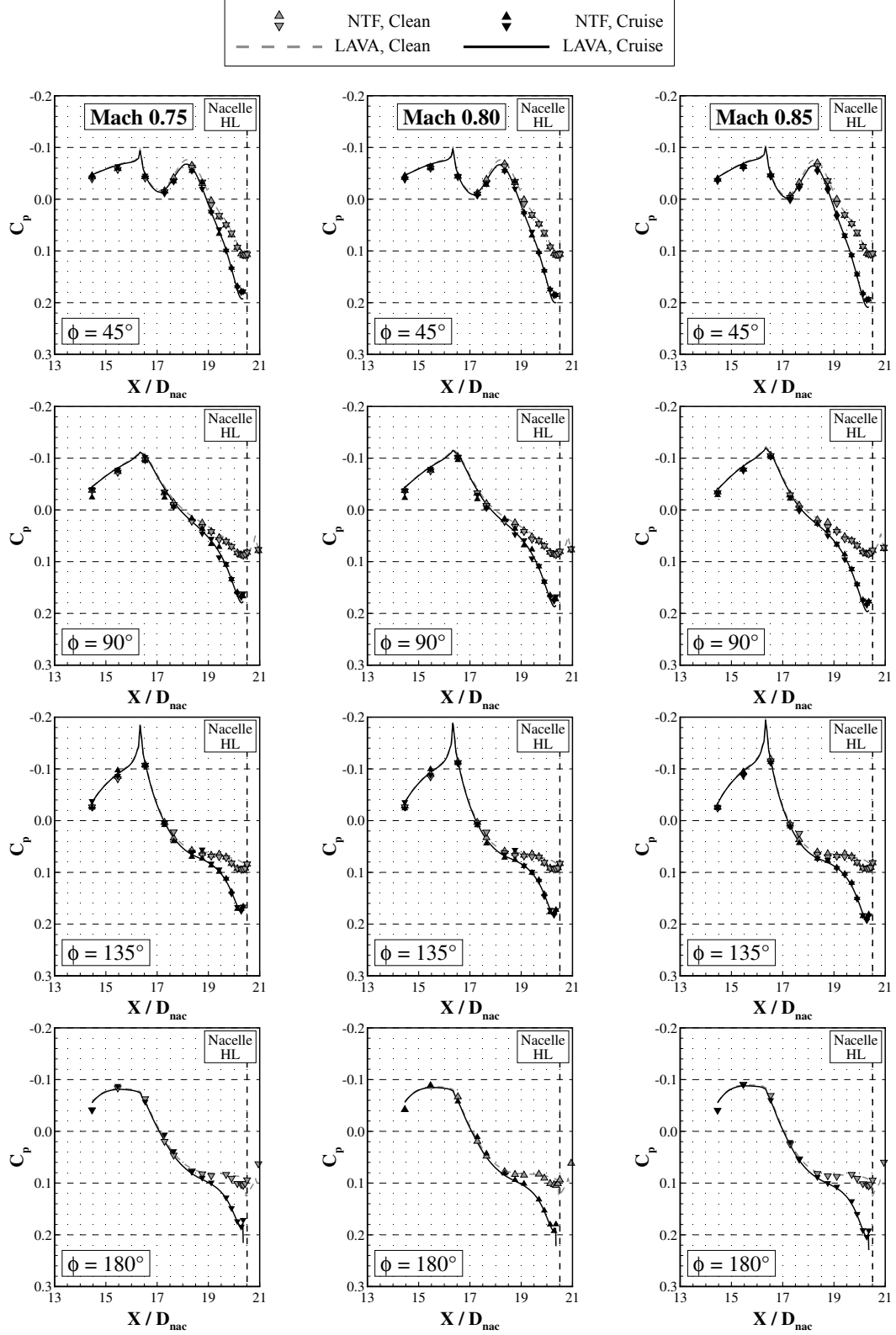


Fig. 36 Comparison of aftbody static pressure coefficients between the *clean* and *flow-through TCT w/ Cruise MFP* configurations. Results shown for the 5-million Reynolds number condition at an angle-of-attack of 2° . Each column represents a different Mach number condition. Experimental results for $\phi = 45^\circ$, 90° and 135° have two points per axial station, representing the pressure coefficient over the port and starboard sides of the model.

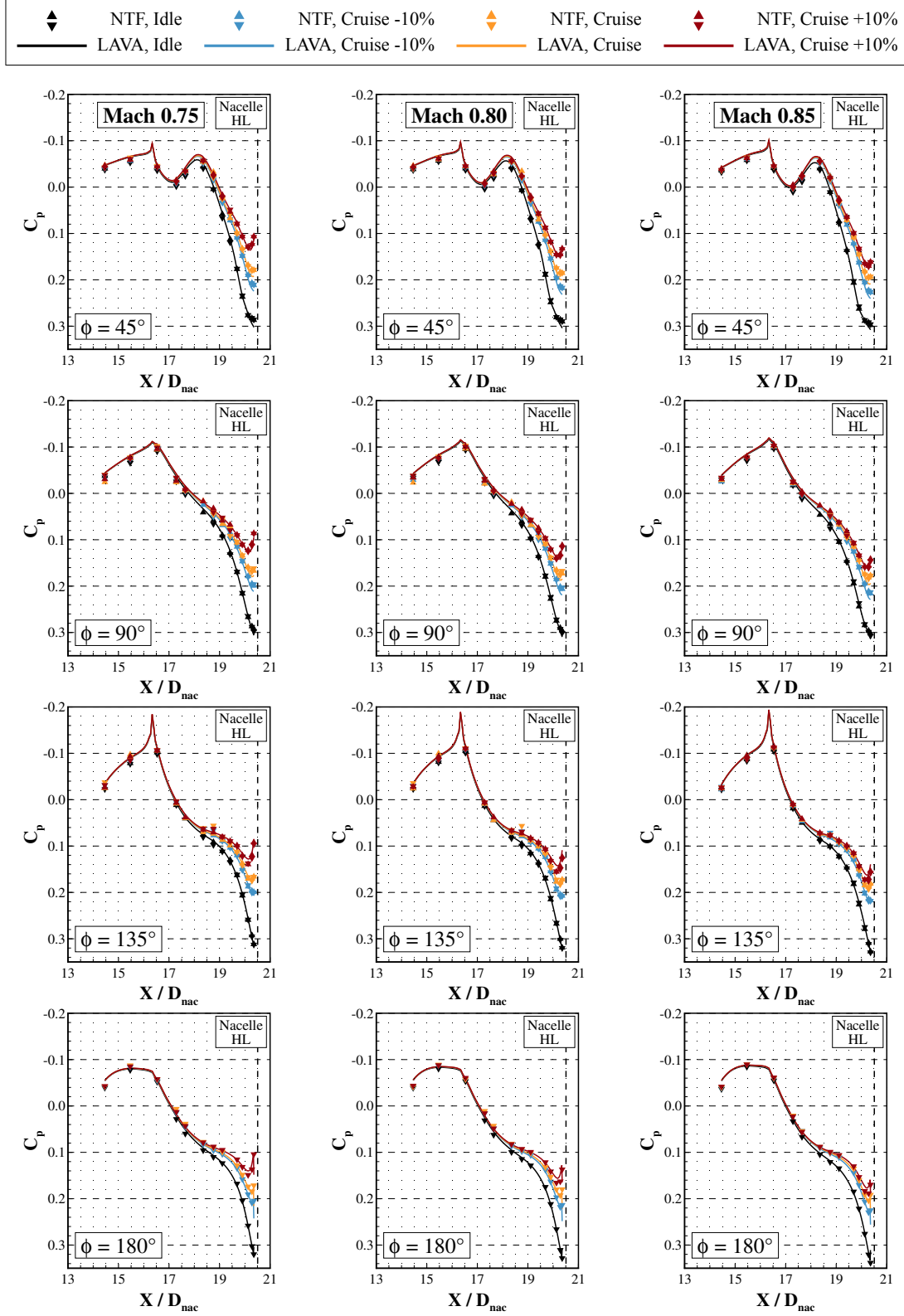


Fig. 37 Comparison of aftbody static pressure coefficients between the different MFPs on the *flow-through TCT* configurations. Results shown for the 5-million Reynolds number condition at an angle-of-attack of 2° . Each column represents a different Mach number condition. Experimental results for $\phi = 45^\circ$, 90° and 135° have two points per axial station, representing the pressure coefficient over the port and starboard sides of the model.

VII. Conclusions

This paper discusses the effort that took place to simulate the CRM-TCT model tested in the NTF using the LAVA solver framework, as part of the NASA AATT Boundary-Layer Ingesting Tailcone System emerging technical challenge. The vast amount of aerodynamic data collected between March and June 2023 was utilized to validate current RANS best-practices within the LAVA curvilinear overset solver when applied to BLI aircraft configurations. An important feature of this dataset is the wide variety of conditions and configurations tested. These include three Mach numbers in the transonic range (0.75, 0.80 and 0.85), three high-Reynolds numbers (5-, 10- and 15-million), and angles-of-attack in the range -3° to 4° . Two main configurations are considered, differing only in the aftbody geometry. A *clean* configuration with a traditional aftbody, and a *flow-through TCT* configuration with an aftbody TCT installed. In the latter, different aftbody propulsor operating conditions are simulated via the introduction of different-diameter mass-flow plugs in the TCT, allowing for a fourth independent parameter to be studied as part of the test.

Preliminary sensitivity studies assessing the influence of grid resolution and turbulence model closure in important flow-field metrics were conducted prior to running the entire database. A grid family consisting of four increasingly-refined grids employing topologies and grid counts following best-practice guidelines from the DPW-VI Workshop was generated. The results supported the use of grids consisting of around 90- and 117-million grid points for the *clean* and *flow-through TCT* configurations, respectively. Results from a turbulence model sensitivity study also showed better agreement with the tunnel data was obtained when using the SA-RC-QCR2000 turbulence model out of all the RANS turbulence model closures tested. These and other preliminary studies performed in the pre-test phase helped inform the numerical setup utilized to populate the final database of cases that was run in the wind tunnel.

A total of 205 steady RANS cases were simulated for the final database. These were meant to validate current RANS methods and best-practices with respect to the accurate prediction of the main features of interest in this BLI configuration, namely, the tailcone inlet flow distortion, aftbody boundary-layer and surface pressures, and airframe load performance. Globally, the steady RANS simulations show good agreement with the wind tunnel data. The aerodynamic performance of the aircraft is well predicted, with the drag coefficient showing excellent agreement with the data and the pitching moment coefficient slightly underpredicted. Overall trends with angle-of-attack, Mach and Reynolds numbers, and engine operating condition are accurately predicted in the simulations. The nacelle mass-flow rate calculation is replicated in the simulation data, and accurately returns values within 3.6 % of the wind tunnel data. Again, trends with propulsor operating condition are well captured by the steady RANS model.

Local flow quantities in the aftbody section of the model also show good agreement with the experiment. Apart from the boundary-layer rake located at the forward station, in the 180° azimuthal position, all other boundary-layer profiles were reproduced numerically. The one positioned at 135° showed the largest sensitivity to the turbulence model closure utilized, with the baseline SA model being the only able to capture the inflection in the profile. Although the SA-RC-QCR2000 model was not able to capture this accurately, the shape of the profile in the outer section of the boundary-layer still matched the experimental data well. Aftbody surface pressure coefficient results also show very good agreement with the tunnel data. The sensitivity to the aftbody geometry is very well captured numerically across the range of Mach numbers tested, and are well within the wind tunnel uncertainty range.

Finally, and perhaps most importantly, the nacelle inlet flow distortion results agree well with the experiment in both a qualitative and quantitative sense. In terms of nacelle total-pressure, the main regions where the model slightly deviates from the wind tunnel data are as follows: close to the hub, the boundary-layer total-pressure is slightly underpredicted in the bottom half of the fuselage; close to the casing, the total-pressure inside the two lobes that result from the effect of the fuselage upsweep vortices is slightly overpredicted, especially in the azimuthal angle range of 150° to 210° . Elsewhere, the pressure recovery profiles at the nacelle highlight are accurately predicted by the simulations. With respect to the flow angularity measured at the nacelle highlight, the predicted profiles match the wind tunnel data within the experimental uncertainty in most regions. Some local regions where the discrepancy is higher are present, especially under the fuselage where the effect of the upsweep vortices is felt, but even here the differences are kept below 4° in terms of the flow angle metric used.

This work demonstrates that RANS-based flow analysis can be used to accurately model TCT configurations using the LAVA curvilinear overset flow solver. Such approach will help in the design of future aircraft configurations employing next-generation BLI technologies, given the low computational cost and short turnaround times for simulation databases composed of hundreds of conditions.

VIII. Acknowledgements

This research is supported in part by the Advanced Air Transport Technology (AATT) project under NASA's ARMD Advanced Air Vehicles Program (AAVP). Computer time was provided by the NASA Advanced Supercomputing (NAS) facility at NASA Ames Research Center. The authors would like to express their sincere gratitude to David Chan and the rest of the National Transonic Facility team for hosting a visit to NASA Langley Research Center while the test campaign was underway, which provided an environment for fruitful discussions on the modeling of certain aspects of the wind tunnel model. Dr. Shishir Pandya provided technical guidance and supervised the coming together of the experimental and simulation teams throughout the project, and Dr. Michael Bozeman actively engaged in discussions of the results obtained with the USM3D code for comparison with the LAVA solver framework. The authors also gratefully acknowledge Daniel Maldonado and Brandon Lowe for the internal review of this abstract.

References

- Celestina, M. L., and Long-Davis, M. J., "Large-Scale Boundary Layer Ingesting Propulsor Research," *International Society for Air Breathing Engines (ISABE) Conference*, Canberra, Australia, 2019. <https://ntrs.nasa.gov/api/citations/20190031845/downloads/20190031845.pdf>.
- Kawai, R. T., Friedman, D. M., and Serrano, L., "Blended Wing Body (BWB) Boundary Layer Ingestion (BLI) Inlet Configuration and System Studies," Tech. Rep. NASA/CR-2006-214534, NASA Langley Research Center, 2006. <https://ntrs.nasa.gov/api/citations/20070006754/downloads/20070006754.pdf>.
- Drela, M., "Development of the D8 Transport Configuration," *29th AIAA Applied Aerodynamics Conference*, AIAA, Honolulu, Hawaii, USA, 2011. doi:10.2514/6.2011-3970.
- Welstead, J., and Felder, J. L., "Conceptual Design of a Single-Aisle Turboelectric Commercial Transport with Fuselage Boundary Layer Ingestion," *54th AIAA Aerospace Sciences Meeting*, AIAA, San Diego, California, USA, 2016. doi:10.2514/6.2016-1027.
- Jansen, R., Kiris, C. C., Chau, T., Machado, L. M., Duensing, J. C., Mirhashemi, A., Chapman, J., French, B. D., Miller, L., Litt, J. S., Denham, C. L., Lynde, M. N., Campbell, R. L., Hiller, B. R., Blaesser, N. J., and Heersema, N., "Subsonic Single Aft Engine (SUSAN) Transport Aircraft Concept and Trade Space Exploration," *AIAA Scitech 2022 Forum*, AIAA, San Diego, California, USA, 2022. doi:10.2514/6.2022-2179.
- Hardin, L., Tillman, G., Sharma, O., Berton, J., and Arend, D., "Aircraft System Study of Boundary Layer Ingesting Propulsion," *48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, AIAA, Atlanta, Georgia, USA, 2012. doi:10.2514/6.2012-3993.
- Ochs, S. S., Tillman, G., Joo, J., and Voytovych, D. M., "Computational Fluid Dynamics-Based Analysis of Boundary Layer Ingesting Propulsion," *Journal of Propulsion and Power*, Vol. 33, No. 2, 2017, pp. 522–530. doi:10.2514/1.B36069.
- Machado, L., Chau, T., and Duensing, J. C., "Aeropropulsive Assessment of a BLI Tail-Cone Thruster Propulsion System for the NASA Common Research Model and a Transonic Truss-Braced Wing Configuration," *AIAA Aviation 2024 Forum*, AIAA, Las Vegas, Nevada, USA, 2024.
- Hall, D. K., Huang, A. C., Uranga, A., Greitzer, E. M., Drela, M., and Sato, S., "Boundary Layer Ingestion Propulsion Benefit for Transport Aircraft," *Journal of Propulsion and Power*, Vol. 33, No. 5, 2017, pp. 1118–1129. doi:10.2514/1.B36321.
- Chan, D. T., Jones, G. S., Langston, S. L., and Kwok, A. K., "Experimental Investigation of a Boundary Layer Ingesting Tailcone Thruster Configuration at the National Transonic Facility," *AIAA Aviation 2024 Forum*, AIAA, Las Vegas, Nevada, USA, 2024.
- Bozeman, M. D., "USM3D-ME Analyses Performed in Support of a Wind Tunnel Test of a Boundary-Layer Ingestion Configuration," *AIAA Aviation 2024 Forum*, AIAA, Las Vegas, Nevada, USA, 2024.
- Wahls, R., "The National Transonic Facility - A Research Retrospective," *39th Aerospace Sciences Meeting and Exhibit*, AIAA, Reno, Nevada, USA, 2021. doi:10.2514/6.2001-754.
- Jones, G. S., Saxer, D. R., and Flamm, J. D., "Model Preparation Areas for Propulsion Airframe Integration Testing at the NASA Langley Research Center," *AIAA SciTech 2021 Forum*, AIAA, Virtual, 2021. doi:10.2514/6.2021-1108.
- S-16 Turbine Engine Inlet Flow Distortion Committee, *Gas Turbine Engine Inlet Flow Distortion Guidelines*, March 1978. doi:10.4271/ARP1420.
- Kiris, C. C., Housman, J. A., Barad, M. F., Brehm, C., Sozer, E., and Moini-Yekta, S., "Computational Framework for Launch, Ascent, and Vehicle Aerodynamics (LAVA)," *Aerospace Science and Technology*, Vol. 55, 2016, pp. 189–219. doi:10.1016/j.ast.2016.05.008.

- Spalart, P., and Allmaras, S., “A One-Equation Turbulence Model for Aerodynamic Flows,” *30th Aerospace Sciences Meeting & Exhibit*, AIAA, Reno, Nevada, USA, 1992. doi:[10.2514/6.1992-439](https://doi.org/10.2514/6.1992-439).
- Shur, M. L., Strelets, M. K., Travin, A. K., and Spalart, P. R., “Turbulence Modeling in Rotating and Curved Channels: Assessing the Spalart-Shur Correction,” *AIAA Journal*, Vol. 38, No. 5, 2000, pp. 784–792. doi:[10.2514/2.1058](https://doi.org/10.2514/2.1058).
- Spalart, P., “Strategies for Turbulence Modelling and Simulations,” *International Journal of Heat and Fluid Flow*, Vol. 21, No. 3, 2000, pp. 252–263. doi:[10.1016/S0142-727X\(00\)00007-2](https://doi.org/10.1016/S0142-727X(00)00007-2).
- Menter, F. R., “Improved Two-Equation k- ω Turbulence Models for Aerodynamic Flows,” Tech. Rep. NASA-TM-103975, NASA Ames Research Center, 1992. <https://ntrs.nasa.gov/api/citations/19930013620/downloads/19930013620.pdf>.
- Housman, J. A., Kiris, C. C., and Hafez, M. M., “Time-Derivative Preconditioning Methods for Multicomponent Flows—Part I: Riemann Problems,” *Journal of Applied Mechanics*, Vol. 76, No. 2, 2009a, p. 021210. doi:[10.1115/1.3072905](https://doi.org/10.1115/1.3072905).
- Housman, J. A., Kiris, C. C., and Hafez, M. M., “Time-Derivative Preconditioning Methods for Multicomponent Flows—Part II: Two-Dimensional Applications,” *Journal of Applied Mechanics*, Vol. 76, No. 3, 2009b, p. 031013. doi:[10.1115/1.3086592](https://doi.org/10.1115/1.3086592).
- Koren, B., “A Robust Upwind Discretization Method for Advection, Diffusion and Source Terms,” *Numerical Methods for Advection-Diffusion Problems*, edited by C. Vreugdenhil and B. Koren, Vieweg, Germany, 1993. <https://ir.cwi.nl/pub/2269/2269D.pdf>.
- Saad, Y., and Schultz, M. H., “GMRES: A Generalized Minimal Residual Algorithm for Solving Nonsymmetric Linear Systems,” *SIAM Journal on Scientific and Statistical Computing*, Vol. 7, No. 3, 1986, pp. 856–869. doi:[10.1137/0907058](https://doi.org/10.1137/0907058).
- Tinoco, E. N., Brodersen, O. P., Keye, S., Laflin, K. R., Feltrop, E., Vassberg, J. C., Mani, M., Rider, B., Wahls, R. A., Morrison, J. H., Hue, D., Roy, C. J., Mavriplis, D. J., and Murayama, M., “Summary Data From the Sixth AIAA CFD Drag Prediction Workshop: CRM Cases,” *Journal of Aircraft*, Vol. 55, No. 4, 2018, pp. 1352–1379. doi:[10.2514/1.C034409](https://doi.org/10.2514/1.C034409).
- Fernandes, L. S., Machado, L. M., Duensing, J. C., and Kiris, C. C., “Computational Aerodynamics Analysis in Support of the CRM Tail Cone Thruster Configuration Wind Tunnel Test,” *AIAA Scitech 2022 Forum*, AIAA, San Diego, California, USA, 2022. doi:[10.2514/6.2022-1171](https://doi.org/10.2514/6.2022-1171).
- Tinoco, E., “Summary Data from the Seventh AIAA CFD Drag Prediction Workshop,” *AIAA Aviation 2023 Forum*, AIAA, San Diego, California, USA, 2023. doi:[10.2514/6.2023-3492](https://doi.org/10.2514/6.2023-3492).