

Flow and Acoustics Simulations of a Supersonic Engine Inlet

David B. Stephens¹ and Alexander A. Svetgoff²

NASA Glenn Research Center, Cleveland, OH, 44135, USA

An inlet for a supersonic aircraft engine was previously studied in a geometrically simplified form. The present paper continues that work and specifically investigates three-dimensional effects on the combined inlet and fan system. Three-dimensional flow, inlet components and viscous effects are studied for their effect on fan response and acoustics. Operating condition was identified as the most important parameter affecting noise, which is discussed in a companion paper. The details of the inlet geometry presented here are a smaller effect. Additionally, the results indicate that an axisymmetric two-dimensional computation is sufficiently accurate for some cases while a three-dimensional computation may reveal particular details.

I. Introduction

The NASA Commercial Supersonic Technology (CST) Project is working to reduce technical barriers to practical commercial supersonic flight. One of these barriers is the noise generated at takeoff and landing by engines designed for efficient supersonic cruise. Engines designed for high-speed flight have specific geometries that are significantly different than conventional subsonic engines. On the intake side of the engine, the inlet has features designed to efficiently decelerate the freestream flow from supersonic to subsonic. The fan system requires a higher pressure ratio than current subsonic engines, resulting in the two stage rotor considered here. The fan and inlet system considered in the present paper is shown in Fig. 1.

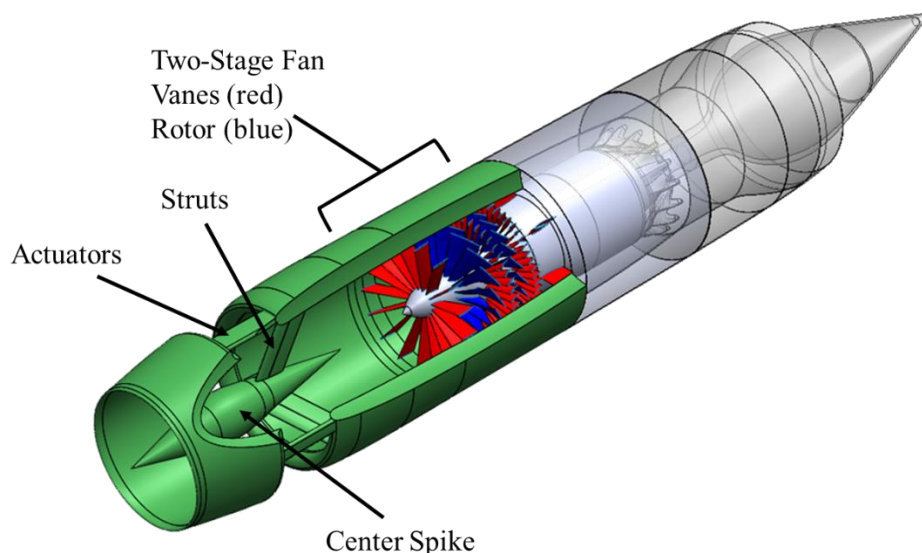


Fig. 1 Inlet and fan system used in this study.

¹ Research Engineer, Acoustics Branch, AIAA Member.

² Research Engineer, Acoustics Branch

The engine and inlet system that is considered in the present work was designed by GE Aviation under contract to NASA. The inlet was designed for supersonic cruise using the NASA SUPIN code [1] and then adapted for feasibility and off-design operation. The forward opening of the inlet is inadequate for the airflow requirement during low-speed flight, so an inlet with auxiliary doors was designed to enable additional airflow. Three struts link the center spike to the nacelle outer wall and four actuators attach the upstream cowl to the downstream part of the nacelle. Additionally, the actuators are of two different sizes, with the upper and lower actuators somewhat larger than the side actuators.

In our previous work, [2] the inlet was considered in a two-dimensional axisymmetric representation. In that work the present authors discussed rotor-stator interaction noise, fan operation conditions, and duct acoustics. The turbomachinery and sound source aspects of that work are continued in a companion paper [3]. In short, FINE™/Turbo CFD software [4] was used in a selective frequency-domain URANS simulation method known as a non-linear harmonic (NLH) method. The advantage of this method is a single blade passage is required for each blade row, greatly reducing the computational cost. The present paper focuses on the inlet flow field and far-field acoustics, using the results from the turbomachinery simulation as a boundary condition.

II. Background

NASA is currently pursuing high-speed flight by developing the X-59 Low Boom Flight Demonstrator. This research aircraft is intended to provide data on shaped sonic booms so overland supersonic flight can be regulated instead of prohibited. The work described here is part of a related task to enable commercial supersonic flight through progress in reducing airport noise. Specifically, the task is to improve landing and takeoff (LTO) noise predictions by using physics-based simulations to provide data and guidance that is useful for engineers studying supersonic aircraft systems. More accurate noise predictions will enable systems engineers to better design aircraft that can meet noise requirements while maintaining a useful mission.

Prior work by the first author [5] includes a survey of supersonic inlet fan acoustics research, including empirical fan noise models and specifically a study on how the shape of inlets affects the radiated sound. One conclusion from that paper is that for a given acoustic input, the shape of the inlet only affects the far-field directivity to the extent the modes propagating in the duct are scattered. Using linear acoustics, sound power is conserved as it propagates through the domain, whatever the inlet shape is. That paper only considered a compressible potential flow field while the present work uses a viscous flow as the background flow for the acoustic propagation.

III. Turbomachinery Acoustics

A major part of this investigation was the evaluation of the 2D axisymmetric method as a simplification for future noise propagation studies. The full set of 3D effects include the flow distortions caused by the struts and actuators, the mixing of flow from the front inlet and the auxiliary inlet, plus the effect of distortion due to the inlet operating at an angle of attack with respect to the freestream flow. All together, these effects produce an inflow distortion into the turbomachinery which affects the generation of noise propagating upstream. The inflow distortion thus changes both the sound source and the propagation, as the sound field must pass through the distorted inflow to reach observers on the ground. In the present paper, the baseline “clean” inflow noise source is used for the duct mode boundary conditions. All the distortion due to the inflow is neglected and the turbomachinery is fed with only a uniform inflow boundary condition. This serves to isolate the effect of noise propagation due to inlet effects from the effect of inflow distortion on the turbomachinery. The purpose of the present paper is inlet propagation assessment (IPA) rather than demonstrating a full noise prediction which we do not currently have data to validate.

The acoustic portion of the sound field is quantified by the Actran utility called iTM using the single-plane pressure-velocity (SPPV) method described in the Actran manual [6]. This utility extracts pressure and velocity from a plane perpendicular to the axis of the turbomachine and intersects the computational domain in the CFD output files. Each computational domain is shaped as a sector of a circle or annulus. In the present case, the inlet guide vanes (IGVs) are represented on a 1/17 sector, corresponding to the number of vanes. The iTM utility uses this sector to reconstruct the solution that would represent a full-wheel solution. The result of this process is shown in Fig. 2, for both approach and lateral engine operating conditions.

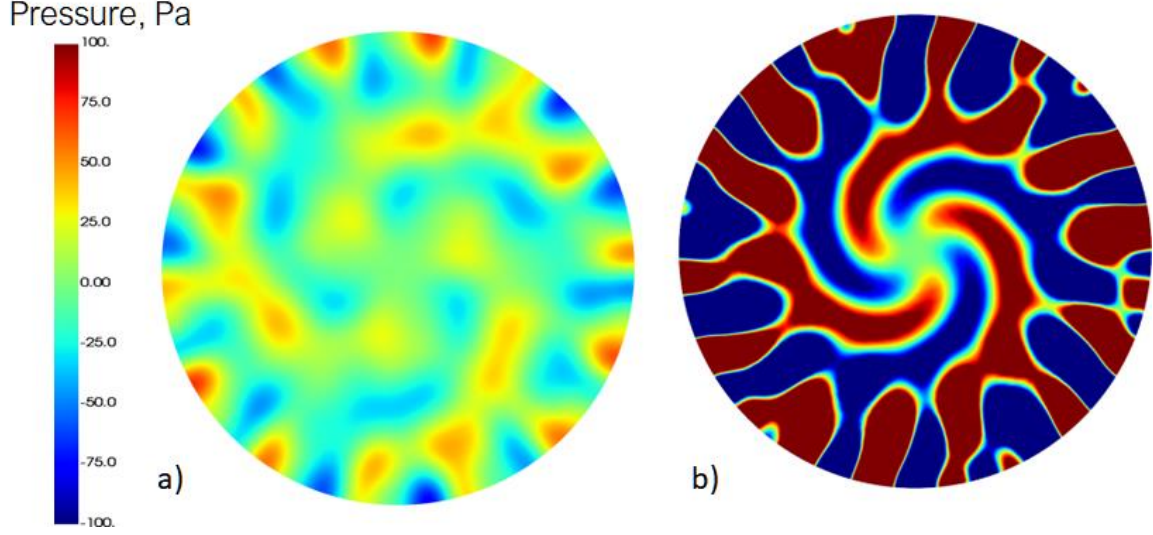


Fig. 2 Full wheel reconstruction of blade passing frequency at a) approach and b) lateral.

Note that the condition *lateral* was labeled *sideline* in the prior report as both terms are generally accepted. As discussed in the companion paper to the present report [3], the most important interaction noise source was determined to be between the IGVs and rotor 1. These blade rows have counts of 17 and 20 respectively. This interaction leads to strong interference patterns of orders 3 and -14, with the negative sign indicating rotation in the opposite direction from the rotor. The patterns of 3 and -14 are even apparent by inspection of Fig. 2. The approach shaft speed is 4534 RPM and for lateral it is 6673 RPM. These correspond to blade passing frequencies of 1511 Hz and 2224 Hz respectively.

The output of Actran iTM is a table of radial and azimuthal duct mode complex pressure amplitudes, including upstream and downstream traveling components. These complex pressures are used at the boundary conditions for the subsequent acoustic simulations. The amplitude of the complex pressures is shown as a bar graph in Fig. 3. The amplitude of the duct modes will linearly affect the amplitude of the radiated sound while the modal distribution of acoustic energy will affect the directivity radiation pattern of the sound.

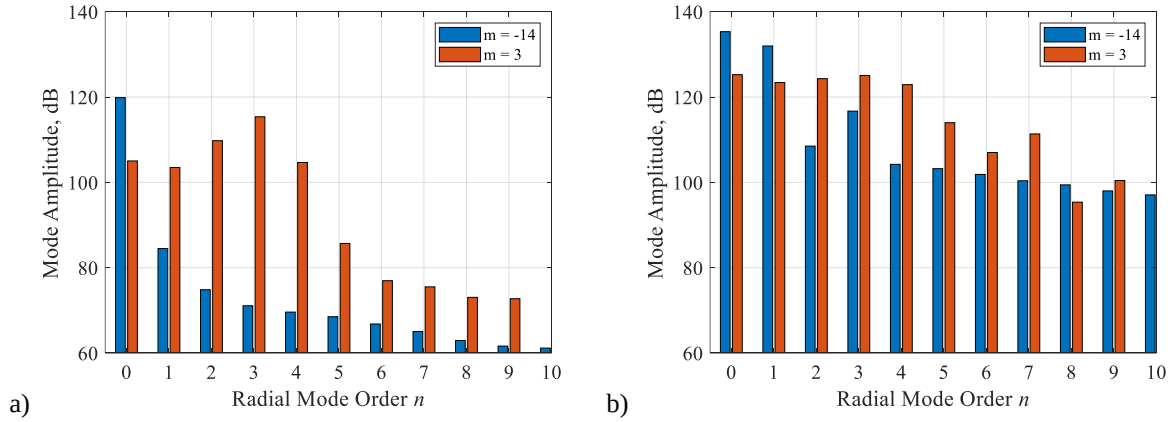


Fig. 3 Duct mode amplitudes for a) approach and b) lateral.

The turbomachinery noise at approach condition is a combination of five significant $m = 3$ modes plus one $m = -14$ mode. On the other hand, at lateral condition, the two $m = -14$ modes are substantially louder than the $m = 3$ but mode (-14,2) is cut-off and (-14,1) is close to being cut-off. The lower order modes propagate closer to the axis of the turbomachine while the higher order modes propagate closer to perpendicular to the axis, especially if they are close to cut-off.

IV. Flow Solution

The flow field for the inlet was computed using the NASA code FUN3D [7]. This code is currently popular as the “U” stands for unstructured, which is needed for complicated geometries that are difficult to mesh using structured grid methods. The code is also computationally efficient when distributed across a large number of CPU cores. The mesh was made using Pointwise [8] and contained 28.4 million nodes. The solution for each set of boundary conditions required around 20 hours on a 32 core AMD Threadripper workstation. The computational domain was roughly cylindrical, with the free-field inlet and outlet measuring 30 and 42 meters, respectively, and a length of 30 meters for an inlet duct diameter of 1.2 m. The computational domain is shown in Fig. 4. The steady RANS was run using the Menter SST two-equation model [7] with the vorticity source term.

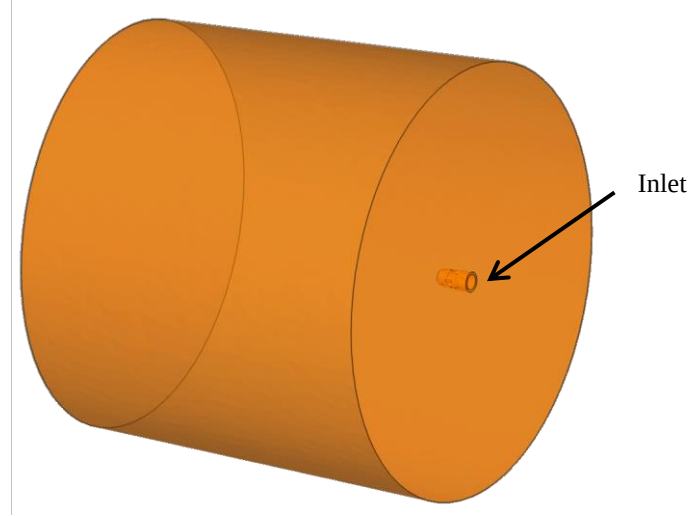


Fig. 4 Computational domain for FUN3D simulation.

Six cases are investigated in this study and outlined in Table 1. This is a subset of all the cases investigated by the team supporting the CST IPA research and was selected as representing the operating conditions required for noise certification.

Table 1 CFD runs and conditions.

<i>Run #</i>	<i>Condition</i>	<i>AoA (α), degrees</i>	<i>Mach Number</i>	<i>Struts & Actuators</i>
1	Approach	5.2	0.261	Yes
2	Approach	0.0	0.261	Yes
3	Lateral	6.7	0.350	Yes
4	Lateral	0.0	0.350	Yes
5	Approach	0.0	0.261	No
6	Lateral	0.0	0.350	No

Results from the FUN3D simulation were examined for flow features. The flow velocity in the axial direction is plotted for each case and streamlines are provided around the nacelle. Additionally, a cut-plane in the normal to the inlet after the center spike and upstream of the inlet guide vanes at an axial distance of -1.3081 m. The design Mach number for approach and lateral is used for the free-stream flow properties. The static pressure ratio is used as input to define the presumed flow rate at the outlet plane inside the duct. Results are presented in Fig. 5 for approach cases 1 and 2.

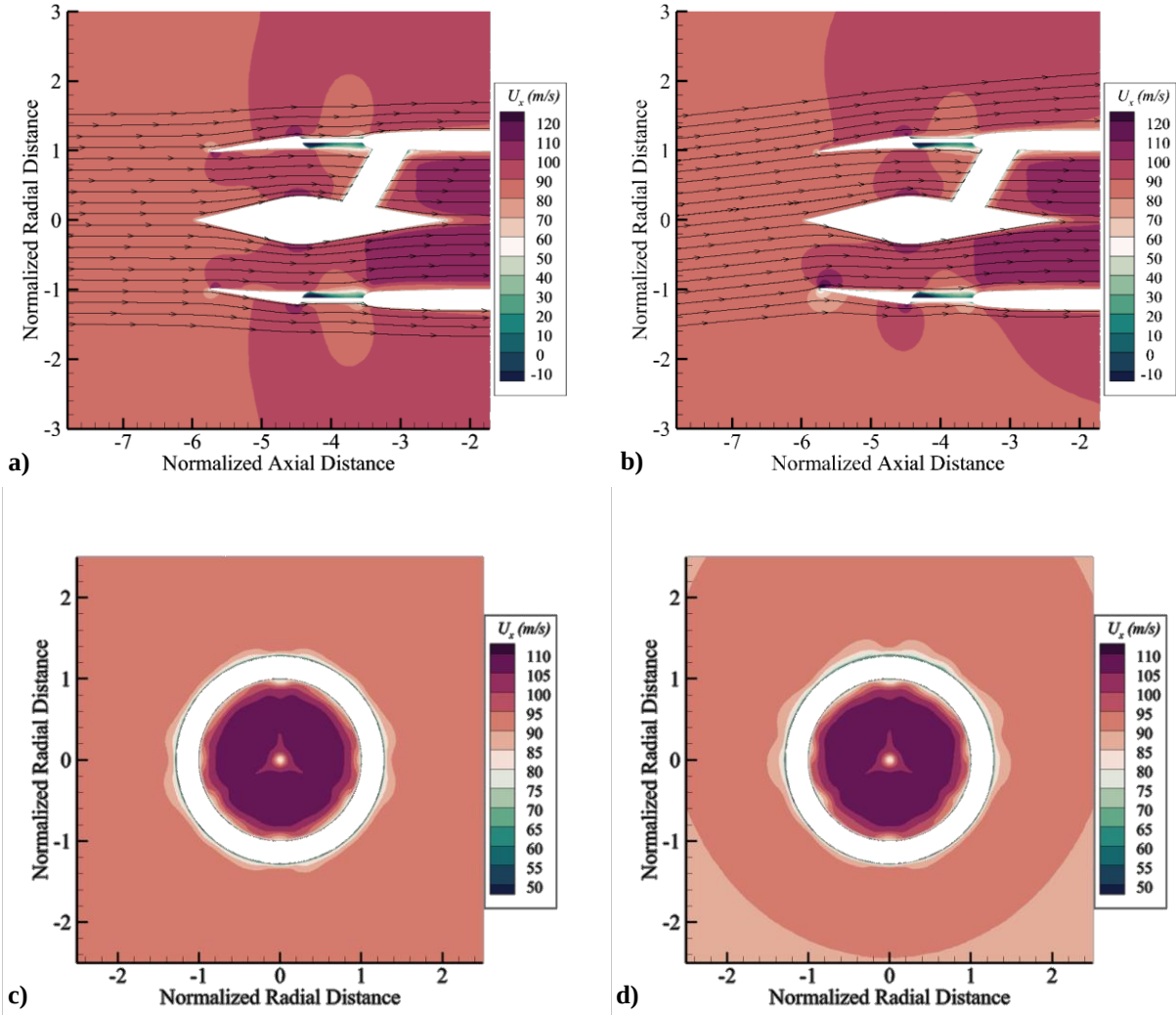


Fig. 5 Velocity contour of the approach case with (a) no angle of attack and (b) at angle of attack using an on-axis cut plane and at no angle of attack (c) and at $\alpha=5.2^\circ$ (d) using a cut plane upstream of the fan.

The flow with and without an angle of attack shows very similar flow characteristics, both inside and outside of the nacelle. When the inlet is at angle of attack, the streamlines show that some of the flow near the lower part of the nacelle inlet is diverted into the duct. Despite the seemingly sharp corner of the inlet cowl, the flow quickly reattaches and there is no large-scale separation. The flow rate inside the duct upstream of the fan stacking axis for both cases is very similar and symmetrical, providing very similar inflow conditions. Both runs show that the flow is largely attached through the inlet. There is very little distortion indicated inside of the engine nacelle on Fig. 5d, if any when compared to Fig. 5c due to the approach angle of attack. This is evidently a benefit of the long inlet. Results are presented in Fig. 6 for lateral cases 3 and 4.

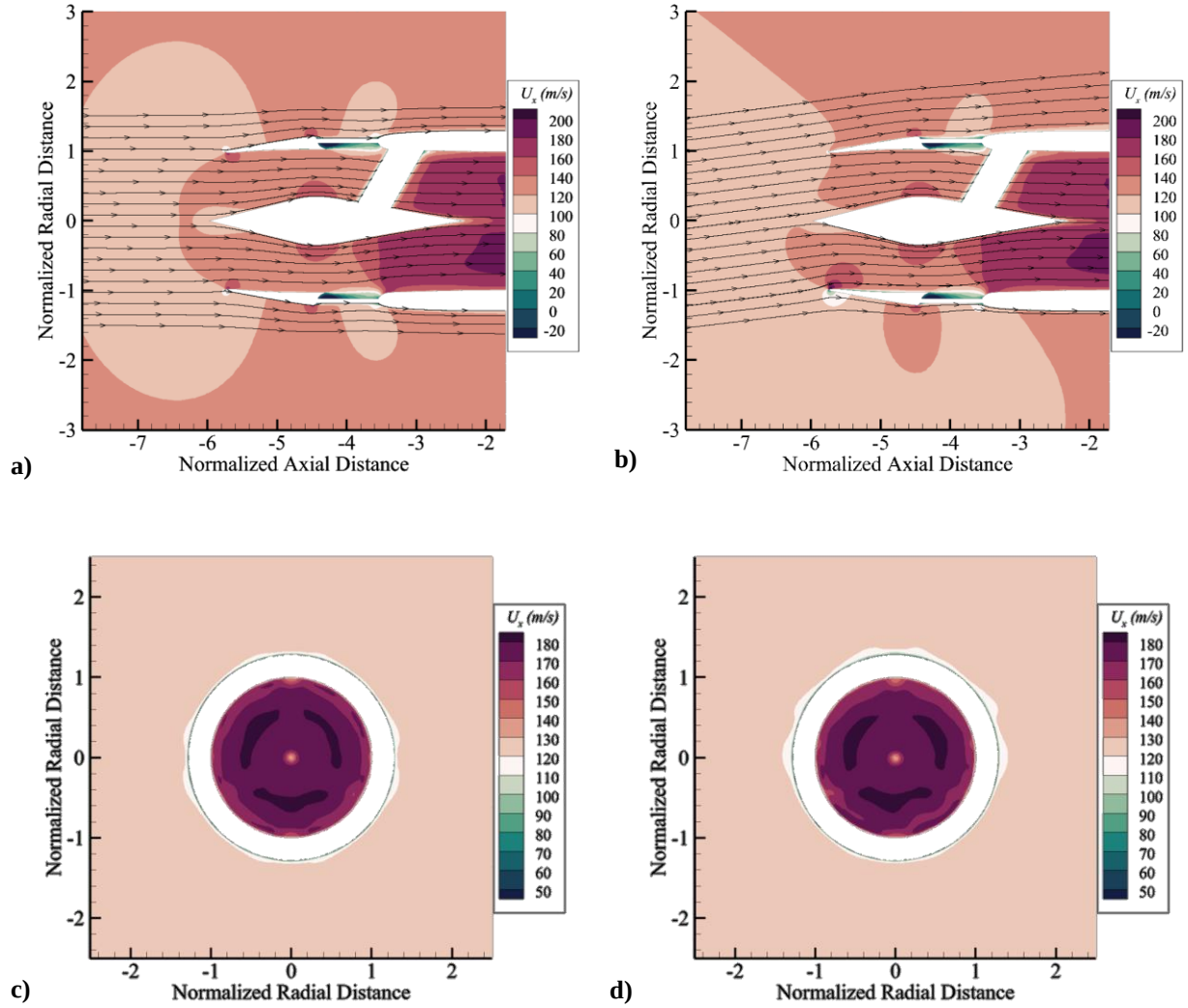


Fig. 6 Velocity contour of the lateral case with (a) no angle of attack and (b) at angle of attack using an on-axis cut plane and at no angle of attack (c) and at $\alpha=6.7^\circ$ (d) using a cut plane upstream of the fan.

Due to the increase in freestream velocity and a more drastic angle of attack, the differences in the axial flow between cases 3 and 4 are much more noticeable. There is a much larger gradient on the inlet lip on the lower part of the nacelle, resulting in more changes in the velocity in the lower half of the duct. However, on the upper half of the nacelle, the flow is much more uniform as it progresses down the duct due to the angle of the inlet spike and outer cowl. Both lateral cases display a non-uniform flow upstream of the fan stacking axis. Even at 6.7 degrees of angle of attack for lateral condition, corresponding to the aircraft climbing after takeoff, there is no significant flow separation. Similar to the previous case, there is very little distortion indicated inside of the engine nacelle on Fig. 6d, if any, when compared to Fig. 6c due to the lateral angle of attack. A pattern of the three strut wakes is also evident. Results are presented in Fig. 7 for axisymmetric Runs 5 and 6 where both actuators and struts have been removed.

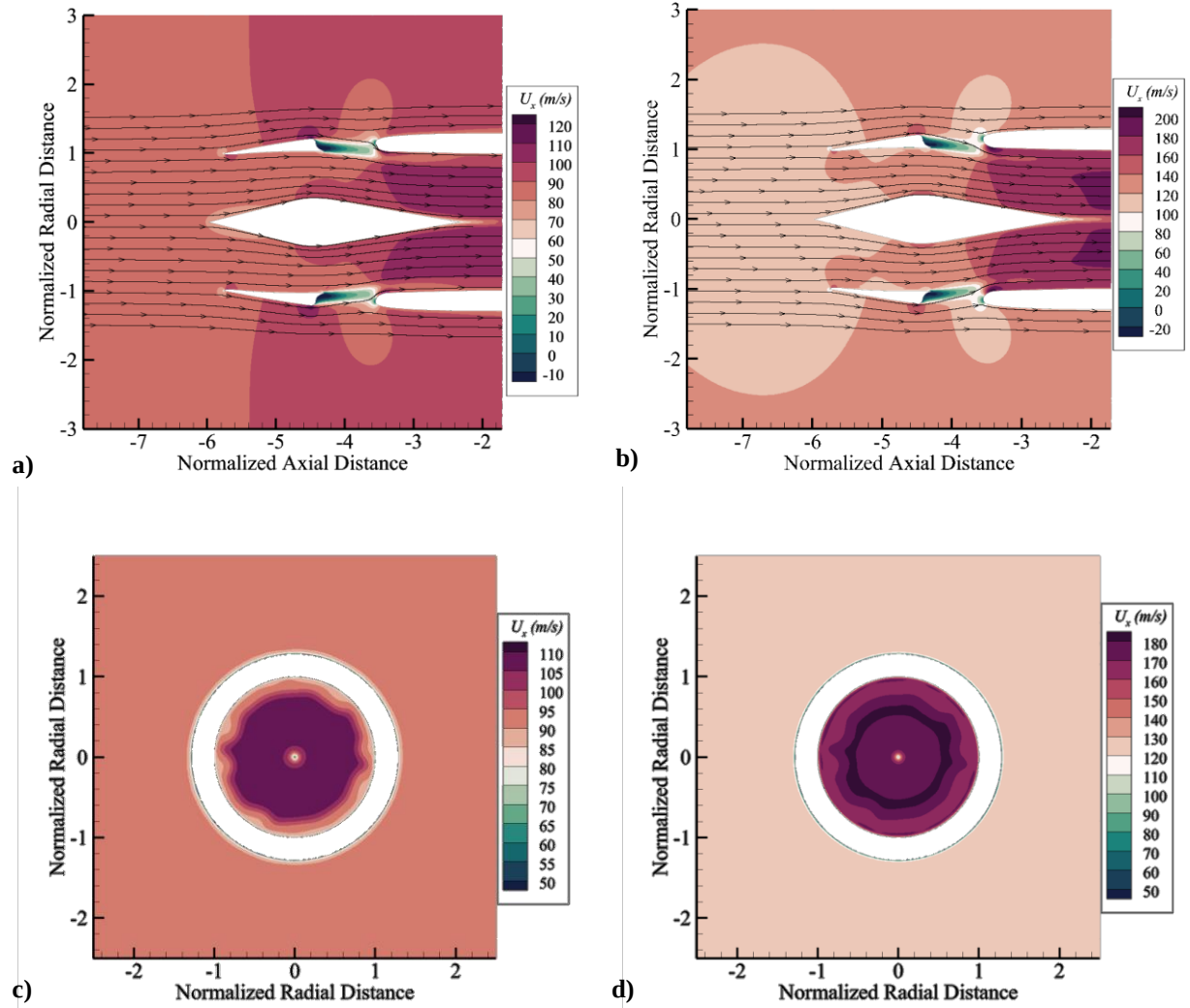


Fig. 7 Velocity contours for the axisymmetric approach (a) and lateral (b) cases depicted using an on-axis cut plane with no angle of attack. The approach (c) and lateral (d) cases using a cut plane upstream of the fan.

The axisymmetric cases display a very symmetric flow inside and outside of the nacelle. The axisymmetric approach solution shows that the flow upstream of the fan stacking axis is very similar to that of runs 1 and 2. The similarities of the inflow may indicate that there might not be a significant change in the acoustic solution. The axisymmetric lateral solution upstream of the fan stacking axis does differ from runs 3 and 4. The lower half of the nacelle in these cases is very similar to run 6. This may indicate nonuniformities in the acoustic solution depending on the observed angle. Like the previous four-run cases, the axisymmetric runs show that the flow is largely attached through the inlet with an exception downstream of the forward cowl. In all cases, there is some distortion from the centerbody spike, but it is not a strong effect.

V. Acoustic Solution

The acoustic domain is sized for solution on a desktop workstation with dual Intel Xeon 8-core CPUs and 1 TB RAM. The present results are from an Actran direct frequency response (DFR) frequency domain solution method. The DFR method requires four quadratic elements per convected wavelength, defined as $\lambda = \frac{c-v}{f}$, where λ is the convected wavelength, c is the speed of sound, v is the local velocity, and f is the frequency. The current results are presented for a uniform grid spacing, based on the smallest necessary length, which was around 1.9 cm. Initial attempts to perform the simulations using a full 3D simulation topology revealed that our computing resources were not quite

adequate for the size of domain required. Improvements in domain size are possible by using the symmetric-antisymmetric method in Actran which allows simulation on a half-domain, so this method was chosen. This is compatible with the present inlet geometry where the top strut is centered along the plane of symmetry. The three-dimensional representation of the acoustic domain is shown in Fig. 8a. The finite element domain has a 1.22 m (48 inch) radius and about 2.9 million nodes with 1.2 million quadratic elements. Larger domains could be solved using single precision arithmetic, which reduces memory requirements, but this was found to affect the solution. The results presented here were computed using the default double precision method with block low-rank factorization set to auto. The physical extent of the domain for the 3D computation is about 1/3 the size used for the 2D axisymmetric results presented previously [2]. The finite element domain uses the infinite element boundary condition for the spherical surface mesh. This is used to affect a non-reflecting boundary condition and also allow propagation of the sound to far-field points. These far-field points were arranged in 12 arcs of 1000 points from directly upstream to 150 degrees from upstream. The arcs are spaced azimuthally at 30-degree intervals and the radius is 46 m (150 feet). Since the eventual output of the present activity is flyover noise, the bottom center arc is highlighted in red in Fig. 8b.

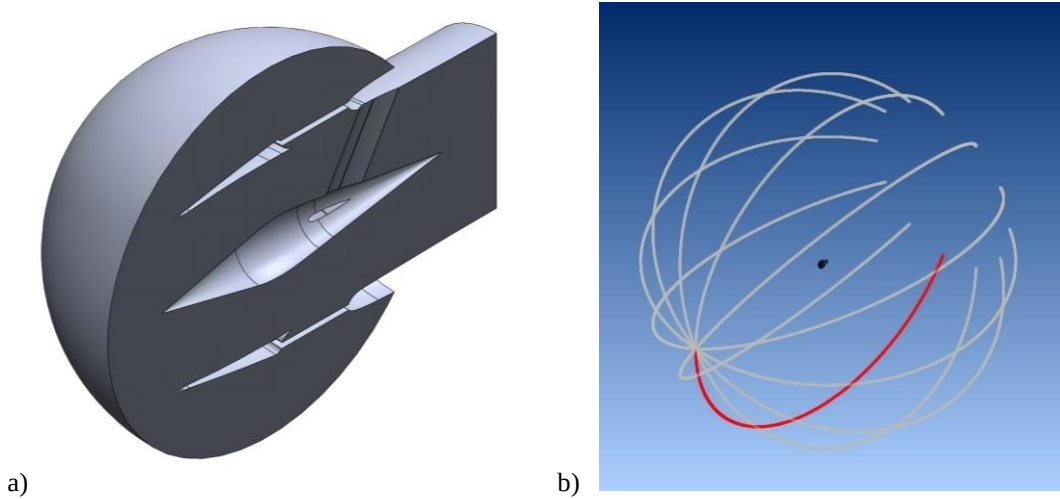


Fig. 8 Acoustic finite element domain for symmetric-antisymmetric solution method. a) CAD model and b) Far-field points around infinite element domain, bottom arc highlighted.

The acoustic solution followed the Actran workflow for a symmetric-antisymmetric inlet. The flow field was calculated using the steady RANS discussed previously. The flow field parameters of velocity, pressure, and density were interpolated onto the acoustic mesh using the Actran iCFD tool. Small pockets of supersonic flow were identified in the lateral and AoA conditions, these were cropped to Mach 0.9 to improve compliance with the assumption of the Möhring convected wave equations. Example results of the acoustic propagation through the domain are given in Fig. 9.

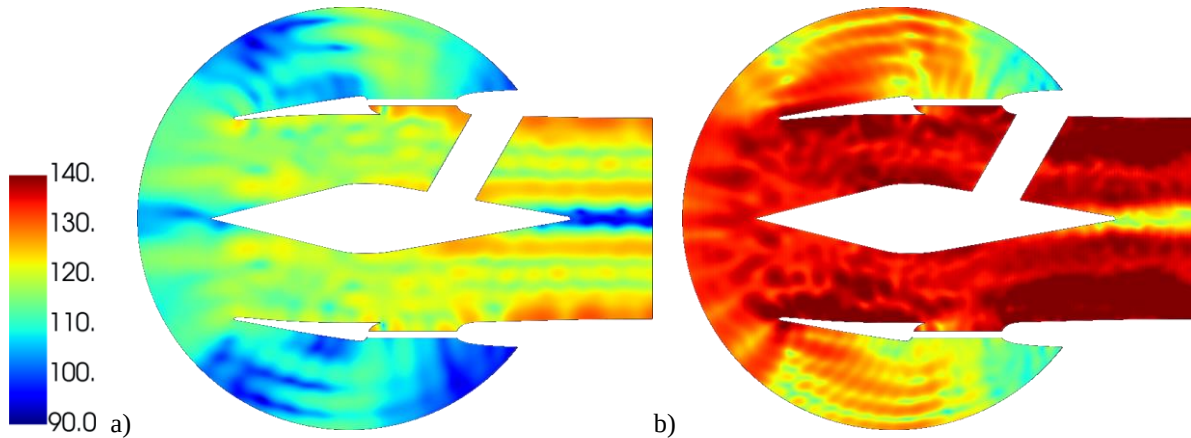


Fig. 9 Pressure map in decibels for a) approach at AoA and b) lateral at AoA.

VI. Importance of 3D Effects

The end goal of these aircraft engine noise simulations is to predict the sound experienced by an observer on the ground. With the simulation workflow shown here, this can be done by extrapolating from the arc arrays shown in Fig. 8 b) to the observer locations. The radiated sound directivity will be discussed by considering the pressure levels observed on the 12 arcs of far field observers and how it changes with the different simulation cases. A simplistic way to compare the six simulation results is to examine the directivity averaged over all 12 of the arcs, giving one sound directivity curve for each run. This is shown in Fig. 10 a) for the three approach condition and b) for the lateral condition. In both cases, the axisymmetric configuration stands out slightly from the other two.

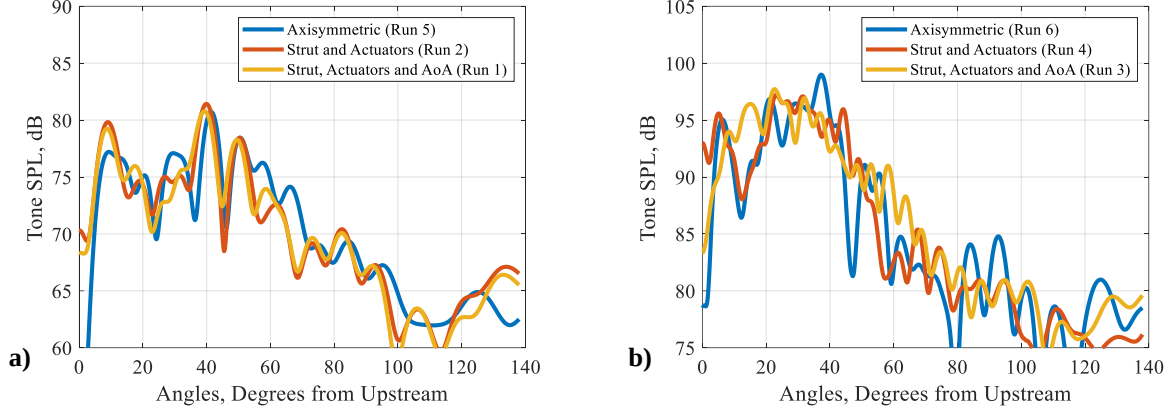


Fig. 10 Average directivity over all azimuthal angles. a) Approach, b) Lateral.

A more focused comparison is to consider only the downward radiated sound, calculated as the average of the bottom center arc and the ones ± 30 degrees azimuthally to each side. These are shown in Fig. 11, which is admittedly extremely similar to Fig. 10. The overall shapes of the directivity are largely unchanged, and the sound power is conserved in this simulation so for the present inlet geometry and operation conditions, it seems like the radiated sound is largely insensitive to the inlet angle of attack and 3D features.

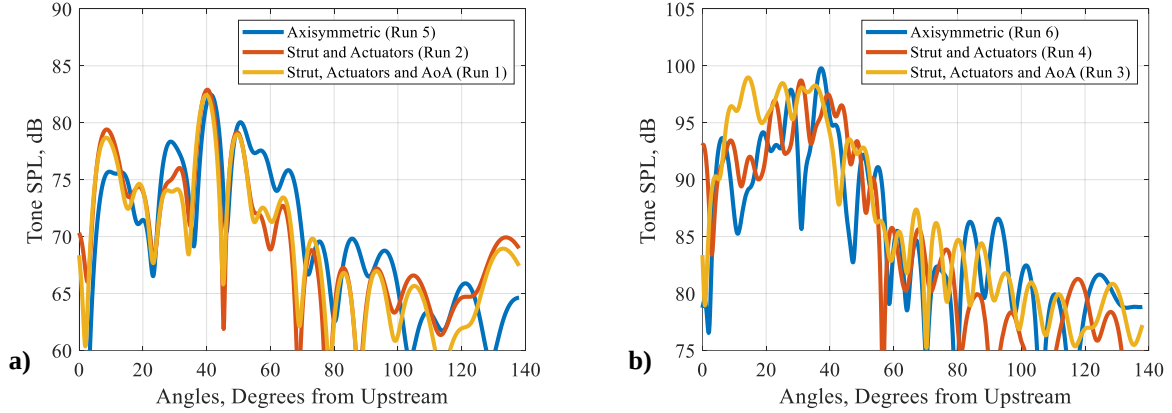


Fig. 11 Average directivity over bottom three azimuthal angles. a) Approach, b) Lateral.

To close the loop on the simulations, a revisit of the 2D axisymmetric solution is merited. The results from the present axisymmetric 3D simulations are compared against the 2D axisymmetric solutions with a potential flow field presented in our paper from last year [2]. For the 3D axisymmetric cases presented here, the geometry is identical and the remaining difference between the two simulations is the flow field. In the present work, the flow field was viscous RANS while compressible potential flow was used in the prior work. Flyover directivity for approach and lateral cases is shown in Fig. 12. The results are qualitatively similar for both flow conditions but the decibel difference for individual angles is quite substantial compared to the effects of the struts and the angle of attack. A side computation (not shown) demonstrated that identical results were found between 2D and 3D inlets with no flow, so the mathematical implementation in Actran is consistent. Further investigation of the results is warranted but outside of

the present scope of work. Until that is completed, the authors concede that using the 2D simulations is extremely efficient but spot-checks with full 3D solutions are needed.

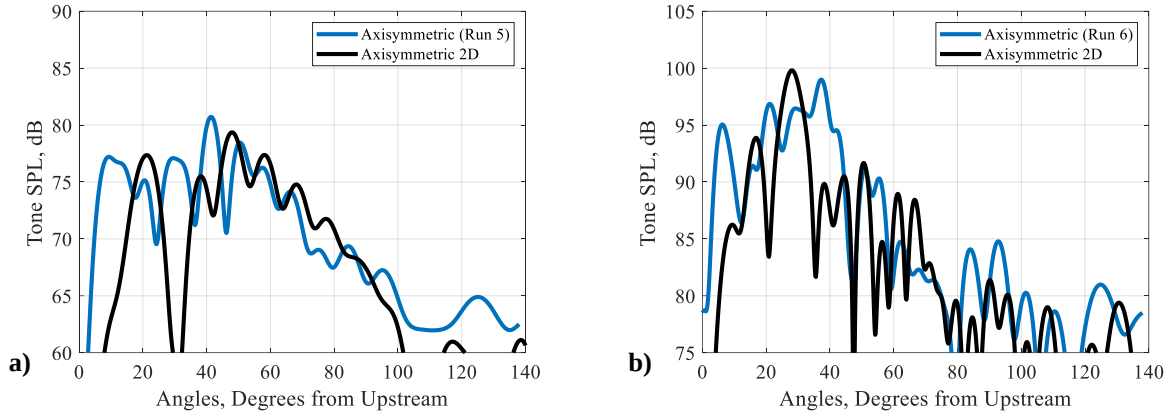


Fig. 12 Two-dimensional vs three-dimensional computation. a) Approach, b) Lateral.

Looking at the results in Fig. 10 to Fig. 12 gives a qualitative comparison of the effect of the different cases on the radiated sound. An overall comparison between a pair of cases was computed by subtracting the directivity curves and taking the standard deviation, which results in a single number expressing how much the two curves match. This calculation was limited to angles between 40° and 120° , as these angles are relevant for flyover noise computations. Ten total comparisons were made, presented in Table 2. Conclusions drawn from this table follow reasonably closely the observations given above. The effect of adding struts and actuators is larger than the effect of angle of attack. The approach case in 2D potential flow agrees better with the 3D viscous flow while the lateral case shows a greater effect.

Table 2 Standard Deviation of Directivity Differences

	Approach	Lateral
Effect of Struts and Actuators, dB. (Ref. Fig. 10)	2.0	3.7
Effect of Angle of Attack, dB. (Ref. Fig. 10)	1.2	2.9
Effect of Struts and Actuators (bottom), dB. (Ref. Fig. 11)	4.4	5.9
Effect of Angle of Attack (bottom), dB. (Ref. Fig. 11)	1.9	4.3
Effect of Viscous Flow Field, dB. (Ref. Fig. 12)	3.4	4.7

VII. Conclusion

The sound propagation from an inlet designed for a commercial supersonic engine was demonstrated. The source sound was computed using a URANS method presented in a companion paper while the present work focused on the effects of inlet flow and geometry on the radiated noise. The inlet in the present study was shown to well tolerate the range of flow parameters, giving attached flow nearly throughout the domain and providing an essentially uniform fan face pressure field. The transition from an annular to a circular duct may contribute to this effect. The long length of the inlet certainly improves the flow, and the rather modest angles of attack are not harmful. The acoustic simulation was for tone noise and linear acoustics required that the sound entering and exiting the domain were equal. With these constraints, it is perhaps not surprising that a fairly small impact on the radiated sound was noted. If resources are limited, the 2D inlet solution method is extremely practical and gives a good answer for the cases shown.

A few recommendations for future work are also identified. A more computationally efficient mesh could be made since the highest speed flow only occurs at certain areas of the domain. Doing this is complicated, as the flow velocity changes throughout the domain and is also dependent on the operating condition. An automated method for generating an improved mesh is being pursued. An alternative computational solution is to use the Actran Discontinuous Galerkin Method (DGM) which is a time domain method and handles shear flow. The acoustics DGM requires only 1.5 points per convected wavelength, which enables simulation with a much smaller memory footprint. This method is computationally expensive, but it is efficient to parallelize.

Acknowledgments

This work was funded by the NASA Commercial Supersonic Technology project.

References

- [1] J. W. Slater, "SUPIN: A Computational Tool for Supersonic Inlet Design," in *SciTech Forum*, San Diego, California, USA, 2016.
- [2] D. B. Stephens, A. A. Svetgoff and E. Envia, "Acoustics of a Supersonic Mach 1.4 Axisymmetric Spike Inlet (NASA TM-20230006147)," in *AIAA AVIATION 2023 Forum*, San Diego, California, USA, 8 June 2023.
- [3] E. Envia, "Prediction of Inlet Tone Noise from Two-Stage Fans," in *30th AIAA/CEAS Aeroacoustics Conference*, Rome, Italy, 2024.
- [4] E. Envia, "Open Rotor Aeroacoustic Modelling," NASA/TM-2012-217740, 2012.
- [5] D. B. Stephens, "Noise Correction for Supersonic Inlet Geometry (TM-20220007033)," NASA Glenn Research Center , Cleveland, Ohio, United States, November 2022.
- [6] F. F. T. SA, "ACTRAN 2021 User's Guide," MSC Software, Mont-Saint-Guibert, Belgium, 2023.
- [7] NASA, "FUN3D Manual: 14.0.1," NASA/TM-20230004, 2023.
- [8] Cadence, "Pointwise User Manual," 2023.
- [9] W. Ng, "Comparison of the Aeroacoustics of Two Small-Scale Supersonic Inlets," NASA CR-96-206507, January 1996.