

Auxiliary Inlet Design Study for Mach 1.4

Layton W. Howerton¹ and John W. Slater²

NASA John H. Glenn Research Center, Cleveland, Ohio 44135

Auxiliary inlet configurations were developed for inlets designed for a NASA Mach 1.4 Supersonic Technology Concept Aeroplane (STCA). The inlets included axisymmetric pitot and axisymmetric spike inlets. The auxiliary inlets were full circumferential slots within the cowl. Computational fluid dynamics (CFD) simulations were performed to obtain inlet performance metrics of total pressure recovery and radial distortion at the engine face for take-off and approach conditions. Methods of design of experiments were used to explore the statistical significance of the design factors for the auxiliary inlets which included the axial location, length of the opening, and auxiliary inlet angle. The results demonstrate the effect of the design factors and show the potential for improved inlet performance with the use of auxiliary inlets.

Nomenclature

A	=	Area
D	=	Diameter
h	=	Altitude
L	=	Length
M	=	Mach number
q	=	Dynamic Pressure
p	=	Pressure
T	=	Temperature
W	=	Flow rate
α	=	Angle-of-Attack
ϕ	=	Auxiliary inlet angle
x, y, z	=	Cartesian coordinates
$()_0$	=	Freestream property
$()_2$	=	Property at the engine-face station
$()_{aux}$	=	Property of auxiliary inlet
$()_{SD}$	=	Property at the subsonic diffuser station

I. Introduction

The success of commercial supersonic aircraft will depend on the good performance of the various subsystems of the aircraft. Within the propulsion system, the inlets should provide the required amount of high-quality airflow to the turbofan engines at all operating conditions of the aircraft. In this paper, we examine inlets for the NASA Supersonic Technology Concept Aeroplane (STCA) [1] designed as a 55-ton business aircraft capable of cruise at Mach 1.4. Of special focus will be the aerodynamic performance of the inlets at the take-off and approach conditions and the effects of auxiliary inlets to provide additional airflow to the engines at this condition.

Section II will discuss the freestream and engine face conditions of the STCA that provide the boundary conditions for the design of axisymmetric pitot and axisymmetric spike inlets. The inlets were designed using the Supersonic Inlet Design and Analysis (SUPIN) tool [2]. The auxiliary inlets were modeled with design factors specifying the location, size of the area opening, and surface angles. The modeling of the inlets and the auxiliary inlets was also performed within SolidWorks. Section II will also discuss the design and modeling of the inlets with the auxiliary inlets.

¹ Pathways Intern, Propulsion Division / Inlets and Nozzles Branch, AIAA Student Member.

² Research Aerospace Engineer, Propulsion Division / Inlets and Nozzles Branch, AIAA Associate Fellow.

Section III will discuss the computational methods for the studies. The methods included a quasi-one-dimensional analysis of the inlet flow rates with the auxiliary inlets. Computational fluid dynamics (CFD) methods are discussed for the analysis of the flow about and through the inlets. The methods were facilitated using the SolidWorks Flow Simulation (SWFS) [3] and Wind-US [4] software. The CFD solutions were processed to obtain the inlet performance metrics as characterized by the inlet flow rates, total pressure recovery, and radial total pressure distortion. To examine the significance and interactions of the various design factors for the auxiliary inlets, methods of design-of-experiments (DOE) were used for statistical analysis of the factors and building of surrogate models for the inlet performance with respect to the design factors.

Section IV will discuss the results of the CFD simulations and DOE analyses for the inlets. The results provide design information for the inlets of this study, as well as, for inlets being designed for similar commercial supersonic aircraft.

II. Inlets and Auxiliary Inlets

The inlets were designed for the conditions of the NASA STCA, which is described in the first subsection. The second subsection discusses the design of axisymmetric pitot and axisymmetric spike inlets using the SUPIN tool. Configurations for the auxiliary inlet and its associated design factors are discussed in the third subsection.

A. Supersonic Technology Concept Aeroplane

The STCA was developed by NASA to explore several technology goals for commercial supersonic flight [1]. Figure 1 shows an image of the STCA that features three turbofan engines with two engines mounted on the side of the rear fuselage and one engine mounted within the vertical stabilizer. The image shows axisymmetric spike inlets.

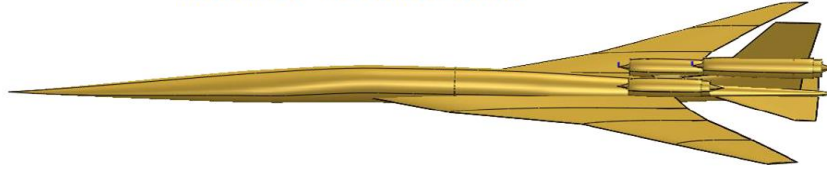


Figure 1. Image of the STCA.

The mission profile for the STCA established the speed and altitude of the aircraft. These provided the freestream conditions that define the state of the uniform flow approaching the inlets. The inlets were assumed to be isolated with the freestream designated as station 0. The freestream conditions provided the upstream boundary conditions for the inlet designs. The freestream was characterized by the Mach number (M_0), altitude (h_0) or dynamic pressure (q_0), and angle-of-attack (α_0). The Standard Atmosphere was used to find the static pressure and temperature that defined the thermodynamic state of the freestream for calculation of the freestream total pressure (p_{t0}) and temperature (T_{t0}). Table 1 lists the freestream conditions across the Mach number range. The take-off condition is at $M_0 = 0.25$. The approach condition is at $M_0 = 0.30$. The cruise condition is at $M_0 = 1.4$.

Table 1. Freestream conditions and turbofan engine-face corrected flow rates and Mach numbers.

M_0	h_0 (ft)	q_0 (psf)	p_{t0} (psf)	T_{t0} (°R)	W_{C2} (lbm/s)	M_2
0.00	0	0.00	2116.23	518.67	424	0.69976
0.25	0	92.59	2210.27	525.15	420	0.68592
0.30	0	133.32	2252.58	528.01	419	0.68256
0.40	5000	197.21	1966.03	516.87	424	0.69976
0.50	15000	209.00	1416.67	488.44	427	0.71062
0.60	20000	245.07	1240.43	479.56	427	0.71062
0.70	30000	215.55	871.71	452.03	427	0.71062
0.80	35000	223.09	759.06	444.27	427	0.71062
0.90	40000	222.08	662.45	453.14	427	0.71062
1.00	45000	215.61	583.04	467.96	427	0.71062
1.10	48000	225.85	569.34	484.34	427	0.71062
1.20	49000	256.17	616.28	502.28	427	0.71062
1.30	50000	286.54	671.11	521.78	423	0.69623
1.40	50000	332.32	770.79	542.84	413	0.66323

The turbofan engine conceptualized for the STCA was developed based on publicly available data related to the CFM International CFM56-7B engine. Details of the engine configuration are discussed in Reference [1]. Figure 2 shows images of the engine models for the CFM56 and the STCA turbofan engines. The engine face is designated as station 2 and modeled as an annular cross-section with an engine-face diameter of $D_2 = 3.625$ feet and a spinner for the hub with a hub-to-tip ratio of $D_{hub}/D_2 = 0.3$. The spinner is modeled with an elliptic profile with an aspect ratio of length-to-diameter of 2.0. The cross-sectional area of the engine face (A_2) is the annular area formed by the circular engine face and spinner hub diameter and is $A_2 = 9.3918$ ft². The engine flow rate is specified by the engine-face corrected flow rate (W_{C2}), which is set by the desired level of thrust at a given point in the mission profile. The engine-face corrected flow rate corresponds to an engine-face mass-averaged Mach number (M_2). Both the engine-face corrected flow rate and Mach number are listed in Table 1 for each of the freestream Mach numbers. The engine-face geometry and flow rates provide the downstream boundary conditions for the inlet designs.

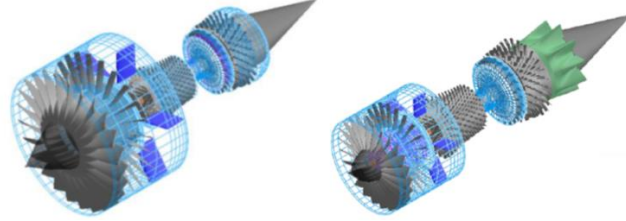


Figure 2. Images of the models for the CFM56 (left) and STCA (right).

B. Inlet Designs

The inlets of this study included axisymmetric pitot and axisymmetric spike inlets designed as external-compression supersonic inlets using the $M_0 = 1.4$ cruise freestream and engine-face conditions as listed in Table 1. The inlet designs were created using the SUPIN tool [2]. SUPIN is a Fortran 95 program that reads in a text-based input data file that provides the values of the design factors. SUPIN uses analytic, empirical, and computational methods to design the inlet and estimate the flow rates, total pressure recovery, and drag for the inlet. SUPIN generates surface grids for the inlet and writes a Plot3D file [5]. SUPIN can also automatically generate a multi-block, structured grid for a flow domain about the inlet for flow analysis using CFD methods.

The axisymmetric pitot inlet provided the simplest inlet and served as a baseline for inlet performance. Figure 3 shows images of the axisymmetric pitot inlet. The inlet had an essentially sharp cowl lip modeled with an elliptical profile with a thickness of 0.002 feet. The cowl interior provided a gradual diffusion from the cowl lip to the engine face over a distance of one engine-face diameter. SUPIN estimated the total pressure recovery of $p_{t2}/p_{t0} = 0.9557$ with most of the total pressure loss being due to the normal shock with $p_{t1}/p_{t0} = 0.9582$. Station 1 was designated as the cowl lip plane.

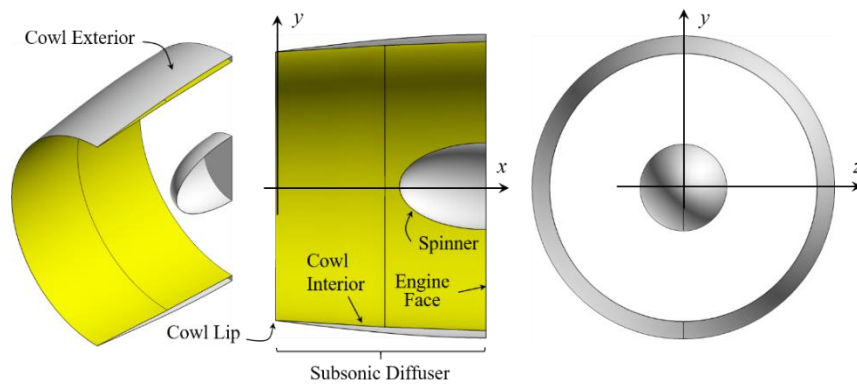


Figure 3. Axisymmetric pitot inlet.

The axisymmetric spike inlet introduced a centerbody spike to an axisymmetric cowl. Struts were needed to support the centerbody. Figure 4 shows images of an axisymmetric spike inlet. The spike consists of a single cone with a conical half-angle of 13 degrees. The spike angle was also chosen to match the conical spike to a centerbody with a uniform diameter within the inlet matching the engine face hub. The constant-diameter centerbody was the result of very little diffusion needed for the subsonic diffuser. The spike resulted in the external supersonic

compression to $M_{EX} = 1.3$ just upstream of the terminal shock located at the cowl lip plane. The conical flow about the spike results in a flow angle of 1.7 degrees at the cowl lip. The cowl lip interior angle matched this local flow angle and allowed a cowl lip exterior angle at the start of the cowl exterior of 7.0 degrees. Four support struts were used for the inlet. The length of the cowl was approximately equal to the engine face diameter, D_2 . SUPIN estimated the total pressure recovery of $p_{t2}/p_{t0} = 0.9761$.

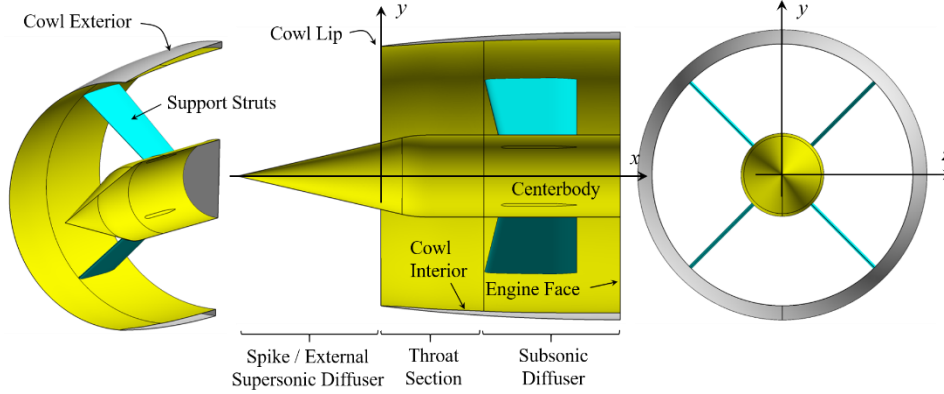


Figure 4. Axisymmetric spike inlet.

A few options existed for importing the inlet geometry into the computer-aided design software SolidWorks. The first option suitable for the axisymmetric inlets was to import a list of planar coordinates for the profiles of the cowl interior, cowl exterior, and spike centerbody. These profiles were converted to curves and then extruded about the axis-of-symmetry to form the solid models for the centerbody and cowl. For the axisymmetric spike inlet, planar curves could also be imported to define the support struts. The second option was to import the Plot3D file for the inlet surface grid into Pointwise and then export an IGES or STL file from Pointwise which could be imported into SolidWorks. A third option was to export the Plot3D file from Pointwise to SolidWorks and rebuild the model within SolidWorks.

C. Auxiliary Inlets

The purpose of the auxiliary inlets is to provide additional passages for external airflow into the inlet at take-off and approach-to-landing conditions for which the engine airflow demand is greater than the freestream capture airflow rate of the inlet. In this case, the internal inlet pressures are lower than the freestream and the engine is drawing airflow into the inlet. The modeling approach of the auxiliary inlets for the inlets of this study was to provide an axisymmetric slot through the cowl of the inlet, as shown in Figure 5. The auxiliary inlets were created in a SolidWorks sketch. Inside the sketch, parallel lines cutting through the cowl in the correct orientation defined the sidewalls of the auxiliary inlet. Rounded corners one-quarter the thickness of the cowl were added to the interior and exterior surfaces of the cowl. The design factors included the axial position of the center of the auxiliary inlet (x_{aux}), the axial length of the slot (L_{aux}), and the angle of the auxiliary inlet walls (ϕ_{aux}).

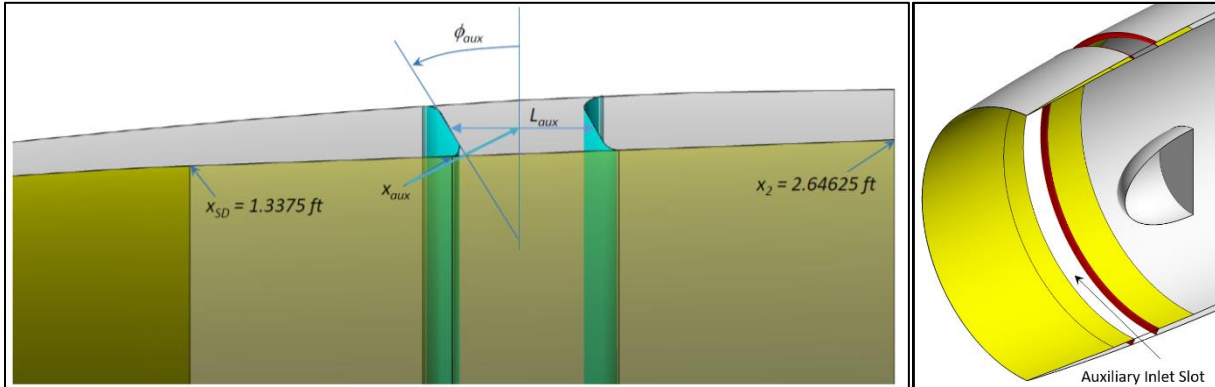


Figure 5. Auxiliary inlet design parameters (left) and auxiliary inlet for an axisymmetric pitot inlet (right).

III. Computational Methods

The computational methods used to simulate the airflow through the inlets and auxiliary inlets are described in this section. The methods include an analytical method based on quasi-one-dimensional assumptions, the SolidWorks Flow Simulation CFD software, and the Wind-US CFD software. The CFD simulations provided for the computation of the inlet performance metrics are described in this section.

A. Analytic Method

An analytic method was developed to estimate the area of the opening for the auxiliary inlets. It was based on a quasi-one-dimensional assumption of the continuity of the airflow through the inlet and auxiliary inlets. The method required specification of the desired Mach number for the inlet at station 1 and the auxiliary inlet opening and estimates for the total pressure losses through the inlet and auxiliary inlets. A simple solution method yielded the required area for the auxiliary inlet opening. The method was incorporated into a MATLAB script, as well as into SUPIN and provided a starting point for the length of the auxiliary inlet for the axisymmetric pitot inlet. The subsequent CFD simulations indicated that the analytic model underestimated the length and CFD simulations were needed to properly design the auxiliary inlets.

B. SolidWorks Flow Simulation

SolidWorks Flow Simulation (SWFS) [3] is an add-in program to SolidWorks capable of running CFD simulations for the models created in SolidWorks. For the CFD simulations, the solid models of the inlets were extended downstream. Downstream of the engine face, a constant area duct was added to move the internal outflow boundary condition downstream of the engine face by one engine face diameter. This reduced possible interference of the internal outflow boundary condition with flow properties at the engine face. A solid disk was created at the engine face plane, as shown in Figure 6, to monitor the flow properties at the engine face as part of the process for achieving the desired engine face Mach number for the simulation. The disk was hidden from the flow solver, so there was no influence of the disk on the flow solution. Downstream of the cowl exterior, a solid cylinder was added to move the external outflow boundary condition further downstream to reduce interference with the auxiliary inlet flow.

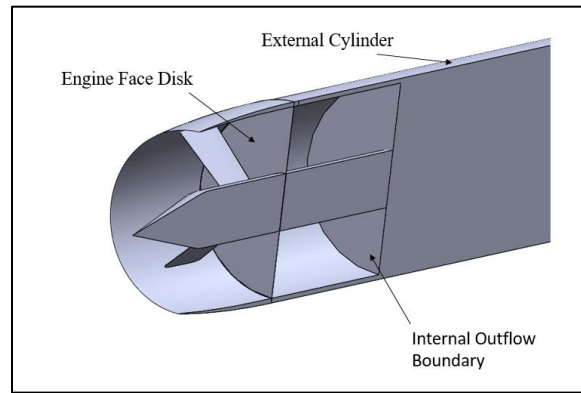


Figure 6. Additional features of the solid model for the SWFS CFD simulations.

The computational flow domain for the simulations of the axisymmetric pitot and axisymmetric spike inlets consisted of a five-degree wedge at the bottom of the inlet as shown in Figure 7. Due to the subsonic freestream conditions, the external boundaries were placed four engine face diameters from the cowl lip as to not cause interference from the boundary conditions on the internal inlet flow.

A Cartesian structured grid was generated for the computational flow domain. The grid defined for a SWFS case is categorized as global or local. There can only be one global grid, but many local grids can be generated. Local grids can be placed in a volume of space or be defined by the shape of a solid body in the model. The global grid defines the number of grid points to be placed in the finite volume of the computational domain. The global grid can also include an aspect ratio factor to create the largest cells away from the model and smallest cells nearest to the model. The local grid was used in this study to resolve the boundary layer flow near the inlets. Both local and global grids are controlled with sliders to determine the characteristics of the grid. Grid generation of the Cartesian grid is then completed for both local and global based on the combination of slider positions. Options are available in SWFS for automatic grid refinement while running SWFS cases, but this option was not used in this study.

SWFS solves the Favre-averaged Navier-Stokes equations. A perfect gas model was used for the air in the computational domain. Turbulence was modeled in the simulations using the $k-\epsilon$ turbulence model. Additionally, a wall function was used to resolve thick and thin boundary layers. A grid study was performed to refine the grid near the inlet surfaces to provide sufficient resolution for the wall functions. The freestream conditions were imposed on the inflow and external outflow boundaries. An internal outflow boundary condition was set to specify the outflow Mach number for the internal flow. The Mach number, total pressure, static pressure, and mass flow rate were monitored by SWFS at the engine face disk to determine iterative convergence of the simulations. Iterative convergence was considered achieved once stable values for all the engine face variables were achieved for one hundred iterations.

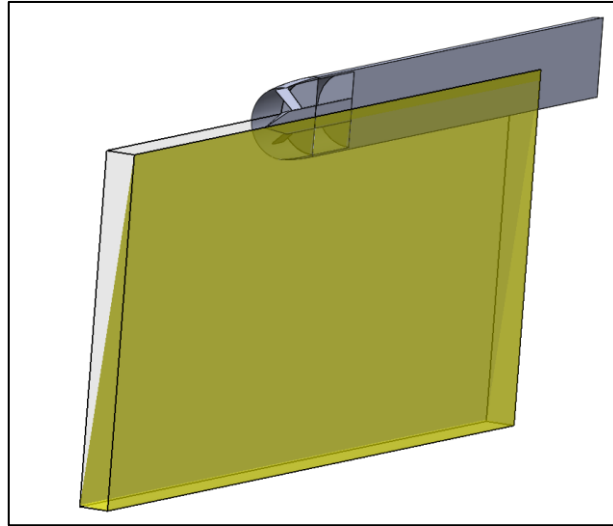


Figure 7. Computational flow domain for the axisymmetric inlets.

C. Wind-US CFD Solver

The Wind-US flow solver [4] solves the steady-state, Reynolds-averaged Navier-Stokes (RANS) equations for the flow properties at the grid points of a multi-block, structured grid defining a flow domain about the inlets. Wind-US used a cell-vertex, finite-volume representation for which the flow solution was located at the grid points and a finite-volume cell was formulated about the grid point. In Wind-US, the RANS equations were solved for the steady-state flow solution using an implicit time-marching algorithm with a first-order, implicit Euler method using local time-stepping from an initial flow solution. All the simulations were performed assuming calorically perfect air. The inviscid fluxes of the RANS equations were modeled using a second-order, upwind Roe flux-difference splitting method. The flow simulations assumed fully turbulent flow in which the turbulent eddy viscosity was calculated using the two-equation Menter Shear-Stress Transport (SST) [6] turbulence model.

Figure 8 shows the computational flow domain and boundary conditions (BC) used for the planar CFD simulations of the axisymmetric pitot inlets. The flow domain defined the control volume in which the RANS equations were solved. The internal and external surfaces of the inlet formed a portion of the boundary of the flow domain where non-slip, adiabatic viscous wall boundary conditions were imposed. The inflow, far-field, and external outflow boundaries of the flow domain had freestream boundary conditions imposed in which the Mach number, pressure, temperature, and angle-of-attack were specified. The inflow, far-field, and external outflow boundaries were positioned about five engine-face diameters from the cowl lip.

Downstream of the engine face, a constant-area outflow nozzle section was added to the flow domain to set the flow rate through the inlet. The nozzle section moved the internal outflow boundary condition downstream of the engine face, which reduced possible interference from the boundary condition on the flow at the engine face. The outflow nozzle is shown in Figure 8. The length of the outflow nozzle section was about 1.7 times the diameter of the engine face. At the internal outflow boundary, the static pressure, Mach number, or corrected flow rate was specified as the boundary condition.

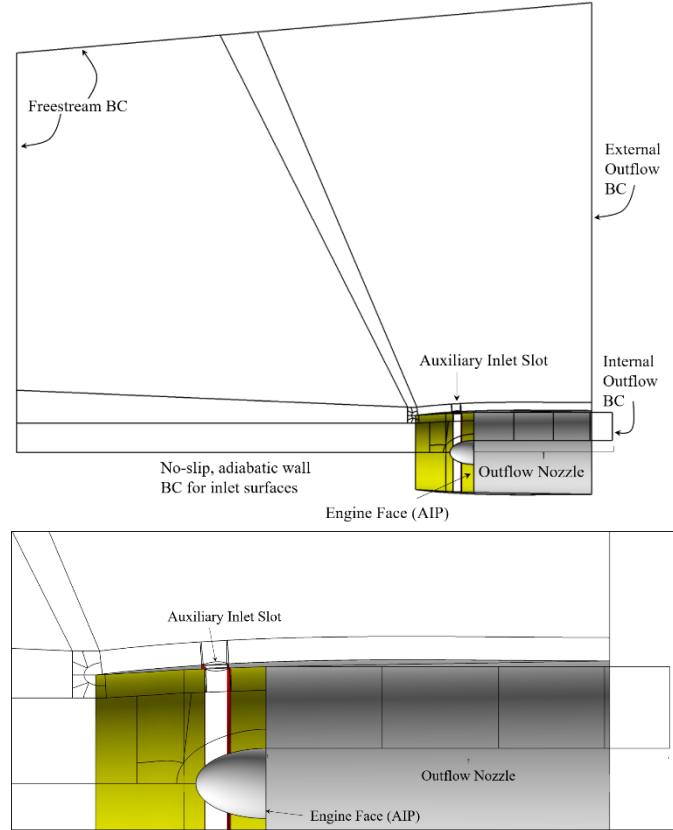


Figure 8. Flow domain and boundary conditions for the axisymmetric pitot inlet CFD simulations.

The computational grid for the flow domain was generated by dividing the flow domain into multiple blocks and generating structured grids for each block. SUPIN was used to generate the blocks and grid points using an automated process. SUPIN also created the boundary condition file for Wind-US. The inputs to the process include some factors to determine the extent of the flow domain and the resolution of the grid points. The grid resolution factors include the grid resolution of the first grid point away from the wall (Δs_{wall}) and the grid resolution within the throat section in the streamwise direction (Δs_x). The wall spacing (Δs_{wall}) was set to resolve the boundary layer with $y^+ = 1$. SUPIN then imposed these grid resolution values along the edges of the inlet geometry and flow domain to compute the required number of grid points along those edges. A grid block topology was assumed for the inlet to form the edges into faces and those faces into blocks. SUPIN generated grids along the edges, on the surfaces, and within the interior volume of each block. The interior block boundaries abutted with other block boundaries. For most blocks, the grid lines were contiguous across block boundaries, but some non-contiguous boundaries were used to facilitate the structured topology. Figure 9 shows examples of the grid lines for the planar grid.

The CFD simulations were initialized with a flowfield set to the freestream conditions. The simulations were started with the first-order form of the Roe flux-splitting method to damp out large initial gradients. Eventually, the second-order flux method was applied as the residuals over the iterations decreased and the boundary layers, shock waves, and subsonic inlet flow took form. At the start of the simulations, the Courant-Friedrichs-Lewy (CFL) number had a value of 0.5 but increased incrementally to a value of 2.5 as the flow solution developed. Local time stepping was used in which the local time step used was computed based on the CFL number and the local grid cell size. The iterative convergence was indicated in part by the reduction of the root-mean-square of the residuals of the conservative variables for each block. Iterative convergence was also evaluated through the monitoring of the convergence of the inlet flow rate, total pressure recovery, and total pressure distortion. The steady-state solution was considered converged when these values varied less than 0.1% of their values over 1000-2000 iterations.

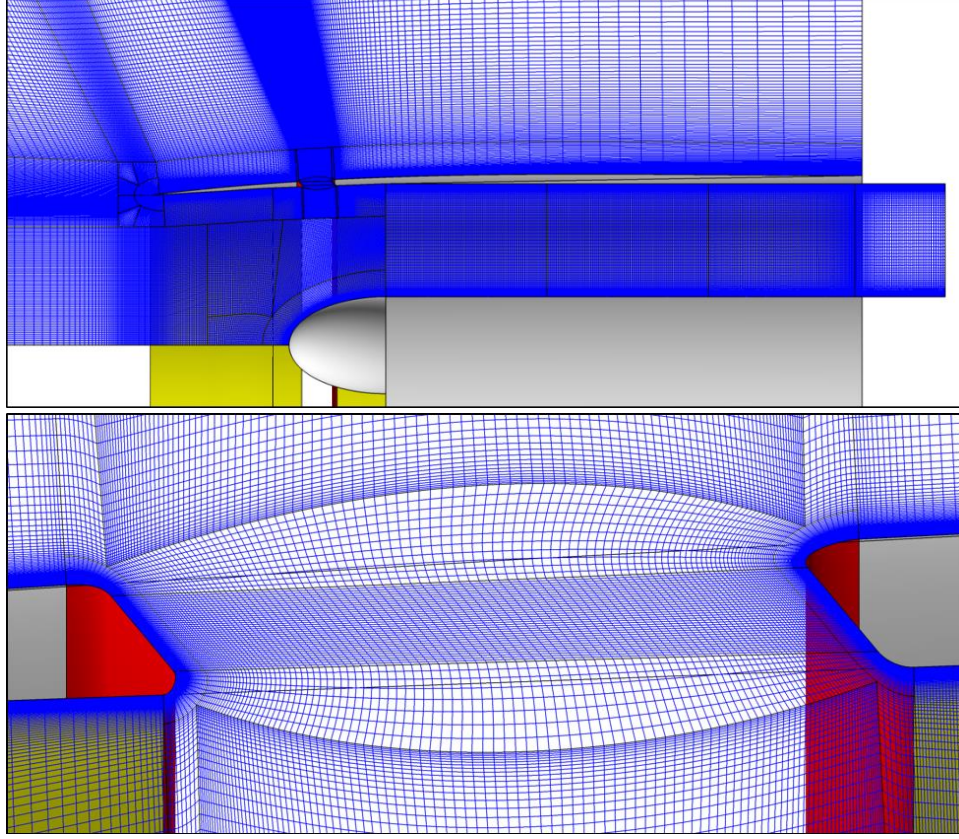


Figure 9. Structured, multi-block, computational grids on the symmetry plane for the axisymmetric pitot inlet.

D. Inlet Performance Metrics

The flow solutions from the CFD simulations were used to obtain the aerodynamic performance metrics for the inlet. The three inlet performance metrics used to characterize the inlet included the inlet flow ratio (W_2/W_{cap}), the inlet total pressure recovery (p_{t2}/p_{t0}), and the SAE ARP 1420 radial (DPRP) distortion index. The inlet flow ratio was defined as the inlet flow rate (W_2) divided by the reference capture flow rate (W_{cap}). The inlet flow rate (W_2) was obtained from the CFD simulation by integrating the rate of flow passing through the cross-stream grid planes of the outflow nozzle. The total pressure at the engine face (p_{t2}) was computed as the mass-average of the total pressures at the grid plane at the engine face. The inlet radial (DPRP) total pressure distortion index at the engine face was defined on a standard 40-probe rake array of the Society of Automotive Engineers (SAE) Aerospace Recommended Practices (ARP) 1420 document [7]. For the planar CFD domain, the rake array consisted of a single rake arm containing five total pressure probes. Each probe was placed radially at the centroid of equal-area sectors of the annular engine face. The flowfield from the CFD simulation was interpolated onto the locations of the probes to obtain the total pressure at the probe location.

E. Design of Experiments

The methods of design-of-experiments (DOE) were applied to statistically examine the three factors for the auxiliary inlet inlets that influenced the inlet performance. The three factors were the position (x_{aux}), the angle (ϕ_{aux}), and the length (L_{aux}) of the auxiliary inlet. The responses were the inlet performance metrics of inlet flow ratio, inlet total pressure recovery, and SAE ARP radial distortion index. The primary DOE design used for this work was the Central-Composite Face-Centered (CCF) design. Three levels of the three factors were used which required 15-factor combinations to build a quadratic model. Each combination involved one CFD simulation at the specified corrected flow rate for the $M_0 = 0.25$ take-off condition. To obtain the simulation at the desired inlet flow rate, several simulations were required while iterating the internal outflow boundary condition. The Design Expert [8] software was used to perform the statistical analysis.

IV. Results

This section discusses the results of CFD simulations that explored the design factors for the auxiliary inlets. The results from the DOE studies will be discussed separately for each inlet.

A. Axisymmetric Pitot Inlet

The axisymmetric pitot inlet provided a baseline and simpler geometry for examining the auxiliary inlet performance. Figure 10 shows Mach number contours and the inlet performance with respect to the range of freestream Mach numbers. At $M_0 = 1.4$, the cruise design condition, the normal terminal shock stood at the cowl lip plane. Most of the total pressure loss was due to the normal shock. At $M_0 = 1.0$, the terminal shock was not present, and the total pressure recovery was nearly unity. At $M_0 = 0.6$, there were some total pressure losses from boundary layer separation at the sharp cowl lip interior as air was being drawn into the inlet. At $M_0 = 0.25$, there was significant separation at the sharp cowl lip and portions of the core inlet flow reached supersonic conditions.

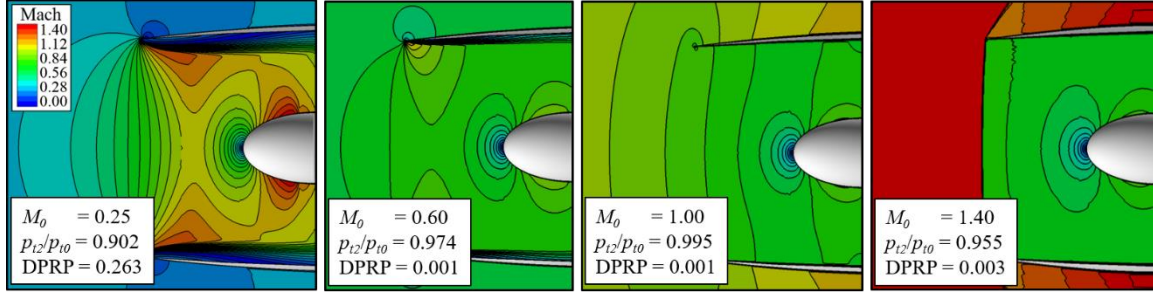


Figure 10. Mach number contours on the symmetry plane and inlet performance metrics of the axisymmetric pitot inlet at various freestream Mach numbers.

An auxiliary inlet consisting of a circumferential slot just ahead of the engine face was included in the axisymmetric pitot inlet. The overall effect of the auxiliary inlet can be seen in the Mach number contours shown in Figure 11. The freestream and engine face conditions corresponded to the $M_0 = 0.25$ conditions. With no auxiliary inlet, the airflow being drawn into the inlet separated at the sharp leading edge and created a separation bubble downstream of the cowl lip on the interior of the cowl. The auxiliary inlet provided an additional passage for airflow and the Mach number of the flow entering the cowl lip station was reduced. The engine face total pressure recovery for the simulation of the inlet with the auxiliary inlet was $p_{t2}/p_{t0} = 0.952$ compared to $p_{t2}/p_{t0} = 0.902$ without. The radial distortion decreases slightly with the use of the auxiliary inlet.

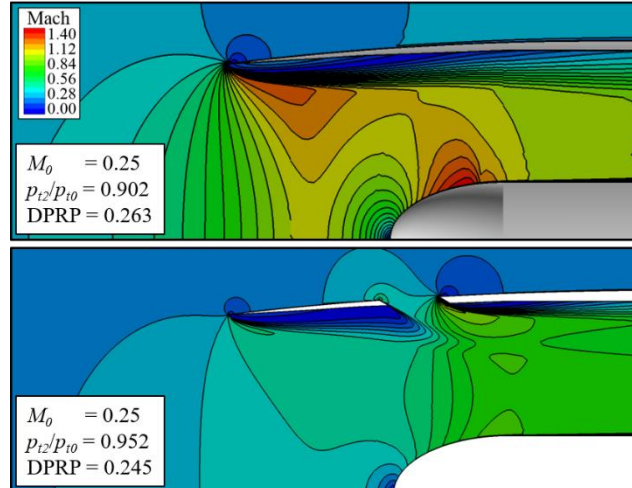


Figure 11. Mach number contours for planar CFD simulations of the axisymmetric pitot inlet without (top) and with (bottom) an auxiliary inlet.

A DOE study was performed to examine the variation of the inlet performance metrics with respect to the three auxiliary inlet design factors. Fifteen combinations of the input factors were used for the study and the resulting performance metrics were obtained from CFD simulations performed using SWFS. The auxiliary inlet location was

varied from 1.665 to 2.319 feet from the cowl lip. The auxiliary inlet angle was varied from 0 to 40 degrees, and the length of the auxiliary inlet was varied from 0.127 to 0.211 feet. The resulting total pressure recovery varied from 0.940 to 0.960. The radial distortion varied from 0.118 to 0.347. Design Expert was used to perform the statistical analysis of the data. With regards to the total pressure recovery, a linear model was suggested to fit the data. The first-order term for each of the three design factors was included. No terms dictating an interaction of design factors were kept in the model. The location of the auxiliary inlet, x_{aux} , had a smaller impact on increasing total pressure recovery than the angle, ϕ_{aux} , or length of auxiliary inlet, L_{aux} . The angle, ϕ_{aux} , was the dominant design factor. The statistical analysis showed that increasing any of the design parameters would increase the total pressure recovery. Increasing the factors translates to a larger angle on a longer auxiliary inlet closer to the engine face plane. The argument agrees with the logic of increasing flow in the slowest region to improve recovery.

In terms of radial distortion, a linear model was used to fit the data. The length, L_{aux} , of the auxiliary inlet was deemed statistically insignificant when compared to the other two parameters. No mixed terms for the interaction of design variables were significant. The angle, ϕ_{aux} , was determined to be the dominant factor for radial distortion. To decrease the radial distortion, the angle of the inlet should be increased, and the location should be moved further forward.

Optimization of both radial distortion and total pressure recovery showed that the largest angle which was furthest aft on the longest inlet was the most desirable. This configuration corresponded to the maximum of each design factor examined for the axisymmetric pitot DOE. This suggested that the optimum was located outside of the bounds examined. Further examination could verify this prediction.

A CFD simulation was performed of the optimum auxiliary inlet configuration as determined by Design Expert as a means to verify the optimum configuration. The predicted total pressure recovery was 0.9567, and the CFD solution gave a total pressure recovery of 0.9600. The radial distortion prediction was 0.1946 with the CFD giving a distortion of 0.1182.

An additional DOE study was performed using Wind-US on the axisymmetric pitot inlet. The Wind-US DOE replicated the geometries from the DOE performed in SWFS. A modified aft cowl lip on the auxiliary inlet was developed to direct the flow more smoothly. An elliptical profile with an aspect ratio of approximately four was also modeled. Design Expert was used to analyze the solutions. For this additional DOE, the total pressure recovery varied from 0.918 to 0.935, and the radial distortion varied from 0.2105 to 0.2787.

Total pressure recovery was modeled in design expert as dependent on all three design factors as well as a second-order position term and mixed terms of position times length and of length times angle. The mixed terms indicate an interaction or coupling of responses of multiple design factors. As with the SWFS DOE, an increase in any design factor would lead to an increased total pressure recovery.

Radial distortion in the inlet was modeled to depend on first and second-order terms related to positioning the inlet, a first-order term related to the length of the auxiliary inlet opening, and a mixed term including first-order terms for length multiplied by position. The angle was determined to not have a significant influence. This contrasts with the SWFS DOE conclusion for the axisymmetric pitot inlet that the angle of the auxiliary inlet dominated. The inclusion in the Wind-US DOE of the elliptical profile on the aft portion of the auxiliary inlet could have eliminated the effectiveness of the change in angle on radial distortion.

The statistical optimum of the three design factors was determined in Design Expert, and this configuration was simulated in Wind-US after the completion of the DOE study. The Design Expert optimum had a position of 1.665 feet, an angle of 40 degrees, and a length of 0.211 feet. Agreement was found of a half percent between the model predicted total pressure recovery and the CFD total pressure recovery for the optimum configuration. The DOE optimum also showed good agreement between model optimum and CFD in radial distortion.

Comparison between the Wind-US and SWFS DOE studies for the axisymmetric pitot inlet show several key similarities and differences. The first difference found between the two DOE studies was a discrepancy between nearly all cases in terms of the specific values for total pressure recovery and radial distortion. The values found by SWFS for all cases was on the order of 3% higher total pressure recovery, and a margin lower radial distortion. The margin by which the radial distortion varied from geometry to geometry. However, the trends remained consistent, especially for total pressure recovery. The qualitative predictions align well with the total pressure recovery for this reason. The recommendations mentioned in both DOE studies were very similar. The recommendations for the radial distortion as well as the qualitative analysis regarding the importance of auxiliary inlet angle could be clouded slightly by the elliptical aft lip included in the Wind-US DOE.

B. Axisymmetric Spike Inlet

The axisymmetric spike inlet provided a higher-performing inlet for a Mach 1.4 aircraft. Figure 12 shows Mach number contours and the inlet performance with respect to the range of freestream Mach numbers. At $M_0 = 1.4$, the

cruise design condition, the normal terminal shock stood at the cowl lip plane. Most of the total pressure loss was due to the normal shock; however, the pre-shock Mach number was lower than for the axisymmetric pitot inlet, and so, the total pressure losses were less for the axisymmetric spike inlet. At $M_0 = 1.0$, the terminal shock was not present, but there was a small region of supersonic flow at the shoulder of the centerbody. At $M_0 = 0.6$, the inlet flow was choked as air was being drawn into the inlet. At $M_0 = 0.25$, there was significant separation at the sharp cowl lip and the flow in the inlet was choked. A normal shock formed within the inlet, leading to greater total pressure losses and distortion.

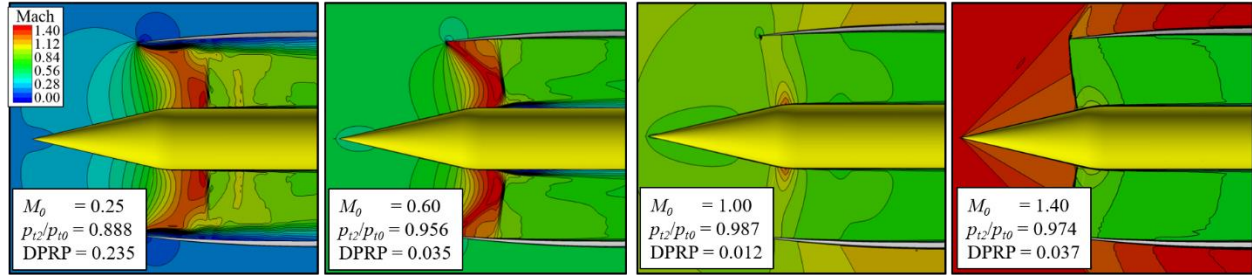


Figure 12. Mach number contours on the symmetry plane and inlet performance metrics of the axisymmetric spike inlet at various freestream Mach numbers.

An auxiliary inlet consisting of a circumferential slot just ahead of the engine face was included in the axisymmetric spike inlet. The first set of CFD simulations considered a planar flow domain that did not include the struts. The effect of the auxiliary inlet can be seen in the Mach number contours shown in Fig. 13. The freestream and engine face conditions correspond to the $M_0 = 0.25$ conditions. The auxiliary inlet provided an additional passage for airflow and the Mach number of the flow entering the cowl lip station was reduced. Compared to the corresponding frame of Fig. 12, the normal shock was eliminated, and the flow separation at the engine face was reduced. The engine face total pressure recovery for the axisymmetric spike inlet with the auxiliary inlet was $p_{t2}/p_{t0} = 0.9329$ compared to $p_{t2}/p_{t0} = 0.888$ without the auxiliary inlet. Radial distortion was reduced with the use of an auxiliary inlet from 0.235 to 0.172.

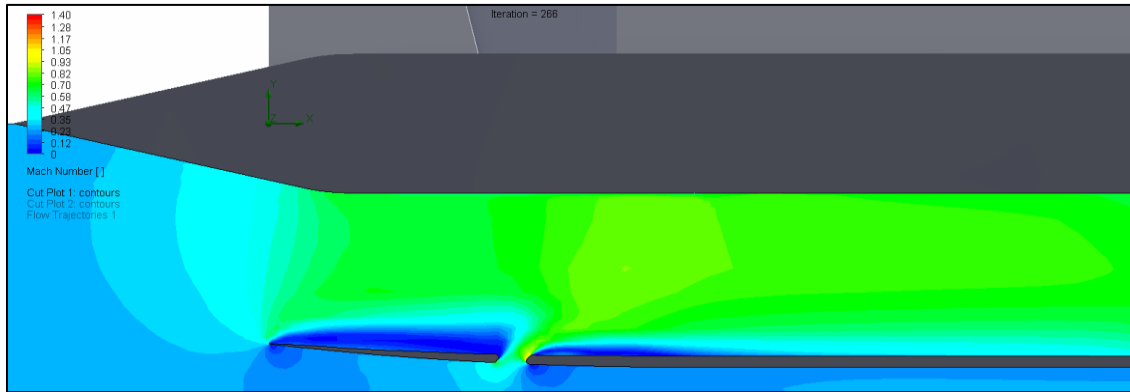


Figure 13. Mach number contours on the symmetry plane of the axisymmetric spike inlet with an auxiliary inlet at a take-off conditions with a freestream Mach number of $M_0 = 0.25$.

Preliminary 3-Dimensional CFD simulations have been performed in SWFS to include the axisymmetric spike strut. These simulations include a domain that includes a full strut and half of an auxiliary inlet. The auxiliary inlet ends before reaching the support strut, so the focus of the simulation will be the interaction between the flow entering the auxiliary inlet and the support strut.

A fifteen case DOE study was performed for three design factors in a similar fashion as the axisymmetric pitot DOE. The auxiliary inlet location was varied from 1.822 to 2.728 feet from the cowl lip. The auxiliary inlet angle was varied from 0 to 40 degrees, and the length of the auxiliary inlet was varied from 0.130 to 0.217 feet. The resulting total pressure recoveries varied from 0.941 to 0.958. The radial distortion varied from 0.193 to 0.335. Design Expert was used to perform the statistical analyses. A quadratic model was developed for the total pressure recovery response to the DOE design factors. All first-order terms for each variable, the second-order term for χ_{aux} , as well as a mixed angle and location term were all included in the quadratic equation for the recovery. The second-order term of position

dominated the model behavior. The mixed term produced the smallest influence. As with the axisymmetric pitot inlet statistical model, the model predicts that total pressure recovery will increase as each of the factors is increased. The mixed term is the only exception as the model predicts that if the mixed term decreases the total pressure will increase.

For radial distortion, a similar model to the axisymmetric pitot was produced. A linear model including first-order terms for position and angle but not length was created. The length was deemed to have insignificant influence on distortion in the axisymmetric spike inlet. Position and angle had similar control over the model behavior. As with the axisymmetric pitot, to decrease radial distortion the angle should be large, and the auxiliary inlet position should be further forward from the engine face plane.

For the optimization of total pressure recovery and radial distortion, a configuration of maximized angle and length but minimized position was produced. The model predicted optimum and corresponding case from the axisymmetric spike DOE represent the same geometry. The predicted total pressure recovery was 0.9542 and the CFD simulation total pressure recovery was 0.9540. This served as a check that the model predicted optimum was very close to the CFD solution. The radial distortion prediction was 0.1805 with the CFD giving a distortion of 0.1953.

The optimum CFD solution compared to the solution of a no auxiliary inlet case again shows benefits to including an auxiliary inlet. Without an auxiliary inlet, the recovery for the axisymmetric spike inlet was 0.9358. The auxiliary inlet improved the recovery to 0.9540. The radial distortion was decreased from 0.2707 to 0.1953 with the auxiliary inlet providing additional flow at the cowl.

An additional DOE study was performed using Wind-US. This study repeated the 15 simulations performed in Solidworks Flow Simulation. A modified aft cowl lip on the auxiliary inlet was used to reduce radial distortion. The elliptical auxiliary inlet aft lip can be seen in Figure 14 below. Some simulations within the DOE displayed unsteady behavior. Five simulations were reduced from second-order to first-order solutions to obtain a steady result. Design Expert was used to analyze the solutions. The results from the additional DOE varied in total pressure recovery from 0.922 to 0.940. The radial distortion varied from 0.250 to 0.365.

Total pressure recovery was determined to be dependent entirely on the length of the auxiliary inlet. The statistical analysis determined that the influence of the angle of the auxiliary inlet, as well as the location, were not significant. The trend displayed that a larger auxiliary inlet would improve the recovery performance of the inlet.

Radial distortion in the inlet was modeled to depend on first and second-order terms related to positioning the inlet, and a first-order term related to the length of the auxiliary inlet opening. The angle was determined to not have a significant influence. Qualitatively, the inlet distortion is reduced if the additional flow from the auxiliary inlet is introduced further from the engine face plane.

The statistical optimum of the three design factors was determined in Design Expert, and this configuration was simulated in Wind-US. The Design Expert optimum had a position of 1.978 feet, an angle of 20 degrees, and a length of 0.217. There was good agreement between the model predicted total pressure recovery and the CFD total pressure recovery for the optimum configuration. The predicted total pressure recovery was 0.9391, and the CFD simulation provided a total pressure recovery of 0.9329. The predicted radial distortion was higher than the CFD simulation result; a predicted value of 0.2704 compared to 0.1720.

Comparison between the Wind-US and SWFS DOE studies show several key similarities and differences. The differences between models, similarities in trends, and potential causes will be laid out. When analyzing the DOE models, the angle was deemed insignificant in the Wind-US DOE but significant by the SWFS DOE. One potential explanation for this change could be the modified aft lip of the auxiliary inlet used in the Wind-US study. The main difference between studies is Wind-US predicted lower total pressure recovery and higher distortion than SWFS for all cases. The cases from Wind-US predicted a total pressure recovery an average of 0.0159 lower, and a radial distortion 0.0473 higher on average. While the target values were shifted, both DOE studies developed similar recommendations to increase recovery and decrease distortion. The two studies agreed that increasing the length of the auxiliary inlet would increase total pressure recovery. Additionally, the dominating factor for radial distortion in both studies was the axial location. Moving the axial location away from the engine face plane reduced the radial distortion.

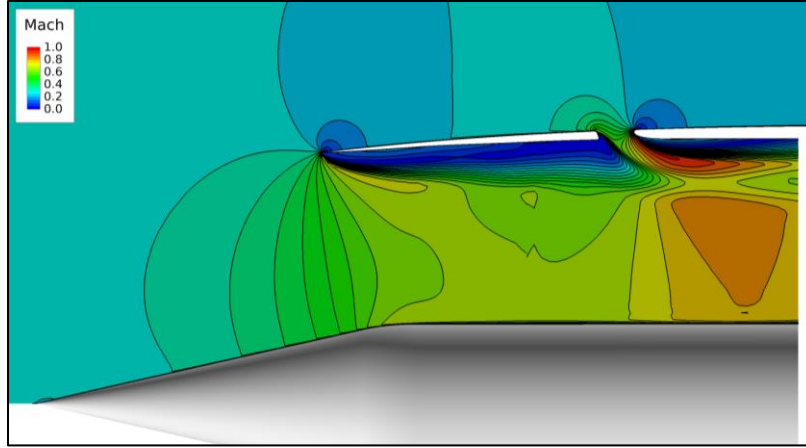


Figure 14. Mach number contours on the symmetry plane of the axisymmetric spike inlet with an auxiliary inlet at a freestream Mach number, $M_0= 0.25$.

Upon the conclusion of the four DOE studies, an additional investigation was done into redesigning the thickness of the cowl and a translated cowl on the axisymmetric spike inlet. The DOE simulations suggested that performance would increase as the auxiliary inlet was moved forward and increased in length. Several simulations were performed with the maximum cowl thickness varied from about 1 inch to about 3 inches. The additional thickness of the cowl would allow greater room for the inlet mechanical hardware and acoustic lining within the inlet. The translation of the cowl forward as opposed to making an opening within the set length of the inlet allowed for greater auxiliary inlet lengths. The longer auxiliary inlet promised improved aerodynamics, and the translation forward allowed for the positioning of the auxiliary inlet to be moved forward. The forward position also allows for more area for acoustic liners. Simulations were completed at take-off, cruise, and approach on the thicker cowl model.

SUPIN design factors were varied and Wind-US simulations were completed to arrive at the three-inch cowl thickness and one-foot auxiliary inlet length for take-off conditions. Wind-US calculations were then completed with an auxiliary inlet at take-off, and with and without an auxiliary inlet at cruise and approach to landing. Because of limitations of SUPIN and Wind-US, the take-off with auxiliary inlet, and approach with auxiliary inlet simulations were repeated with a translated forward cowl in SWFS. An image of the translated geometry is shown below in Figure 15 alongside an image of the untranslated SUPIN model. The constant-area downstream duct section shown in the translated model between the blue lines is not a part of the inlet geometry. In Figures 16 and 17, the Mach contour results are shown for the take-off condition. The much larger 1-foot slot produces much better radial distortion and total pressure recovery than previously attained. Wind-US provided a total pressure recovery of 0.989 for the simulation shown in Figure 16. In contrast, during the DOE study, the optimum was found to be 0.932. The same simulation saw a drop in radial distortion from 0.172 to 0.026. The larger auxiliary inlet produced significant aerodynamic improvements.

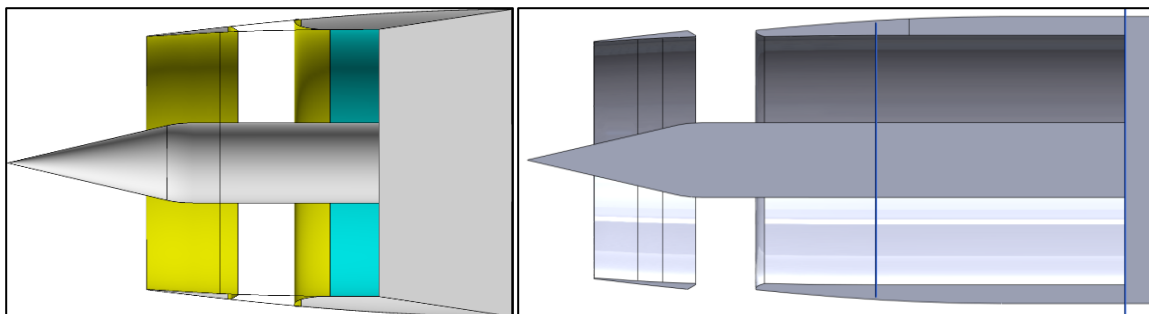


Figure 15. Cowl thickness redesign shown translated forward (right) and untranslated (left) with 1-foot auxiliary inlet for take-off conditions.

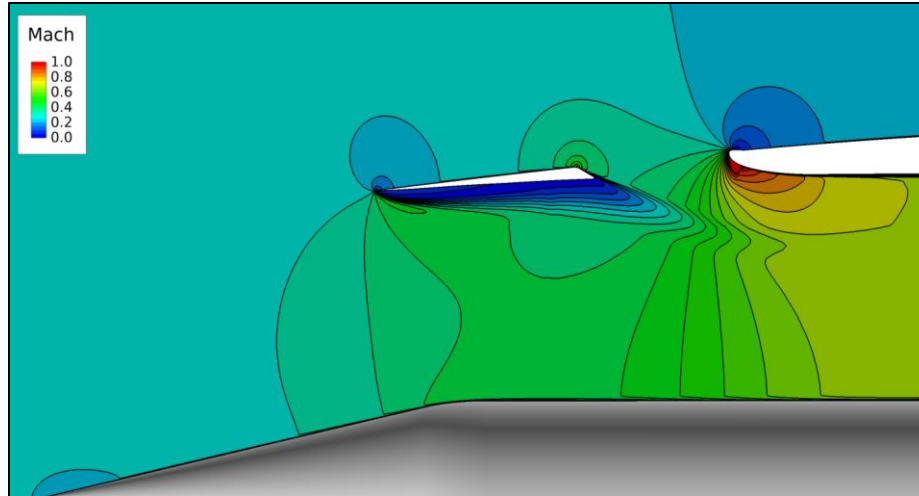


Figure 16. Wind-US untranslanted cowl redesign Mach contours with 1-foot auxiliary inlet for take-off conditions.

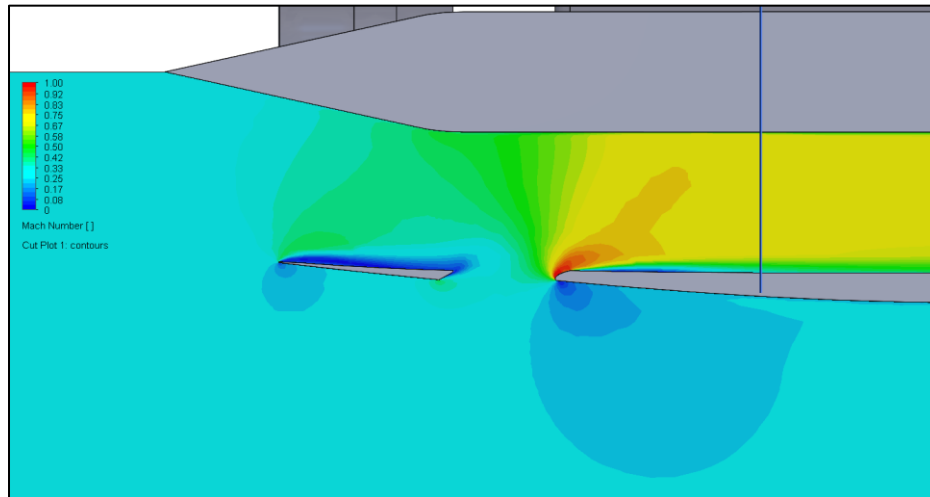


Figure 17. SWFS translated cowl redesign Mach contours with 1-foot auxiliary inlet for take-off conditions.

For approach to landing, additional SUPIN design and Wind-US simulations were completed to choose a 0.3 foot length auxiliary inlet to provide the appropriate amount of flow for the reduced engine throttle settings during approach to landing. The translated and untranslanted models for the approach to landing can be seen in Figure 18. The constant-area downstream duct section shown in the translated model between the blue lines is not a part of the inlet geometry. In Figures 19 and 20, the Mach contour results are shown for the approach condition. At the reduced engine face Mach number required at approach to landing, the inlet requires much less auxiliary flow. Thus, the slot length is 0.3 feet. Approach condition were not simulated in the DOE studies, so there is not a point of comparison except between Wind-US and SWFS. Wind-US provided a total pressure recovery of 0.993 for the simulation shown in Figure 19. The SWFS also gave a result of 0.993 for total pressure recovery. Wind-US had a radial distortion of 0.0218 and for SWFS a value of 0.0111. The agreement of both simulations verifies this configuration should provide good performance at approach to landing.

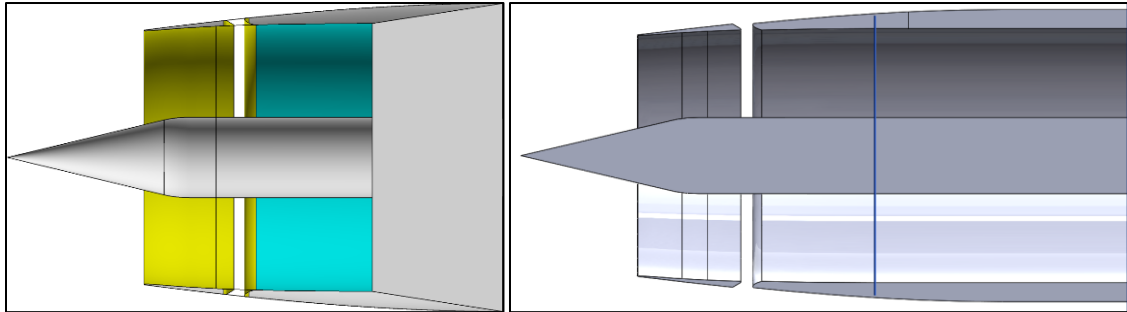


Figure 18. Cowl thickness redesign shown translated forward (right) and untranslated (left) with 0.3-foot auxiliary inlet for approach-to-landing conditions.

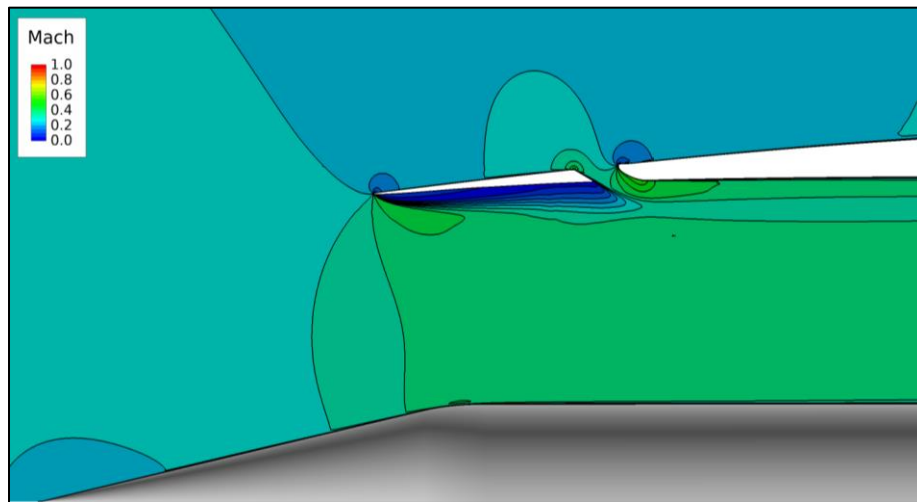


Figure 19. Wind-US untranslanted cowl redesign Mach contours with 0.3-foot auxiliary inlet for approach-to-landing conditions.

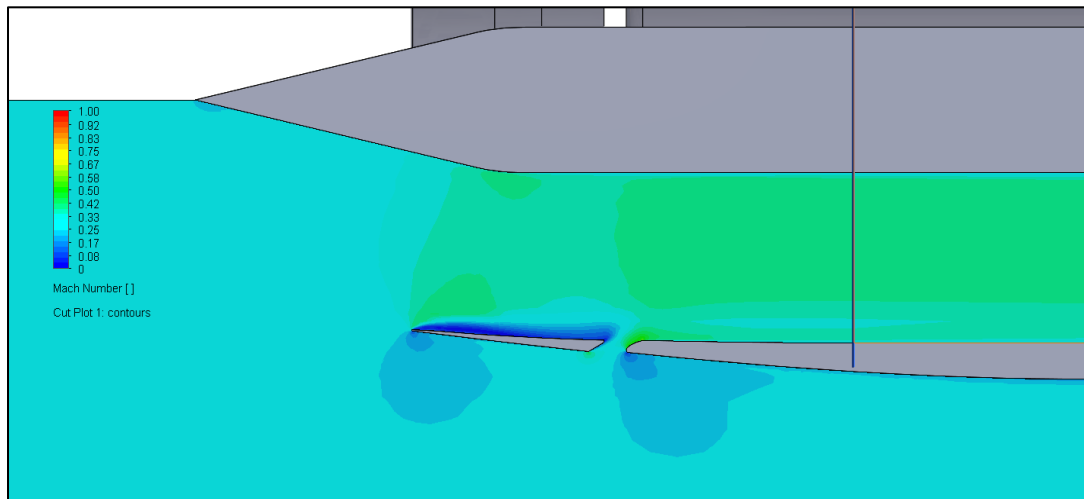


Figure 20. SWFS translated cowl redesign Mach contours with 0.3-foot auxiliary inlet for approach-to-landing conditions.

V. Conclusion

Inlets designed in SUPIN were modeled in SolidWorks. SolidWorks Flow Simulation and Wind-US were used to analyze the flow through auxiliary inlets in the SUPIN models. Statistical analysis in Design Expert was completed on the resulting data. Consistent recommendations regarding how to best reduce radial distortion and increase total pressure recovery were developed. Statistically significant improvements to inlet performance with the inclusion of the auxiliary inlet align with the expected result. The cowl thickness redesign, as well as translation option, represent the culmination of the lessons learned throughout the design process. A few of the most important qualitative lessons will be shared here. Moving auxiliary inlets forward produces a better aerodynamic performance as the flow has time to settle and become uniform at the engine face plane. The thicker cowls that allow for the possibility of mechanical translation and acoustic lining also help limit flow separation. The larger auxiliary inlets help provide a more uniform auxiliary flow and increase performance. The auxiliary inlets discussed provided improved flow quality at the engine face of the SUPIN inlets designed for the STCA engines.

Acknowledgments

This work is supported by the NASA Commercial Supersonic Technology Project.

References

- [1] Berton, J. J., Huff, D. L., Seidel, J. A., and Geiselhart, K. A., “Supersonic Technology Concept Aeroplanes for Environmental Studies,” AIAA Paper 2020-0263, January 2020.
- [2] Slater, J. W., “SUPIN: A Computational Tool for Supersonic Inlet Design,” AIAA Paper 2016-0532, January 2016.
- [3] Dassault Systemes, “Technical Reference SolidWorks Flow Simulation 2019”, 2019.
- [4] Yoder, D. A., ‘Wind-US User’s Guide: Version 3.0,’ NASA TM 2016-219078, March 2016.
- [5] Walatka, P. P., Buning, P. G., Pierce, L., and Elson, P. A., “PLOT3D User’s Manual,” NASA-TM-101067, March 1990.
- [6] Menter, F. R., “Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications,” *AIAA Journal*, Vol. 32, No. 8, 1994, pp. 1598–1605. doi: 10.2514/3.12149.
- [7] Society of Automotive Engineers (SAE), “Gas Turbine Engine Inlet Flow Distortion Guidelines,” SAE ARP 1420, Rev. B, February 2002.
- [8] Design Expert, “Version 7.0.0. Stat-Ease,” Design Expert Inc., Minneapolis, 2005.