

Some Design Considerations for Two-Dimensional, Mixed-Compression Inlets for Mach 2.5 to 3.5

Christine M. Pastor-Barsi¹ and John W. Slater²

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

A computational design study was performed to explore design factors for two-dimensional, mixed-compression inlets designed for cruise conditions at Mach 2.5, 2.75, 3.0, 3.25, and 3.5 and their influence on the aerodynamic performance of the inlets. The design factors included the split of external and internal compression, capture aspect ratio, throat section and subsonic diffuser geometry, and location and rates of porous bleed regions. The performance of the inlet was measured by the total pressure recovery, cowl wave drag, bleed flow rates, and total pressure distortion at the engine face. The results provide useful information for the design of inlets for future supersonic aircraft.

Nomenclature

Roman Symbols

A	Area
D	Diameter or drag
DPR/P	Radial distortion index
DPC/P	Circumferential distortion index
h	Height or altitude
L	Length
M	Mach
p	Pressure
q	Dynamic pressure
T	Temperature
w	Width
W	Flow rate

Greek Symbols

δ	Boundary-layer thickness
Δ	Increment
θ	Total temperature ratio, surface angle, or streamwise angle

Subscripts

0	Freestream or reference conditions
1	Entrance to interior duct at the cowl lip
2	Engine face
bleed	Bleed
C	Corrected
cap	Capture
cbsh	Bodyside shoulder
cbTX	Bodyside at station TX
clin	Cowl lip interior
comp	Compressive
hub	Hub

¹ Research Aerospace Engineer, Propulsion Division / Inlets and Nozzles Branch, AIAA Senior Member.

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

SD	Start of the subsonic diffuser
shock	Shock
shTX	End of shoulder to station TX
spillage	Spillage
t	Total or stagnation conditions
TX	Start of the after section of the throat
TXSD	Station TX to station SD
wave	Wave

I. Introduction

Future supersonic and hypersonic aircraft are envisioned to provide commercial flight at speeds up to Mach 5 with an attractive design point near Mach 3 [1], [2]. The design of the propulsion system for this class of aircraft is a key aspect of the development of a high-speed vehicle. The work of this paper explores the design of two-dimensional mixed-compression inlets which is a critical element of the propulsion system. Mixed-compression inlets capture a portion of the supersonic flow approaching the aircraft and decelerate the flow in a stable and efficient manner, to subsonic conditions, for intake by a turbofan or turbo-ramjet engine. The scope of this work focuses on the performance of isolated inlets and defers consideration of the integration of the inlet with an aircraft to later stages of aircraft and propulsion system development. This paper focuses on the design factors of two-dimensional mixed-compression inlets for Mach 2.5 to 3.5 and develops an understanding of their effect on the aerodynamic performance as characterized by the inlet flow rates, cowl wave drag, and the total pressure recovery and distortion at the engine face. Consideration is also given to the size of the inlet, which directly impacts vehicle design. This study continues and explores many of the same issues discussed in previous reports on inlet design [3], [4]. This investigation also builds upon previous work exploring design factors for two-dimensional external-compression inlets from Mach 1.4 to 2.0 [5]. The results provide guidance for the design of two-dimensional, mixed-compression inlets for future supersonic aircraft.

Section II discusses the approach for modeling the inlets using the Supersonic Inlet Design and Analysis (SUPIN) tool being developed at the NASA Glenn Research Center. The study involves segmenting the inlet flowpath that includes the approaching freestream, external supersonic diffuser, throat section, subsonic diffuser, and engine face. SUPIN establishes the geometry of the inlet and provides estimates of flow rates, total pressure recovery and cowl wave drag. Section III discusses the computational fluid dynamic (CFD) methods that are used to obtain higher fidelity estimates of flow rates, total pressure recovery, and cowl wave drag. Section IV presents the results of studies of the design factors for the external supersonic diffuser, throat section, and subsonic diffuser. The studies of the external supersonic diffuser mainly explore the split between external and internal supersonic compression and the capture aspect ratio for the inlet. The studies of the throat section explore the lengths, internal supersonic compression, and the location and rates of porous bleed. The studies of the subsonic diffuser explore the behavior of the terminal shock, diffusion of the subsonic flow, and total pressure recovery and distortion at the engine face.

II. Inlet Modeling and Design Considerations

The inlets discussed in this paper were modeled using the Supersonic Inlet Design and Analysis (SUPIN) tool [6], which is discussed in the next subsection. Subsequent subsections describe the model for the two-dimensional, mixed-compression inlet and the input design factors for the inlet. The last subsection discusses the measures of inlet performance.

A. SUPIN Supersonic Inlet Design and Analysis Tool

SUPIN is a Fortran 95 program that reads in a text-based input data file which specifies the design factors that describe the geometry and aerodynamic conditions of the inlet. SUPIN uses analytic, empirical, and computational methods to design the inlet and estimate the flow rates, total pressure recovery, and drag for the inlet. To facilitate the simulation of the inlet flow using CFD methods, SUPIN generates multi-block planar and three-dimensional structured grids for a flow domain about the inlet and writes a Plot3D file [7] for the grids.

B. Two-Dimensional, Mixed-Compression Inlet

The two-dimensional, mixed-compression inlet is represented by the images of Fig. 1. The inlet is divided into the external supersonic diffuser, throat section, and subsonic diffuser. The numeric stations shown in Fig. 1 are in accordance with the Society of Automotive Engineers Standard [8]. Considering an isolated inlet, station 0 represents

the uniform inflow to the inlet at the freestream conditions. The external supersonic diffuser extends from station 0 to station 1 and includes one to three stages over which external supersonic compression occurs through oblique shock and Mach waves. Station 1 is at the interface of the external supersonic diffuser and throat section and marks the entrance to the internal ducting of the inlet. The throat section turns the flow toward the engine face and performs the internal supersonic compression. Station TX indicates the expected end of the internal supersonic compression. Station SD denotes the interface between the throat section and subsonic diffuser. Station 2 is the end of the subsonic diffuser, which is also the engine face.

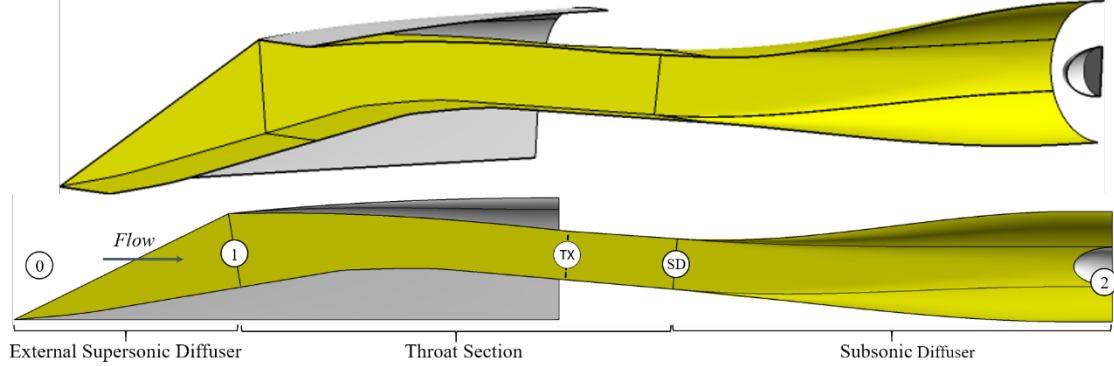


Figure 1. Two-dimensional, mixed-compression inlet.

C. Freestream and Engine Face Conditions

The freestream conditions are those of the flow approaching the isolated inlet and are designated as station 0 within the inlet flowpath, as shown in Fig. 1. The freestream conditions provide the upstream boundary conditions for the inlets. The freestream Mach numbers for this study ranged from $M_0 = 2.5$ to 3.5 as listed in Table 1. The dynamic pressures listed in Table 1 for each freestream Mach number were selected as representative values based on the authors' experience. For reference, the Lockheed SR-71 had a maximum speed of $M_0 = 3.3$ at an altitude of $h_0 = 80,000$ feet, which results in a dynamic pressure of $q_0 = 445$ psf. The higher values of the dynamic pressures listed in Table 1 reflect the use of more advanced technology than was used for the SR-71. It is believed that different values of the dynamic pressure would not seriously alter design guidelines regarding the inlet design factors. The 1976 U.S. Standard Atmosphere [9] was used to establish the altitude and thermodynamic properties for the freestream conditions.

Table 1. Freestream and engine face conditions for the inlet design studies.

M_0	q_0 (psf)	h_0 (ft)	p_{t0} (psf)	T_{t0} (°R)	M_2	W_{C2} (lbm/s)	$(p_{t2}/p_{t0})_{MIL}$
2.50	700	58627	2733.75	877.43	0.350	306.5	0.8660
2.75	800	59815	3799.25	979.80	0.325	287.4	0.8362
3.00	900	60985	5247.53	1091.9	0.300	267.7	0.8048
3.25	970	62758	6978.91	1213.8	0.275	247.5	0.7720
3.50	1000	65207	8894.73	1345.4	0.250	226.8	0.7379

The geometry and flow conditions at the engine face, station 2, are specified by the choice of engine and the desired engine flow rate, thereby forming the downstream boundary condition for the inlet design. A representative low-bypass ratio (0.5 to 1.0) turbofan engine is assumed for this study, with size and flow rates estimated. For this study, the engine-face diameter was specified as $D_2 = 4.0$ feet and the engine face included a spinner with a hub-to-tip ratio of $D_{hub}/D_2 = 0.35$. For the specified diameter and hub-to-tip ratio, the annular area of the engine face was $A_2 = 11.027$ ft².

The engine-face flowrate was specified in terms of the corrected flow rate (W_{C2}), which can equivalently be expressed as the engine-face Mach number (M_2), for a specified engine-face area. Table 1 lists the engine-face Mach numbers and corrected flow rates assumed for each of the freestream Mach numbers. The decrease in the engine-face Mach number as the freestream Mach number increases corresponds to likely temperature limits on the engine.

D. Inlet Design Factors

The design factors that establish the external supersonic diffuser, throat section, and subsonic diffuser are discussed in this subsection. The external supersonic diffuser incorporated three stages, as indicated in Fig. 2. The first stage

was a ramp that formed an oblique shock at the leading edge of the inlet. The second stage was a smoothly curved section that created a Mach wave system that was focused onto the cowl lip. The third stage was a ramp with a slope equal to the slope at the end of the second stage. The third stage created the uniform flow of Mach M_1 , which remains uniform through the inlet entrance station 1. Thus, one design factor for the external supersonic diffuser is the Mach number (M_1). The spatial dimensions of the external supersonic diffuser are established by the sizing of the rectangular capture area defined by $A_{cap} = w_{cap} h_{cap}$, which is shown in Fig. 2. The capture width w_{cap} is measured horizontally across the span of the external supersonic diffuser and is a design factor specified as a normalized capture width, w_{cap}/D_2 . The capture height, h_{cap} , is the vertical dimension measured from the inlet axis to the cowl lip and calculated from $h_{cap} = A_{cap} / w_{cap}$. The location of the leading edge of the first stage is established through specification of the design factor of the supersonic spillage ratio ($W_{spillage}/W_{cap}$). Figure 2 illustrates a streamline that results in supersonic spillage, which is calculated as $W_{spillage} = W_{cap} - W_1$.

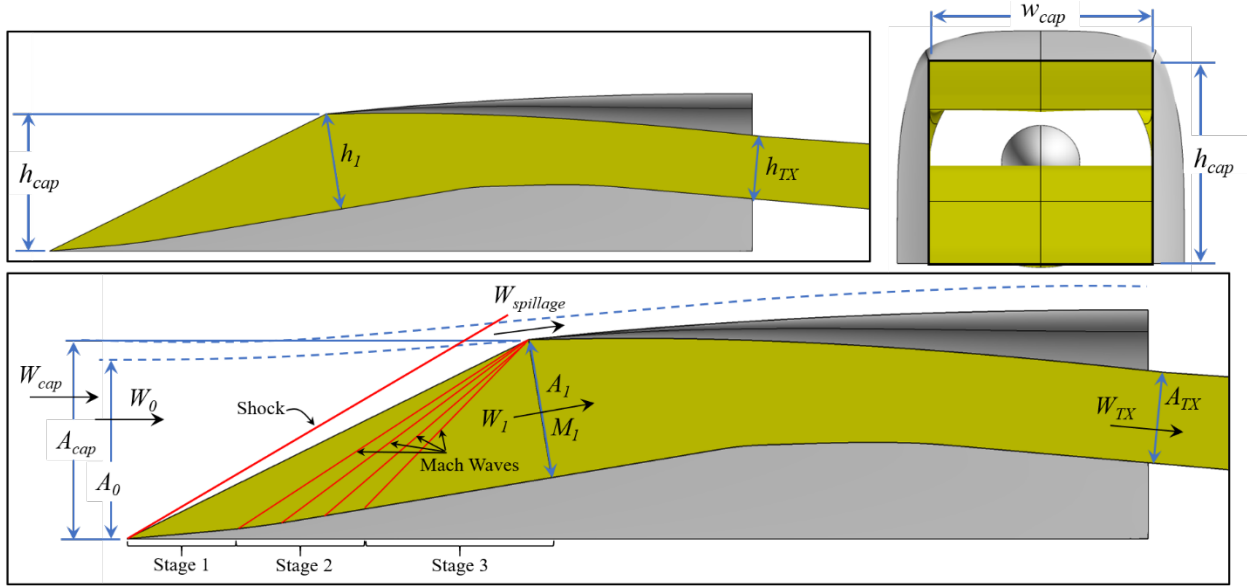


Figure 2. Two-dimensional, mixed-compression inlet through station TX with the inlet streamtube and flow rates.

The uniform flow at station 1 is deflected by the cowl lip interior, which has a slope of θ_{clin} , and an oblique cowl shock is formed as shown in Fig. 3. The cowl lip interior angle (θ_{clin}) is a design factor. The inlet is designed such that the start of the shoulder is placed at the point of intersection of the cowl shock with the bodyside. The design method results in a bodyside deflection at the start of the shoulder that cancels the cowl shock. The bodyside downstream of the cowl shock is defined by the design factors for the axial length of the shoulder (Δx_{cbsh}), slope of the bodyside downstream of the shoulder (θ_{cbTX}), length of the throat section from the shoulder to station TX (L_{shTX}), and length of the throat section between stations TX and SD (L_{TXSD}). The cowl interior is designed using method-of-characteristics to create a series of Mach waves that perform internal supersonic compression to a desired bodyside Mach number of M_{cbTX} , which is also a design factor. Figure 3 illustrates geometry definitions for the throat section for two-dimensional inlets.

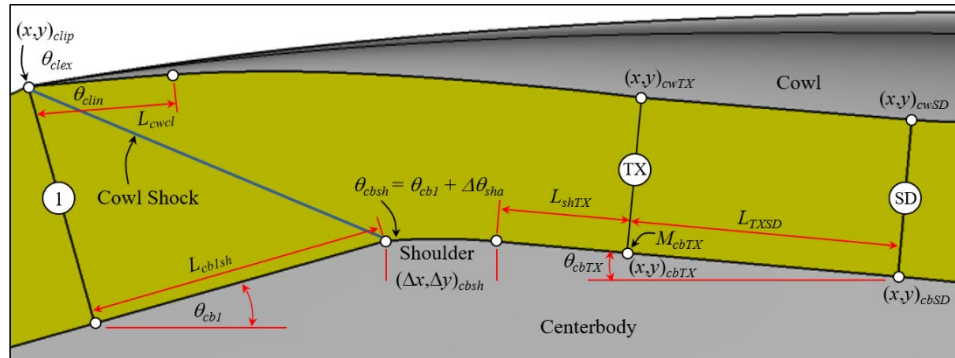


Figure 3. Geometry model for the parametric throat section for two-dimensional inlets.

The primary design factor for the subsonic diffuser is its length (L_{subd}) and vertical placement of the engine face axis (y_{EF}). The axis of the engine face is assumed to be parallel to the inlet axis. The length establishes the rate of diffusion for the subsonic diffuser and the vertical placement affects the curvature within the subsonic diffuser.

The split between the external and internal supersonic compression is expressed in terms of area ratios shown in Eq. 1.

$$\begin{aligned} \text{External Compression \%} &= \left(\frac{A_0 - A_1}{A_0 - A_{TX}} \right) 100\% \\ \text{Internal Compression \%} &= \left(\frac{A_1 - A_{TX}}{A_0 - A_{TX}} \right) 100\% \end{aligned} \quad (1)$$

Within this paper, the compression split is denoted in the form of %external/%internal, such as, 55/45 to designate 55% external supersonic compression and 45% internal supersonic compression.

E. Inlet Performance Metrics

The inlet performance metrics include the flow rates, total pressure recovery, total pressure distortion, and cowl wave drag. The inlet flow rates consist of the spillage flow rate ($W_{spillage}$), the bleed flow rate (W_{bleed}), and the engine-face flow rate (W_2). The inlet total pressure recovery (p_{t2}/p_{t0}) is the mass-averaged total pressure at the engine face normalized by the freestream total pressure. SUPIN calculates an estimate of the total pressure recovery using analytical and empirical methods. Total pressure recovery is also computed from the CFD simulations using a mass-average of the total pressure at the engine face. The column of Table 1 labeled with $(p_{t2}/p_{t0})_{MIL}$ lists the inlet total pressure recoveries as computed from MIL-E-5007E [10] as a function of M_0 . These values provide reasonable expectations of inlet total pressure.

The inlet total pressure distortion is characterized by the Society of Automotive Engineers (SAE) Aerospace Recommended Practices (ARP) 1420 radial (DPR/P) and circumferential (DPC/P) distortion indices [11]. These indices are computed from total pressure values interpolated from CFD simulations of the inlet flowfield onto a standard 40-probe rake described in the SAE ARP 1420 document. The rake array consists of eight radial rakes each containing five total pressure probes. In the circumferential direction the eight probes are located at a constant radius and form a ring about the circumference of the engine face. Each ring was placed radially at the centroid of equal-area sectors. The DPC/P indices reported in this paper are computed as the average of the DPC/P indices computed on the two inner or outer rings depending on whether the greatest total pressure distortion is at the hub or tip, respectively. The limits to the distortion indices depend on the stability of the specific turbofan engine, but for this study, the reasonable goals were set to $DPR/P < 0.05$ and $DPC/P < 0.08$ based on past inlet studies.

The cowl wave drag was estimated by SUPIN using analytical methods to compute pressure distributions on the cowl exterior and then performing an integration of the pressure over the cowl exterior. Likewise, the pressure distribution can be obtained from a CFD simulation. However, the CFD simulations reported in this paper didn't model the full cowl exterior, and so, no cowl wave drag coefficients were computed from the CFD simulations.

III. CFD Simulation Methods

The CFD simulations were performed using the Wind-US and FUN3D flow solvers, which are discussed in the following sub-sections. Both flow solvers were utilized to develop and exercise tools as part of inlet development. Subsequent sub-sections discuss the modeling of the flow domain and boundary conditions for the two-dimensional, mixed-compression inlets, generation of the grid, refinement of the grids, flowfield initialization, and monitoring of the flow solution for iterative convergence.

A. Wind-US Flow Solver

The Wind-US flow solver [12] was used to solve 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 uses a cell-vertex, finite-volume representation for which the flow solution is located at the grid points and a finite-volume cell is formulated about the grid point. In Wind-US, the RANS equations are solved for the steady-state flow solution using an implicit time-marching algorithm with a temporally first-order implicit Euler method using local time-stepping from an initial flow solution. All the simulations were performed assuming calorically perfect gas air. The inviscid fluxes of the RANS equations were modeled using a spatially second-order upwind Roe flux-difference splitting method. The flow simulation can assume inviscid flow, as well as, viscous, laminar or turbulent flow. For turbulent flow, the turbulent eddy viscosity was calculated using the two-equation Menter Shear-Stress Transport (SST) [13] turbulence model. Porous bleed was modeled using the Slater bleed model

[14] in which portions of the inlet surface were defined as bleed regions. For surface grid points falling within the bleed regions, a normal velocity was computed using empirical methods that required specification of bleed hole properties and bleed plenum static pressure.

B. FUN3D Flow Solver

The FUN3D flow solver [15] is a node-centered, unstructured, Reynolds-averaged Navier-Stokes (RANS) solver developed and maintained by NASA Langley Research Center (LaRC). All the simulations performed using this code were run as steady-state and assumed calorically perfect gas air. Flow simulations were performed both using inviscid and viscous turbulent flow. For viscous simulations a turbulence model of Spalart-Allmaras (SA) was used with a flux construction of LDFSS and the flux limiter was “the stencil-based van Albada limiter augments with a heuristic pressure limiter” [15]. FUN3D is unable to run a planer grid, so for two-dimensional simulations that were performed, Pointwise Mesh Generation software was used to add 1 step of width/depth to the grid. This two-dimensional grid configuration was utilized for all the FUN3D analysis reported in this paper. When simulating bleed, a porous boundary condition was used to simulate a normal velocity exiting a boundary. The porous boundary computational approach is further detailed in reference [16].

C. Computational Flow Domain and Boundary Conditions

Figure 4 shows the computational flow domain and boundary conditions (BC) used for the CFD simulations of the inlets. The two-dimensional flow domain for Wind-US was a single plane, while the two-dimensional flow domain for FUN3D was a plane extruded a single step. Both codes can use the three-dimensional flow domain, which contained only the starboard half of the inlet since the inlet had geometric symmetry about the vertical plane through the center of the inlet and flow symmetry was assumed. Figure 4 also notes the different boundary conditions applied for the two CFD codes that were used. Blue text and bold font refer to Wind-US boundary conditions while green text and italic font indicates FUN3D boundary conditions. This flow domain assumed that at the supersonic flow condition, there was no spillage past the sidewall. This approximation of the cowl exterior geometry was meant to simplify the CFD simulations. The external portions of the flow domain were of constant width equal to the capture width of the inlet

The internal and external surfaces of the inlet formed a portion of the boundary of the flow domain where no-slip, adiabatic viscous wall boundary conditions were imposed. The inflow and farfield boundaries were positioned just upstream of the leading-edge oblique shock so that the uniform freestream conditions could be imposed on those boundaries. At the downstream end of the cowl exterior, the domain had an external outflow boundary in Wind-US (extrapolate boundary in FUN3D) where both codes extrapolate for supersonic outflow.

For three-dimensional CFD simulations, downstream of the engine face, a converging-diverging nozzle section with sonic throat conditions was added to the flow domain to set the flow rate within the inlet. This has been found to be the most accurate way to computationally back-pressure an 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 Fig. 4. The converging-diverging portion is preceded by a portion of constant-area. The length of the outflow nozzle section was twice the diameter of the engine face, which was found to be sufficient for this inlet study. A longer length may be required for an inlet in which significant boundary-layer separation extends into the engine face. The cross-sectional area of the throat was set by specifying the ratio of the diameter of the nozzle throat to the diameter of the engine face (D_{noz}/D_2) and was set to form choked flow at the throat. Upstream of the nozzle throat and into the subsonic diffuser, the flow was subsonic and created the necessary back-pressure to support the terminal shock at the inlet throat section. Reducing the outflow nozzle throat area increased the back-pressure, and so, reduced the inlet flow rate. Downstream of the outflow nozzle throat, the flow was supersonic, and so, an extrapolation boundary condition could be applied at the outflow boundary.

Some of the CFD simulations explored only the supersonic flow through the throat section without formation of the terminal shock. For these simulations, the flow domain was truncated at the end of the throat section and an internal outflow boundary (Wind-US) or back-pressure (FUN3D) was specified to establish a supersonic outflow.

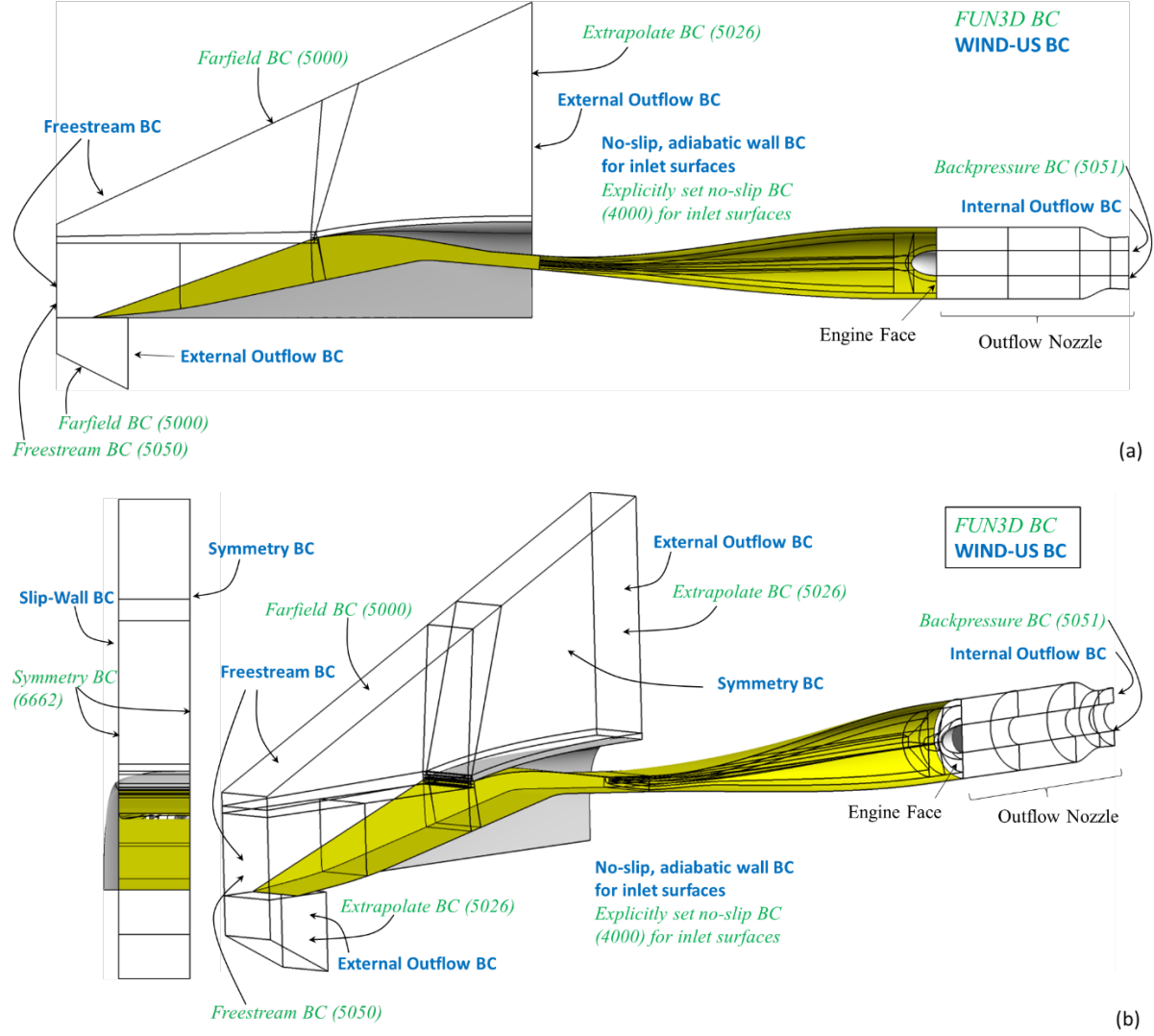


Figure 4. Flow domain and boundary conditions for the two-dimensional inlet CFD simulations. (a) two-dimensional domain (b) three-dimensional domain

D. Computational Grid

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 FUN3D simulations used the same base grid as the Wind-US simulations and were then manipulated accordingly to reflect the needs of that code. The inputs to the process include some factors to determine the extents 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}), the grid resolution within the throat section in the streamwise direction (Δs_x), and the grid resolution at the symmetry boundary (Δs_{sym}). 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.

E. Initial Flow Solution and Solution Monitoring

The CFD simulations were initialized with a flowfield set to the freestream conditions. The simulation 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 starting at a low value and is increased 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 L2 norm 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.

F. Grid Sensitivity Study

A grid sensitivity study was performed using Fun3D. The inlet case selected for this study was a Mach 3.0, 55/45 inlet configuration that was developed prior to studying the inlet factors. The flow domain was truncated to end at station SD and a supersonic outflow boundary condition was imposed to create supersonic flow through the throat section. A 1-ft in length bleed was placed at the shoulder to evaluate sensitivity of mass flow exiting a bleed patch in addition to the throat boundary condition. The grid was varied through the SUPIN grid-spacing scaling factor (Fgdds). The factor Fgdds is a multiplying factor on the specified spacing within the throat. Table 2 below lists the values that were compared in assessing the level of grid. The 0.8x grid spacing scaling factor was chosen as it was a good balance between compute time and consistency of mass flow being computed.

Table 2. Grid Sensitivity Parameters

Fgdds	Bleed Mass flow (lbm/s)	Throat Mass flow (lbm/s)	Grid size	CPU time used
0.6x	0.979519	105.037	2,611,680	121:58:41
0.7x	0.994594	105.001	2,138,390	100:44:25
0.8x	1.00393	104.976	1,775,620	84:53:43
0.9x	1.00318	104.97	1,499,916	74:11:11
1.0x	1.01748	104.934	1,293,118	70:29:42
1.1x	1.02006	104.921	1,147,406	64:01:48

IV. Results of the Inlet Design Studies

A series of studies were performed to examine the effect of the inlet design factors on the performance of the two-dimensional, mixed-compression inlets operating from $M_0 = 2.5$ to 3.5. Figure 5 summarizes and presents in a roadmap, the studies that were performed. Due to the streamwise character of supersonic flow through the inlet, the studies started with the design of the external supersonic diffuser and cowl exterior, using SUPIN and both flow solvers. External supersonic diffuser outflow Mach number and normalized capture width values were then selected and held constant moving into the study of the throat section. The throat section studies minimized length without bleed, and then using these minimal lengths determined bleed locations and rates. Using this information, the flow through the subsonic diffuser was then simulated with Wind-US CFD to evaluate the behavior of the terminal shock, diffusion of subsonic flow, and total pressure distortion.

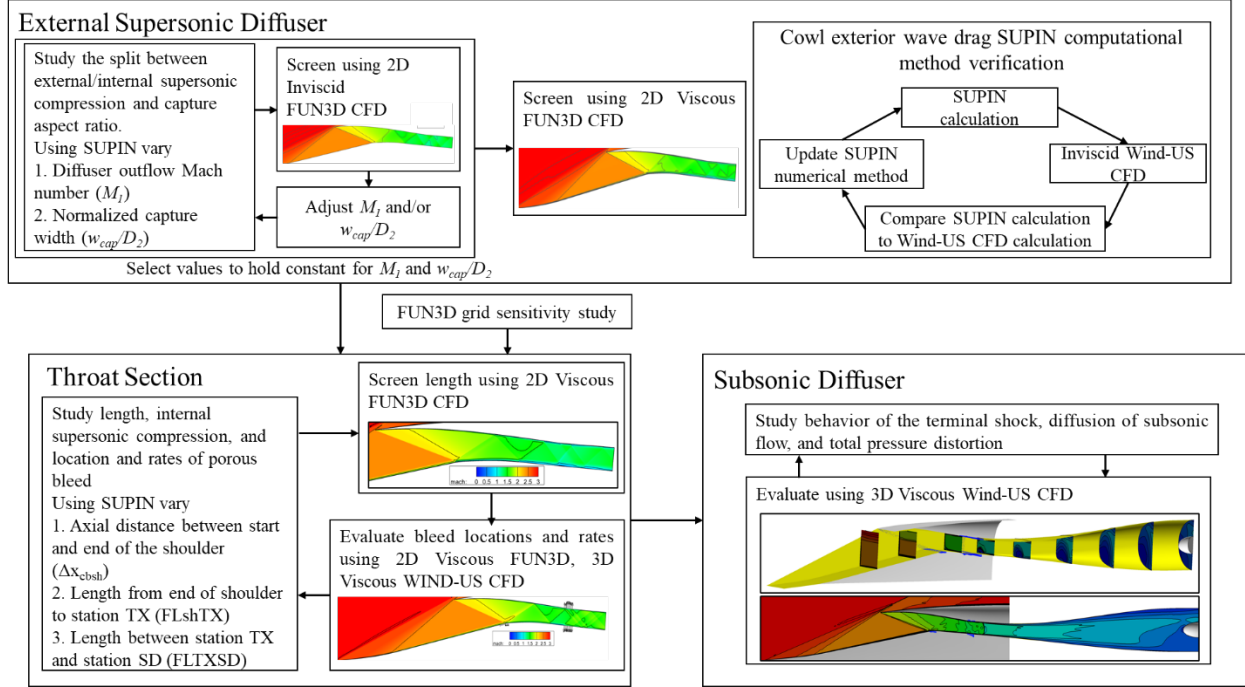


Figure 5. Road map illustrating inlet design studies performed. The flow of the study is forward to aft following the streamwise character of supersonic flow through an inlet.

A. External Supersonic Diffuser

The study of the design of the external supersonic diffuser for the inlets provided an understanding of the effects of the external / internal compression split and the capture width on the inlet performance and size. The downstream-directed characteristics of the supersonic flow allowed the design of the external supersonic diffuser and cowl exterior to be decoupled from the design of the throat section and subsonic diffuser. The study primarily involved the variation of the entrance Mach number (M_I) and the normalized capture width (w_{cap}/D_2).

This study was performed entirely using SUPIN. For each value of M_0 , the value of M_I was varied to obtain compression splits ranging anywhere from 30/70 to 70/30. For each choice of compression split, the value of w_{cap}/D_2 was varied from values from 0.7 to 3.50 with the lowest increment being $\Delta w_{cap}/D_2 = 0.05$. During the study, it was observed that the choice of w_{cap}/D_2 had little effect on the compression split. The specific range of values for w_{cap}/D_2 depended upon the values of M_0 and M_I and the intent of modeling the inlet configuration with the lowest cowl wave drag. The supersonic spillage ratio was set to $W_{spillage}/W_{cap} = 0.01$, thereby providing a small margin to prevent the leading-edge shock from entering the internal ducting of the inlet. The external supersonic diffuser for most of the inlets was specified to consist of three stages with the first stage having a slope of 5.0 degrees and the second stage specified to be an isentropic contour generating Mach waves. The third stage had a slope that matched the slope at the end of the second stage. The number of stages was reduced to two for those inlets with an external compression of 40% and lower. For these inlets, there was not sufficient external compression to allow three stages.

The value of the cowl lip interior angle (θ_{clin}) established the strength of the oblique cowl shock as measured by the rise in pressure across the shock, $(dp/dx)_{shock}$. The values of θ_{clin} were selected to impose the condition that $(dp/dx)_{shock} \approx 1.72$. This condition was based on the guideline that $(dp/dx)_{shock} > 1.80$ was expected to result in boundary layer separation [17]. As M_I was varied, the value of θ_{clin} also had to be varied until the desired compression split and shock pressure rise condition was achieved.

The study involved modeling the internal supersonic compression of the throat section to obtain the compression split for an inlet. The input factors for the lengths of the shoulder and segments of the throat section downstream of the shoulder were fixed at $\Delta x_{csh}/D_2 = 0.5$, $L_{shTX}/D_2 = 1.0$, and $L_{TXSD}/D_2 = 1.0$. The slope of the throat section downstream of the shoulder was set to $\theta_{cbTX} = -5.0$ degrees. The Mach number at the end of the internal supersonic compression at station TX on the bodyside was specified to be $M_{cbTX} = 1.3$. The expectation was that a normal terminal shock would be positioned at station TX, and so, a value of $M_{cbTX} = 1.3$ satisfied the condition that $(dp/dx)_{shock} < 1.8$. These

values of the input factors for the throat section resulted in a conservative throat section that provided sufficient length for the internal supersonic compression while allowing the turning of the flow toward the engine face.

The change in the wave drag (C_{Dwave}) and inlet size with respect to the normalized capture width (w_{cap}/D_2) can be illustrated through consideration of inlets designed for a compression split of 50/50. The plots on the left-hand-side of Fig. 6 show the variation of C_{Dwave} with w_{cap}/D_2 . An observation was that for each freestream Mach number (M_0), there existed a value of w_{cap}/D_2 that resulted in the lowest cowl wave drag. In some cases, a range of values of w_{cap}/D_2 provided near the lowest value of C_{Dwave} . Another observation was that the values of w_{cap}/D_2 for the lowest wave drag increased with increased freestream Mach number. The right plots of Fig. 6 present the same data with respect to the capture aspect ratio (w_{cap}/h_{cap}) and suggests that across the Mach number range, the best capture aspect ratio for these inlets for minimum wave drag could be summarized as $w_{cap}/h_{cap} \approx 1.5$.

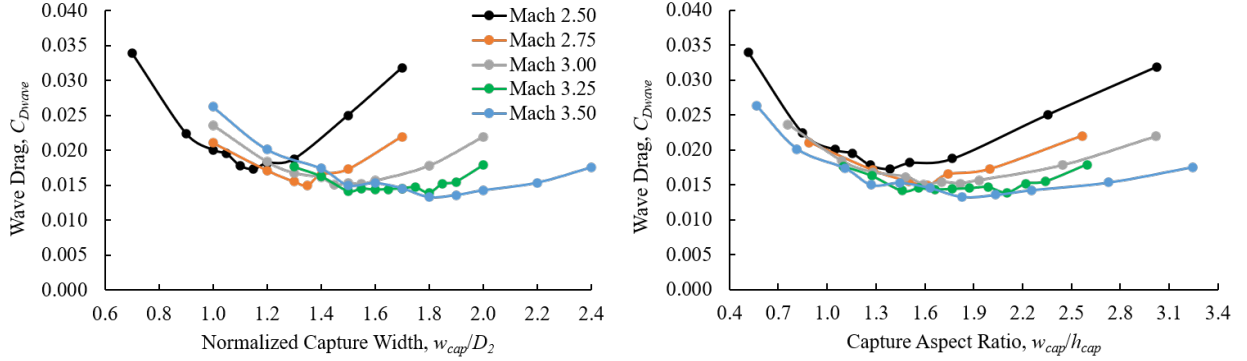


Figure 6. The variations as obtained from SUPIN of the cowl wave drag with respect to the normalized capture width (left) and the capture aspect ratio (right) for inlets with 50% external compression.

The variation of the normalized capture width (w_{cap}/D_2) influenced the size of the inlet as measured by the normalized compressive surface area (A_{comp}/A_{cap}) and the normalized compressive length (L_{comp}/D_2). Both metrics provided an indication of the relative size of the inlets, and thereby, the relative weight of the inlets. The value of A_{comp}/A_{cap} was the sum of the internal surface areas of the external supersonic diffuser and throat section normalized by A_{cap} . The L_{comp}/D_2 was the sum of the axial lengths of the external supersonic diffuser and throat section normalized by the engine face diameter (D_2). The plots of Fig. 7 show the variation of A_{comp}/A_{cap} and L_{comp}/D_2 with respect to the w_{cap}/D_2 for the inlets with 50% external compression. The plots on the left in Fig. 7 indicate that for the inlets for a specific M_0 , there was a value of w_{cap}/D_2 that minimized A_{comp}/A_{cap} and those values of w_{cap}/D_2 are close to those that minimize the cowl wave drag. The plots on the right of Fig. 7 indicate that the values of L_{comp}/D_2 decreased as w_{cap}/D_2 increased.

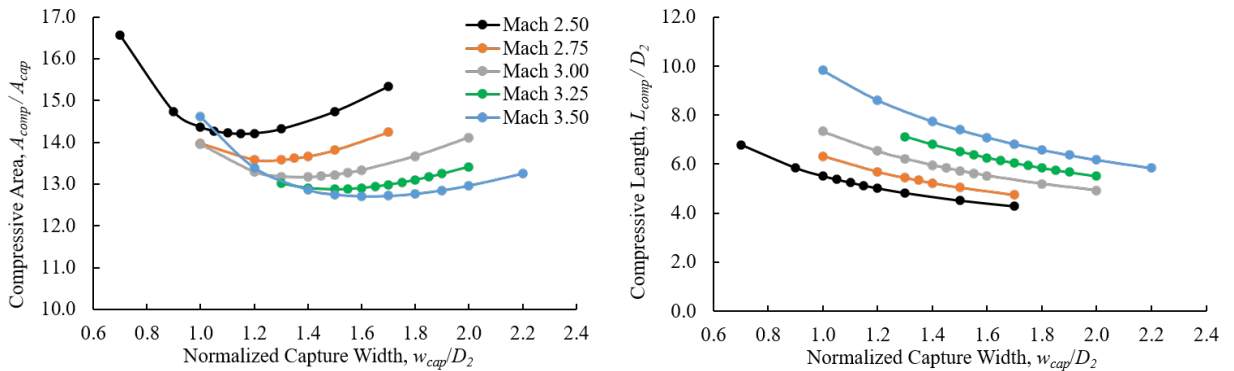


Figure 7. The variations as obtained from SUPIN of the compressive surface area (left) and the compressive length (right) with respect to the normalized capture width for inlets with 50% external compression.

The wave drag, compressive surface area, and compressive length varied with the level of compression split, as well as, the capture width. Figure 8 shows variations of wave drag and compressive surface area of the $M_0 = 3.0$ inlets with respect to the capture width for levels of compression split from 40/60 to 60/40. The plots of the left-hand-side of Figure 8 show that the wave drag was greater as the level of external compression increased. This was expected since for a greater level of external compression, the slope of the third stage and cowl lip exterior angle were greater.

The plots also indicate that as the level of external compression increased, the capture width that resulted in the lowest wave drag for that compression split decreased. The plots on the right-hand-side of Fig. 8 suggest that the capture widths yielding the lowest compressive surface areas were about $w_{cap}/D_2 \approx 1.5$ for all of the compression splits.

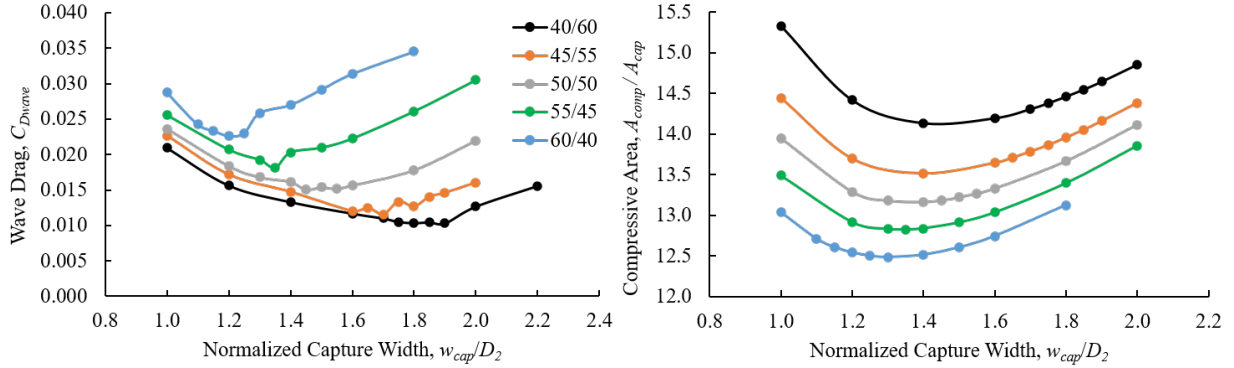


Figure 8. The variations as obtained from SUPIN of the cowl wave drag (left) and compressive surface area (right) with respect to the normalized capture width and compression split for $M_0 = 3$ inlets.

The change in the wave drag and the compressive area with respect to the level of external compression for inlets over the range of freestream Mach numbers are shown in Fig. 9. The capture width for each inlet was set to the value that produced the lowest wave drag for the respective freestream Mach number. The plots on the left-hand-side of Fig. 9 show that as the level of external compression is increased, the wave drag increased. As mentioned previously, a higher value of external compression was accompanied by an increase in the angle of the cowl lip, which lead to greater wave drag. The plots on the right-hand-side of Fig. 9 show that the compressive area decreased as the level of external compression increased. A higher value of external compression resulted in a shorter external supersonic diffuser, which also had less compressive area. A possible guideline from Fig. 9 is that levels of external compression from 40% to 60% provide a reasonable compromise between lower wave drag and lower compressive area and weight.

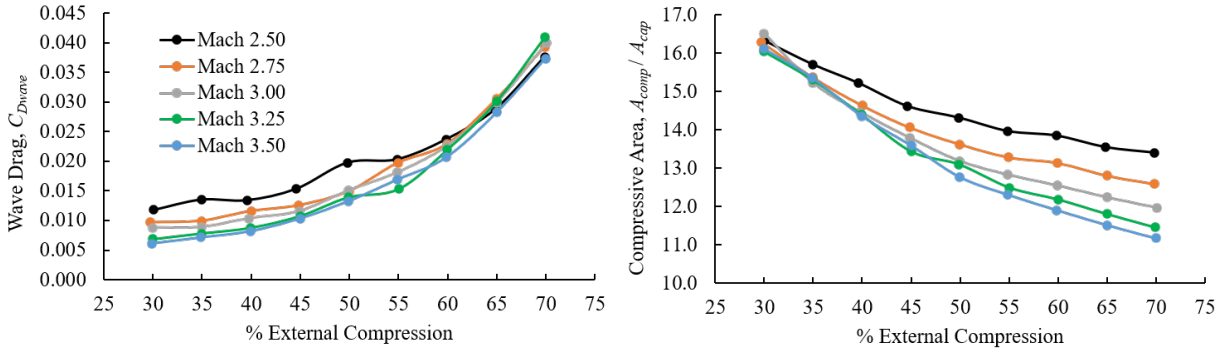


Figure 9. The variations as obtained from SUPIN of the cowl wave drag (left) and compressive surface area (right) with respect to the compression split.

Some interesting observations are noted for the variations of w_{cap}/D_2 , M_1 , and θ_{clin} with respect to M_0 . The plots of Fig. 10 show these variations for inlets with various compression splits for inlet configurations that provided the lowest values of C_{Dwave} . The plots on the left-hand-side show the variations of w_{cap}/D_2 . Since the choice of w_{cap}/D_2 for the lowest C_{Dwave} may be a range of values, the plots suggest a linear behavior of w_{cap}/D_2 with respect to M_0 . The plots of the image in the middle of Fig. 10 show a clear linear behavior of M_1 with M_0 . This linear behavior can be used to determine M_1 for a specified value of M_0 and compression split. The plots of the image on the right-hand-side of Fig. 10 suggest that as M_0 and the level of external compression increases, the change in θ_{clin} becomes less. The information of Fig. 10 could provide useful guidance for inlet design.

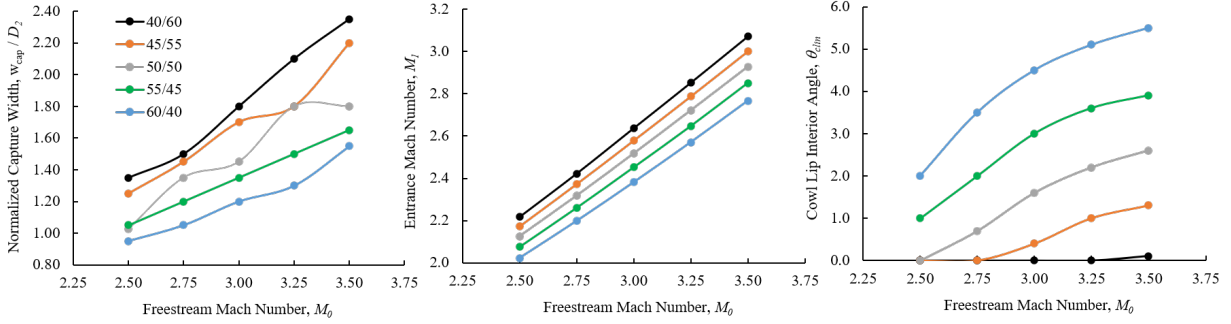


Figure 10. The variations as obtained from SUPIN of w_{cap}/D_2 for the minimum cowl wave drag, M_1 , and θ_{clin} with respect to M_0 for the inlets with 50% external compression.

B. Verification of the Inlet Design Methods

The verification of the SUPIN inlet design methods for the supersonic compression was explored through inviscid Wind-US CFD simulations of the inlets. As described above, SUPIN used the method-of-characteristics (MOC) approach, which assumed inviscid flow, to design the external supersonic diffuser and the throat section. Wind-US CFD simulations were performed using planar, two-dimensional flow domains that included the external flow and supersonic flow through the throat section. The internal outflow boundary was placed at station SD and an extrapolation boundary condition was applied for the supersonic outflow. The initial flow solution was specified as the freestream conditions. The simulations assumed inviscid flow. The Mach contours of Fig. 11 show the flow solution for 50/50 inlets at $M_0 = 2.5$, 3.0, and 3.5. The images show the leading-edge oblique shock passing slightly forward of the cowl lip to create the 1% supersonic spillage. The Mach lines of the isentropic second stage converge at the cowl lip. The flow was mostly uniform over the third stage leading into station 1. The cowl shock was formed and intersected the bodyside at the start of the shoulder. The cowl shock was mostly cancelled at the start of the shoulder, but weak oblique shocks do form and reflect through the remainder of throat section.

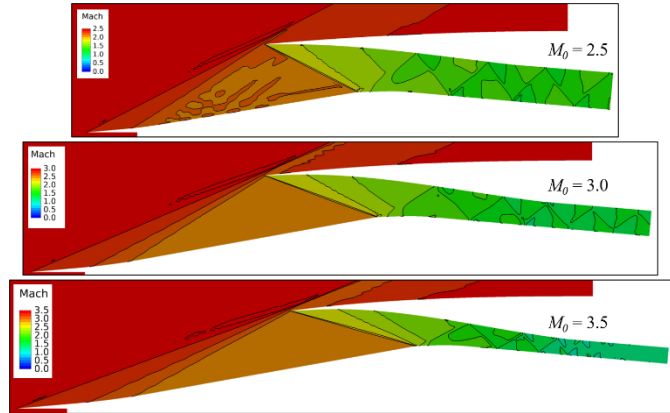


Figure 11. Mach number contours from inviscid Wind-US simulations for inlets with compression splits of 50/50.

Some verification of the methods for computing the cowl exterior wave drag were performed using the Wind-US inviscid simulations. SUPIN computed a Mach number and static pressure distribution on the profile of the cowl exterior and used the static pressure distribution to compute the wave drag. Figure 12 shows comparisons of the Mach numbers and static pressures on the cowl exterior from SUPIN and the Wind-US CFD simulations. There was generally good agreement of the results.

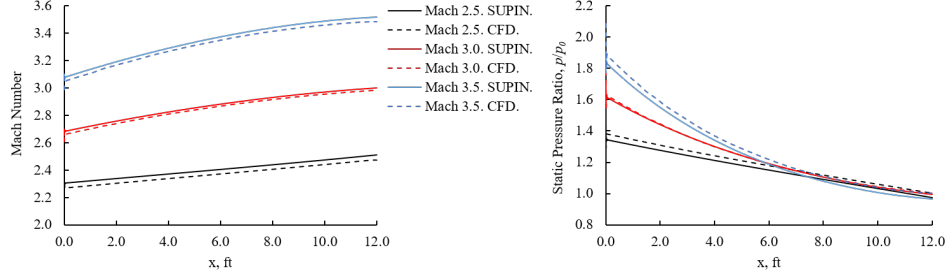


Figure 12. Comparison of Mach numbers and static pressures on the cowl exterior from SUPIN and inviscid Wind-US CFD simulations.

C. Throat Section without Bleed

The design of the throat section had the goal of providing the shortest length to efficiently perform internal supersonic compression while turning the flow toward the engine face. Designing the shortest throat section was consistent with the goal of reducing the compressive surface area, compressive length, and weight of the inlet. An efficient internal supersonic compression involved reducing or eliminating adverse interactions of the shock and Mach waves with the boundary layers, such as boundary layer separation. The design factors for the throat section included the cowl lip internal angle (θ_{clin}), normalized axial length of the throat section shoulder ($\Delta x_{cbsh}/D_2$), normalized length of the aft supersonic compression segment of the throat section (L_{shTX}/D_2), and slope of the aft bodyside segment of the throat section (θ_{cbTX}). These factors are illustrated in Fig. 3. As discussed previously, the factor θ_{clin} established the oblique cowl shock and was selected to result in the pressure rise of $(dp/dx)_{shock} \approx 1.72$. The forward part of the throat section was defined by the length L_{cb1sh} , as illustrated in Fig. 3, and was established by the intersection of the cowl shock with the bodyside, which defined the start of the shoulder. The factors $\Delta x_{cbsh}/D_2$ and L_{shTX}/D_2 established the length over which the internal supersonic compression was performed between the cowl shock and station TX. The factor θ_{cbTX} established the turning of the throat section. Most of the turning was performed over the shoulder length $\Delta x_{cbsh}/D_2$. The value of θ_{cbTX} was set by the condition that the flow at station TX should be directed toward the engine face. Most of the inlets used $\theta_{cbTX} = -5.0$ degrees.

The above factors defined the geometry of the bodyside of the throat section. The MOC methods within SUPIN were used to design the geometry of the cowl interior for the throat section and establish the Mach waves for the internal supersonic compression. The MOC methods required a Mach number distribution on the bodyside between the start of the shoulder to station TX. This distribution was defined as a linear deceleration of the Mach number from just downstream of the cowl shock / bodyside intersection to the specified bodyside Mach number at station TX (M_{cbTX}). As discussed previously, a value of $M_{cbTX} = 1.3$ was used for this study.

The study involved the use of Wind-US and FUN3D CFD simulations to properly capture the internal supersonic compression with the cowl shock and Mach waves and their interaction with the turbulent boundary layers. The study assumed supercritical flow conditions with supersonic flow through the entire throat section. The computational flow domain used a supersonic outflow boundary condition at station SD. The CFD simulations started from an initial flowfield defined by the freestream conditions and continued until iterative convergence was obtained to result in the steady-state flowfield. The design process involved selecting values of the design factors and then performing CFD simulations to obtain the flowfield for the resulting inlet configuration. The flowfield was assessed and the design factors modified for a subsequent CFD simulation.

The boundary layers on the surfaces of the throat section altered the area distribution of the inlet streamtube through the throat section. Within SUPIN, the design factor F_{cwnds} allowed a means to account for the displacement thickness and recover the intended area distribution of the inviscid MOC design method. The value of F_{cwnds} indicated an adjustment of the height at station TX from the bodyside to the cowl interior and was intended to be approximately equal to the boundary layer displacement thickness at station TX in units of feet. The CFD simulations for an inlet started with a value of F_{cwnds} that would be oversized to ensure supersonic flow through station TX without choking of the throat section. The value of F_{cwnds} was then intended to be reduced to achieve $M_{TX} \approx 1.3$.

The process for the design of the throat section using viscous CFD simulations can be summarized as follows:

1. The initial values for the factors were set to provide a conservative throat section that was intended to ensure supersonic, viscous flow through the throat section. These values corresponded closely to those used for the study of the external supersonic diffuser. The value of F_{cwnds} was set high to ensure that the flow at

station TX did not choke. The CFD simulations were performed for these inlet configurations to verify that supersonic flow through the throat section was established.

2. The cowl lip interior angle (θ_{clin}) was adjusted as needed to ensure little or no boundary layer separation occurred in the intersection of the cowl shock with the bodyside. Increasing the value of θ_{clin} lessened the pressure rise across the shock and helped to reduce boundary layer separation. Increasing θ_{clin} also decreased the angle of the cowl shock and resulted in larger values of L_{cb1sh} , which resulted in a longer inlet. Thus, an objective was to decrease θ_{clin} until just a very small separation bubble was observed at the point of intersection of the cowl shock with the bodyside.
3. The axial length of the shoulder ($\Delta x_{cbsh}/D_2$) was reduced to explore its effects on the formation of the Mach wave system responsible for turning the flow about the shoulder. The angle θ_{cbTX} also affected the formation of the Mach waves by setting the amount of turning. The objective was to reduce $\Delta x_{cbsh}/D_2$ until excessive thickening of the boundary layer or separation was encountered.
4. The length of the throat section from the end of the shoulder to station TX (L_{shTX}) was reduced until boundary layer separation or choking was observed. For some simulations, both $\Delta x_{cbsh}/D_2$ and L_{shTX} were reduced at the same time.
5. The factor F_{cws} was reduced to lower M_{TX} toward the desired value of $M_{TX} \approx 1.3$. This adjustment was sometimes performed along with reduction of $\Delta x_{cbsh}/D_2$ and L_{shTX} .

In summary, this process involved adjustment of the respective factors and executing the CFD simulation for that inlet configuration. Once the flowfield was assessed, another iteration of the process may be performed to further shorten the throat section. Eventually, the shortest throat section that provided for an efficient internal supersonic compression that avoided choking was obtained.

The process was applied to the design and analysis of throat sections for inlets for $M_0 = 2.5, 3.0$, and 3.5 with a compression split of 55/45. Table 3 lists the values of the respective factors for the shortest throat sections achieved for the inlets for each M_0 . The length of the shoulder was shortened significantly from the length used for the studies of the external supersonic diffuser. When lengths were shortened below those listed in Table 3, the internal supersonic compression caused the throat sections to choke and the inlets to unstart. For the $M_0 = 3.0$ inlet, the value of θ_{cbTX} was varied to explore its effect on the internal supersonic compression. When $\theta_{cbTX} \leq -7$ degrees, the increased amount of turning about the shoulder resulted in some boundary layer separation on the bodyside downstream of the shoulder. Thus, the value of $\theta_{cbTX} = -5$ degrees was used for the final configurations for all the inlets, which roughly directed the flow at station TX toward the engine face.

Table 3. Properties of the throat section without bleed.

M_0	w_{cap}/D_2	θ_{clin}	$(dp/dx)_{shock}$	$\Delta x_{cbsh}/D_2$	L_{shTX}/D_2	θ_{cbTX}	F_{cws}
2.5	1.0	0.0	1.77	0.2	0.6	-5.0	0.10
3.0	1.2	2.2	1.78	0.2	0.7	-5.0	0.15
3.5	1.6	6.0	1.50	0.3	0.7	-5.0	0.16

The Mach number contours from FUN3D simulations for the inlets described by Table 3 are shown in Fig. 13. The images show the internal supersonic compression formed with the oblique cowl shock and some weak reflected shocks. The boundary layers show some low momentum regions as indicated by the darker blue colors. The contours for the $M_0 = 3.5$ inlet, show a small region of separated flow at the intersection of the cowl shock with the start of the shoulder. The key observation was that the internal supersonic compression with the boundary layers was able to form without choking of the flow and the flow remained started for all inlets. The distribution of Mach number from the FUN3D simulations, across station TX for each inlet is presented in Fig. 14. The distance coordinate indicates the distance from the bodyside to the cowlside units of feet. The distributions show that the core Mach number was above the Mach 1.3 goal and there was a sizable boundary layer thickness on the bodyside for all the inlets. The flowfields indicated the need for bleed to reduce any separation bubbles and reduce the boundary layer thickness at station TX, which will be addressed in the next subsection.

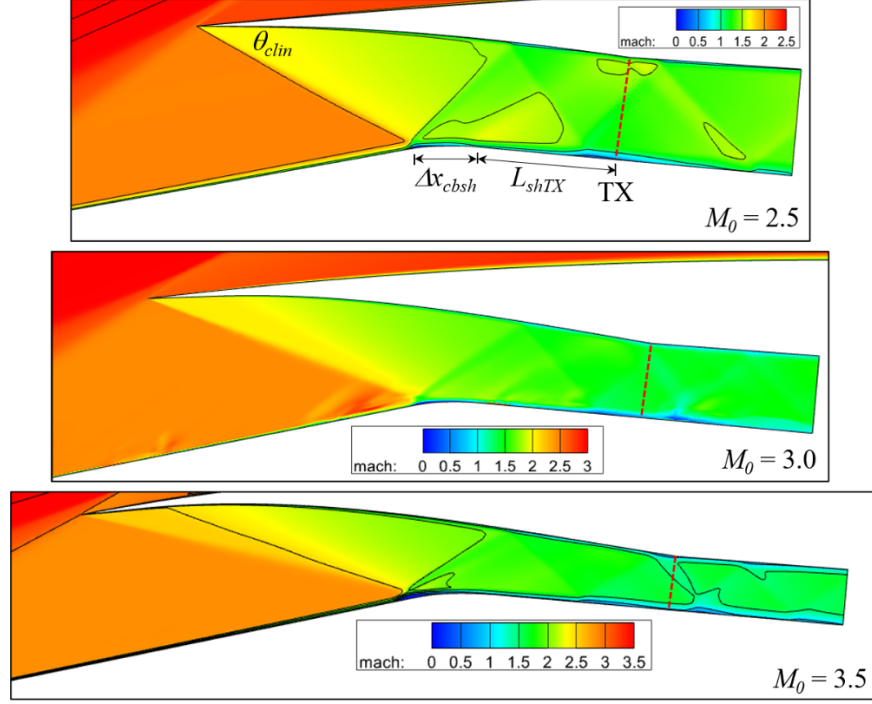


Figure 13. Mach number contours from planar FUN3D CFD simulations for $M_0 = 2.5$ (top), 3.0 (middle), and 3.5 (bottom).

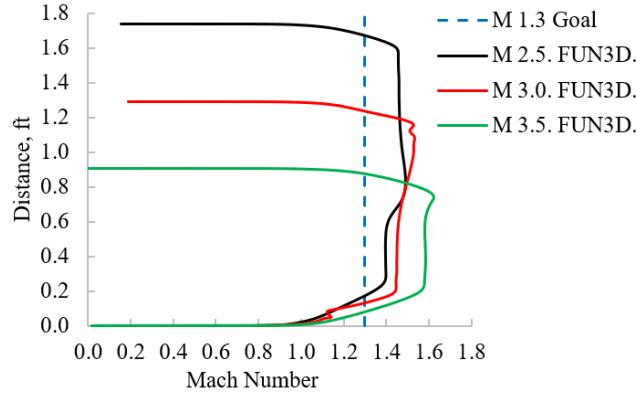


Figure 14. Mach number distribution at station TX from the bodyside to the cowl at the inlet symmetry plane for FUN3D CFD simulations of the throat sections with no bleed.

D. Throat Section with Bleed

The use of porous bleed regions within the throat section was studied. The purpose of the bleed regions was to minimize adverse interactions of the cowl shock with the inlet boundary layers and provide uniform conditions for a normal, terminal shock to be established just downstream of station TX. The inlets for this study were designed using information learned from the studies discussed in the previous subsections. Table 4 summarizes the values of some of the key design factors. All the inlets were designed to provide a compression split of 55/45 with the internal compression resulting in $M_{TX} = 1.3$. Table 4 also lists the static pressure rise across the cowl shock, $(dp/dx)_{shock}$, as obtained from SUPIN.

The study involved specifying the initial locations, extents, and plenum pressures for each bleed region based on experience with past inlet designs. Wind-US CFD simulations were then performed to obtain the flow through the throat section and assess the performance of the inlet and bleed regions. Modifications to the inlets or bleed regions were made and the CFD simulations were repeated until a desirable throat section configuration was obtained. The goals of the design process included establishing boundary layers with little or no separation, minimizing the extent

of the bleed regions, minimizing the rate of bleed flow, and establishing flow conditions at station TX with mostly uniform Mach 1.3 flow with relatively thin boundary layers.

The study assumed five bleed regions for the inlets, as labeled in Fig. 15. Bleed region 1 was located at the shoulder on the bodyside and placed downstream of the intersection point of the cowl shock with the bodyside. Bleed region 2 was located on the sidewall and extended from the cowl lip to the start of the shoulder. This placed bleed region 2 to be roughly aligned with the cowl shock and was meant to reduce adverse effects of the interaction of the cowl shock with the sidewall. Bleed regions 3, 4, and 5 were placed about station TX with the expectation that they would reduce the interactions of the terminal shock. Bleed regions 3, 4, and 5 were placed on the bodyside, sidewall, and cowl interior, respectively. All bleed regions were assumed to contain circular bleed holes oriented normal to the inlet surface. The porosity of the bleed holes was assumed to be 40%, which meant that the area of the hole opening occupied 40% of the surface area of the respective bleed region. The Slater bleed model was used to establish the local normal velocity at the bleed region surface based on the local flow conditions, bleed region properties, and the plenum pressure.

One design factor for the throat section involved a modification of the height of the cross-section at station TX through the SUPIN input factor F_{cwnds} to accommodate for the blockage of the boundary layer. The value of F_{cwnds} roughly corresponds to the displacement thickness of the boundary layer at station TX. During the design process, a conservative value of F_{cwnds} was initially chosen. As the CFD simulations were performed and the flow at station TX evaluated, the value of F_{cwnds} was reduced to approach $M_{TX} = 1.3$. In cases where F_{cwnds} was reduced too much, the inlet would choke at station TX. Table 4 lists the final values of F_{cwnds} used for the inlets.

The design process was performed for inlets for $M_0 = 2.5, 3.0$, and 3.5 . Figures 16, 17, and 18 show images of the Mach number contours on the inlet symmetry planes and at axial stations along the inlets. The blue arrows in the images indicate local momentum vectors within the bleed regions at the individual grid points within the bleed regions. The lengths of the arrows vary over the bleed regions to indicate varying levels of local bleed due to variations of the local static pressure. The inlets shown reflect the result of several design iterations performed for each inlet to refine the inlet performance. The Mach number contours show that acceptable internal supersonic compression was obtained. The cowl shocks were mostly cancelled at the shoulder and a distributed compression due to Mach waves occurs downstream of the cowl shock. The inlets had essentially no boundary layer separation through the throat section. It was found that the best location for bleed region 1 was just downstream of the cowl interaction point. For the $M_0 = 3.5$ inlet, the bleed region 6 was added on the bodyside just upstream of the cowl shock interaction point to reduce the thickness of the boundary layer to help eliminate the boundary layer separation due to the cowl shock. Bleed regions 1 and 6 worked together but needed to be separated with individual plenum pressures since the local pressures downstream of the cowl shock were greater than those upstream of the shock. The $M_0 = 3.5$ inlet did indicate a weak shock being reflected from the shoulder. The axial Mach number contour planes show the development of a vortex at the lower corner of the throat section. It is believed this vortex forms as part of the boundary layer on the sidewall is directed downward. Upon reaching the corner, the flow is turned, which forms the vortex. This vortex is a common feature of two-dimensional inlets with internal supersonic compression and its development has implications when it interacts with the normal terminal shock, which will be discussed in the next subsection.

The total amount of bleed used for each inlet is listed in Table 4. It is expected that a greater amount of bleed is needed as M_0 increases, which is not the case for the inlets for $M_0 = 2.5$ and 3.0 . This likely indicates that the design process could use further iterations to refine the throat section and bleed regions.

The primary goal of the throat section design is to obtain a uniform profile at station TX with $M_{TX} \approx 1.3$. Figure 19 shows the variation of the Mach number at station TX on the inlet symmetry plane for each of the inlets. The distance plotted along the ordinate is the distance from the bodyside to the cowl at station TX. As M_0 increases, the height of the throat at station TX decreases to perform greater internal supersonic compression, which is reflected in Fig. 19. The design process was successful in achieving the desired conditions at station TX; however, further design iterations are needed to reduce the core Mach numbers for the $M_0 = 3.5$ inlet, as well as, the peaks in the Mach number near the bodyside and cowlside caused by the bleed regions.

Table 4. Properties of the throat section with bleed.

M_0	w_{cap}/D_2	θ_{clin}	$(dp/dx)_{shock}$	$\Delta x_{cbsh}/D_2$	L_{shTX}/D_2	θ_{cbTX}	F_{cwnds}	W_{bleed}/W_{cap}
2.5	1.0	0.0	1.79	0.2	0.6	-5.0	0.00	0.0667
3.0	1.2	2.2	1.78	0.2	0.7	-5.0	0.01	0.0598
3.5	1.6	6.0	1.50	0.3	0.7	-5.0	0.06	0.0879

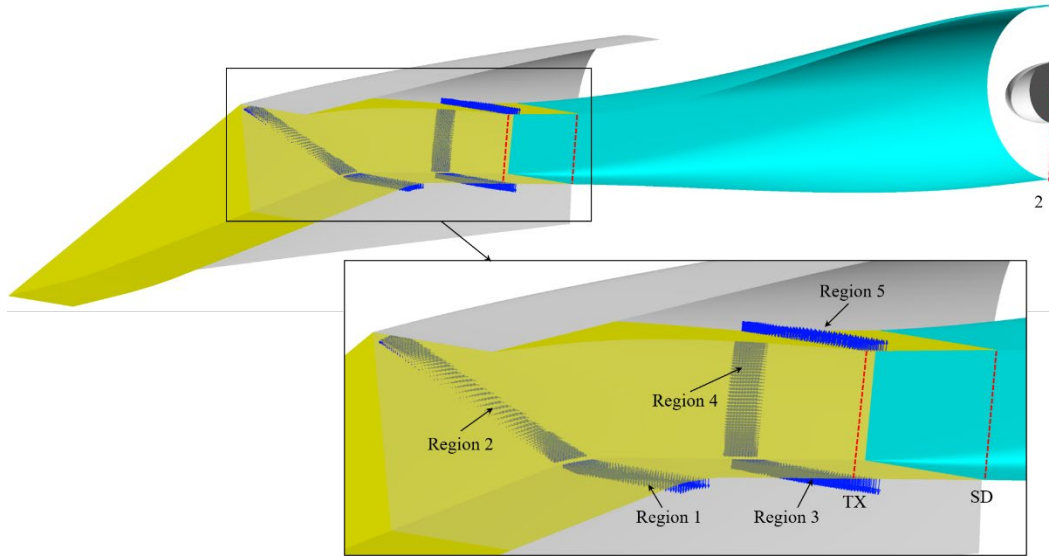


Figure 15. Bleed regions for the Mach 2.5 inlet.

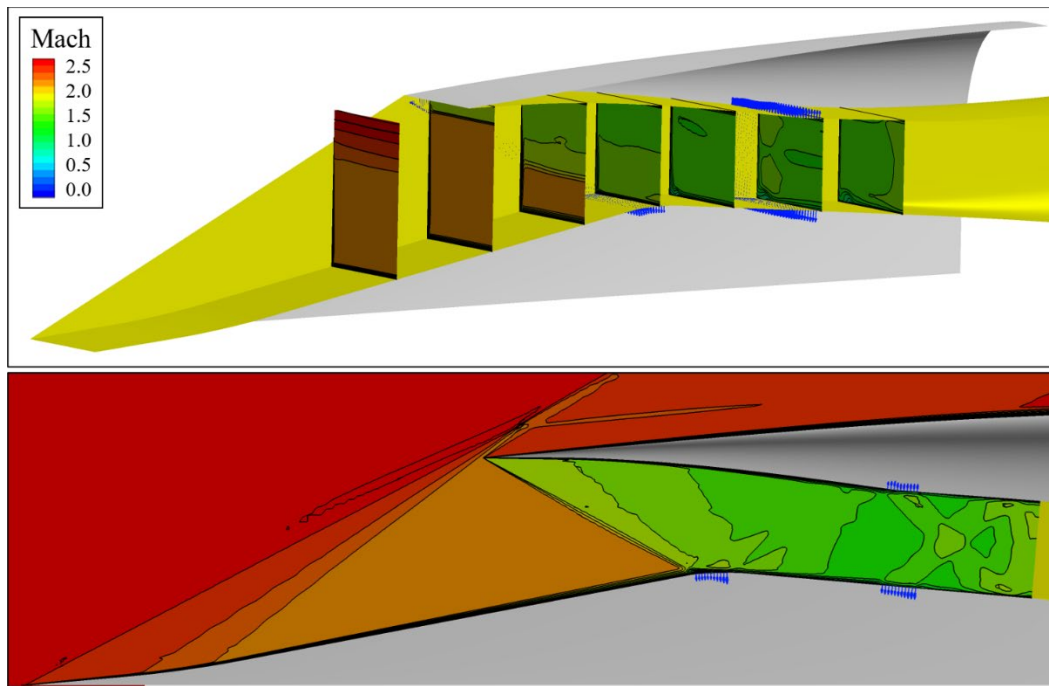


Figure 16. Mach contours from a Wind-US simulation of the flow within the throat section of a $M_0 = 2.5$ inlet with porous bleed regions.

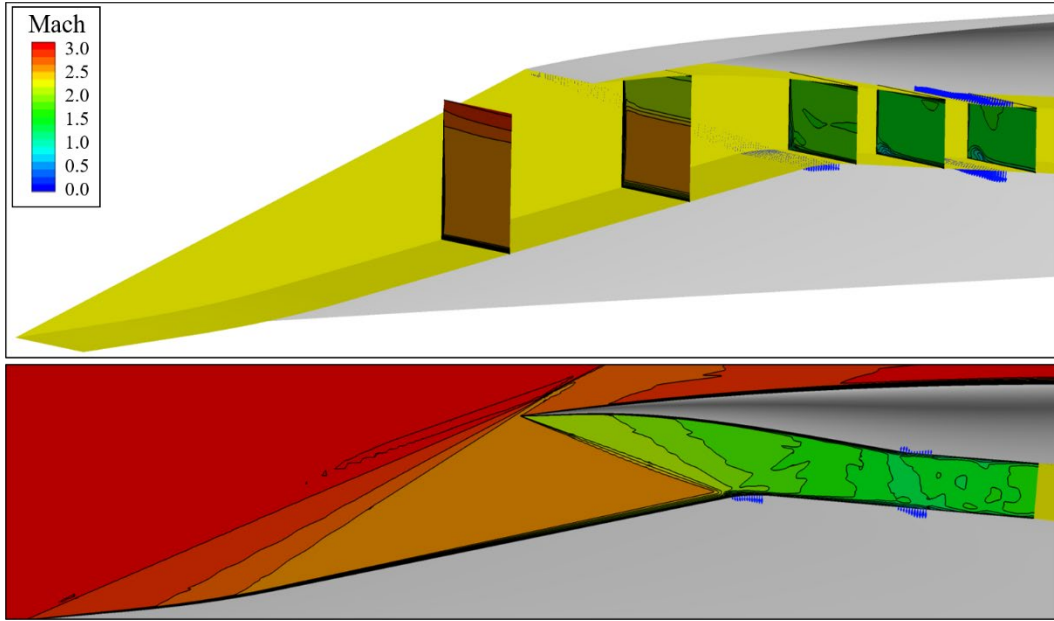


Figure 17. Mach contours from a Wind-US simulation of the flow within the throat section of a $M_0 = 3.0$ inlet with porous bleed regions.

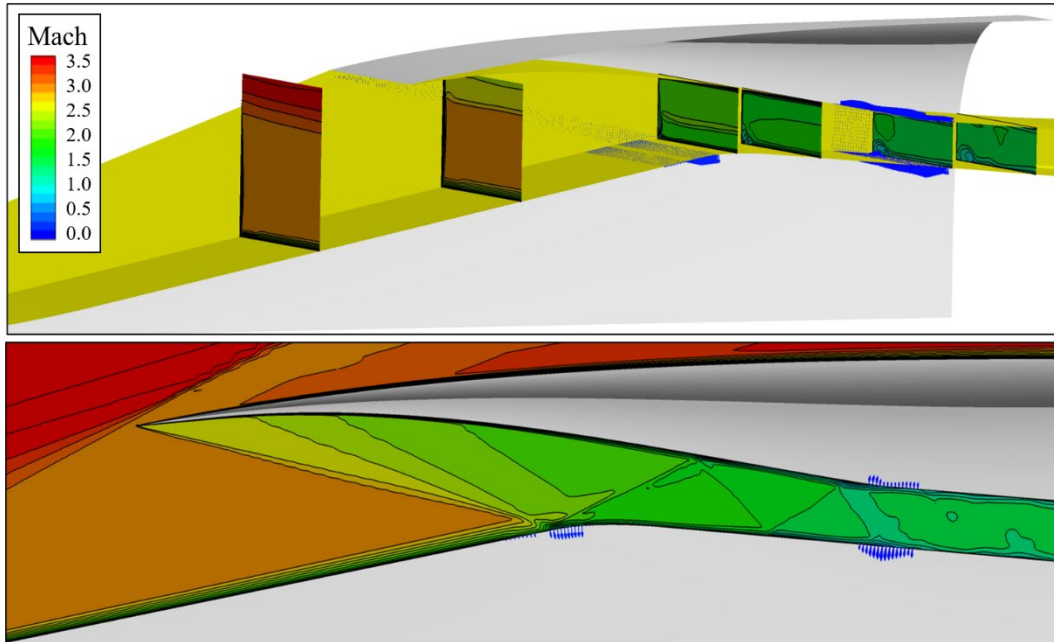


Figure 18. Mach contours from a Wind-US simulation of the flow within the throat section of a $M_0 = 3.5$ inlet with porous bleed regions.

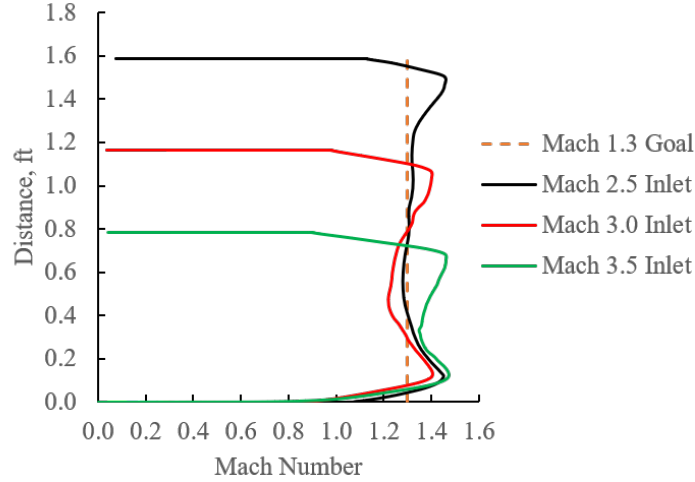


Figure 19. Mach number distribution at station TX from the bodyside to the cowl at the inlet symmetry plane.

E. Subsonic Diffuser

The design of the subsonic diffuser involved establishing the terminal shock system to create subsonic flow within the subsonic diffuser with the inlet flow at the critical design condition with the terminal shock near station TX. The primary design factor for the subsonic diffuser was the length from station SD to station 2. The approach of this study was to establish the length based on an equivalent conical diffusion angle using the known cross-sectional areas at stations SD and 2. An equivalent conical angle of 2 degrees was chosen as a conservative choice for the subsonic diffuser lengths. The lengths likely can be shortened with further design iterations and CFD simulations. The vertical placement of the engine face was specified to be aligned with the vertical position of the midpoint of station SD. This reduced the amount of turning required for the subsonic diffuser. The surfaces of the subsonic diffuser were created within SUPIN as a network of NURBS curves from station SD to station 2. As a result, the transition from a rectangular cross-section at station SD to a circular cross-section at station 2 was formed. This design process was applied to the $M_0 = 2.5$, 3.0, and 3.5 inlets of the previous section. Table 5 lists the resulting lengths of the subsonic diffusers for these inlets. Figure 20 shows the three inlets and illustrates the relative size between the inlets. Figure 21 shows various views of the inlet for $M_0 = 3.5$ and illustrates the transition in the width of the subsonic diffuser from the capture width of $w_{cap}/D_2 = 1.6$ used for the throat section up to station SD to the width of the engine face, which was D_2 .

CFD simulations were performed to analyze the flow through the subsonic diffuser and allow computation of the inlet total pressure recovery and distortion. Subsonic flow and the terminal shock structure were formed within the subsonic diffuser using the choked outflow nozzle. The simulations involved adjusting the area of the nozzle throat to provide the necessary back-pressure to position the terminal shock close to station TX and approach the design corrected flow rate at the engine face as listed in Table 1. The simulations of the $M_0 = 2.5$ inlet were able to reach the critical operating condition with the engine-face corrected flow of the simulation approximately equal to the value listed in Table 1. The simulations of the $M_0 = 3.0$ and 3.5 inlets were still slightly supercritical.

The Mach number contours on the symmetry plane and along axial stations through the inlets are shown in Figure 22. All the inlets indicated a low momentum region that formed at the lower corner downstream of station TX and grew through the subsonic diffuser. The $M_0 = 2.5$ inlet shows a small low-momentum region forming at the upper corner. The inlets have some separation in the subsonic diffuser; however, there is no separation at the engine face. The blue arrows showing the local bleed momentum shows the desired behavior in that in regions of high pressure, such as downstream of the terminal shock, more bleed is extracted, which helps stabilize the terminal shock within the bleed region. The bleed plenum pressure remains constant at the specified value, which suggests that the bleed system would incorporate a control mechanism to vary the exit of the bleed system to maintain the desired bleed plenum pressure. Further refinement of the inlets and bleed regions are needed to improve the inlet performance.

The performance of the inlets as obtained from the CFD simulations is summarized in Table 5. The engine-face Mach number (M_2) listed in Table 5 for the $M_0 = 2.5$ inlet is still slightly above the respective value listed in Table 1, which indicates an adjustment to the sizing of the inlet might be needed. The values of M_2 listed in Table 5 for the $M_0 = 3.0$ and 3.5 inlets reflect the supercritical operating condition of these inlet simulations. The spillage, bleed, and

engine-face flow ratios are listed for each inlet along with the inlet total pressure recoveries and the distortion indices. The bleed rate for the $M_0 = 2.5$ inlet is likely high compared to past inlet studies, but the rates for the $M_0 = 3.0$ and 3.5 inlets are likely close to those of past studies. The total pressure recoveries listed in Table 5 were all within 2 to 4 percent of the respective values for the goals listed in Table 1. All the inlets indicated hub distortion, which means DPC/P was calculated as the average of the two rings nearest the hub.

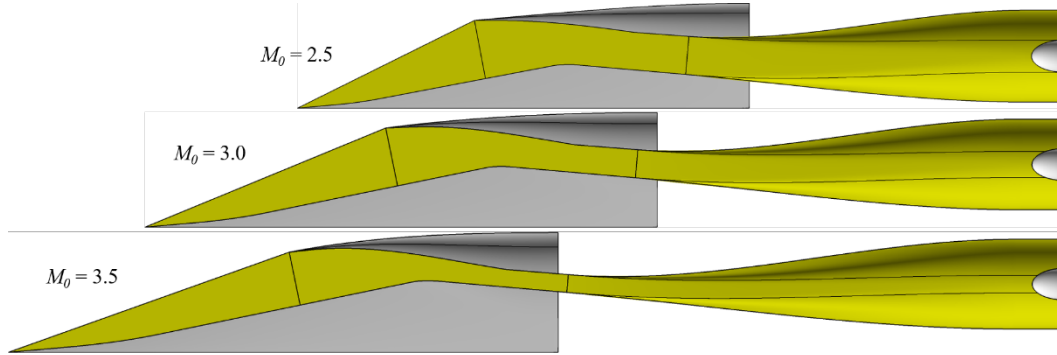


Figure 20. Comparison of inlet size for $M_0 = 2.5$, 3.0, and 3.5.

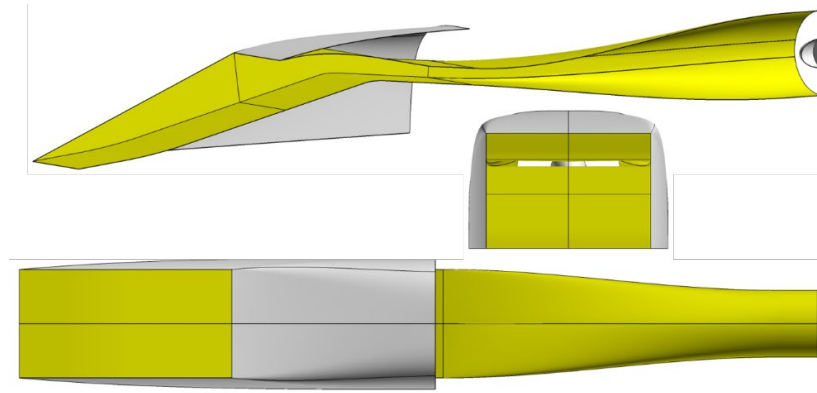


Figure 21. Inlet for $M_0 = 3.5$.

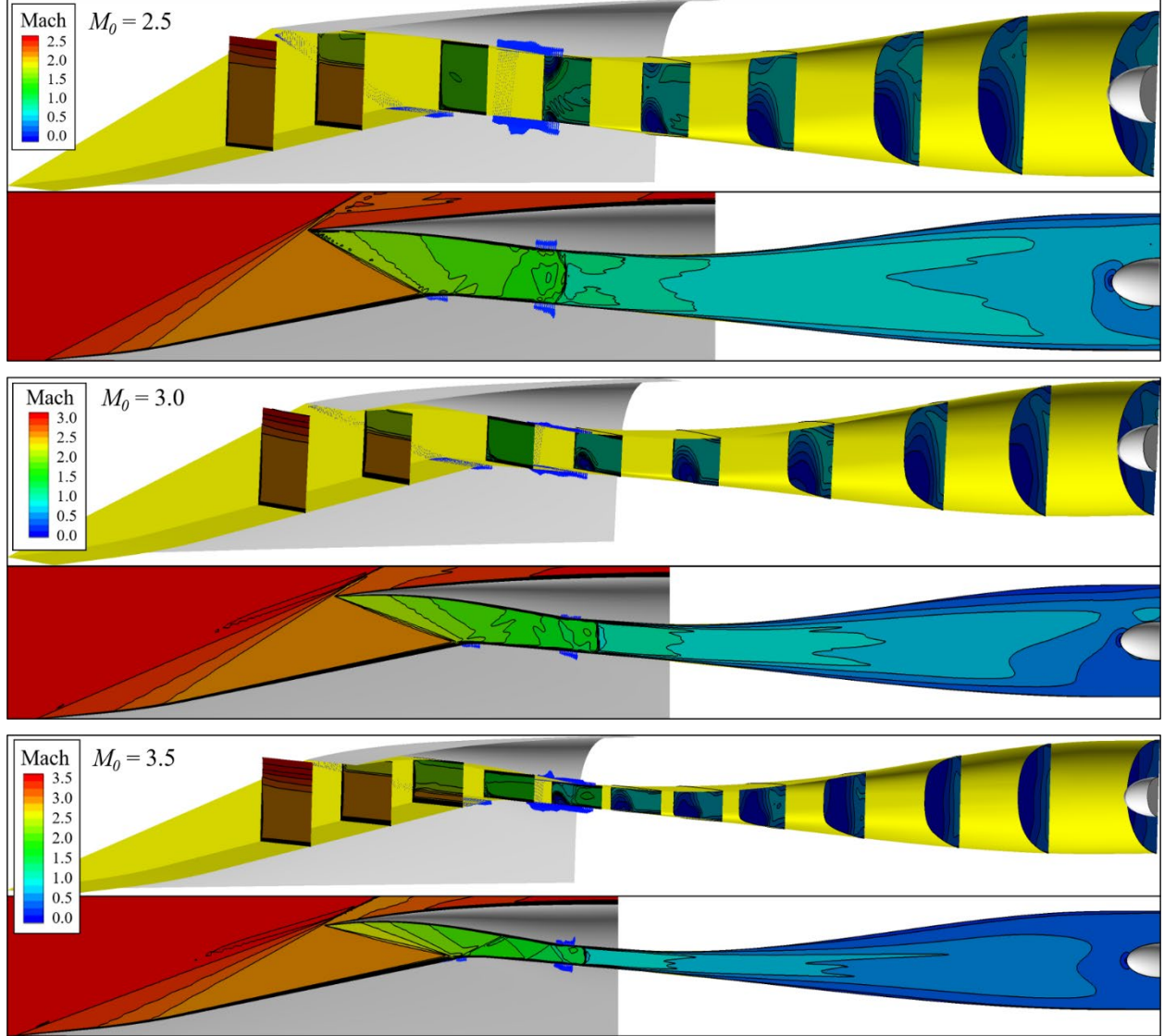


Figure 22. Mach number contours on the symmetry plane and along axial cross-sections for inlets for $M_0 = 2.5$ (top), 3.0 (middle), and 3.5 (bottom).

Table 5. Inlet properties.

M_0	L_{SD}/D_2	W_{spit}/W_{cap}	W_{bleed}/W_{cap}	W_2/W_{cap}	p_{t2}/p_{t0}	M_2	DPR/P	DPC/P
2.5	3.95	0.0197	0.0871	0.8932	0.8320	0.3572	0.0359	0.0510
3.0	4.61	0.0236	0.0577	0.9187	0.7752	0.3352	0.0427	0.0353
3.5	5.44	0.0266	0.1094	0.8640	0.7231	0.2811	0.0295	0.0331

V. Conclusions

This paper examined some design factors for two-dimensional mixed-compression inlets for Mach 2.5 to 3.5 to develop general guidance for the selection of the values for those factors. Of interest was to understand the effects of design factors on the inlet aerodynamic performance as characterized by the inlet flow rates, cowl wave drag, and the total pressure recovery and distortion at the engine face. The inlet was examined in three parts, the external supersonic diffuser, the throat section, and the subsonic diffuser.

For the external supersonic diffuser, the key inlet characteristics were the external / internal compression split and the width of the capture cross-section. It was found that compression splits between 40/60 to 60/40 using a three-

stage external supersonic diffuser provided a good compromise between lower wave drag coefficients and smaller external supersonic diffusers. The normalized capture widths affected the wave drag and compressive area and preferred values were found to range from $w_{cap}/D_2 = 1.0$ for $M_0 = 2.5$ to $w_{cap}/D_2 = 1.6$ for $M_0 = 3.5$. One observation was that the lowest wave drag coefficients were found with a capture aspect ratio of $w_{cap}/h_{cap} \approx 1.5$. Another observation was that the lowest compressive surface areas were found with a normalized capture width of $w_{cap}/D_2 \approx 1.5$.

The factors for the throat section mainly affected the length of the throat section. A goal was to reduce the lengths, but that there seemed to be a minimum length needed to allow for sufficient length to distribute the internal supersonic compression. The use of porous bleed was beneficial for removing boundary layer separation and for establishing a uniform distribution of $M_{TX} = 1.3$ flow at the inlet throat station, TX. Further study is needed to refine the placement and properties of the bleed regions while developing an understanding of how much bleed is needed.

The primary factor for the subsonic diffuser was its length, but while the studies demonstrated acceptable subsonic diffuser performance, further study is needed on reducing the length of the subsonic diffuser. In addition, the refinement of the area distribution through the diffuser and use of vortex generators need further study for improving the flow through the subsonic diffuser.

There is interest in expanding the study to inlets for Mach numbers up to $M_0 = 5$. In addition, a companion project has been exploring design factors for axisymmetric spike inlets and the comparison between axisymmetric spike and two-dimensional inlets would be informative.

Acknowledgments

This work was funded by the Hypersonic Technology Project (HTP) under NASA's Advanced Air Vehicles Program (AAVP). Also, the authors would like to acknowledge the NASA Advanced Supercomputing (NAS) Division at NASA Ames, who maintain the High-End Computing Capability (HECC) that was utilized in completing this work.

References

- [1] Deloitte, "Commercial Hypersonic Transportation Market Study," April 2021.
<https://www2.deloitte.com/us/en/pages/energy-and-resources/articles/rise-of-hypersonics.html>
- [2] SAIC and Bryce Tech, "Independent Market Study, Commercial Hypersonic Transportation," April 2021.
<https://brycetech.com/reports>
- [3] Bowditch, D.N., "Some Design Considerations for Supersonic Cruise Mixed Compression Inlets," NASA TM X-71460, November 1973.
- [4] Connors, James F., Meyer, Rudolph C., "Design Criteria for Axisymmetric and Two-Dimensional Supersonic Inlets and Exits," NACA TN-3589, January 1956.
- [5] Slater, John. W., "Design Factors for Two-Dimensional, External-Compression Supersonic Inlets," AIAA 2020-2090, January 2020.
- [6] Slater, J. W., "SUPIN: A Computational Tool for Supersonic Inlet Design," AIAA Paper 2016-0530, January 2016.
- [7] Walatka, P. P., Buning, P. G., Pierce, L., and Elson, P. A., "PLOT3D User's Manual," NASA-TM-101067, March 1990.
- [8] Society of Automotive Engineers (SAE), "Aircraft Propulsion System Performance Station Designation and Nomenclature," Aerospace Standard (AS) 755, December 1997.
- [9] Mattingly, J.D., W.H. Heiser, and D.H. Daley, Aircraft Engine Design, AIAA Education Series, New York, 2002.
- [10] MIL-E-5007E, "Engines, Aircraft, Turbo-Jet, and Turbofan, General Specifications For," September 1983.
- [11] Society of Automotive Engineers (SAE), "Gas Turbine Engine Inlet Flow Distortion Guidelines," SAE ARP 1420, Rev. B, February 2002.
- [12] Yoder, D. A., "Wind-US User's Guide: Version 3.0," NASA TM 2016-219078, March 2016.

- [13] 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.
- [14] Slater, J.W., “Improvement in Modeling 90-degree Bleed Holes for Supersonic Inlets,” *AIAA Journal of Propulsion and Power*, Vol. 28, No. 4, pp 773-781. July-August 2012. Also, AIAA-2009-0710, January 2009.
- [15] Biedron, Robert T., Carlson, Jan-René, Derlaga, Joseph M., et. al., “Fun3D Manual: 13.7,” NASA/TM-20205010139, November 2020.
- [16] Baurle, R. A. and Norris, A. T., “A Source-Term Based Boundary Layer Bleed/Effusion Model for Passive Shock Control,” JANNAF, 2011.
- [17] Nussdorfer, T. J., “Some Observations of Shock-Induced Turbulent Separation on Supersonic Diffusers,” NACA RM E51L26, May 1954.