

# Planar Inlet Design and Analysis Process (PINDAP)

John W. Slater\*

*NASA Glenn Research Center, Brook Park, Ohio, 44135*

and

Christopher R. Gruber†

*University of Notre Dame, South Bend, Indiana, 46556*

**The Planar Inlet Design and Analysis Process (PINDAP) is a collection of software tools that allow the efficient aerodynamic design and analysis of planar (two-dimensional and axisymmetric) inlets. The aerodynamic analysis is performed using the Wind-US computational fluid dynamics (CFD) program. A major element in PINDAP is a Fortran 90 code named PINDAP that can establish the parametric design of the inlet and efficiently model the geometry and generate the grid for CFD analysis with design changes to those parameters. The use of PINDAP is demonstrated for subsonic, supersonic, and hypersonic inlets.**

## I. Introduction

THE aerodynamic design of inlets for air-breathing propulsion systems across the Mach number range have traditionally relied on basic planar flow assumptions and analysis. Planar flow includes both two-dimensional and axisymmetric flow. Many of the inlets for subsonic commercial transport aircraft have an axisymmetric shape. Many of the inlets for supersonic military and commercial aircraft (production and conceptual) have either an axisymmetric or two-dimensional shape. Many of the inlets for vehicles envisioned for hypersonic speeds have either an axisymmetric or two-dimensional shape. The use of a planar flow assumption simplifies the geometry and flow analysis, which is desired during the preliminary design phase when several configurations may be studied. The planar flow analyses may also serve as reference flows for more complex inlet shapes. Thus, planar flow analysis remains a part of inlet design.

The methods of computational fluid dynamics (CFD) are being used routinely for the aerodynamic analysis of inlet flow fields. The work here uses the Wind-US CFD code of the NPARC Alliance<sup>1</sup>. Two of the most time-consuming aspects of CFD analysis are the modeling of the geometry and the generation of the grid. For planar flow, the geometry is relatively simple and consists of lines and curves. The grids are also planar. The work here uses structured, multi-zone grids. From the simulated flow field, the performance of the inlet can be determined. These performance measures include total pressure recovery, distortion, engine flow, bleed flow, and spillage.

In response to the need to efficiently conduct planar inlet aerodynamic design and analyses, a small-scale project has been conducted to develop the Planar INlet Design and Analysis Process (PINDAP). PINDAP mainly collects software tools that work together with Wind-US. The next section will discuss all of the elements. A major element in PINDAP is a Fortran 90 code named PINDAP that establishes the parametric design of the inlet and efficiently models the geometry and generates the grid. The design and analysis process will involve changes in the design parameters and grid properties. As these are changed, PINDAP quickly regenerates the geometry and grid for further flow analysis. PINDAP also helps set up boundary condition information for the Wind-US CFD code. Section 3 will provide an overview of PINDAP.

Examples of the use of PINDAP for subsonic, supersonic, and hypersonic inlets are provided in Section 4. The cases are fairly generic and focus on the process rather than the design results. One aspect of PINDAP is the use of methods of statistical design of experiments (DoE) and response surface methods (RSM) to allow the intelligent examination of the design factors of the parameterized inlet design to approach an optimum inlet design. This aspect is only examined here in the example of the hypersonic inlet design.

The development of PINDAP is continuing and the last section discusses some future directions for PINDAP. PINDAP will eventually be made available to interested US citizens.

---

\* Aerospace Engineer, Inlet Branch, MS 86-7. AIAA Senior Member.

† LERCIP Summer Intern, Inlet Branch, MS 86-7. AIAA Student Member.

This material is declared a work of the U.S. Government and is not subject to copyright protection in the United States.

## II. Software Elements of PINDAP

The Planar INlet Design and Analysis Process (PINDAP) is a collection of loosely-coupled software elements that work together to improve the flow of information during the aerodynamic design of an inlet system. Central to PINDAP is a Fortran 90 code named PINDAP. The software elements of PINDAP include:

- 1) PINDAP is a Fortran 90 code that encodes the inlet design parameterization and creates the geometry, grid, and boundary condition information for the inlet geometry. The inputs to PINDAP are contained within an ASCII input file. PINDAP outputs a multi-zone, structured grid file (run.x) in Plot3d format and a boundary condition file (gman.com) for use by GMAN.\*
- 2) \*CFCNVT converts the Plot3d file (run.x) to the common grid (run.cgd) file format.
- 3) GMAN executes in batch mode to quickly set the boundary conditions within the common grid file (run.cgd) using the boundary condition file (gman.com) as input.
- 4) Wind-US is the CFD flow solver. Wind-US solves the Reynolds-Averaged Navier-Stokes equations for turbulent, compressible flow using a cell-vertex, finite-volume, time-marching approach. Spatial accuracy is formally second-order using the Roe flux-difference splitting upwind formulation. Turbulence is modeled using the Spalart-Allmaras or SST models. Wind-US is capable of solving for flows of speeds ranging from low subsonic to hypersonic. Wind-US is distributed by the NPARC Alliance.
- 5) \*CFPOST performs some simple visualization of the flow field and extracts flow field information such as mass flows, total pressure recoveries, distortion, and other derived flow field information.
- 6) Design-Expert is a statistical design of experiments (DoE) software package that performs statistical analysis of the DoE design and can apply response surface methods (RSM) to examine design factors.<sup>2</sup>

\*The CFCNVT, GMAN, and CFPOST elements are utilities that are distributed with the Wind-US CFD code.

## III. PINDAP

The operations of the PINDAP code include 1) form the geometry model of the inlet, 2) establish the grid topology, 3) generate the structured, multi-zone grid, and 4) output files for use with the Wind-US CFD code. The input to PINDAP is an ASCII text file.

### A. PINDAP Geometry Model

The PINDAP geometry model was established to provide a simple description of the geometry of planar inlets while allowing a wide variety of inlet shapes. The geometry model assumes that the inlet consists of combinations of a body and a cowl. The shape of the body and cowl can vary based on the design conditions, such as Mach number. Both the body and cowl can have an interior and exterior component. In addition to the inlet surfaces, the geometry model describes the boundaries of the enclosed flow domain upon which the flow equations are solved. For a planar analysis, the flow domain is a planar area. The flow domain is divided into zones. A structured grid is then generated within each zone. The PINDAP geometry model consists of the following five *components*:

- 1) **Body** (BOD). This component models the geometry such as the spinner of a subsonic inlet, the centerbody of a supersonic inlet, and the body of a 2D hypersonic inlet / vehicle. The start of the body component is referred to as the nose.
- 2) **Body Exterior** (BEX). This component models the geometry of the body that is external to the inlet flow. One example is the external (top) surface of a 2D hypersonic body. This external component is needed for simulations at non-zero angle-of-attack. The body external component also starts at the nose. If the end of the body exterior connects with the end of the body component, the point is referred to as the tail.
- 3) **Cowl Interior** (CIN). This component models the geometry of the interior of a cowl. The cowl interior is wetted by flow entering the inlet. The start of the cowl interior is the cowl lip.
- 4) **Cowl Exterior** (CEX). This component models the geometry of the exterior of a cowl. The cowl exterior also starts at the cowl lip. If the end of the cowl exterior connects with end of the cowl interior, the connecting point is referred to as the trailing edge.
- 5) **Domain** (DOM). This component models the geometry that defines the boundaries of the flow domain and the boundaries of the grid zones. The flow domain must be an enclosed area for planar domains.

Modeling the geometry in terms of components allows a logical grouping of the geometry that can be used to simplify other tasks of PINDAP. An inlet geometry model can have each of these components in a variety of

combinations. An inlet is not required to have all of these components. It is possible to generate a geometry and grid with only one of these components. The examples of Fig. 1 illustrate a small variety of inlet geometries that can be created with this geometry model.

Each component consists of one or more of *entities*. Each entity has its unique mathematical description and can be mathematically described as a line, conic, non-uniform rational B-spline (NURBS), or cubic spline. Each entity has a set of control points. The control points are defined with respect to the Cartesian (x,y) frame for two-dimensional shapes and the cylindrical (x,r) frame for axisymmetric shapes. A line entity is described by two control points which are the endpoints of the line. Various options exist for determining the two endpoints that include specifying the length or angle of the line. A conic entity is a section of a circle or ellipse. A NURBS curve entity is defined by its control points. The NURBS curve does not interpolate through the control points, but rather the control points control the local curvature of the NURBS curve. The slopes of the ends of the NURBS curve are tangent to the respective lines connecting the two control points at the respective ends of the NURBS curve. A cubic-spline curve entity is defined by a set of control points through which the curve interpolates. Each entity has a parameter associated with the distance along the entity. The parameter is directed from the first control point to the last control point along the entity. At any point along the entity a tangent vector is defined as being in the direction of the parameter. The normal to the entity is the cross product of the unit vector normal to the plane to the tangent vector. The entities of a component are not required to be contiguously connected.

Each entity can also be divided into *segments*. The points at which an entity is broken into segments are called *breakpoints*. The locations of the breakpoints are defined by the normalized distance along the entity. Dividing the entity into segments makes it easier to construct the flow domain and divide the flow domain into zones for grid generation. The segments also allow greater flexibility in the generation of the grid along the entities.

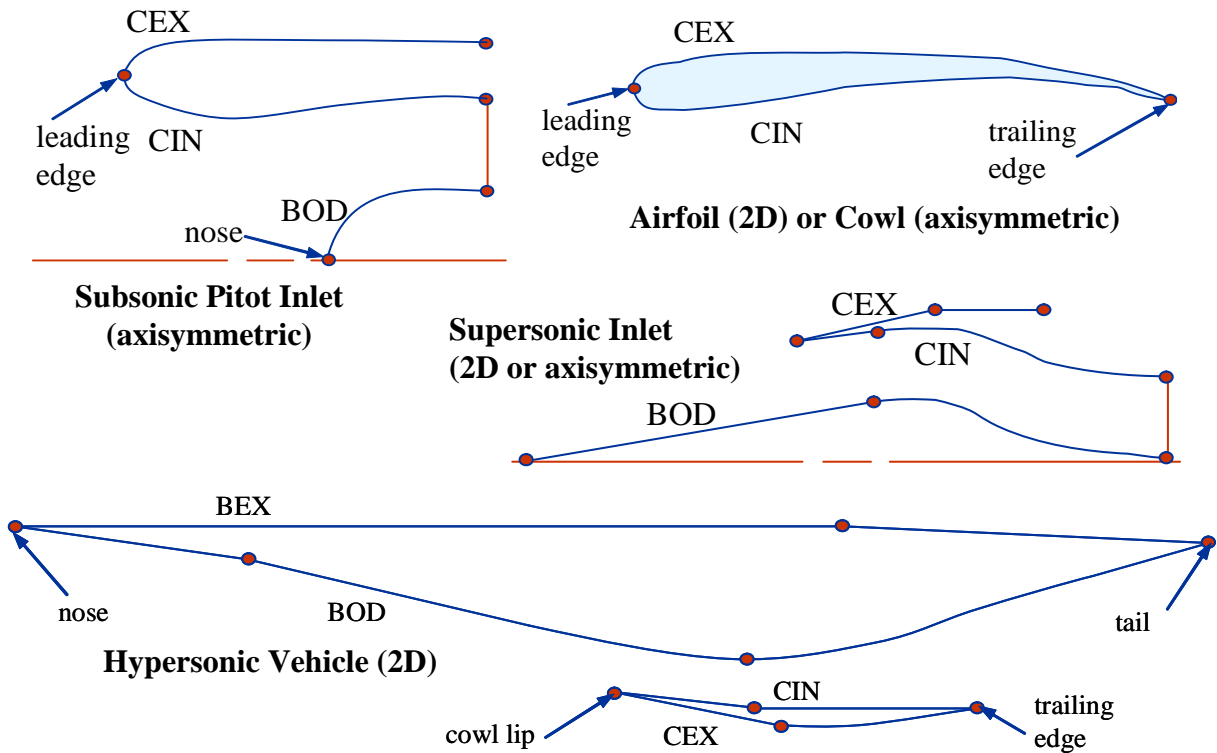


Figure 1. Examples of subsonic, supersonic, and hypersonic inlet geometries that can be modeled with PINDAP.

## **B. PINDAP Grid Zone Topology and Grid Generation**

The CFD analysis requires that a multi-zone, structured grid be generated about the inlet shape and within the flow domain. The flow domain is first divided into abutting zones. Each zone is a topological quadrilateral with four *edges*. The shape of each edge is defined by specifying the segments of the entities that comprise the edge. A structured grid zone has  $i$  and  $j$  coordinates associated with the edges. The direction of the grid coordinate must be specified with respect to the direction of the entity. Generating the grid for the zone involves generating the grid along each segment of each entity. The basic grid generation method for generating a grid on a segment is the hyperbolic tangent method in which one specifies the grid spacing at the ends of the entity segment and the number of grid points for the segment. PINDAP has the ability to determine the appropriate number of grid points along a segment based on the grid spacing and other grid quality parameters. A structured grid for a zone requires that opposite edges of the zone have the same number of grid points, and so, the procedure for specifying the number of grid points for each entity segment must provide this match. Once the grid points are distributed on the edges of the zone, the zonal grid is computed using a basic trans-finite interpolation method. PINDAP outputs the grid (run.X) in multi-zone, whole, unformatted Plot3d file format. The Plot3d grid file is converted to a common grid format (run.cgd) using the CFCNVT utility.

## **C. Boundary Conditions**

The common grid file (run.cgd) for input to the Wind-US code requires that boundary condition information be specified. The PINDAP code creates a file named `gman.com` that is used as the input for the GMAN utility. GMAN reads the common grid file (run.cgd) to get the grid coordinates and reads the `gman.com` file for information on how to set the boundary conditions.

## **D. Direct Specification of the Geometry and Grid and the Concept of the Design Module**

The basic method of input of information into PINDAP is the direct specification of the geometry and grid in the ASCII input data file. For each entity, all of the information is specified in the input data file, such as, the geometry coordinates, number of segments, segment end-point grid spacing, number of grid points of each segment, and more. The direct specification mode is flexible, but can be tedious to enter the complete information for every entity and segment for even simple shapes. Further, information is often duplicated within the input file. Thus PINDAP allows one to code a design module, which is a Fortran 90 subroutine, to allow the input of a minimum set of information to describe the inlet shape and generate the grid. As an example, a design module was written to generate a grid about a NACA 4-digit airfoil such that a user is only required to input the three digits describing the airfoil, distances defining the extent of the flow domain, and select grid spacings, such as those at the leading and trailing edges and the grid spacing at the airfoil surface. The design module then computes the remaining information to generate the geometry and generate the grid. The design module can encode assumptions about the geometry, topology, and grid so that a designer does not have to get bogged down with the details, and so, can focus on the design of the inlet. The design module can be designed to be applicable for a certain class of inlets, such as supersonic, axisymmetric, mixed-compression inlets.

## **E. Isolator and Nozzle Zone**

The CFD analysis of inlet flows requires a boundary condition to be specified at the engine-face outflow boundary. One successful approach has been to attach at the engine-face a grid zone which contains an isolator and nozzle. The isolator is a straight duct that gives the flow time to mix out prior to reaching the nozzle. The nozzle is constricted such that its throat is choked. The boundary conditions at the outflow of the nozzle are simple extrapolation boundary conditions. The back-pressure at the engine-face is set by the height of the throat. This approach has been found to be fairly non-reflective, which improves iterative convergence of the simulation.

## **F. Three-Dimensional Grids**

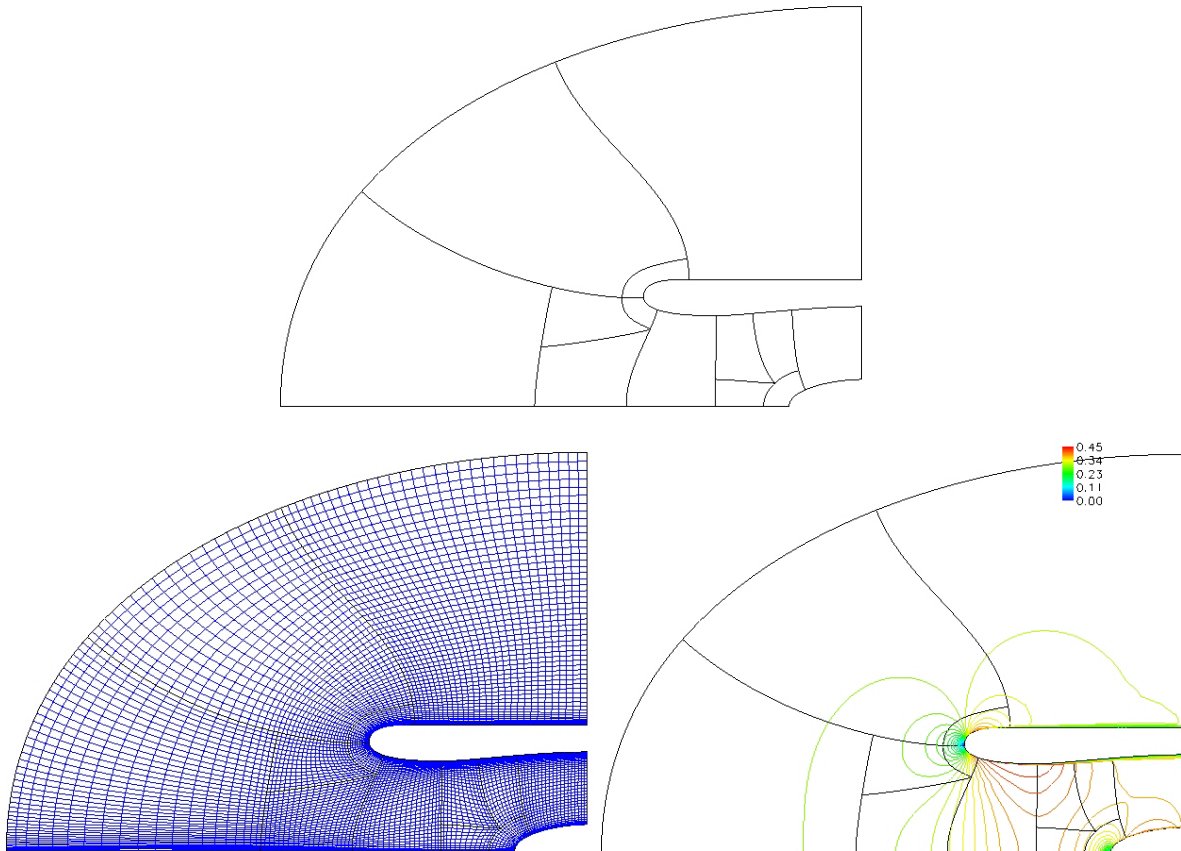
PINDAP has the capability of generating a three-dimensional grid obtained by extruding the planar grid in the  $z$ -coordinate or circumferential direction. The three-dimensional grid can then be used for simulations at non-zero angles-of-attack or angles-of-sideslip.

## IV. Examples

Several examples illustrate the use of PINDAP across the Mach number range.

### A. Subsonic Pitot Inlet

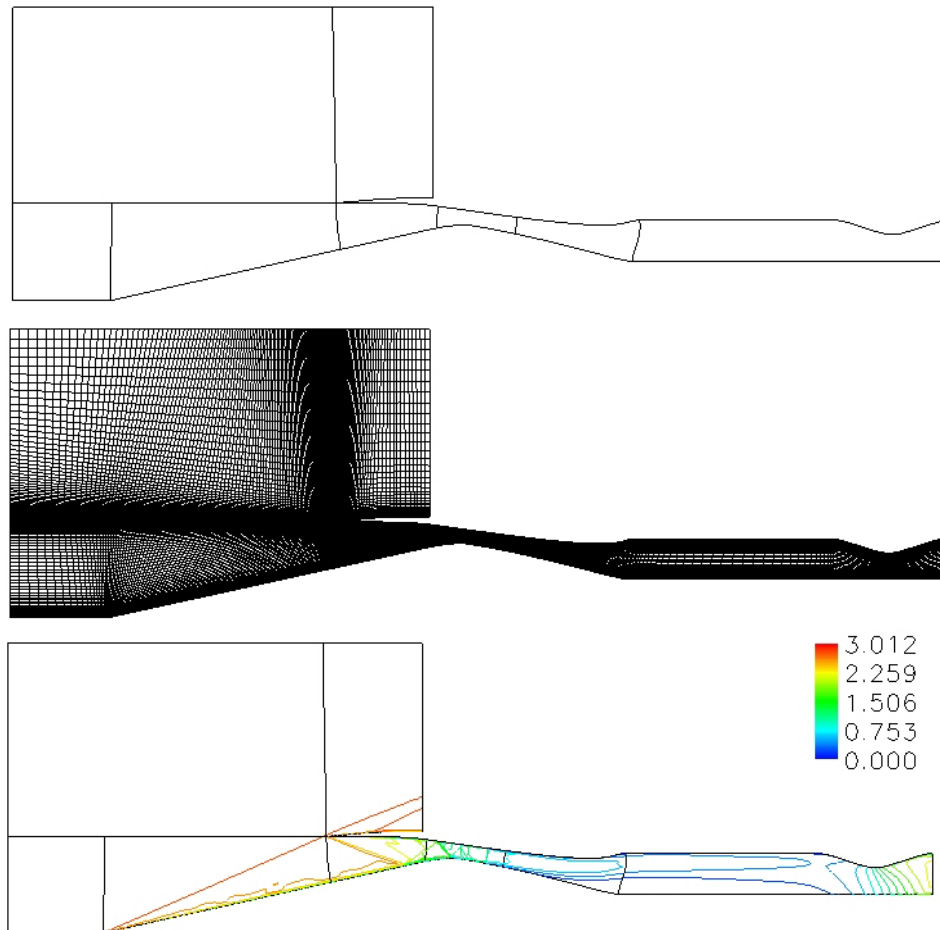
A common inlet for subsonic transport aircraft with high-bypass turbo-fan engines is a pod-mounted, pitot inlet. The cowl is typically axisymmetric. The engine has a spinner to provide for smooth flow about the hub of the fan. Figure 2 shows a generic planar geometry for a subsonic pitot inlet. The blunt-nosed spinner is the body component of the geometry model and contains a single entity of a NURBS curve. The cowl consists of a cowl interior component and a cowl exterior component. The cowl has a blunt cowl lip. The cowl interior consists of two entities of NURBS curves. The cowl exterior consists of two entities – the first a NURBS curve and the second a line. The flow domain boundaries consist of lines and NURBS curves to divide the flow domain into 13 zones. The use of NURBS curves for zonal boundaries allows the generation of orthogonal grids within the zones. Note that some of the zonal boundary curves connect at breakpoints in the body and cowl entities. The distribution of the grid is guided by the normal grid spacing at the surfaces to resolve boundary layers on the cowl and spinner and the desired axial and radial grid resolution in the core flow. PINDAP generates the grid and boundary condition file for the Wind-US simulations. Figure 2 shows the grid and Mach number contours for a representative simulation. Once the inputs are established, PINDAP can be used to generate new grids quickly if modifications are made to the cowl shape or if the grid properties are modified to provide increased resolution. PINDAP can also extrude the grid circumferentially over a 180-degree or 360-degree section for flow simulations at angle-of-attack and / or angle-of-sideslip.



**Figure 2. Planar geometry model, grid, and Mach number contours for the subsonic pitot inlet.**

## B. Supersonic, Mixed-Compression NASA-Ames Inlet

PINDAP was used to model the geometry and generate the grid for a CFD validation study involving a NASA-Ames, supersonic, mixed-compression inlet.<sup>3</sup> The inlet was axisymmetric with a centerbody and cowl. Figure 3 shows the planar geometry. The centerbody had a conical nose, which was modeled as a line entity. The rest of the centerbody was modeled with a cubic spline to interpolate the  $(x,r)$  geometry coordinates published in the test report. Similarly a cubic-spline entity was used to model the cowl interior to interpolate the published coordinates. The cowl lip was modeled as being sharp. The first entity of the cowl exterior was a line. The second entity was a NURBS curve to provide a smooth shape to the third entity, which was a constant-radius line. The cowl exterior geometry was not provided in the test report, and so, was approximated, which is acceptable since at supercritical, supersonic flow conditions, the cowl exterior should not affect the flow through the inlet. Figure 3 shows the grid and Mach contours for one of the simulations. A useful feature in PINDAP is the ability to translate or rotate geometric entities. For the NASA-Ames inlet, the cowl translated to alter the internal area distribution of the inlet for start-up and for off-design conditions. Thus, once the inlet is modeled within PINDAP, one can quickly regenerate the grid when the cowl needs to be translated for another configuration. This example also shows the use of the isolator and nozzle. The isolator allows the separated flow from the subsonic diffuser to settle before reaching the nozzle. The nozzle radius can be varied to vary the back-pressure of the inlet. For a mixed-compression inlet, the back-pressure adjusts the location of the terminal shock.



**Figure 3. Planar geometry model, grid, and Mach number contours for the NASA-Ames mixed-compression inlet.**

### C. Supersonic Business Jet Inlet

PINDAP was used for a preliminary study for the design and analysis of an inlet for a supersonic business jet.<sup>4</sup> The business jet was designed to carry six passengers with a range of 4000 nautical miles at a cruise speed of Mach 2.0 at an altitude of 50,000 feet. For this design study, it was assumed that the turbo-fan engine had a fan diameter of 60 inches and with a fan face Mach number of 0.6 at cruise. Further, it was assumed that the inlet used up to 3% bleed with 2% spillage. A two-dimensional, external-compression inlet was modeled. The geometry model consisted of a body, cowl interior, and cowl exterior components. Figure 4 shows the geometry model of the inlet.

The body consisted of three entities. The first entity was a line representing the nose of the compression ramp defined by the nose angle. The second entity was a compression ramp defined by five control points that were interpolated using a cubic spline. The method of characteristics was used to determine the placement of the control points such that the Mach lines of the compression surface intersected at the focal point. The third entity was a NURBS curve defining the body surface of the internal duct leading to the fan. At the junction of the compression ramp and the internal body surface a “break” existed that resulted in a discontinuity of the slope of the body surface. The NURBS curve was defined by eight control points. The first three were collinear and defined the angle and strength of the start of the curve. The last three control points were also collinear and defined the angle and strength of the end of the curve. Two control points on the interior of the curve allowed local shaping of the middle of the curve.

The cowl interior consisted of two entities. The first entity started at a sharp cowl lip and was a line entity whose angle was a design factor. The second entity of the cowl interior defined the rest of the interior cowl surface of the inlet. The second entity was a NURBS curve consisting of eight control points in the same fashion as the body curve. The cowl interior curve ended at the fan face.

The cowl exterior contained three entities. The first entity was a line that started at the cowl lip with an angle. The second entity was a NURBS curve with four control points that matched the angles of the first and third entities. The third entity was a straight line that was flat.

The flow domain boundaries consisted of an inflow boundary, farfield boundary, and outflow boundary for the exterior boundaries. The interior outflow boundary consisted of a straight line at the fan face connecting the body and cowl interior. The fan face was defined by the position of the axis of the fan at the fan face and the height of the fan. The fan face could also be at an angle. The interior boundaries extended downstream with the addition of an isolator and outflow nozzle.

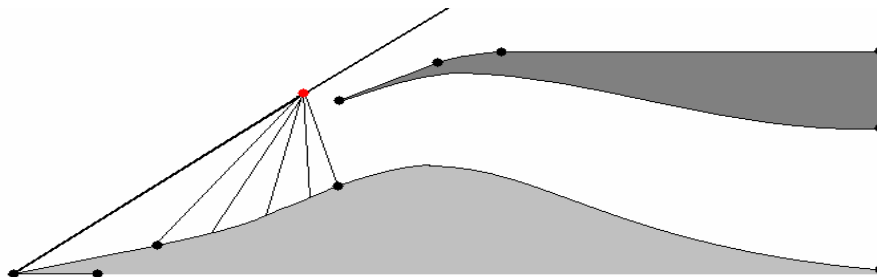


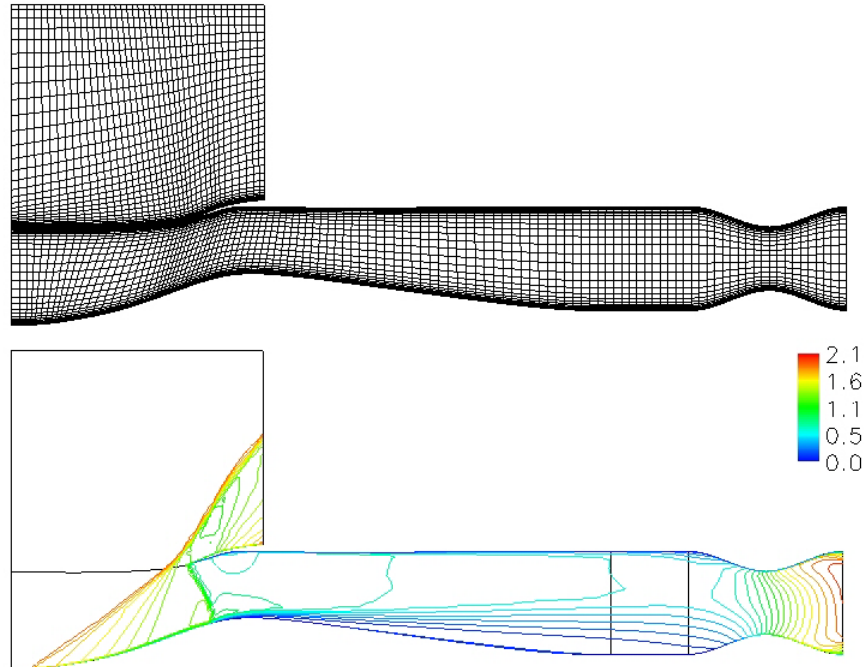
Figure 4. Geometry model for the 2D SBJ inlet.

The main design factors for the inlet design included the factors for the shaping of the NURBS curves for the body and cowl interior, placement of the cowl lip, slope of the interior of the cowl lip, placement of the fan relative to the cowl lip. A design module was created for PINDAP that allowed the input of these design factors along with other geometric factors that define the geometry and domain.

The design module also establishes the grid topology as a 2-zone grid. Zone 1 starts just ahead of the leading edge of the body nose and extends through the inlet to the fan face. Zone 2 encompasses the exterior of the inlet. The design module also reads in the grid quality information. Figure 5 shows an example of the grid for the inlet. The grid is clustered along the inlet surfaces. The grid also contains zones for an isolator and outflow nozzle attached to the fan face to apply outflow boundary conditions.

The sizing of the inlet provided the location of the focal point. The inlet was sized for cruise conditions. With the diameter of the fan and the average Mach number at the fan face provided, the corrected weight flow through the fan was computed. With an estimate of the total pressure loss and assuming constant total temperature through the inlet, the actual weight flow through the fan face at the design condition was computed. With an assumption of the allowable bleed and spillage, the design capture weight flow was computed. With a supersonic inlet, the captured

weight flow was the amount of flow through the area enclosed by the focal point. The cowl lip was then placed to capture the sum of the assumed fan-face and bleed flow. Actual conditions caused the amounts of weight flows to vary, but PINDAP allowed the cowl lip position and bleed region design factors to be varied to adjust these values to desired levels. Figure 5 shows the Mach number contours for one of the simulations.



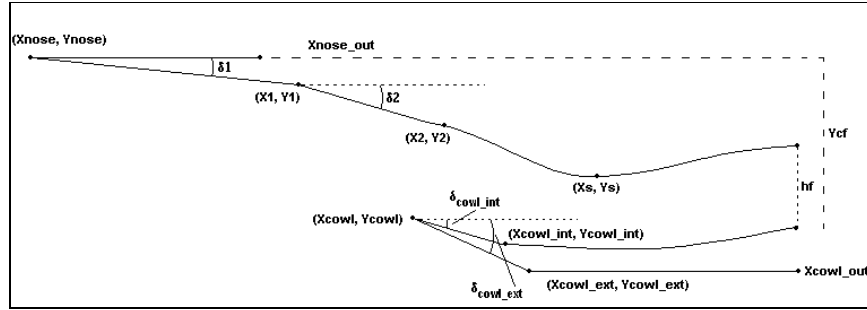
**Figure 5. Grid and Mach number contours for a 2D SBJ inlet.**

#### **D. Hypersonic Inlet**

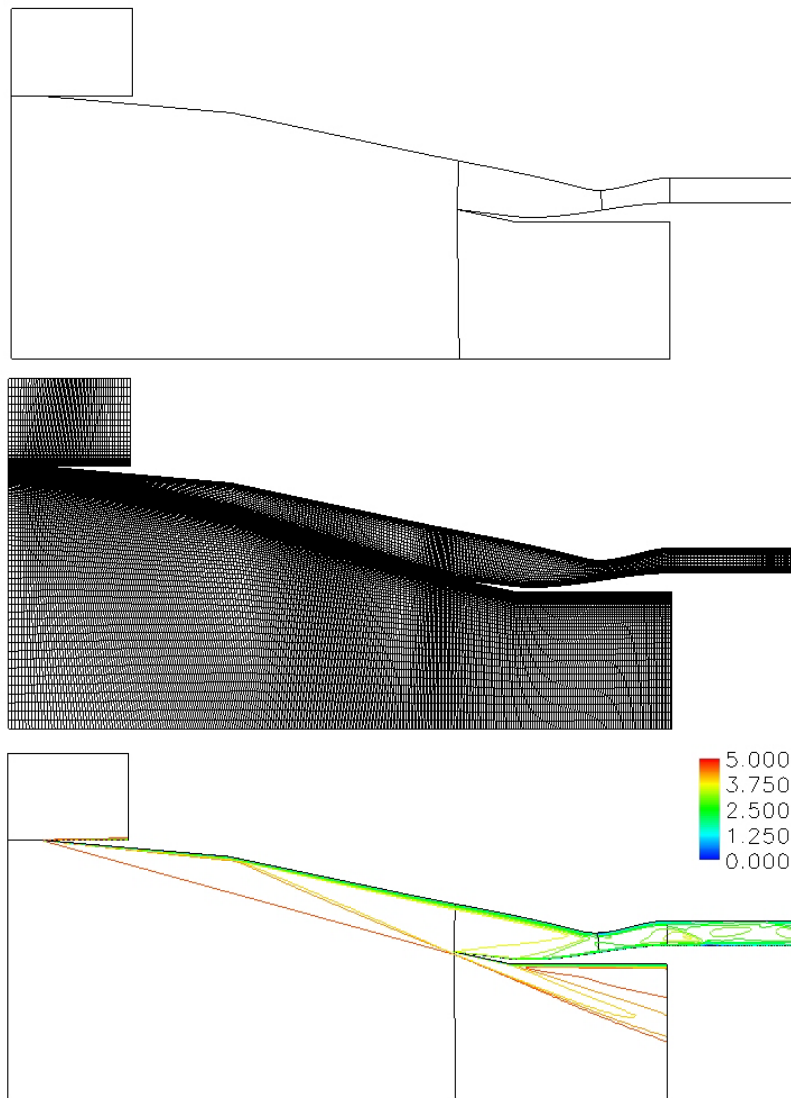
The PINDAP design process was exercised during the design of a generic, two-dimensional hypersonic inlet for Mach 5 flow. The design point was an altitude of 65000 feet and angle-of-attack of 0 degrees. Figure 6 shows the design factors for this study. The design requirements included a specified engine weight flow with an average Mach number into the isolator of approximately Mach 2.5. For this study, the geometry was parameterized with respect to a number of design factors. A Microsoft Excel spreadsheet was created to generate the geometry coordinates for the inlet based on the set of design factors and some calculations based on oblique shock theory and the engine weight flow requirement. Figure 6 shows a diagram of the design factors and the entities modeled for the body and cowl. The body component consisted of four entities. The first two were the ramps modeled as line entities. Within the diffuser, the body was modeled with NURBS curve entities. The position of the shoulder ( $x_s$ ,  $y_s$ ) was a variable design factor. A body exterior component was specified with a single line entity. The cowl lip was modeled as being a sharp point located at ( $x_{cowl}$ ,  $y_{cowl}$ ) whose position was determined by requirements that the oblique shocks from the ramps intersect near the cowl lip with a specified spillage and inlet capture flow. The cowl interior component consisted of two entities. The first was a line that established the interior angle of the cowl. This angle set up the oblique cowl shock. The remainder of the cowl interior was modeled with a NURBS curve entity. The engine-face height ( $h_e$ ) was mostly determined by the requirement of approximately Mach 2.5 at the entry to the isolator. The coordinate ( $x_{cf}$ ,  $y_{cf}$ ) set the position of the engine face. The cowl exterior was modeled with two line entities. The geometry coordinates produced from the spreadsheet for a specific design were then input into PINDAP to generate the grid. Figure 7 shows an example grid. CFD simulations were then performed. Figure 7 shows the sample Mach number contours. Based on preliminary CFD simulations, two of the design factors were selected to vary for the DoE study – the axial position of the shoulder ( $x_s$ ) and the vertical position of the bottom of the engine face ( $y_{cf}$ ). A statistical design was established to generate a quadratic response surface of the total pressure recovery with respect to the two design factors. This required a full-factorial design requiring 9 CFD simulations (2 factors with 3 levels each). Figure 8 shows a typical response surface generated during the process. The optimum set of design factors was obtained from examining the response surfaces during the design process.



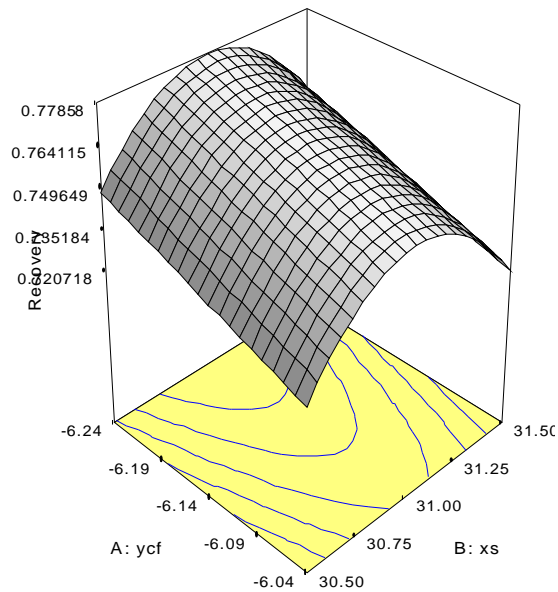
The PINDAP code was useful since each set of design factors requires a slightly different geometry and the grid generated for that geometry. With simple changes to the input file, the grids were quickly generated.



**Figure 6. Design factors and geometry entities for the hypersonic inlet.**



**Figure 7. Planar geometry model, grid, and Mach number contours for the hypersonic inlet.**



**Figure 8. Response surface for total pressure recovery for the hypersonic inlet design.**

## V. Summary and Conclusion

PINDAP provides a framework for efficient design-based CFD analysis of planar inlet systems. By automatically generating the geometry, grid topology, and grid, it removes some of the most tedious and demanding aspects of the use of CFD for inlet design. This will allow designers and engineers possessing only fundamental knowledge of CFD to use a CFD tool for their aerodynamic analysis of inlet systems.

## VI. Future Work

The development of the PINDAP code is ongoing. Further work is proceeding to improve the geometry modeling and grid generation capabilities and develop design modules for certain classes of inlets. PINDAP will be developed to generate geometry and grids for three-dimensional inlets with flow control devices such as vanes and jets. PINDAP will also be able to generate grids for a turbo-fan simulation module.

## Acknowledgments

The authors would like to thank the NASA-Glenn Lewis' Educational and Research Collaborative Internship Program (LERCIP) for support of the second author.

## References

- <sup>1</sup>NPARC Alliance, "The Wind-US Code", [www.grc.nasa.gov/www/Wind-US\\_docs](http://www.grc.nasa.gov/www/Wind-US_docs) [cited 3 November 2004].
- <sup>2</sup>State-Ease, Inc. [www.stat-ease.com](http://www.stat-ease.com). [cited 3 November 2004].
- <sup>3</sup>Sorensen, Norman E.; and Smeltzer, Donald B.: Investigation of a Large-Scale Mixed Compression Axisymmetric Inlet System Capable of High Performance at Mach Number 0.6 to 3.0. NASA TM X-1507, 1968.
- <sup>4</sup>Bruckner, R.J., "Conceptual Design of a Supersonic Business Jet Propulsion System," AIAA Paper 2002-3919, July 2002.