NASA Technical Memorandum 106693

Investigation of Methods for the Structural Weight Analysis of a Mach 2.4 Axisymmetric Inlet

Shari-Beth Nadell
Lewis Research Center
Cleveland, Ohio

August 1994

National Aeronautics and
Space Administration

SUMMARY

Structural design and analysis tools appropriate for estimating the structural weight of an axisymmetric inlet designed for Mach 2.4 cruise were evaluated. The weight estimates are used to aid in inlet concept evaluation in the preliminary design phase. Little information regarding the inlet mechanical design is available in this design phase, so it is necessary to first develop a reasonable structural design before estimating the inlet weight. The Internally Pressurized Structure Synthesis and Optimization (IPSSO) program was chosen for evaluation due to its combined design and analysis capabilities. In addition, the analytical approach employed by IPSSO provides a relatively quick and simple tool for inlet design and analysis. The inlet design produced by IPSSO was then analyzed using the NASTRAN finite element program. The finite element analysis was performed to help identify the limitations of the analytically based code as well as to evaluate NASTRAN for this application. The weight calculated using IPSSO was compared to the Mixed-Compression Translating Centerbody axisymmetric inlet developed by the Boeing Commercial Airplane Group. It was found that IPSSO predicted an inlet weight approximately 51 percent less than the weight predicted by Boeing analysis methods. This difference was due in part to geometric modeling limitations of the IPSSO code. Finite element analysis methods, such as NASTRAN, which provide greater flexibility in modeling an inlet geometry, require structural design details that are usually not available during this stage of the design. In addition, these tools do not generally incorporate the ability to easily optimize a design for minimum weight. The combined use of IPSSO to create an initial design and NASTRAN to perform a numerical analysis would provide the capability to evaluate a limited number of inlet designs. The development of a new tool for the minimum weight design and analysis of inlet structures, however, would be required for greater flexibility in evaluating inlet conceptual designs.

INTRODUCTION

One of the important parameters used to evaluate inlet concepts during the preliminary design phase is the structural weight. Before the structural weight of the inlet can be estimated, a reasonable structural design must be developed. Structural design and analysis tools appropriate for estimating the structural weight of an inlet were evaluated. The Internally Pressurized Structure Synthesis and Optimization (IPSSO) program, developed at the NASA Langley Research Center, was chosen for evaluation due to its combined design and analysis capabilities (ref. 1). The IPSSO analysis is based on beam theory and was developed specifically for the minimum weight design of pressurized ducts. The inlet design produced by IPSSO was then analyzed using the NASTRAN finite element analysis program coupled with the PATRAN pre- and post-processor (ref. 2). The NASTRAN analysis was performed to help identify the limitations of IPSSO. In addition, the applicability of finite element analysis to the structural design and weight analysis of the inlet was evaluated.

The inlet model chosen for the evaluation was an axisymmetric inlet designed for Mach 2.4 cruise. The baseline chosen for the overall design of the inlet was the Mixed-Compression Translating Centerbody (MCTCB) axisymmetric inlet developed by the Boeing Commercial Airplane Group (ref. 3). Normal operating pressure and temperature distributions for the axisymmetric inlet were calculated at the NASA Lewis Research Center. Geometry, loading, and material properties were as similar between the IPSSO and NASTRAN analyses as possible, given the different input requirements of each method. The weight calculated using IPSSO was compared with the weight calculated by Boeing for the MCTCB inlet and is presented below. The stress and deflection results are also presented. Modeling issues encountered with IPSSO and NASTRAN are discussed along with the assumptions, benefits, and deficiencies of each analysis tool.

BASELINE INLET MODEL

The baseline inlet chosen for this study was the Mixed Compression Translating Centerbody (MCTCB) inlet developed by the Boeing Commercial Airplane Group under the Propulsion Airframe Integration Technology (PAIT) contract (ref. 3). The MCTCB inlet is shown in Figure 1. The inlet is axisymmetric and consists of a cowl and a centerbody; the centerbody is translated on a support tube, which in turn is supported by six struts at the back of the inlet. Overall dimensions of the MCTCB inlet are indicated in Figure 1a. Inlet length from centerbody tip to fan face is 185.5 in. The capture diameter is 58.4 in. The inlet was mounted to the wing box via a forward mount and an aft mount. The forward mount provides support for vertical and side loads, as well as thermal expansion. The aft mount provides support for vertical, side, and thrust loads. Provision for bolting the inlet to the engine forward flange is provided by the aft cowl bulkhead.

The structural design of the MCTCB inlet is described in detail in reference 3 and summarized below. The main structural concept used to design the MCTCB inlet was honeycomb panels supported by rings and longerons (Fig. 4). Skin/stringer type construction was utilized where honeycomb construction was not applicable (support struts, throat plenums, nose cone, leading edge, etc.). Figure 1a shows the various structural components used in the MCTCB inlet design. The translating centerbody consists of the translating tube, the exterior shell, and bulkheads, all of honeycomb construction. The cowl is divided into a forward cowl and an aft cowl. The forward cowl consists of honeycomb panels supported by six longerons, bulkheads, a spacer cowl, and the leading edge lip. The aft cowl consists of honeycomb panels, seven longerons, an acoustic panel, and bulkheads. The six support struts are swept airfoils that perform the dual role of providing structural support and ducts for electrical and hydraulic lines and throat bleed air. A cross-section view of the inlet near the engine face (Figure 1b) shows the asymmetric placement of the support struts with respect to the horizontal axis of the inlet. Each of the bottom five struts attach to a longeron; the top strut attaches to a bridge that spans the remaining two longerons.

The major structural materials used in the Boeing study were Ti-3Al-2.5V for the honeycomb core and Ti-6Al-4V for all other structure. The material properties for these Titanium alloys, taken from MIL-HDBK-5E, are given in Appendix 1. The cruise operating temperatures of the inlet were determined by Boeing to be 330 degrees Fahrenheit in the inlet and 380 degrees Fahrenheit at the leading edge. Material properties at 350 degrees Fahrenheit were used in the MCTCB analysis. Honeycomb core depth was limited to 3/16 in. The maximum core depth was 2.0 in. The minimum gage value for the panel skin was 0.012 in.

The critical loading condition that sized the structure of the MCTCB inlet was pressure loading due to the hammershock condition. Hammershock loads result from the back pressure produced by a compressor stall. The loads produced in this situation are generally higher than those produced under normal inlet operation. Various other loading conditions were considered as well, including landing loads, maneuver loads, gust loads, and normal operating loads. Acoustic limits and foreign object damage were also considered in the design of the MCTCB inlet. A weight factor of approximately 20% was applied to the structural weight results (1.20 multiplied by the structural weight) to account for non-optimum items such as fasteners, brackets, seals, bleed assemblies, manufacturing joints, etc.

(ref. 3). In addition, a stress concentration factor of 3.0 was applied to the longerons to account for fasteners (ref. 3).

ANALYSIS MODELS

Two analysis models were developed for this study, one for use in the IPSSO program and one for the finite element analysis performed with NASTRAN. The IPSSO model was derived from the MCTCB axisymmetric inlet described above. The design resulting from the IPSSO analysis was then used to develop the NASTRAN inlet model. Due to the particular modeling and analysis capabilities of each tool, some variations between the models existed. These variations and the particular assumptions made for each analysis are described below.

IPSSO Inlet Model

The IPSSO program, described in reference 1, was developed at the NASA Langley Research Center to size an internally pressurized shell for minimum weight. In order to use IPSSO to analyze an axisymmetric inlet based on the MCTCB inlet model, certain assumptions had to be made. One assumption was that the inlet was constructed from honeycomb panels stiffened with rings and longerons of I-beam cross-section. While similar to the MCTCB design, the limitations of the IPSSO code prevented specifying the location of these members to accurately reflect the MCTCB design. Thus, the IPSSO inlet design may be considered to be more "generic" compared with the more specific "point design" of the MCTCB inlet. In addition, due to the limitations of the IPSSO code, the inlet cowl and centerbody were modeled and analyzed as separate components. No interactions between the inlet and centerbody with respect to transferring loads or affecting airflow were taken into account. The support tube and struts were not modeled, again due to the geometric modeling limitations of the IPSSO code. Weight for the external cowl skin, which is subjected to the external aerodynamic load on the inlet, was estimated based on the average internal skin unit weight calculated by IPSSO. At the Mach 2.4 design condition assumed for this inlet, the external aerodynamic load is small compared with the inlet internal pressures; therefore the unit weight for the inlet internal skin should be sufficient to carry the external pressure load.

One important aspect of the IPSSO design was that both major structural components (the cowl and centerbody) of the inlet were modeled as if containing an internal pressure. This assumption was accurate for the cowl, but resulted in a reverse loading condition for the centerbody. It was assumed that the resulting external ring structure would be adequate to support the centerbody panels internally when subjected to an external load. This assumption was discussed with and approved by the developers of the IPSSO code (ref. 4). During the analysis, however, it was realized that this was not a valid assumption. The centerbody design resulting from this analysis is most likely optimistic. This is a result from the different primary stresses that result from each type of loading. The primary stresses in an internally pressurized shell will be tensile; failure of the shell will result when the hoop stress or longitudinal stress exceeds the yield stress of the material. The primary stresses in same shell subjected to external pressure will be compressive. Depending on the shell geometry, these stresses could cause the shell to fail by buckling. If this type of failure occurs, the failure stress may be lower than the yield stress of the material (ref. 5). Therefore, an externally pressurized shell would require more material to withstand the same pressure load applied internally to a shell of the same overall dimensions.

The axisymmetric cowl and centerbody as modeled in IPSSO are shown in Figures 2a and 2b, respectively. The cowl was modeled as a cylinder made up of honeycomb panels stiffened with rings and longerons. The radius of the cowl cylinder was varied along the length of the inlet to model the MCTCB geometry. The centerbody was modeled as a cone and cylinder, also with varying radius as appropriate. Six longerons were placed symmetrically around the perimeter of the cowl along its entire length. Ring spacing for both the cowl and centerbody was determined from a parametric analysis performed with IPSSO on axisymmetric inlets of varying capture areas. From the results, shown in Figure 5, it can be seen that, for a 30 in panel width (dictated by the MCTCB design and inlet circumference) the optimum ring spacing is approximately 8 in (heavy dashed curve). A 12 in spacing (i.e., 12 in panel length) does not lead to a significant (approximately 10 percent) increase in the assembly weight and may be more realistic given the overall size of the inlet. Therefore, rings were placed at 12 inch intervals along the lengths of both the cowl and centerbody where possible. Variations in the ring spacing resulted from separate part definitions due to varying pressures and temperatures (for a more detailed description of how component geometry is defined in IPSSO, see ref. 1).

Temperatures representative of the maximum total values that would be encountered throughout the flight path were determined at the NASA Lewis Research Center and input to the IPSSO analysis. These temperatures are shown in Figure 2. Material properties, however, were entered for only one temperature condition (500 degrees Fahrenheit), so temperature effects are not accounted for in the IPSSO analysis. The temperature chosen was approximately equal to the maximum total temperature (497 F) predicted for the inlet. While it is acknowledged that the thermal stresses caused by the temperature distribution in the inlet could impact the inlet weight, an investigation of this impact was beyond the scope of this study. The material properties used in the IPSSO analysis for Ti-6Al-4V appear in Appendix 1. This material was used to model both the panel skins and the honeycomb core material.

Internal pressures were chosen to represent the maximum static pressure each component would be exposed to under normal operating conditions throughout the flight path (see Figure 2). Different flight conditions, therefore, are represented in one load case. Although this does not represent an actual loading condition, it is one method of designing a structure; each part of the structure is thus designed to withstand the maximum load it will be subjected to throughout the entire flight path. Note that the internal pressure applied to the centerbody was equal to the actual external maximum pressure.

The IPSSO assembly weight prediction includes a close-out/joint factor of 1.3 for honeycomb and single sheet panels, which accounts for fasteners, panel close-outs, overlaps, doublers, etc., in the finished panel design. This factor is multiplied by the unit weight of the bare panel skins. A non-optimum factor of 1.4, which accounts for minimum gauge thicknesses, growth margins, hatches, reinforcements, etc., is multiplied by the final ring, panel, and longeron weights. These values were chosen based on similar studies performed at NASA Langley Research Center with the IPSSO code. A factor of safety of 1.5 for the cowl and 2.0 for the centerbody multiplies each load within IPSSO. No stress concentration factors were applied.

NASTRAN Inlet Model

The design results from the IPSSO analysis were used as input to the NASTRAN inlet model. One significant difference between the IPSSO and NASTRAN analyses was the ability to model the entire inlet structure as one component in NASTRAN. This included not only the cowl and centerbody, but the support tube and the support struts as well. The support tube was modeled only from the centerbody connection to the aft end of the inlet. The support struts were modeled as flat plates rather than airfoils. This allowed the inclusion of centerbody/support/cowl interactions without complicating the model unnecessarily. The thickness of the strut plates was chosen to maintain an estimated stiffness of the actual airfoil struts. Details of this calculation appear in Appendix 2. No cowl external skin was modeled in the NASTRAN analysis. The weight for the external skin was added to the inlet structural weight after the finite element analysis, as was done during the IPSSO analysis.

The NASTRAN inlet geometry is shown in Figure 3a (note that the rings and longerons do not show up in the shaded image due to their small width). The honeycomb panels in the cowl and support tube and the single sheet panels in the centerbody and support struts were modeled using plate (QUAD) elements. The rings and longerons were modeled using BEAM elements. The total number of elements used to model the inlet was 2308 (QUAD and BEAM elements). Figure 3b shows the inlet model elements (the cowl is shown separately for visual clarity only). Honeycomb and I-beam properties were modeled on the NASTRAN property cards by calculating the appropriate moments of inertia and reference points (see Appendix 3 for more details). All face sheet thicknesses, cap thicknesses, core depths, etc., for the honeycomb and I-beam components were average values taken directly from the IPSSO analysis results.

The materials modeled were similar to those used in the MCTCB model: Ti-6Al-4V for panel skins and beams and Ti-3Al-2.5V (approximate data based on reference 5 assuming a core foil thickness of .0015 in, 3/16 in cell wall, and a 5 lb/ft^3 core density) for honeycomb core. Material properties for the Ti-6Al-4V were chosen at an operating temperature of 500 degrees Fahrenheit, as in IPSSO. The material density was input as zero for the support tube and support struts to obtain a weight calculation that included only the material in the cowl and centerbody. The material properties modeled in NASTRAN appear in Appendix 1.

The pressure distributions applied to the cowl and centerbody were the same as those applied to the IPSSO model. (Pressures were applied to the external surface of the centerbody, as appropriate.) Boundary conditions for the NASTRAN inlet modeled the symmetry of the structure about the vertical axis along the length of the inlet (see Fig. 1b). In addition, displacement constraints were placed on the cowl at approximate positions of the forward and aft mounts to the wing box. The aft cross section of the inlet was further constrained in the longitudinal direction to model a bolted connection to the engine.

The structural weights calculated within NASTRAN were multiplied by non-optimum factors identical to those applied in IPSSO. No consideration for joint-closeout factors was made. This was due to the fact that the inlet weight produced by NASTRAN was not divided into structural components (i.e., rings, panels, etc.), but was given as one total weight result. The application of a joint close-out factor to only panel weights was, therefore, not possible. In addition, the loads applied to the NASTRAN model were not increased by a factor of safety as in the IPSSO analysis. Since the NASTRAN geometry and dimensions were input directly from IPSSO, it was not expected that the exclusion of factors of safety would affect the weight of the structure. However, the lower pressures would be expected to produce lower stresses and, therefore, smaller deflections than those predicted by IPSSO.

ANALYSIS RESULTS

A comparison of the weight results of the IPSSO inlet analysis with the Boeing analysis results for the MCTCB axisymmetric inlet is given below. A comparison of the stresses calculated in IPSSO with those calculated in NASTRAN is also discussed. Finally, a discussion of the deflections calculated in all three analyses is given.

Weight Comparison

A comparison of the structural weights calculated from the IPSSO inlet model and the MCTCB inlet is shown in Table 1. The weight calculated by NASTRAN is also given. An external cowl skin weight of 198.64 lb, found by multiplying the average unit skin weight for the internal cowl skin (as designed in IPSSO) by the estimated cowl surface area, was added to both the IPSSO cowl weight and the NASTRAN structural weight, as described above. The weight of the support tube and support struts was estimated from the MCTCB weight breakdown as 600 lb and added to the IPSSO structural weight.

The structural weight of the MCTCB inlet (cowl, centerbody, and support structure) was calculated by Boeing to be 1620 lb. Comparing this weight to the structural weight of the IPSSO inlet model, 1103.30 lb, shows a decrease by 32 percent . Since the weight of the support structure for the IPSSO model is equal to that of the MCTCB, the weight difference between the inlets appears in the cowl and centerbody structures. As can be seen in Table 1, the IPSSO cowl and centerbody weight is approximately 51 percent less than that for the MCTCB. One reason for these large differences is that the loading condition applied to the IPSSO model was not as severe as the hammershock loading used to design the MCTCB inlet. In addition, the differences in the structural designs of the MCTCB inlet and the IPSSO model with respect to ring location, cross-section of longerons, etc., would contribute to a different structural weight estimate.

The inlet weight calculated by NASTRAN, 1280.41 lb, is approximately 16 percent heavier than the IPSSO prediction. A comparison of the cowl and centerbody weights only (excluding the cowl external skin) for the IPSSO and NASTRAN analyses, 304.66 lb versus 202.96 lb respectively, shows a 33 percent lighter weight calculated in NASTRAN for these structural components. NASTRAN calculates the structural weight from the material properties and the structural dimensions, both of which are input to the analysis and, for this study, both of which were input directly from the design results of the IPSSO analysis. It was expected, therefore, that the weights of the inlet centerbody and cowl would be similar between the two analysis models. The discrepancy in the calculated weights are in part due to the use of averaged dimension values from the IPSSO inlet design in the NASTRAN model. A second cause for the difference may have been the addition of Ti-3Al-2.5V properties for the honeycomb core material in the NASTRAN analysis, whereas the IPSSO analysis contained only Ti-6Al-4V properties. Slight modeling differences may have also contributed to the different weight estimations. For example, a portion of the support tube (Part 11 in Figure 2b) was included in the centerbody model analyzed in IPSSO. Slight geometry differences in the NASTRAN model eliminated this part of the support tube. In addition, the centerbody ring located at the position where the centerbody attaches to the support tube (Ring 15 in Figure 2b) was omitted in the NASTRAN analysis due to its interference with the support tube structure.

Stress Comparison

The maximum tensile stresses calculated in the IPSSO and NASTRAN analyses were compared. For both methods, the maximum tensile stress values were well below the yield stress of the materials used. Only ring stresses were compared; the determination of the stress values of the panels in the NASTRAN model for comparison would have been complex given the number of elements used to model the panels.

Maximum stresses on the cowl rings as calculated in IPSSO and extreme (minimum and maximum) stresses calculated in NASTRAN are plotted in Figure 6a. The first significant feature is that, while the NASTRAN results indicate both compressive and tensile stresses, the IPSSO results indicate only tensile stresses. Therefore, only a comparison of the tensile stresses can be made. The second thing to notice is the different trends indicated in the results. The maximum tensile stresses calculated in NASTRAN decrease fairly steadily along the length of the cowl, except for the initial ring located on the cowl lip. The maximum stresses calculated in IPSSO, however, are fairly constant in the forward cowl and increase erratically in the aft portion of the cowl. The percentage by which the NASTRAN results differ from the IPSSO results is indicated in Figure 6b. (A negative percent indicates a NASTRAN stress prediction lower than the corresponding IPSSO stress.) The stresses in the forward cowl, excluding the first two rings, differ by only -2 to 12 percent. The largest difference occurs in the final ring, where the NASTRAN stress calculation is 60 percent lower than that of the IPSSO analysis. The different boundary conditions modeled in each analysis contributed to this large difference.

The maximum stresses on the centerbody rings as calculated in IPSSO and extreme stresses calculated in NASTRAN are plotted in Figures 7a and 7b, respectively. The centerbody as modeled in IPSSO is shown to be subjected to only tensile stresses; when modeled in an actual loading condition (external load versus internal load) in NASTRAN, the centerbody is shown to be subjected mainly to high compressive stresses. No meaningful comparison of these results can be made. This emphasizes the limitations of the IPSSO program in the design of an externally pressurized structure such as an axisymmetric inlet centerbody.

The stresses on the MCTCB inlet were not available for comparison with the IPSSO and NASTRAN results.

Displacement Results

For an axisymmetric inlet with a centerbody, the deflection of the structure can have a direct impact on the inlet performance. In some cases, the deflection requirements placed on the inlet can therefore impact its structural design and weight. Both the Boeing analysis and the IPSSO program place deflection limits on the structure during design. NASTRAN, however, calculates the deflections resulting from the specified loading and boundary conditions on the structure.

The deflection limits placed on the MCTCB inlet in the Boeing analysis are summarized as follows: the internal duct was designed not to exceed localized deflections measuring a depth of .5 percent of the cowl lip (approximately .15 in) in the supersonic region and 1.0 percent of the cowl lip (approximately .3 in) in the subsonic region (ref. 3). The deflection of the centerbody centerline relative to the cowl centerline was limited to plus or minus 1 degree at the throat (ref. 3).

The IPSSO program automatically sizes the panels and rings to a deflection limit of .5 percent of panel width and the longerons to a deflection limit of .1 percent of span unless a deflection limit is specified (ref. 1). For the cowl, this translated to a maximum panel deflection of .1 in. (The deflections of the rings are not output when no limit is specified by the user.) The maximum deflection of the cowl longerons was .0132 in. Since the centerbody was not designed with longerons, no deflection data is output for either the panels or longerons.

The deformed NASTRAN inlet model is shown in Figure 8 (note that the deflections are grossly exaggerated in the figure). The maximum calculated displacement for the NASTRAN inlet was .844 inches in the axis of symmetry. This value exceeds the deflection limit the inlet was designed for in IPSSO. This difference is a result of different boundary modeling, constraints, loading conditions, and the ability to model the inlet as one component, instead of two separate structures (cowl and centerbody) within NASTRAN. It should be noted that these results do not reflect the previous assumption based on the absence of factors of safety in the NASTRAN model. This may also be a result of the different modeling constraints found in each analysis tool.

DISCUSSION

The IPSSO program as a structural design and weight analysis tool is evaluated below. A comparison of the IPSSO program with NASTRAN is also made with respect to flexibility, modeling issues, and the time required to perform each analysis. Although NASTRAN is not a design tool, the comparison allows further exploration of the structural design and weight analysis issues for an inlet. Finally, recommendations for the use of the analysis methods explored in this study are discussed.

The IPSSO program was chosen for evaluation in part due to its design and weight analysis capabilities. As discussed above, IPSSO was developed to solve one particular problem, that being the minimum weight design of an internally pressurized duct. Any deviation from this problem, for example, modeling a structure that carries an external load as opposed to one with an internal load, will produce results that are suspect. In addition, while IPSSO can model shell-type structures with complex cross-sections, it cannot model structures that are a concentric combination of shell-type structures, such as a cowl and centerbody. In the case of the axisymmetric inlet, the inability of IPSSO to model the cowl and centerbody together introduced error into the results by not allowing the interactions of these two major structures to be included. These limitations restrict the usefulness of the IPSSO program for inlet concept evaluations that include axisymmetric designs.

IPSSO is also more constrained than a finite element analysis with respect to the type of structural components it can model. Built into the IPSSO code is the assumption that the structure is constructed from honeycomb panels stiffened with rings and longerons of I-beam cross-section. The only variation that can be made in this design is the deletion of the frame, leaving a structure of single sheet panels. The generality of the finite element model, however, enables the modeling of any structural component that can be described using the available elements. The drawback of this flexibility is that it requires the engineer to have more precise information regarding the structural design then is generally available at the preliminary design stage. Assumptions regarding the design, therefore, are made. Each assumption, however, decreases the accuracy of the analysis results. An example of this was the assumption that the support struts, originally designed as airfoils, could be modeled as flat plates with equivalent stiffness. The calculation to determine the thickness of the flat plates was approximate, and yielded plates with a very high stiffness. This can be seen in Figure 8, where the cowl and centerbody deformations are very small near the location of the support struts. A slight "downward" shift of the inlet in the direction where the struts are in a greater concentration (similar to a cantilevered beam) may also be a result of the high strut stiffness. The high stiffness of this region prevented deformation; more deformation occurred in the region of lower stiffness, resulting in the downward shift of the inlet. Different deformation results may have been obtained had the actual airfoil shape been modeled for the support struts.

The benefit of IPSSO over a program such as NASTRAN, however, is that it designs and optimizes the structure for minimum weight. Any weight optimization and redesign with the finite element model would need to be performed manually. One further advantage of IPSSO over a finite element analysis is the amount of time required to create a model and obtain results. The model definition in IPSSO is much simpler than that required for a finite element analysis. An IPSSO model requires on the order of days to weeks to create, as opposed to a finite element model, which can require on the order of months for completion. Run-times for the analyses vary according to the computing power available. For this study, actual turnaround time for the IPSSO program on a Silicon Graphics Indigo R4000 workstation was less than one minute; turnaround time for the NASTRAN analysis was anywhere from 5 to 20 minutes on a Cray XMP, depending on the wait in the queue . A more complex finite element model could require one or more hours to run. In general, the finite element analysis also requires more than one iteration to generate a working model and produce reasonable results.

Although IPSSO would not be a useful tool for designing and analyzing many inlet geometries due to its limited modeling capabilities and lack of buckling analysis, it could be used for the design and analysis of straight, internally pressurized ducts found in some inlet designs. The initial structural design for this type of structure could be developed using the IPSSO program; the IPSSO design could then be used as a basis for developing a finite element model for NASTRAN analysis.

CONCLUSION

The Internally Pressurized Structure Synthesis and Optimization program and NASTRAN finite element analysis were evaluated for use in inlet concept evaluation. IPSSO, developed to size an internally pressurized duct for minimum weight, includes both design and analysis capabilities. Discussion has shown that the IPSSO program is useful only for the specific type of structural design problem for which it was developed. Use of the code for any problem other than that for which it was developed, for example, the design of an externally pressurized structure, produces results that may contain some erroneous data. This lack of flexibility precludes the benefits of IPSSO, which are its ease of use, quickness, and optimization capabilities, for many of the high speed inlet concepts. NASTRAN analysis, while providing the flexibility needed to analyze various inlet designs, requires more design detail than is generally available for conceptual studies. The assumptions that must be made to make up for the lack of information reduce the accuracy of the analysis. In addition, the time required to set up a good finite element model may be prohibitive when performing parametric trade studies.

A method is needed that contains enough design and analysis flexibility that any inlet model can be accurately analyzed in the preliminary design phase. Such a method would be analytically based, like the IPSSO program, and flexible, like the finite element method. Development of a method is proceeding at the NASA Lewis Research Center. Until this method is available, however, both of the evaluated methods could be used where appropriate. Specifically, the IPSSO program could be used to perform parametric trade studies on a simple two-dimensional or other symmetrical cross-section shell-type inlet. Finite element analysis could be used for more complex inlet designs where time permits and some amount of design data is available.

REFERENCES

1. Moses, P.L.; Jones, S.C.: IPSSO Version 1.1. Internally Pressurized Struture Synthesis and Optimization Computer Program Documentation. P.R.C. Kentron, Inc. Prepared for NASA Langley Research Center. April, 1987.

2. MSC/NASTRAN Version 6.7 User's Manual. The MacNeil-Schwendler Corporation, Los Angeles, CA.

3. Koncsek, J.L.; Gerstle, J.H.; MacKinnon, M.I.K.: Inlet Type Screening for High Speed Civil Transport with Turbine Bypass Engine. The Boeing Commercial Airplane Company. NAS3-25963. CR191006. March, 1993.

4. Jones, Stuart., NASA Langley Research Center, personal communication with Christopher Kelble, OAI Summer Intern, NASA Lewis Research Center, Summer, 1991.

5. Baker, E.H.; Kovalevsky, L.; Rish, F.L.: *Structural Analysis of Shells*. McGraw-Hill Book Company. 1972.

6. Design Data for Brazed Honeycomb Strutures. Aeronca, Inc. Middletown, Ohio.

7. Shanley, F.R.: Weight-Strength Analysis of Aircraft Strutures. Dover Publications, Inc. New York. 1960.

APPENDIX 1. Material Properties

Ti-6Al-4V:

F_{tu} (350 F)	107.2 Ksi
F_{ty} (350 F)	94.5 Ksi
F_{ty} (500 F)	84.5 Ksi
E (350 F)	14,720 Ksi
E (500 F)	13,600 Ksi
E_c (350 F)	15,088 Ksi
μ	.31
ρ	.160 lb/in^3

Ti-3Al-2.5V:

G (honeycomb core)	26.0 Ksi
μ	.31
ρ	.162 lb/in^3

APPENDIX 2. Support Strut Plate Calculations

The actual support strut geometry appears in Figure A2a. Each strut is a swept airfoil with the middle a constant thickness. The struts are hollow with a panel gage thickness of .25 in (ref. 3). Strut parameters are as follows:

L_1	strut chord at the center tube
L_2	strut chord at the cowl
h	strut height
t	strut skin thickness
d_1	maximum external airfoil depth
d_2	minimum internal airfoil depth

From Shanley (ref. 7), the moment of inertia of the cross-section of a solid airfoil section is given by:

$$I = \frac{4}{105} H^3 C$$

where H is the maximum wing depth and C is the wing chord. Applying superposition, for a hollow airfoil:

$$I = \frac{4}{105}\left[d_1^3 L - d_2^3 (L - 2t)\right]$$

where L is the minimum strut chord for the minimum moment of inertia. (This yields a reasonable estimate for the flat plate thickness of equivalent stiffness.) For the airfoil depicted in Figure A2a,

$$d_2 = d_1 - 2t$$

Therefore,

$$I = \frac{4}{105}\left[d_1^3 L_1 - (d_1 - 2t)^3 (L_1 - 2t)\right]$$

For actual strut parameter values (estimated from Boeing MCTCB model -- note that all six struts were assumed to be of the same dimensions whereas in the actual MCTCB design, the "top" strut is slightly larger than the other five struts) of

L_1	=	27.078 in
L_2	=	37.064 in
h	=	16.494 in
t	=	.25 in (gage)
d_1	=	4.0513 in
d_2	=	3.5513 in
I	=	23.24 in^4

the moment of inertia of a flat plate (about the edge) is given by:

$$I_p = \frac{1}{12} T^3 C$$

Substituting $I=I_p$ and $C=L_1$,

$$T = 2.2 \text{ in}$$

The thickness of a flat plate with the same chord lengths and height as the strut airfoil and an equivalent moment of inertia (therefore, an equivalent stiffness) is 2.2 in. Figure A2b depicts the approximate flat plate strut geometry as modeled in NASTRAN.

APPENDIX 3. NASTRAN Property Cards for Honeycomb and I-Beam Elements

General Modeling Methods

Honeycomb

Honeycomb panels are modeled in NASTRAN as plate elements with equivalent stiffness to the honeycomb. The face sheets are assumed to carry the membrane loads; transverse shear loads are carried by the core material. It is assumed that the bending loads are carried by a single sheet with a thickness equal to twice the face sheet thickness. The bending stiffness of the panel is then modified by a bending stiffness parameter to account for the depth of the core material. Similarly, the shear thickness of the panel is increased by an input parameter as described below.

The property card for plate elements (PSHELL) in NASTRAN includes the following information (ref. 2):

MID1	Material identification number for the membrane
T	Default value for the membrane thickness
MID2	Material identification number for bending
I/I_m	Bending stiffness parameter
MID3	Material identification number for transverse shear
T_c/T	Transverse shear thickness divided by membrane thickness
Z_1, Z_2	Fiber distances for stress computation

The geometric parameters for the honeycomb panel are shown in Figure A3 and are defined as:

T_s	Face sheet thickness
T_c	Core depth

When modeling a honeycomb panel, the plate thickness (T) is set equal to twice the face sheet thickness ($T=2T_s$). The bending stiffness parameter is found by comparing the bending stiffness of the plate modeled without the core depth with that of the actual honeycomb plate. The bending stiffness (moment of inertia) of the modeled plate is given by:

$$I_m = \frac{1}{12}T^3$$

The bending stiffness of the actual honeycomb panel is given by:

$$I = 2T_s\left[\frac{(T_s + T_c)}{2}\right]^2$$

The bending stiffness parameter is then found from

$$\frac{I}{I_m}$$

The transverse shear thickness of the panel is increased by the amount

$$\frac{T_c}{T}$$

Finally, the fiber distance for stress data recovery is input as the neutral surface of the honeycomb panel,

$$Z_1 = Z_2 = \frac{1}{2}T_c + T_s$$

The material identification number for the face sheet material is input for both MID1 and MID2. The material identification number for the core material is input for MID3.

I-Beam

The rings and longerons of I-beam cross-section are modeled in NASTRAN as beam elements. The characteristics of the I-beam cross-section are input through the definition of the moments of inertia.

The property card for beam elements (PBEAM) in NASTRAN includes the following information (ref. 2):

MID	Material identification number
A(A)	Area of beam cross section at end A
I1(A)	Area moment of inertia at end A for bending in plane 1 about the neutral axis (Izz in the element coordinate system)
I2(A)	Area moment of inertia at end A for bending in plane 2 about the neutral axis (Iyy in the element coordinate system)
I12(A)	Area product of inertia at end A (Izy in the element coordinate system)
J(A)	Torsional stiffness parameter at end A (Ixx in the element coordinate system)

The geometric parameters for the I-beam are shown in Figure A3 and are defined as:

Tcap1	Thickness of inside (bottom) cap
Tcap2	Thickness of outside (top) cap
Wcap	Cap width
Tweb	Web thickness
Depth	Beam depth measured from the centerline of cap 1 to the centerline of cap 2
C	Beam centroid

The moments of inertia for a rectangular cross-section is:

$$I_{zz} = \frac{1}{12}bh^3$$

$$I_{yy} = \frac{1}{12}b^3h$$

$$I_{xy} = 0$$

$$I_{xx} = \frac{1}{12}\left(b^3h + bh^3\right)$$

where b is the width and h is the height of the cross-section.

The Parallel Axis Theorem states that

$$I = I' + Ad^2$$

where I' is the moment of inertia of the section about its own centroid, A is the section area, and d is the distance of the section's centroid to the centroid about which the inertia is being calculated. The moments of inertia for the I-beam cross-section are calculated using these definitions.

Axisymmetric Inlet Property Data

Specific data for the cowl, centerbody, centerbody tube, and support struts appears below. All dimensions are average values taken directly from the results of the IPSSO analysis.

Cowl

Honeycomb Panels

T_S	=	.01 in
T_C	=	.3363 in
MID1	=	Ti-6Al-4V
T	=	.02 in
MID2	=	Ti-6Al-4V
I/I_m	=	900
MID3	=	Ti-3Al-2.5V
T_c/T	=	16.815
Z_1, Z_2	=	.17815 in

Ring I-beams

Tcap1	=	.02 in
Tcap2	=	.02 in
Wcap	=	1.0 in
Tweb	=	.02 in
Depth	=	.20 in
MID	=	Ti-6Al-4V
A(A)	=	.0436 in^2
I1(A)	=	.000411 in^4
I2(A)	=	.003333 in^4
I12(A)	=	0.

J(A)	=	.003745 in^4

Longeron I-beams

Tcap1	=	.1524 in
Tcap2	=	.1524 in
Wcap	=	1.0 in
Tweb	=	.02 in
Depth	=	1.8231 in
MID	=	Ti-6Al-4V
A(A)	=	.3382 in^2
I1(A)	=	.2616 in^4
I2(A)	=	.025401 in^4
I12(A)	=	0.
J(A)	=	.28703 in^4

Centerbody

Panels (single sheet)

MID1	=	Ti-6Al-4V
T	=	.01 in
MID2	=	Ti-6Al-4V
I/I_m	=	1.0

Ring I-beams

Tcap1	=	.02 in
Tcap2	=	.02 in
Wcap	=	.75 in
Tweb	=	.02 in
Depth	=	.75 in
MID	=	Ti-6Al-4V
A(A)	=	.0446 in^2
I1(A)	=	.004868 in^4
I2(A)	=	.001407 in^4
I12(A)	=	0.
J(A)	=	.006275 in^4

Centerbody Tube

Honeycomb Panels

T_s	=	.02 in
T_c	=	.20 in
MID1	=	Ti-6Al-4V
T	=	.04 in
MID2	=	Ti-6Al-4V
I/I_m	=	90.75

MID3	=	Ti-3Al-2.5V
T_c/T	=	5.0
Z_1, Z_2	=	.12 in

Support Struts

Panels (single sheet)

MID1	=	Ti-6Al-4V
T	=	2.2 in
MID2	=	Ti-6Al-4V
I/I_m	=	1.0

APPENDIX 4. List of Symbols

A	area of beam cross-section
b	width of a rectangular cross-section
C	wing chord; beam centroid
d	distance of section centroid from inertia axis
d_1	maximum external airfoil depth
d_2	minimum internal airfoil depth
Depth	beam depth measured from the centerline of the inside cap (cap 1) to the centerline of the outside cap (cap 2)
E	elastic tensile modulus
E_c	compressive modulus
F_{ty}	tensile yield strength
F_{tu}	ultimate tensile strength
G	shear modulus
h	strut height; height of a rectangular cross-section
H	maximum wing depth (thickness)
I	moment of inertia
I'	area moment of inertia with respect to the centroid axis
I1	area moment of inertia at end A of a beam for bending in plane 1 about the neutral axis (I_{zz} in the element coordinate system)
I2	area moment of inertia at end A of a beam for bending in plane 2 about the neutral axis (I_{yy} in the element coordinate system)
I12	area product of inertia at end A of a beam (I_{zy} in the element coordinate system)
I_m	bending stiffness of a modeled plate
I_p	moment of inertia of a flat plate
J	torsional stiffness parameter at end A of a beam (I_{xx} in the element coordinate system)
L	minimum strut chord for minimum moment of inertia
L_1	strut chord length at the center tube
L_2	strut chord length at the cowl
MID	material identification number
MID1	material identification number for the membrane
MID2	material identification number for bending
MID3	material identification number for transverse shear
t	strut skin thickness
T	membrane thickness
T_c, Tc	core depth
T_s, Ts	face sheet thickness
Tcap1	thickness of inside (bottom) cap of I-beam
Tcap2	thickness of outside (top) cap of I-beam
Tweb	web thickness
Wcap	cap width
Z_1, Z_2	fiber distances for stress computation
ρ	density
μ	Poisson's ratio

APPENDIX 5. List of Tables and Figures

	Weight (lb)			
	Cowl	Centerbody	Support Structure	Total Structure
IPSSO Inlet	451.95**	51.35	600.00	1103.30
NASTRAN Inlet	401.60***		569.73	1280.41
MCTCB*	800.00	220.00	600.00	1620.00

* Ref. 3
** Includes 198.64 lb for external skin
*** Cowl and centerbody; includes 198.64 lb for external skin.

Table 1. Inlet Weight Comparison

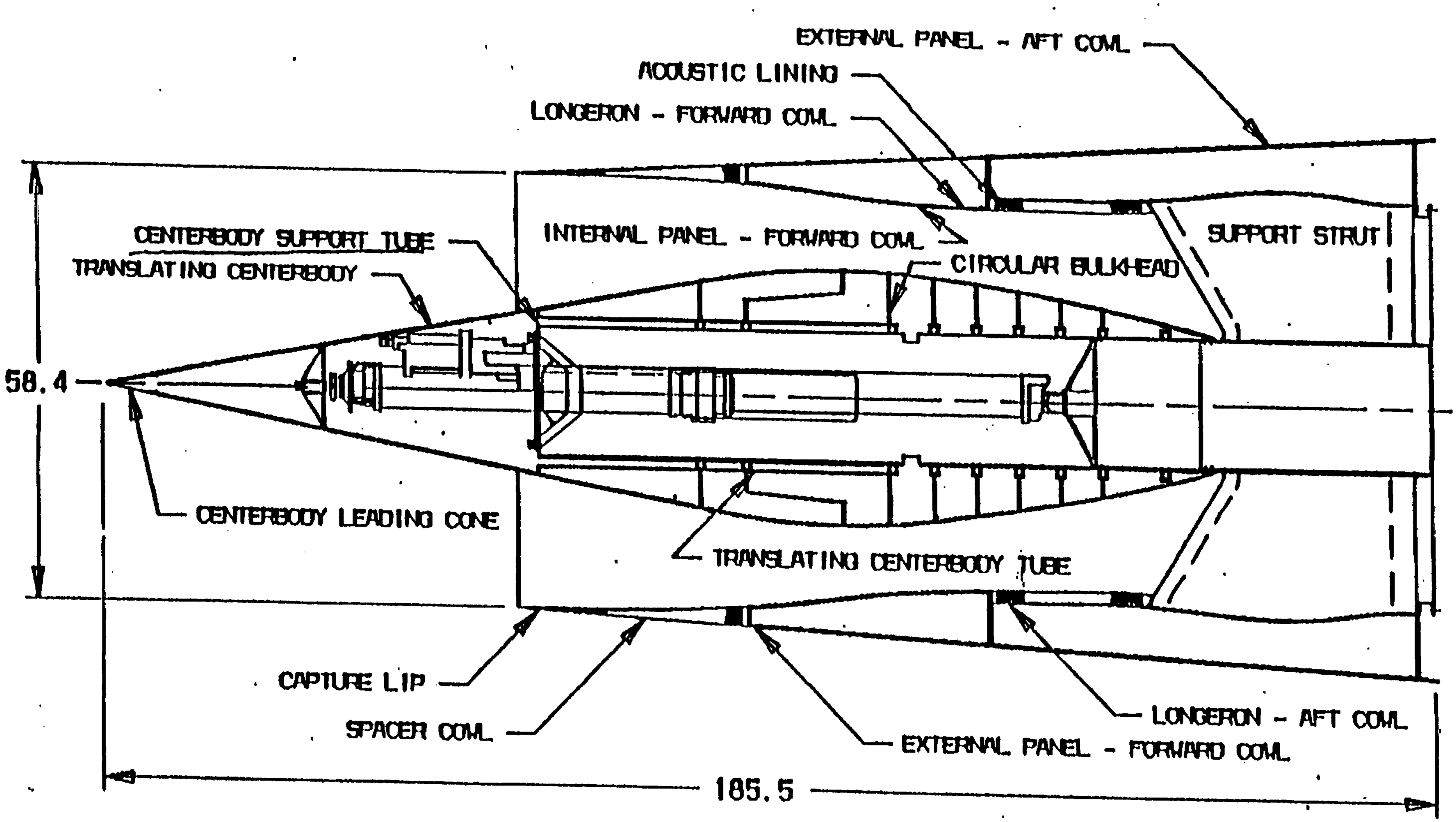


Figure 1a. Mixed-Compression Translating Centerbody Inlet (ref. 3)
(dimensions in inches)

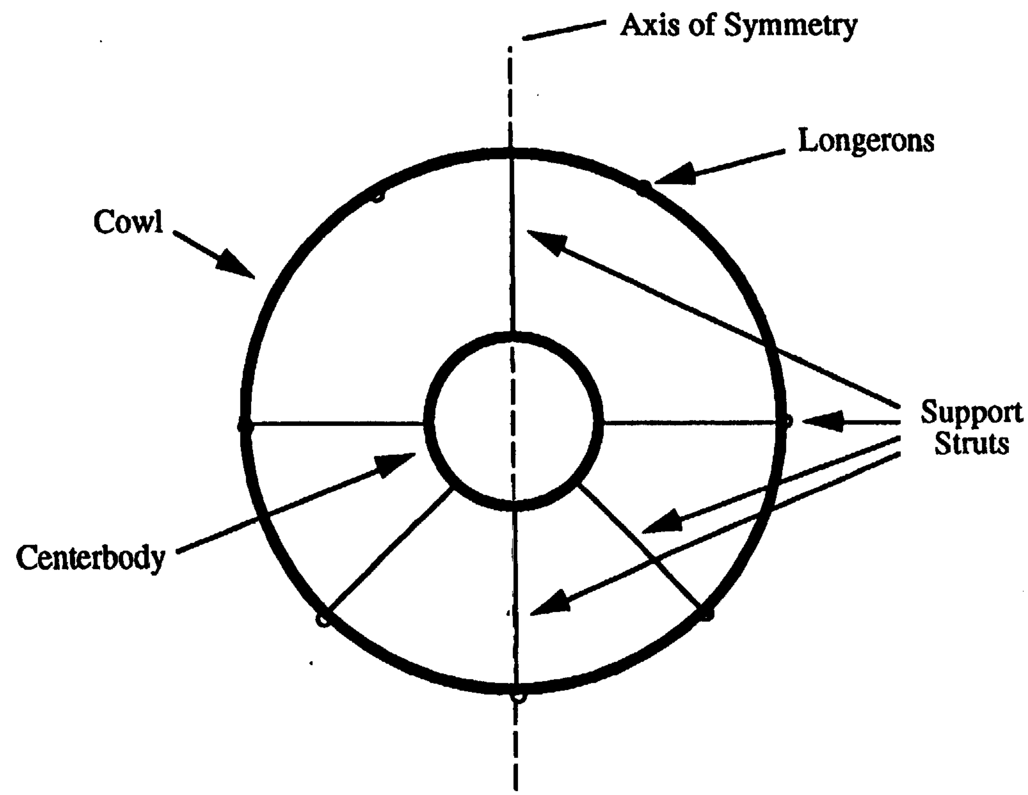


Figure 1b. Mixed-Compression Translating Centerbody Inlet -- Aft Cross-Section

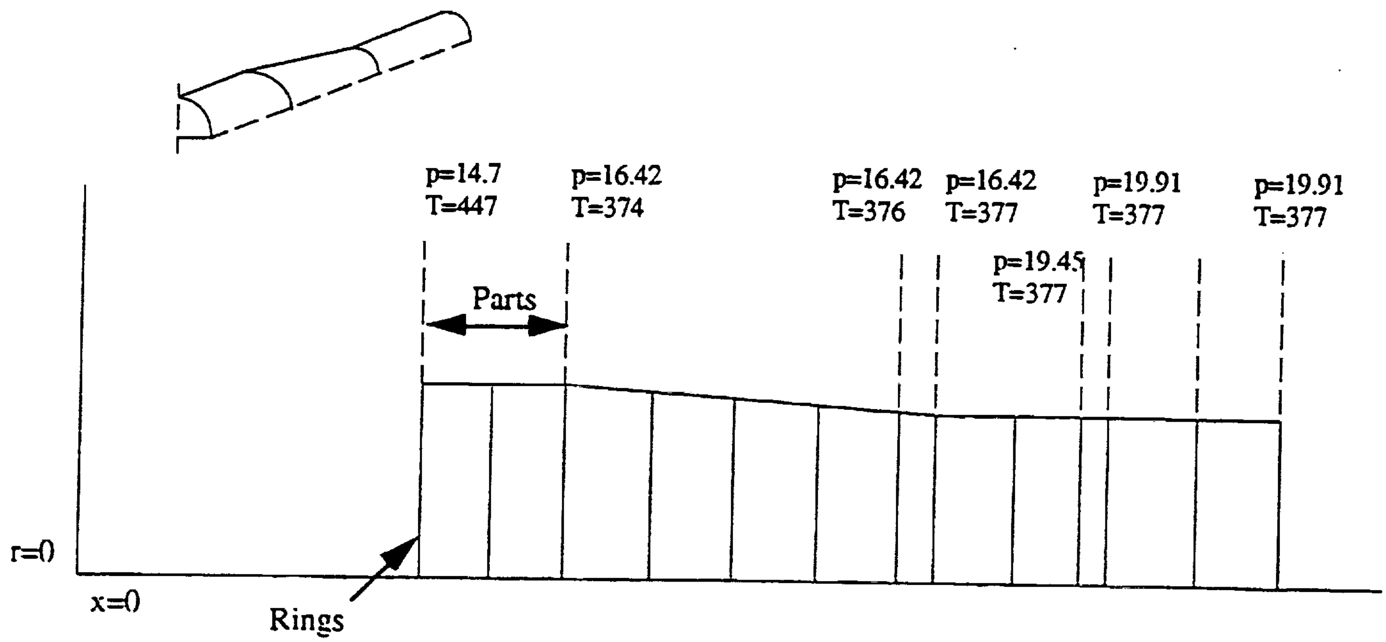


Figure 2a. IPSSO Model of the Inlet Cowl
(p=psi, T=degrees F)

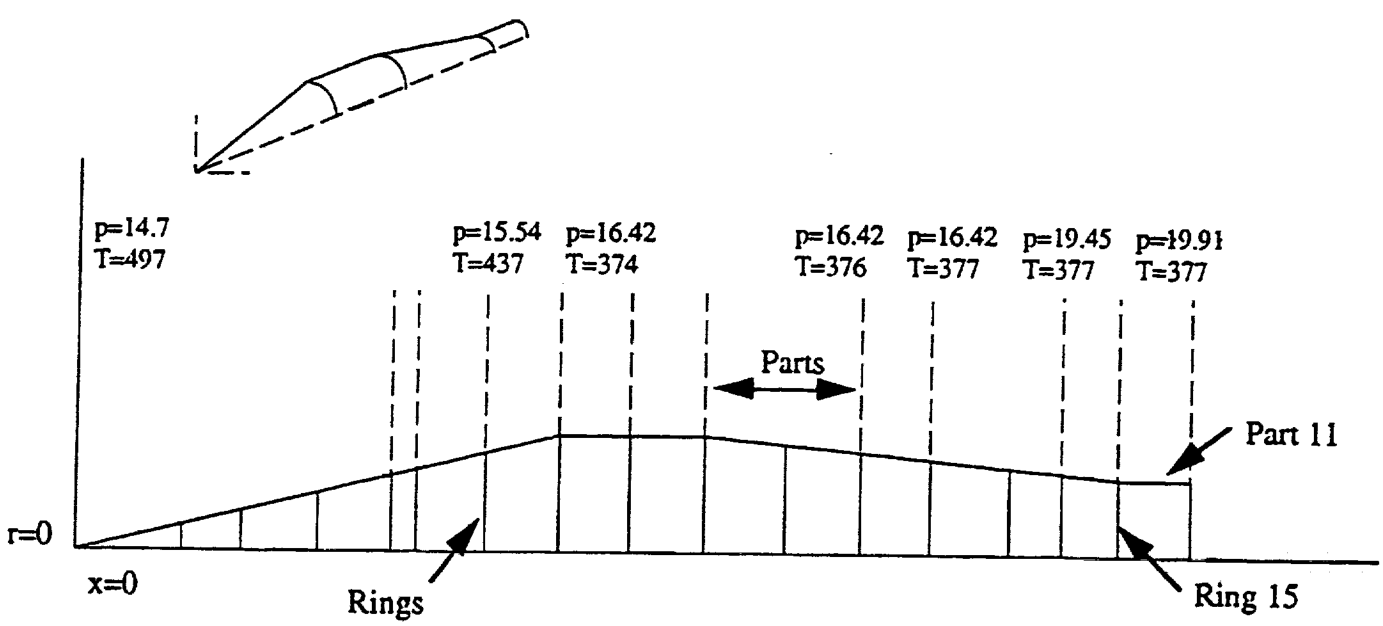


Figure 2b. IPSSO Model of the Inlet Centerbody
(p=psi, T=degrees F)

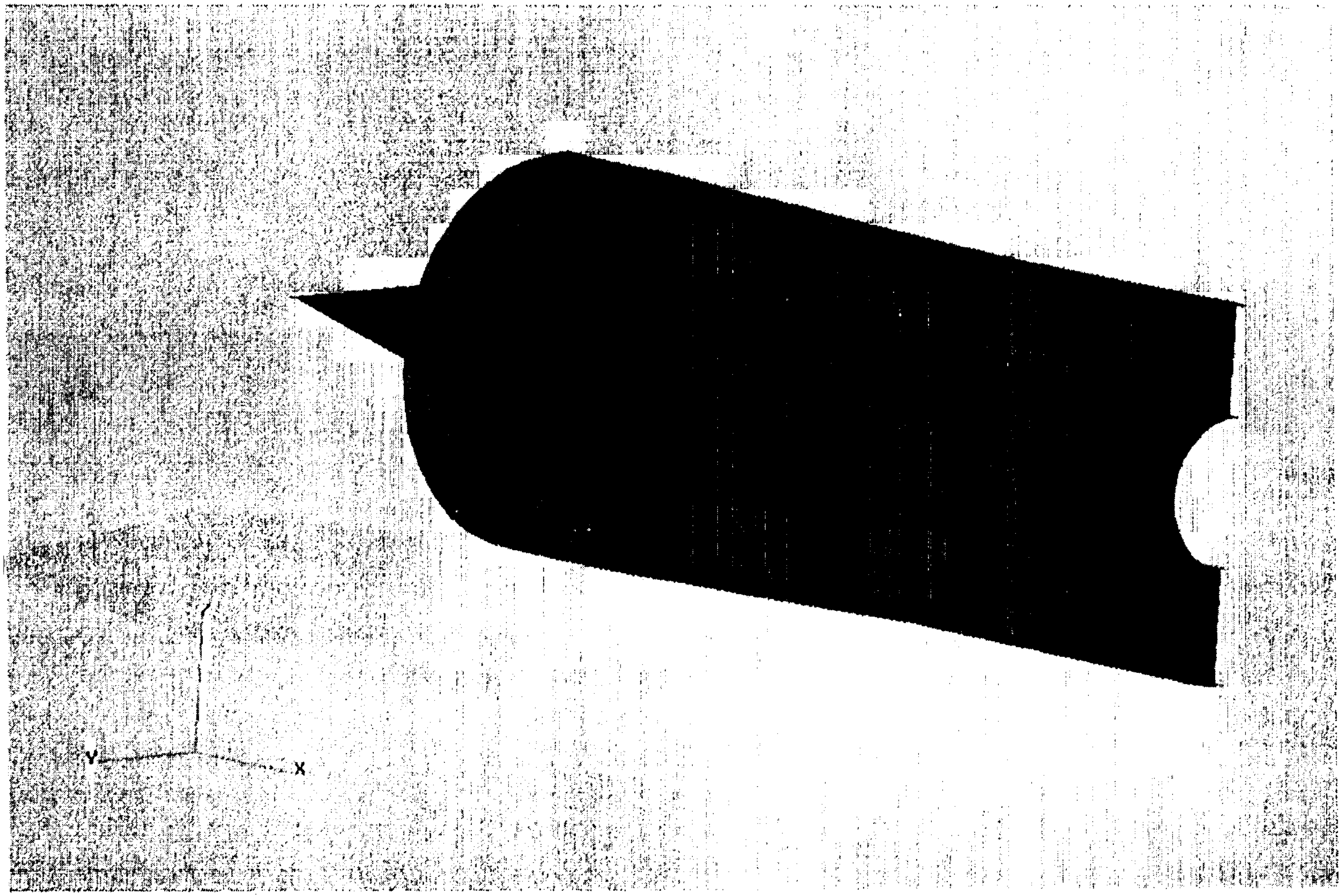


Figure 3a. NASTRAN Inlet Geometry

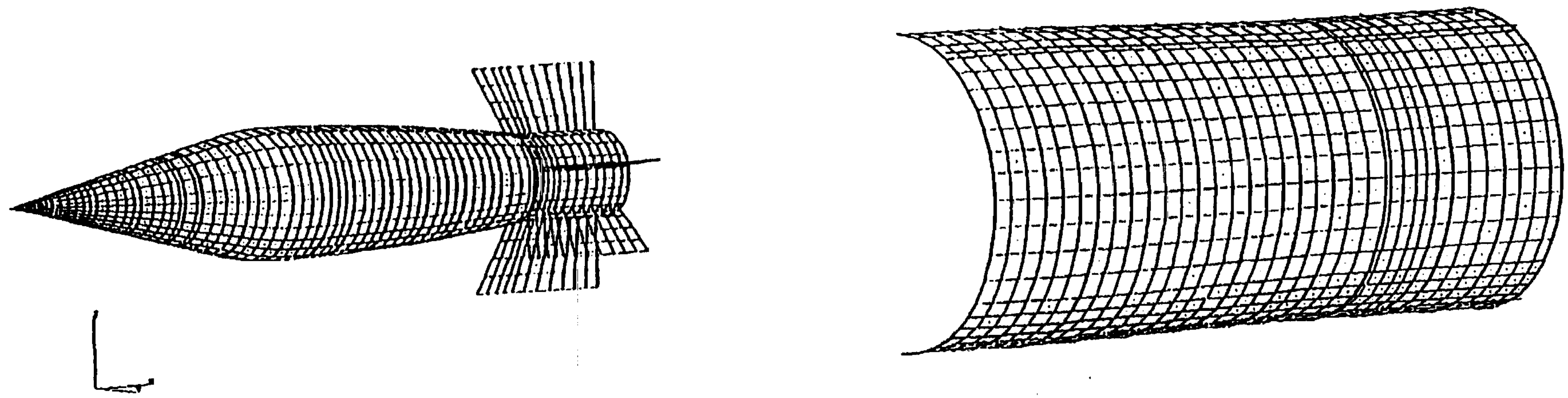


Figure 3b. NASTRAN Inlet Finite Element Model

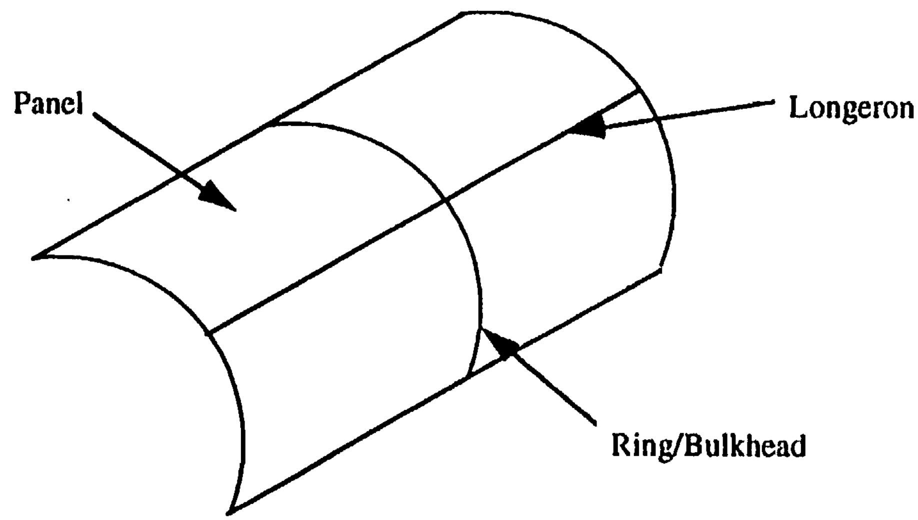


Figure 4. Structural Components

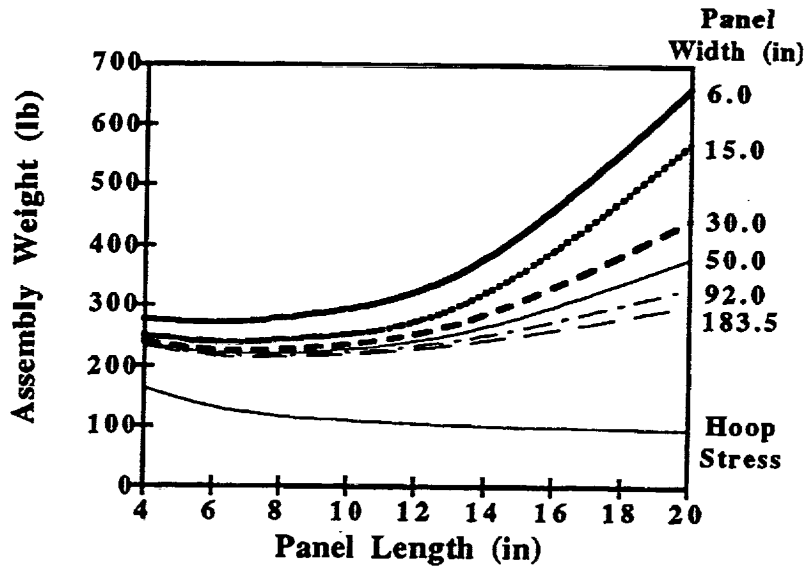


Figure 5a. IPSSO Inlet Cowl Weight for Various Panel Lengths and Widths (diameter=58.4 in)

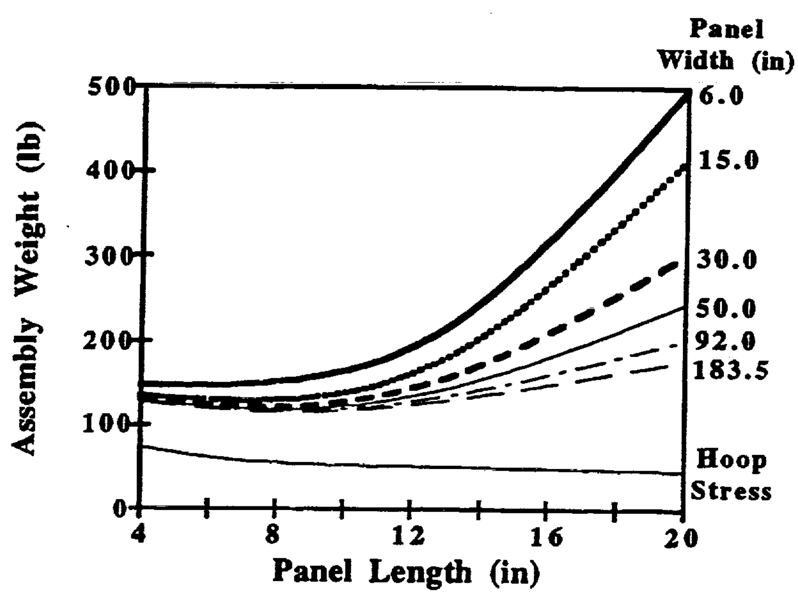


Figure 5b. IPSSO Inlet Centerbody Weight for Various Panel Lengths and Widths (diameter=58.4 in)

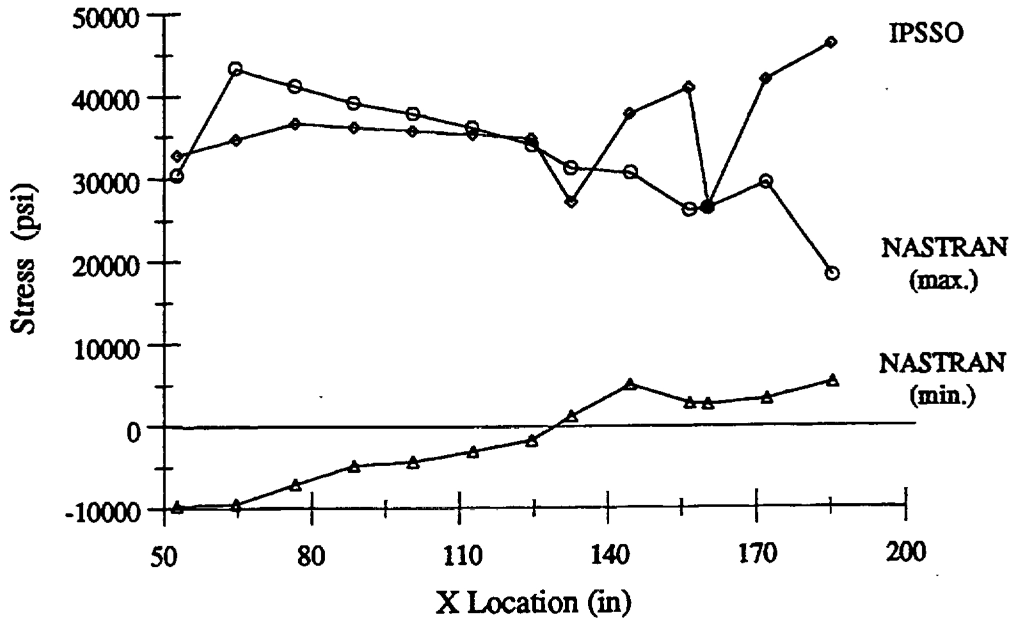


Figure 6a. Cowl Ring Stresses

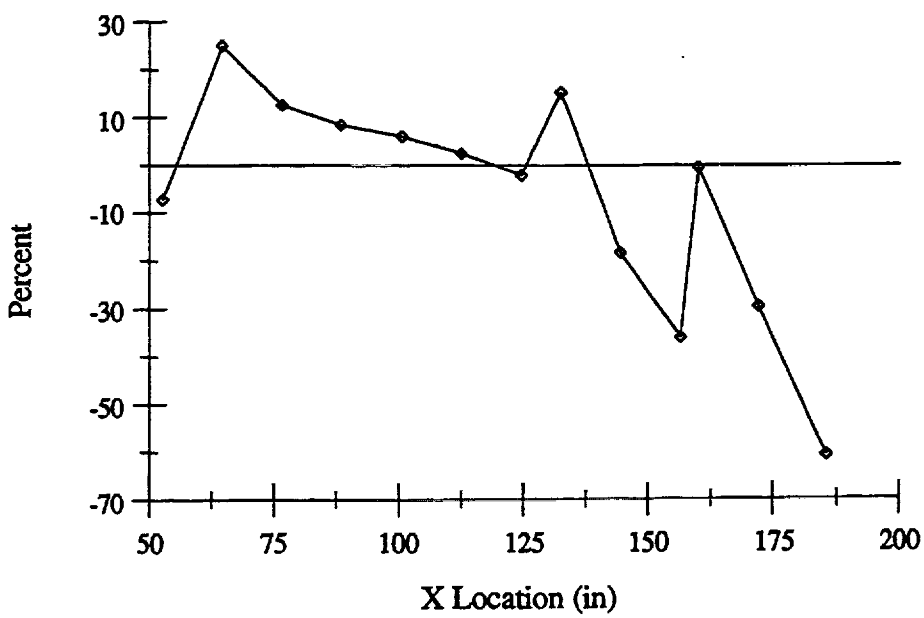


Figure 6b. Difference Between NASTRAN Max. Tensile Stress and IPSSO Max. Stress for Inlet Cowl

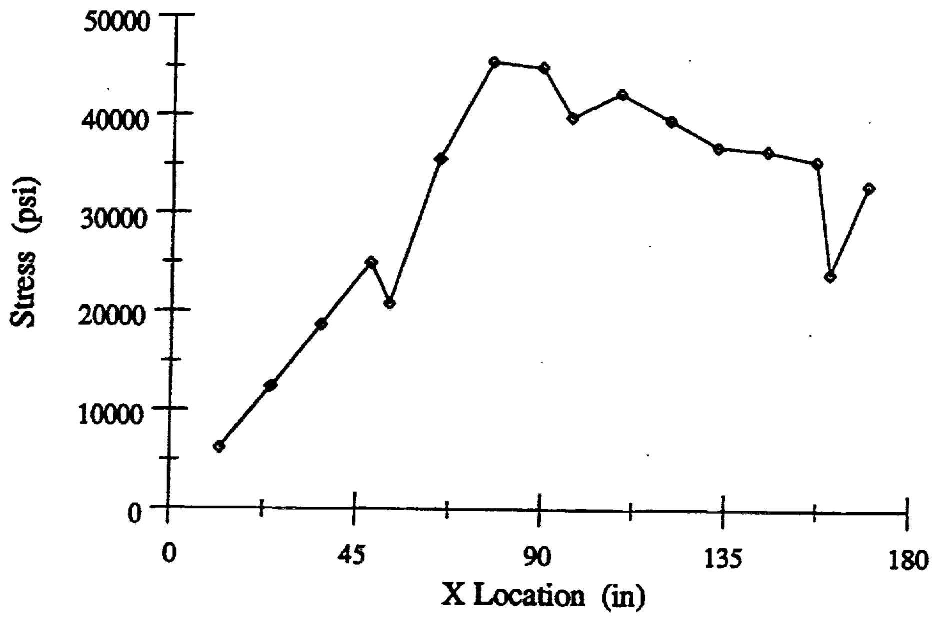


Figure 7a. Max. IPSSO Ring Stresses for Inlet Centerbody

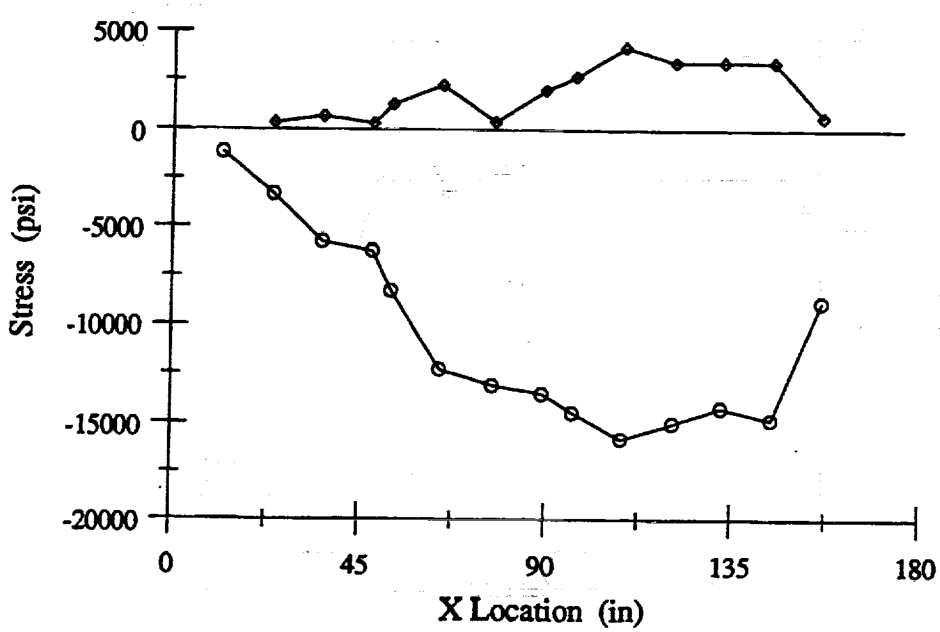
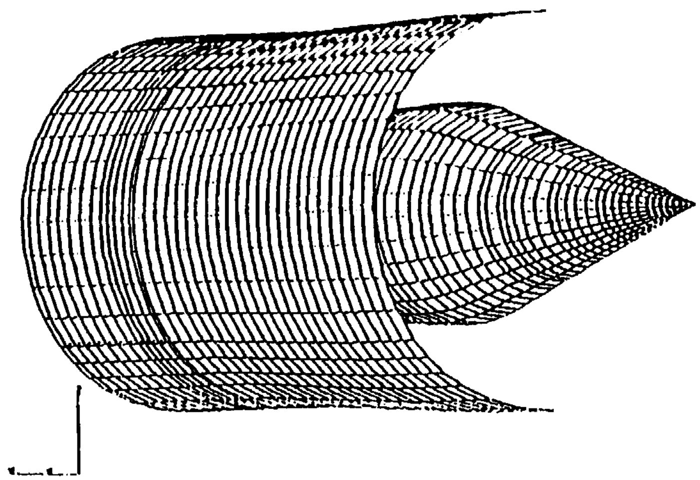
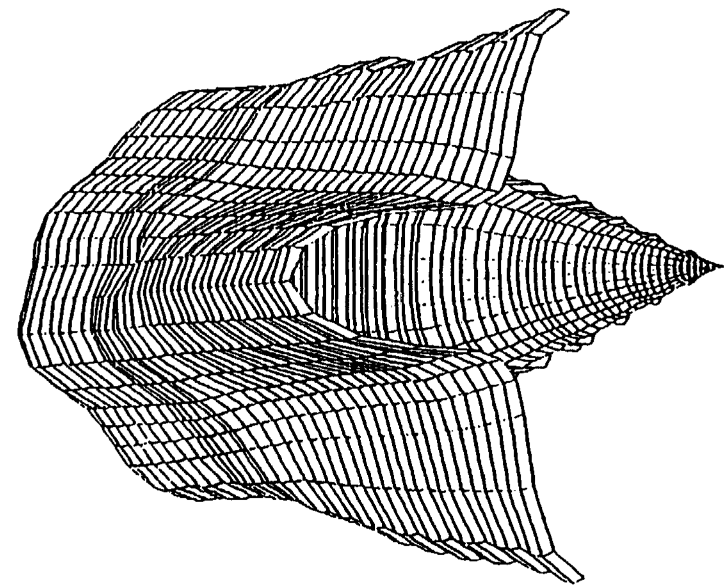


Figure 7b. Max. and Min. NASTRAN Ring Stresses for Inlet Centerbody



Undeformed



Deformed

Figure 8. Deformed Inlet From NASTRAN

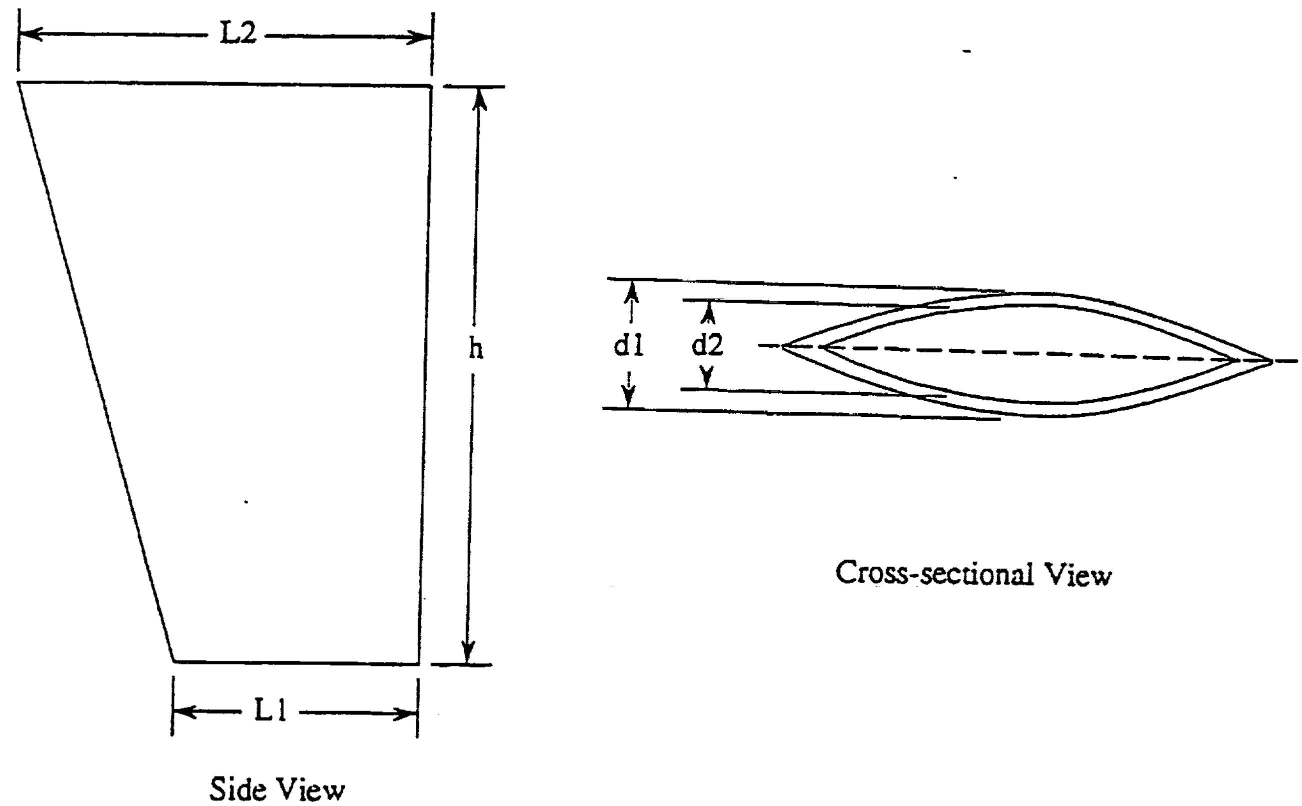


Figure A2a. Actual Support Strut Geometry

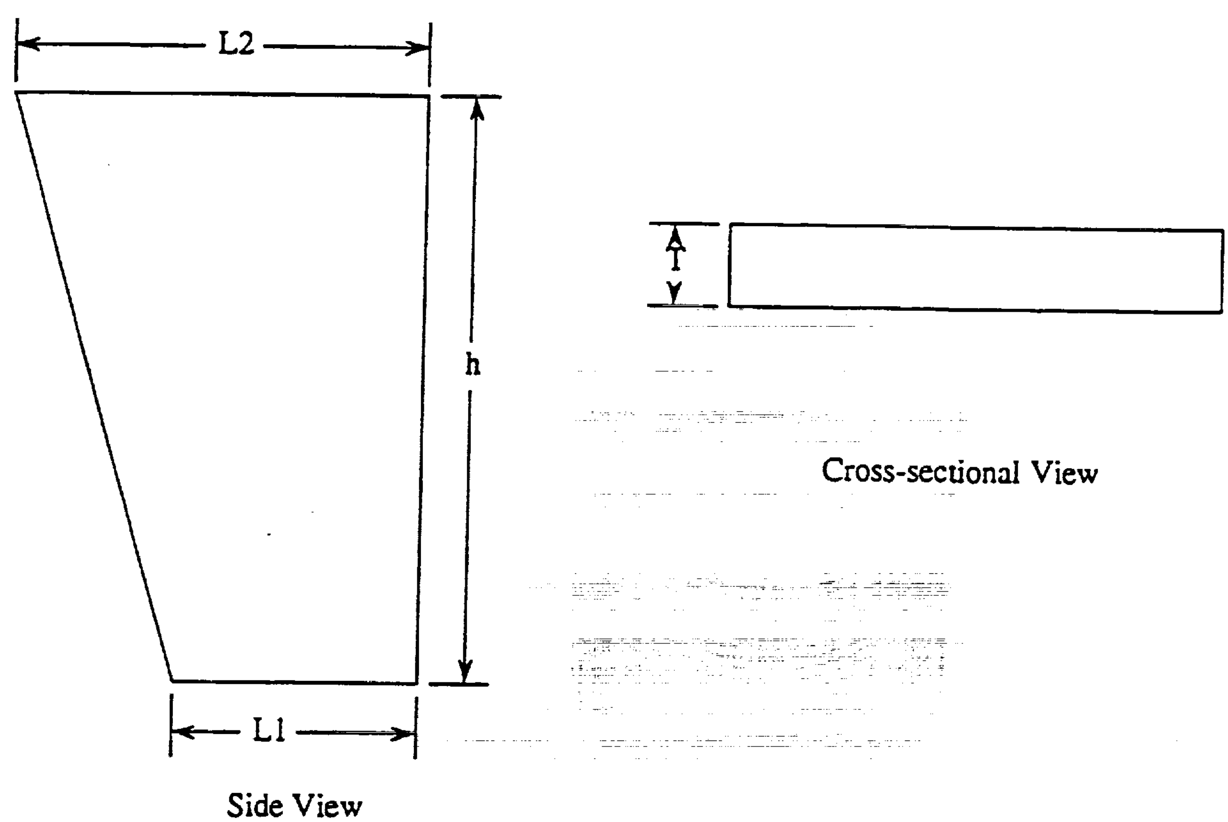


Figure A2b. Flat Plate Strut Geometry

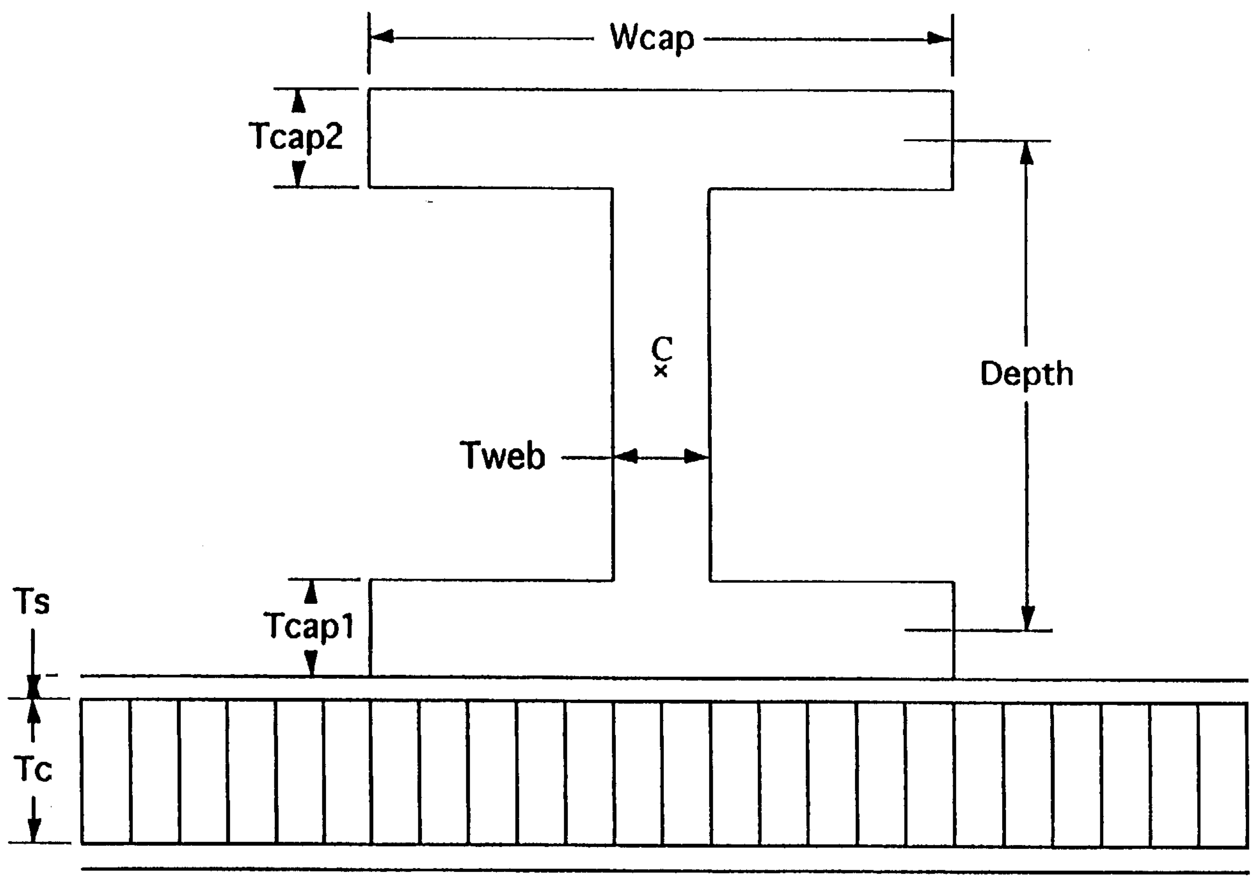


Figure A3. Honeycomb Panel and I-Beam Cross-Section

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY (*Leave blank*)

2. REPORT DATE
August 1994

3. REPORT TYPE AND DATES COVERED
Technical Memorandum

4. TITLE AND SUBTITLE
Investigation of Methods for the Structural Weight Analysis of a Mach 2.4 Axisymmetric Inlet

5. FUNDING NUMBERS
WU-505-69-50

6. AUTHOR(S)
Shari-Beth Nadell

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135-3191

8. PERFORMING ORGANIZATION REPORT NUMBER
E-9048

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, DC 20546-0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA TM-106693

11. SUPPLEMENTARY NOTES
Responsible person, Shari-Beth Nadell, organization code 2400, (216) 977-7035.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified - Unlimited
Subject Category: 07 Distribution: Nonstandard

This publication is available from the NASA Center for AeroSpace Information, (301) 621-0390.

12b. DISTRIBUTION CODE

13. ABSTRACT ***(Maximum 200 words)***

Structural design and analysis tools appropriate for estimating the structural weight of an axisymmetric inlet designed for Mach 2.4 cruise were evaluated. Little information regarding the inlet mechanical design is available in the preliminary design phase, so it is necessary to first develop a reasonable structural design before estimating the inlet weight. The Internally Pressurized Structure Synthesis and Optimization (IPSSO) program, employing an analytical approach, was chosen for evaluation due to its combined design and analysis capabilities. The inlet design produced by IPSSO was then analyzed using the NASTRAN finite element program. The finite element analysis was performed to help identify the limitations of the analytically based code as well as to evaluate NASTRAN for this application. Comparison between the IPSSO inlet weight and that of a similar inlet developed by the Boeing Commercial Airplane Group was also made. Program evaluation concluded that the combined use of IPSSO to create an initial design and NASTRAN to perform a numerical analysis would provide the capability to evaluate a limited number of inlet design The development of a new tool for the minimum weight design and analysis of inlet structures would be required for greater flexibility in evaluating inlet conceptual designs.

14. SUBJECT TERMS
Supersonic inlets; Engine design; Axisymmetric inlets; Weight analysis

15. NUMBER OF PAGES
35

16. PRICE CODE
A03

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	