

DEVELOPMENT AND ANALYSIS OF THE LEARJET 54/55 FUSELAGE
NASTRAN MODEL USING SUBSTRUCTURE TECHNIQUES

BY

ROBERT R. BOROUGHS, SIVAM PARAMASIVAM, AND JOANNA WERNER

SUMMARY

Development and analysis of the Learjet 54/55 fuselage Nastran model presented a real challenge considering the size of the task and the resources that were available at the beginning of this project. Consequently, this structure was broken down into several substructures to make the modeling and analysis effort more manageable. Since the geometry was fairly complex in some areas and in order to provide flexibility for future configuration studies, a series of local coordinate systems were used to describe the model. This work was accomplished using several different computer systems, the more recent of which were connected by a high speed data communications lines to perform the tasks of model generation and Nastran analysis.

INTRODUCTION

The Learjet Model 50 series aircraft has become the latest and largest member of the Learjet product line (see Figure 1). This airplane was an almost completely different aircraft from earlier Learjet models. The wing has been extended six feet from the original Learjet wing configuration, and the tip tanks have been removed and replaced by winglets. A paper describing the Nastran finite element analysis of this wing which was also used on the 28/29 airplanes can be found in another NASA document (Ref. 1). Discussion here will be primarily directed toward the Nastran analysis of the fuselage and vertical fin.

The fuselage on the Model 54/55 aircraft has been increased in diameter as well as length over previous models. Construction in the fuselage was the typical skin stringer arrangement with frames located at given increments to provide rigidity to this shell structure. The windshield was similar to other Learjet windshields with the large unsegmented stretched acrylic panels, except that these panels were bigger and two small side windows were installed on the aft perimeter of the windshield. Attachment of the wing to the fuselage was accomplished through eight fittings, four on each side of the fuselage, which provided fail safe capability in this area. The fuselage structure was cut out at the wing-fuselage juncture to permit wing continuity through the fuselage. A keel beam was installed beneath the wing to provide a more uniform internal load path in this region. This keel beam structure was also continuous forward and aft of the wing attachment.

Pressure bulkhead construction consisted of either sheet metal webs reinforced with stringers and support beams or honeycomb panels reinforced with structural beams. The aircraft was powered by two Garrett AiResearch TFE 731-3 turbofan engines mounted on each side of the aft fuselage section. The engine support structure consisted of a forward and aft box beam which were also integral to two partial bulkheads. Configuration of the vertical tail included five spars covered by aluminum sheet and stiffened by formed sheet metal ribs.

BACKGROUND

Initial studies on the 50 series aircraft started in 1976. Nastran was used as preliminary design tool to study various structural arrangements and determine which configuration was more advantageous. As these investigations matured and more parameters became defined, a transition from a conceptual finite element fuselage model to a final configuration model was necessary. The final configuration fuselage model proposed was to have a fine enough mesh to accurately define the internal loads distribution throughout the structure for all load cases, yet not be overly complex so that the lead times and costs would be excessive. With these guidelines in mind a model of approximately 20,000 degrees of freedom was recommended. Development of this size model still presented real challenges considering the computer resources that were available at the beginning of this effort. There was a good deal of concern about the logistics of model generation, management of the resulting data base, and the method of analysis. The computer system initially considered to carry out these functions was an in-house IBM 370-145, and the method of analysis proposed was static substructuring. If the Nastran analysis were to become too large for the IBM 370-145 to handle effectively, an alternate course of action was planned where the Nastran model would be run on a Cyber computer at a service bureau.

SUBSTRUCTURE DEFINITION

Based on previous runs on the IBM 370-145 computer, a maximum substructure size of 3500 degrees of freedom was established. Each fuselage node point was considered to have the capability of transferring all 6 degrees of freedom. Grid points on the outer surface were to be located at the intersection of the frame and stringer members, and grid points in the fuselage interior were to be defined by the intersection of primary structural beams and stiffeners. These constraints and the basic characteristics of the structure resulted in dividing the Nastran model into seven substructures (see Figure 2). Before the modeling ever began a node and element numbering system was established for the entire structure. Element and grid point numbers were assigned to each substructure in a progressive manner so that one substructure could later be easily merged or subdivided with adjacent substructures. Node numbers were assigned in a sequence that attempted to minimize the matrix bandwidth. Element numbering was keyed to the grid point numbering so that both the grid points

and connection members had a similar numbering sequence.

After studying the pros and cons of various coordinate systems, a geometric definition using a series of local cylindrical coordinate systems in conjunction with the basic rectangular coordinate system was selected. In this system the grid points at each frame location on the outer surface were described by a local cylindrical coordinate system unique to that frame. Grid points interior to the fuselage model were defined in a rectangular coordinate system. There were several advantages in using this type of approach. First of all, this made the modeling of some of the more complex geometry much easier, and also simplified the determination of offsets to be used with the BAR elements which were to define the frame members. Consideration was also made for the incorporation of a fuselage stretch and other possible modifications that would impact future modeling and analysis. Thus, the use of a local cylindrical coordinate system for each frame would permit a fuselage stretch by inserting a fuselage plug and redefining the affected coordinate cards with little or no changes to existing grid or element cards.

COMPUTERS USED FOR MODEL GENERATION

Several in-house computers were used in different phases of the model generation. The first phase of the modeling was the generation of the connectivity cards. Simple routines were written for CONROD and CQDMEM2 elements (Ref. 2) which given the bay ID numbers generated each connection element. These routines were originally run on an IBM 1130 but all these programs were eventually converted to run on an IBM 370-145 or on a PDP 11/70. A similar situation occurred with some frames which were constant in section around the circumference. A program was used that generated all four cards necessary for each BAR element, complete with connectivity, cross-section area, X and Y moments of inertia, and stress recovery coefficients, but with all offsets equal to zero. These values had to be added manually, but even after adding these values by hand, much time was saved using these routines.

The second phase of model generation was the section property calculation. This was done with two different routines. The first routine used ran on the IBM 370-145 and was primarily designed to calculate the section properties for a channel section made of bent-up sheet metal. This routine could also be used to calculate the properties for an angle section. A second routine that was used calculated the properties for any extruded section used such as channels, I-sections, T-sections, and angles. Special provisions for a general section built up of rectangles and fillets were also incorporated in this program. This routine ran interactively on the PDP 11/70 in-house computer.

In some cases where a frame section was comprised of two or more bent up sheet sections which were fastened together, the combined section properties had to be computed. Using the appropriate section property program to find the properties of each component, these properties were then input to a transformation program which converted all the properties with respect to the composite centroid and combined these values. This program was set up to run

on the in-house IBM 370-145.

FORWARD SECTION

The forward section began at frame 1 and extended to frame 16 just behind the cabin door (see Figure 2). This portion of the model was divided into two substructures which were identified as the nose substructure and crew substructure. The nose substructure extended from frame 1 to frame 6 (see Figure 3) and the crew substructure extended from frame 6 through frame 16 (see Figure 4). The components of the forward fuselage consisted of circumferential frames oriented in a fuselage station reference system, stringers located approximately normal to the frames, and the skins which covered this framework. Along the bottom centerline of the fuselage was the keelbeam which runs almost the complete length of the fuselage to give additional longitudinal stiffness and to provide a more continuous load path around the wing. Other components of the forward section included a forward pressure bulkhead, a windshield, and a cabin door.

The grid points defining the outside contour for these substructures as well as the other substructures were located at the frame stringer intersections. These grid points were defined in a local cylindrical coordinate system coincident with each frame. The local coordinate systems were defined with reference to a basic rectangular system where the X axis runs longitudinally aft, the Y axis left hand outboard, and the Z axis down. The local systems were oriented with R radially outward from the centerline, θ counterclockwise from the basic Y coordinate, and positive Z oriented forward. The grid points were numbered with even values on the left hand side and odd values on the right hand side. These numbers were keyed off the local system identification numbers to give unique ID's for each grid point. Grid points in the fuselage interior utilized a similar technique.

Stringer members in the forward section were modeled using CONRODS, since these elements had small cross-sectional areas that acted primarily in axial load transfer. In a few cases a torsional stiffness value was included to allow some torsional loading. The CONROD IDs directly keyed off the grid point ID's. The frames were modeled as BARS (Ref. 2) to carry axial and bending loads. The BAR orientation was defined by a \bar{v} vector at the element origin pointing radially outward. QDMEM2 elements were used to model the skin, since these panels have small thicknesses and generally do not carry significant local bending loads. The keelbeam was modeled using CONRODS along the four corners with SHEAR elements (Ref. 2) on vertical sides. The bottom was coincident with the outer skin and consequently was already defined by QDMEM2 panels.

Stretched acrylic material was used for windshield and cabin windows. These members were generally quite thick and were modeled using QUAD2 elements (Ref. 2) which had bending capability. The forward end of the pressure vessel was located at frame six in the form of a pressure bulkhead. This bulkhead was comprised of a thin web which was supported by vertical stiffeners on the forward side and horizontal stiffeners on the aft side. The thin web was modeled with

QDMEM2 elements, and the stiffeners were modeled with BAR elements.

The cabin door structure consisted of a framework of intersecting frames and stringers with skin and doublers on both the inner and outer sides of this framework. A double set of grid points was used to define the perimeter of the door. The first set of grid points defined the door frame or cutout in the fuselage while the second set of grid points defined the edges of the door. Once again the frames were modeled with BARS, the stringers were modeled with CONRODS or BARS depending on the depth of the member, and the skin panels and doublers were modeled using QDMEM2 elements. The entire door was connected to the fuselage by the use of RIGID (Ref. 2) elements which transferred the appropriate degrees of freedom.

MID SECTION

The fuselage mid section began at frame 16 and continued through frame 31 just aft of the wing trailing edge (see Figure 2). This section was divided into two substructures at frame 24 which was just in front of the wing leading edge. The substructure between frames 16 and 24 was identified as the cabin substructure (see Figure 5), and the substructure between frames 24 and 31 was called the center substructure (see Figure 6). The shell structure for these sections consisted of frame-stringer-skin type construction and contained the keel beam, the escape/baggage door, passenger windows, the frame 24 partial bulkhead, a pressurized baggage floor, the aft pressure bulkhead, and a portion of the fuselage fuel cell bay.

The grid point locations and modeling of the fuselage shell structure for the mid section was accomplished in basically the same manner as described in forward section modeling discussion. The keel beam which runs almost the entire length of the fuselage was an open box type structure forward of frame 24, but aft of frame 24 and the wing leading edge region the keel beam has become a completely closed box. Modeling of the keel beam in this region was accomplished using two dimensional Nastran elements. CONROD members were used to model both the upper caps, lower caps, and vertical stiffeners. SHEARS were used to model the vertical webs while QDMEM2 elements were used to model the horizontal webs.

Window cutouts were reinforced with doublers as well as frame members in certain locations. The doubler panels were modeled using QDMEM2 elements and were connected to the same grid points as the outer skin panels. The windows were made of the same stretched acrylic material as the windshield but only thinner, and these panels were also modeled with QDMEM2 elements.

Between frames 22 and 25 and stringers 6 through 15 on the right side of the fuselage was located the escape/baggage door. A double set of grid points was used to define the perimeter of the escape/baggage door similar to the modeling performed on the main cabin door. Apart from the two close out frames at each end of the door, there were also two inner frames which were adjacent to the window in the door. The door cutout in the fuselage was

reinforced with doublers and additional frame members and intercostals. The door frame and intercostal members were modeled using BAR elements, and the door skin was modeled using QDMEM2 elements. Connection of the escape/baggage door to the fuselage was achieved by using RIGID (Ref. 2) elements between the appropriate degrees of freedom at the door hinge and pinned attachments.

The partial bulkhead at frame 24 served as a close out for the wing cutout in the fuselage. This structure consisted of a thin web supported by horizontal and vertical stiffeners. QDMEM2 elements were used to model the web, and BAR elements were used to model the stiffeners. The top edge of this bulkhead provided the forward support point for the pressurized baggage floor which was made of sandwiched honeycomb plate. Due to the thickness and bending characteristics of the honeycomb, this structure was represented in the Nastran model by QUAD1 elements (Ref. 2). The baggage floor was attached on both sides to a longeron which ran the length of the wing cutout in the fuselage. These longerons also helped to close out the wing cutout and provided a redistribution path for the internal loads. Since the longeron was a large member, BAR elements were used to simulate this structure in the model instead of CONRODS.

Attachment of the wing to the fuselage was accomplished through four fittings on each side of the fuselage in the wing cutout region. The frames located through this section were all double frames. This was done to provide increased stiffness and an adequate load path for the wing reactions. These double frames were all modeled with a single ring of BAR elements to conserve degrees of freedom. Since the wing structure was not simulated in this model, the model was constrained at each of these wing attachment points. The frame 28 structure also provided the support point for the aft pressure bulkhead. This bulkhead also served as the forward retainer for the fuselage fuel cell, and was constructed of sandwiched honeycomb plate. QUAD1 elements were used to model this bulkhead as was previously done for the pressurized baggage floor.

AFT SECTION

The aft fuselage section began just behind the trailing edge of the wing and was divided into two substructures (see Figure 2). These substructures were referred to as the fuel cell (see Figure 7) and tailcone substructures (see Figure 8) and had a mutual boundary at frame 39 which was between the two aft baggage doors. Some of the basic assemblies of the fuel cell section included the fuel cell support structure and aft fuel cell bulkhead, the engine support structure, and the aft keel beam structure. Major features in the tailcone substructure consisted of the aft baggage compartment and the vertical tail support structure. Frames in both substructures were generally oriented in a vertical position with the exception of the frames that attached to the vertical fin which were oriented parallel to the spars in the vertical fin.

Nastran elements used to model the skin, stringer, and frame elements were the same as those used in the other substructures. There was one full

bulkhead in the aft section and several partial bulkheads. The full bulkhead served as the rear boundary of the fuselage fuel bay and was located just in the front of the forward engine beam. This bulkhead was constructed of flat plate reinforced by structural beams. QUAD2 elements (Ref. 2) were used to model the plate structure, and BAR elements were used to model the structural reinforcement beams. One partial bulkhead was located at the aft end of the wing fuselage cutout at frame 31 and served as a close out as well as a fuel retainer for the fuselage fuel bay. Construction of this partial bulkhead was also a flat panel with structural support beams, but the reinforcement beams were generally smaller and there were more of these members than in the aft fuel bulkhead. The webs for this partial bulkhead were modeled with QUAD2 elements, and the support beams were modeled with BARS.

Since grid points in the bulkhead mesh did not always match the grid points on the fuselage outside contour in an even manner, the interface of cross support beams on the bulkheads with the fuselage frames was not always easy to simulate in the Nastran model. MPC equations were used initially to relate the displacement of the grid points in the bulkhead which were adjacent to the outside fuselage contour to the displacement of the grid points on the outside contour. The results of the first Nastran debugging runs with these MPC equations revealed the reactions did not satisfy equilibrium conditions. However, when the MPC equations were removed from the model, then equilibrium was satisfied. This problem was reported to COSMIC, and a sample Nastran run with the MPC equations and faulty equilibrium conditions was sent to the COSMIC staff. After reviewing the Nastran run, COSMIC indicated that the problem appeared to be related to writing an MPC equation for grid points which were in different coordinate systems whose degrees of freedom were not on parallel reference axes. The coordinate system used for the grid points on the fuselage outside contour was a local cylindrical coordinate system with the origin at the maximum breadth line of the fuselage while the coordinate system used for the grid points on the fuselage interior was a rectangular coordinate system. Consequently, MPC equations were discarded for this application and tailored BAR elements were used instead.

Two of the other partial bulkheads were located at the forward and aft engine beam supports. These engine beams were box beams that were continuous through the fuselage from right to left. Both beams were curved so that the outboard ends, or engine attach points, were higher than that portion of the engine beam on the aircraft center line. A double frame was also installed at each of these locations to provide greater support for the engine loads. In order to simplify the modeling and conserve degrees of freedom, the combined section properties of the double frame were calculated and one ring of BAR elements was used to model both frames. The partial bulkheads in this region extended from the engine beam on the bottom to the double frame on the sides and on the top. BAR elements were used to model the engine beams, and QUAD2 members were used to model the web panels in the partial bulkhead. There were only a few reinforcement members in the partial bulkhead, and these elements were modeled using BAR elements.

The intersection of the engine beams and the stringer members on the fuselage contour were such that the grid points for these elements did not

coincide at the true intersection point. Since there was internal load transfer capability of all six degrees of freedom at this point, a method of relating this interaction was necessary. RIGID elements (Ref. 2) were used to connect these degrees of freedom, and the results appeared to be satisfactory.

VERTICAL FIN

The seventh substructure in the fuselage model was the vertical fin installation which also included a model of the rudder (See Figure 9). A model of the horizontal stabilizer was constructed, but was not included in this data base in order to minimize the degrees of freedom and cut down on computer run costs. Horizontal stabilizer loads were applied to the vertical fin at the attach points where the horizontal stabilizer and vertical fin joined. The vertical fin consisted of five spars and eight ribs covered with aluminum sheet while the rudder contained one spar and eleven ribs with an aluminum covering. The vertical fin was joined to the tailcone at each of the five spars as well as through attachments on the skin. Spar connections were made at frames 43, 45, 46, 47 and 48 which were canted to accommodate this interface.

A separate local rectangular coordinate system was adopted for the grid locations for both the vertical fin and rudder to facilitate the modeling. The spar caps in the region of the fuselage frames were modeled using BAR elements while the remainder of the spar caps and rib caps were modeled using CONROD elements. Skin panels were modeled using the QDMEM2 member, and SHEAR elements were used to represent the spar and rib webs. Attachment of the rudder to the vertical fin was accomplished through two hinges which were modeled using CONRODS. The torque load in the rudder was restrained by a torque tube attached to the bottom of the rudder which in turn attached to fuselage frame 48 and was modeled using BAR elements. The spar caps and rib caps were again modeled using CONROD elements while the skin was modeled using QDMEM2 panels. CSHEAR elements were used to model the rudder leading edge ribs while TRMEM elements were used for the ribs between the hinge line and trailing edge of the rudder.

GRAPHICS SYSTEMS

Plots for each of the completed substructures were obtained using an in-house plot package developed for Nastran on an in-house PDP 11/70 mini computer. Since this program was originally written for plots with grid points in the basic coordinate system, an expansion to the software was made to incorporate multiple local coordinate systems. The user could access this routine through a Tektronix CRT, and the substructure would be displayed in a three-dimensional view on the screen. Different views could be selected by specifying the appropriate rotational angles about each of the three principal axes. Hard copy plots of very fine resolution then could be obtained by spooling the desired view to a Versatec plotter. This approach allowed the user to correct any noticeable geometry or connectivity errors before going

to the initial Nastran finite element analysis.

LOADS

Several different load conditions and combinations of load conditions were applied to the model. The basic types of loads used during this analysis were pressure loads, down bending loads, up bending loads, and side bending loads. An ultimate and limit internal pressure was applied separately as well as in conjunction with the ultimate down bending and ultimate up bending load cases. Side bending loads were applied to the vertical fin and the rudder.

COMPUTERS USED FOR ANALYSIS

Further error correction was achieved by running the substructures with the Nastran software on the IBM 370-145. However, after the Nastran debugging process was completed, the first few substructures ran so long that even over night turn around became a problem. This difficulty appeared to be significantly influenced by increased usage of the computer not only by structures personnel but also by other departments. One of the alternatives considered as a solution for this situation was to break the model into smaller substructures and increase the number of substructures from seven to eleven substructures. A second alternative was to run the model on a more powerful outside computer. At that point in time the second of the two alternatives seemed to be more satisfactory in order to meet schedule requirements. Several different outside sources were examined, and the selection process finally narrowed down to Control Data's Cybernet System running Nastran on a Cyber 175 computer.

The first complete computer runs for the 54/55 Nastran fuselage analysis were made successfully on the Cybernet System using Nastran Superelement analysis. In order to cut down on the transmission of data over the phone lines, a data base was set up for the fuselage model at the computer site. This data base permitted the running of several different load cases at reduced cost by transmitting only the JCL and loads data between the Learjet terminal and the Cyber 175.

EXPERIMENTAL RESULTS

Only a portion of the static test program had been completed, and the correlation of experimental data with analytical results had been under way a short time when this paper had to be submitted to the publisher. Consequently, just a few figures were available showing the comparison between the Nastran results and the static test data. A sample of two load conditions have been shown on the following pages. These two load conditions were the vertical fin side bending load case and the limit pressure loading. Plots for the side

bending load case have been shown in figures 10 and 11, and the correlation between the Nastran values and experimental data for the limit pressure load case have been shown in figure 12.

CONCLUDING REMARKS

Modeling of the Learjet 54/55 fuselage was based upon use of a multiple local coordinate system which proved to have significant advantages and worked very satisfactorily. The fuselage was analyzed using Superelement analysis where the structure was divided into seven superelements. This approach made the analysis of this size structure much more manageable and easier to perform.

REFERENCES

1. Boroughs, Robert R.: Development of The Learjet 28/29 Wing Using Nastran Analysis, pp. 11-32, EIGHTH NASTRAN USER'S COLLOQUIUM, NASA CP 2131, OCTOBER 1979.
2. THE NASTRAN USER'S MANUAL (Level 17.0), NASA SP-222 (04), Washington, D.C., December, 1977.



FIGURE 1 - LEARJET 54/55 AIRCRAFT

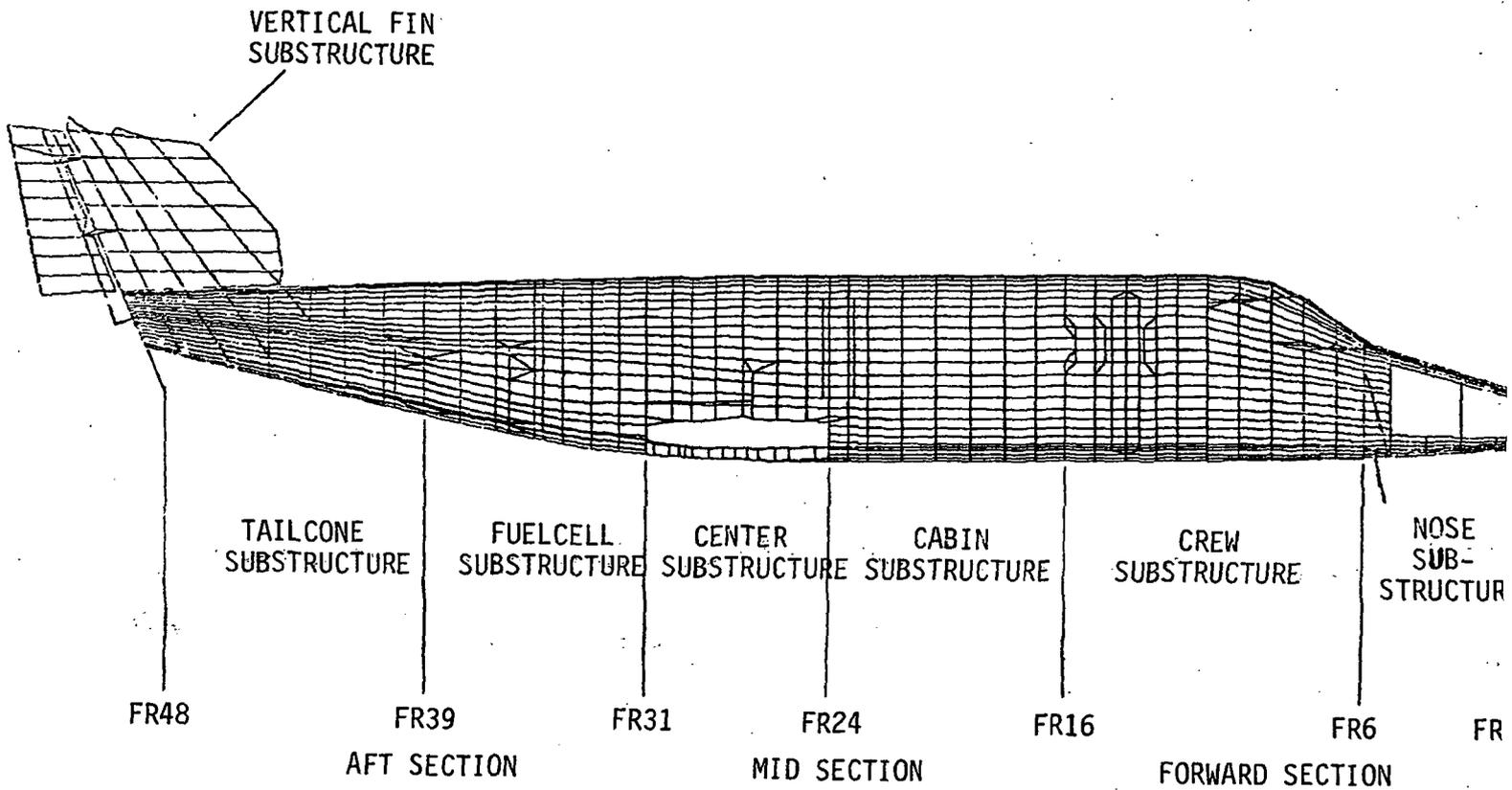


FIGURE 2 - NASTRAN 54/55 FUSELAGE MODEL

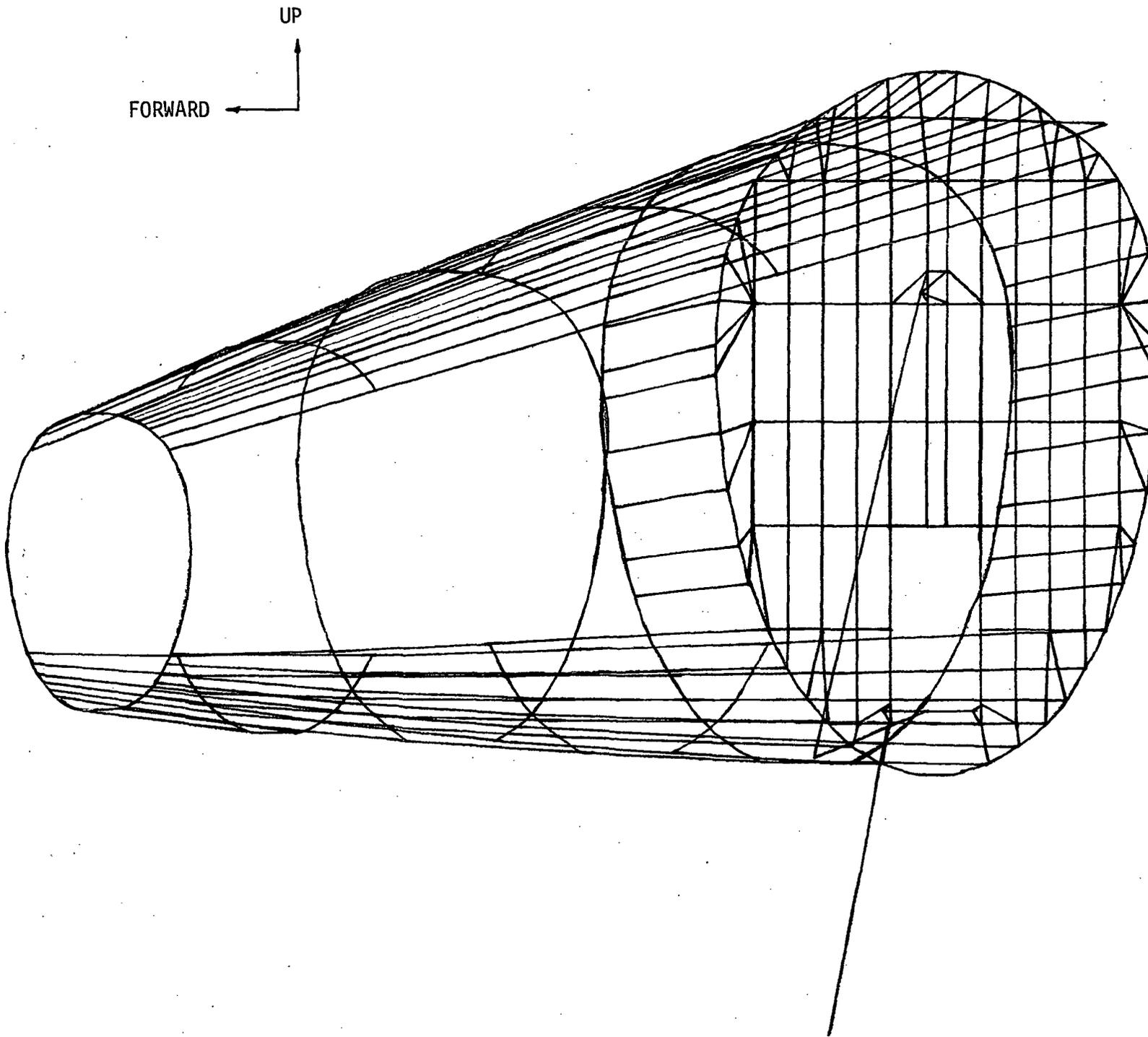


FIGURE 3 - NOSE SUBSTRUCTURE

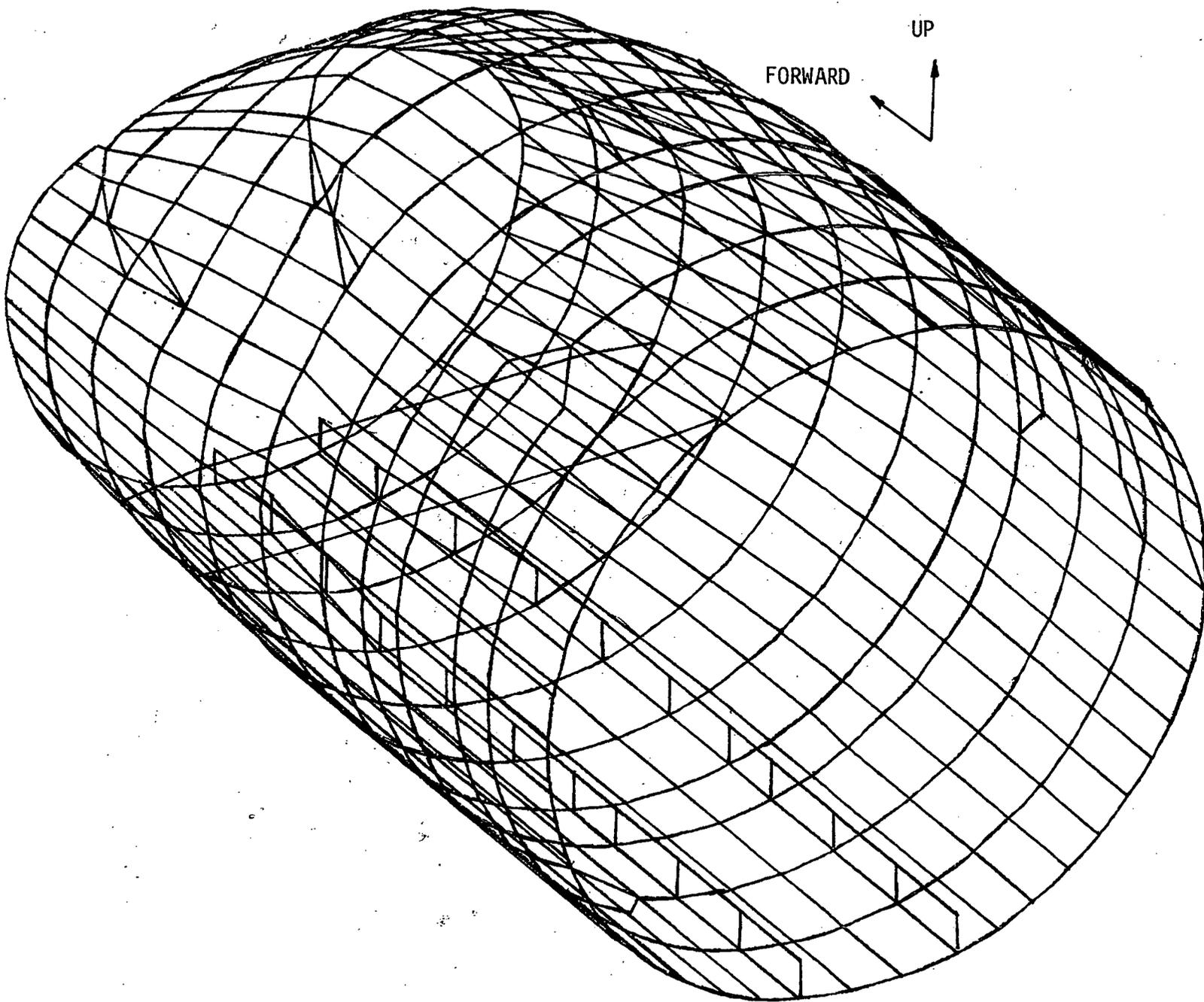


FIGURE 4 - CREW SUBSTRUCTURE
215

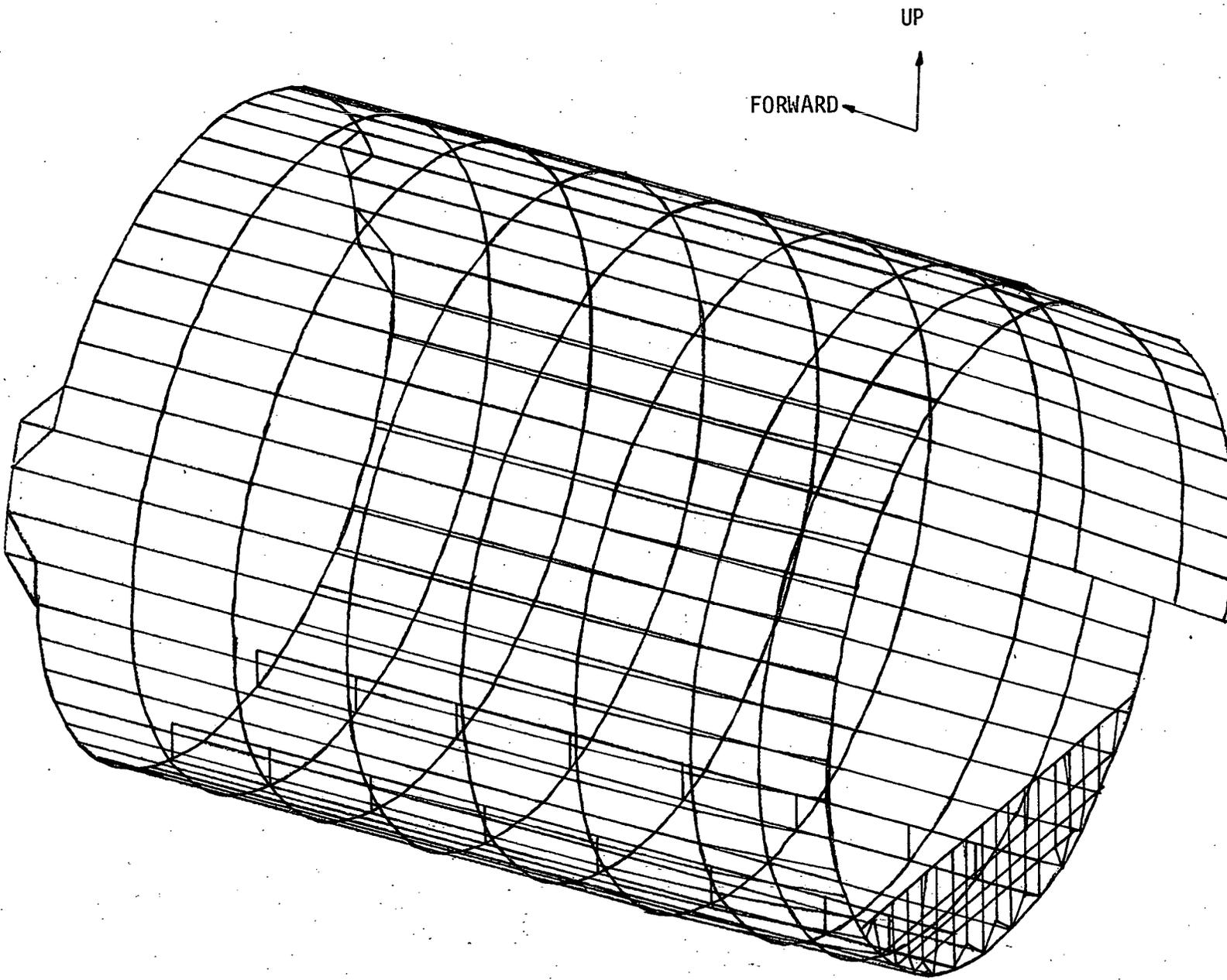


FIGURE 5 - CABIN SUBSTRUCTURE

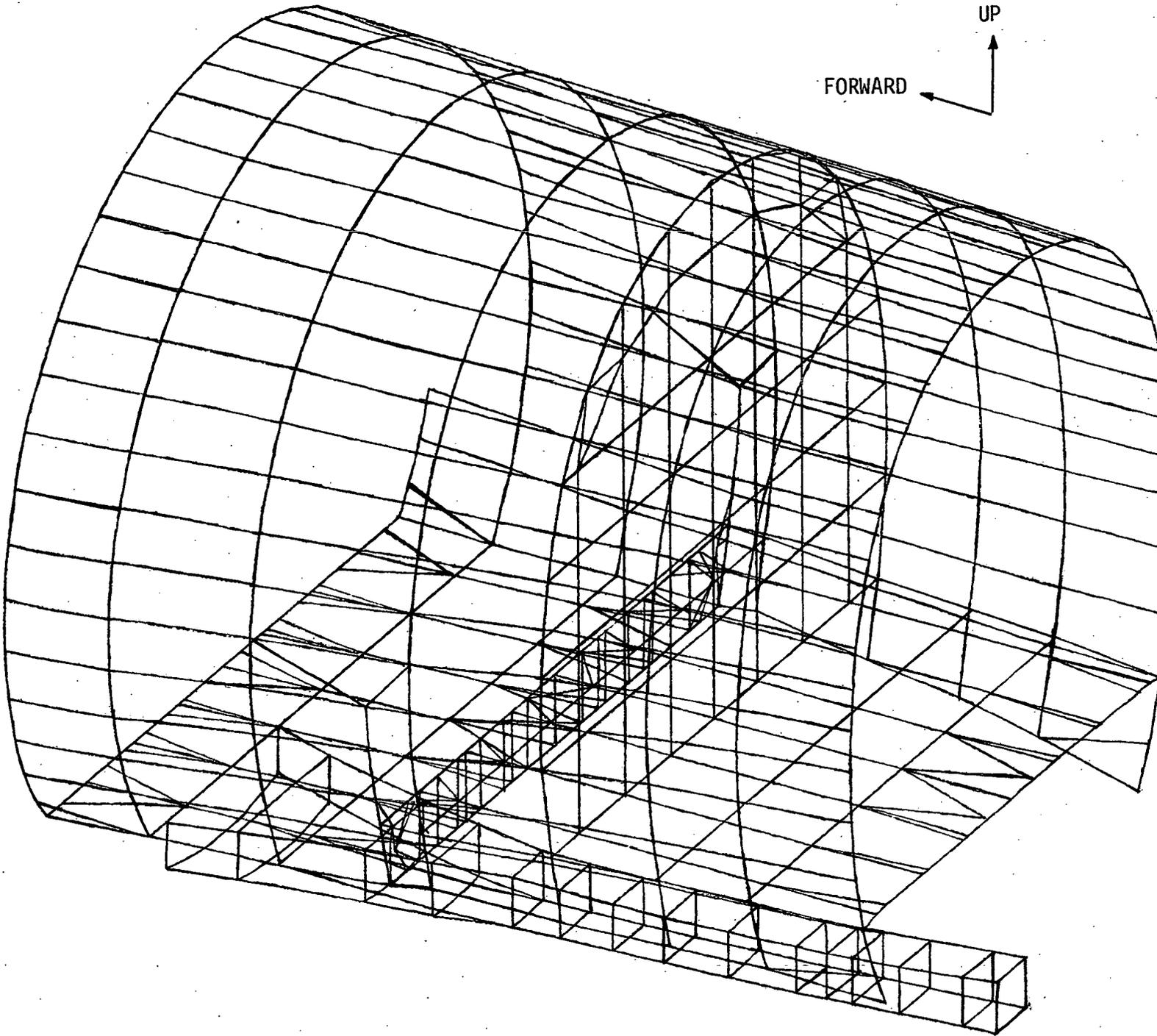


FIGURE 6 - CENTER SUBSTRUCTURE

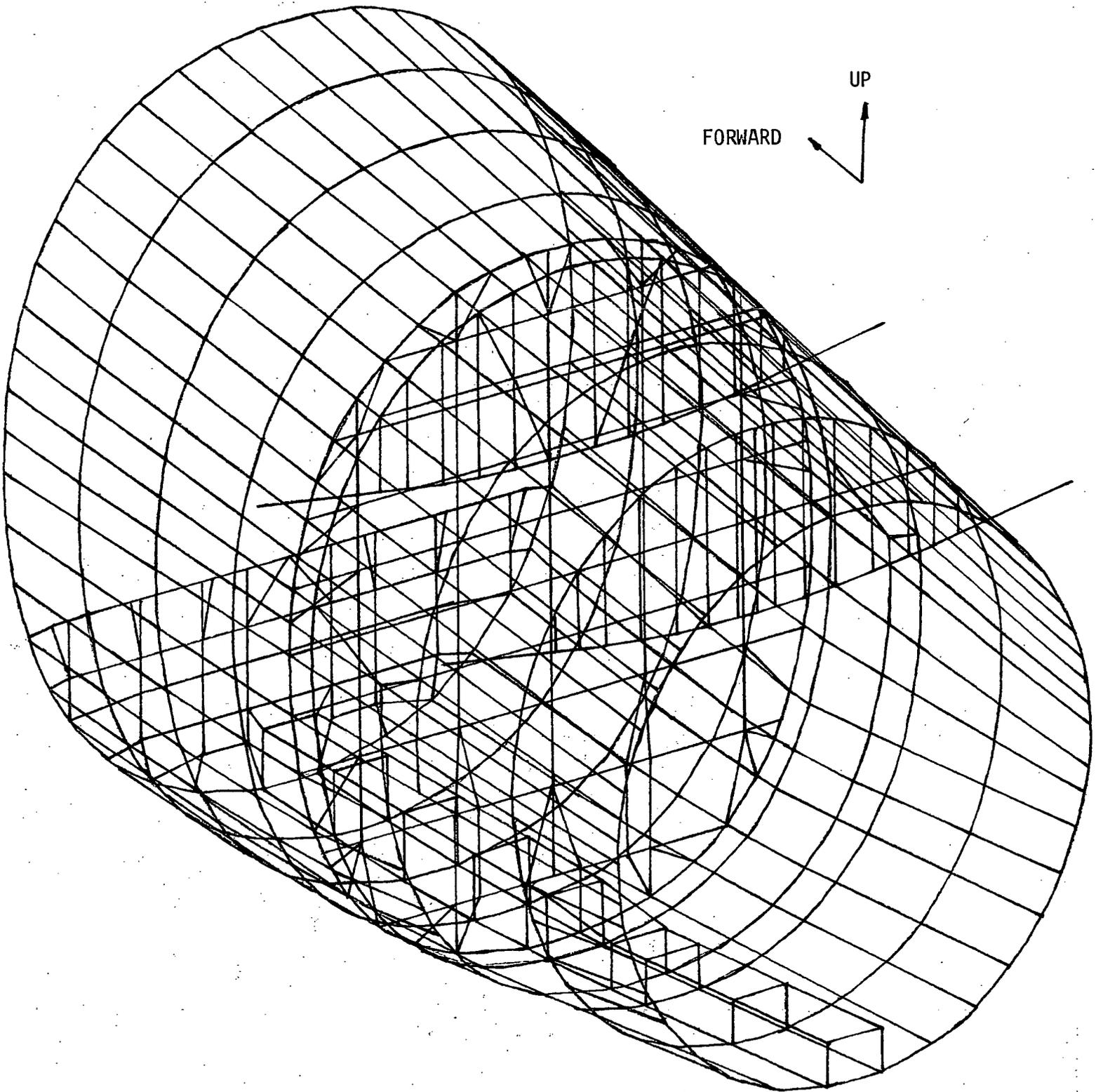


FIGURE 7 - FUELCELL SUBSTRUCTURE

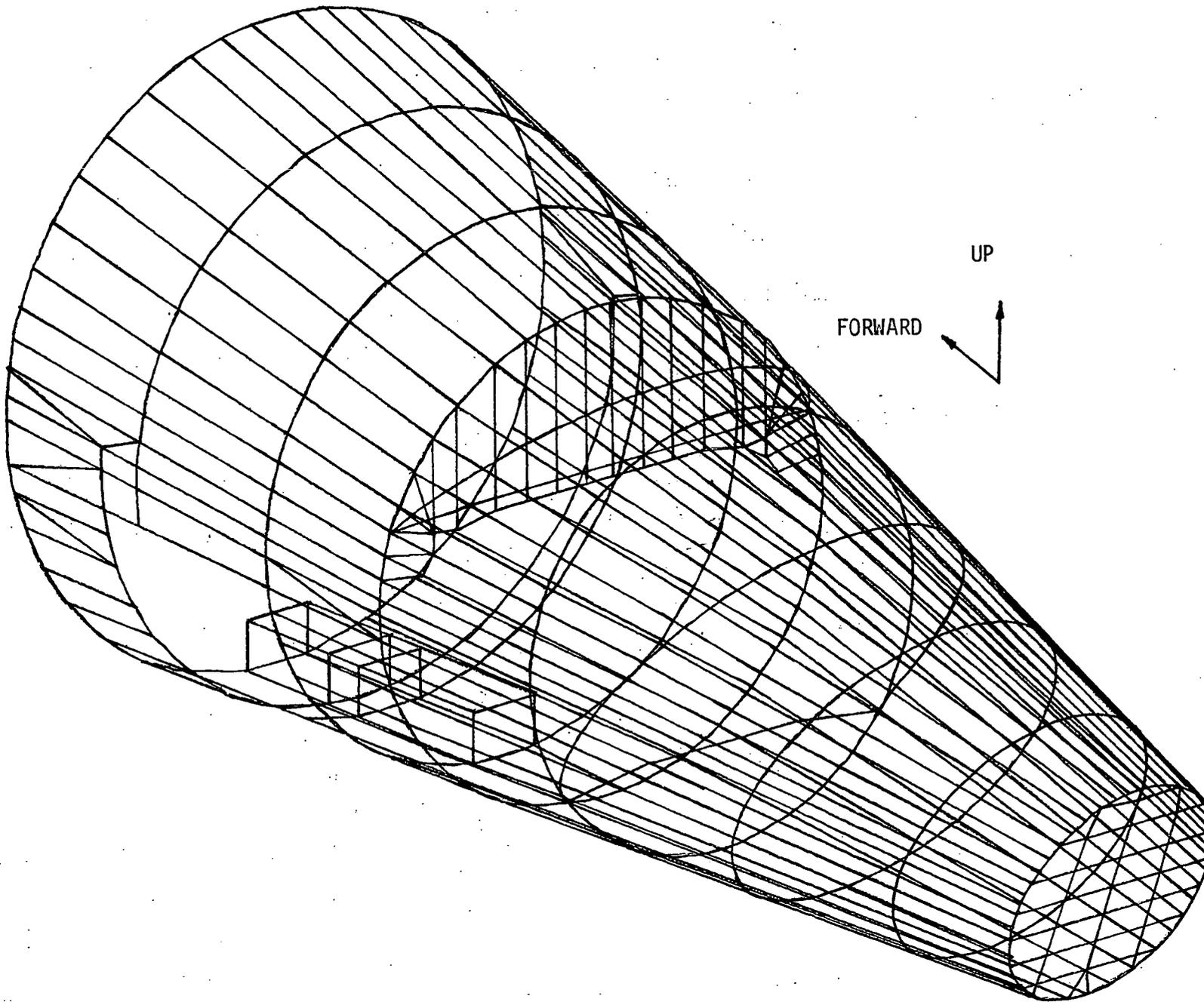


FIGURE 8 - TAILCONE SUBSTRUCTURE

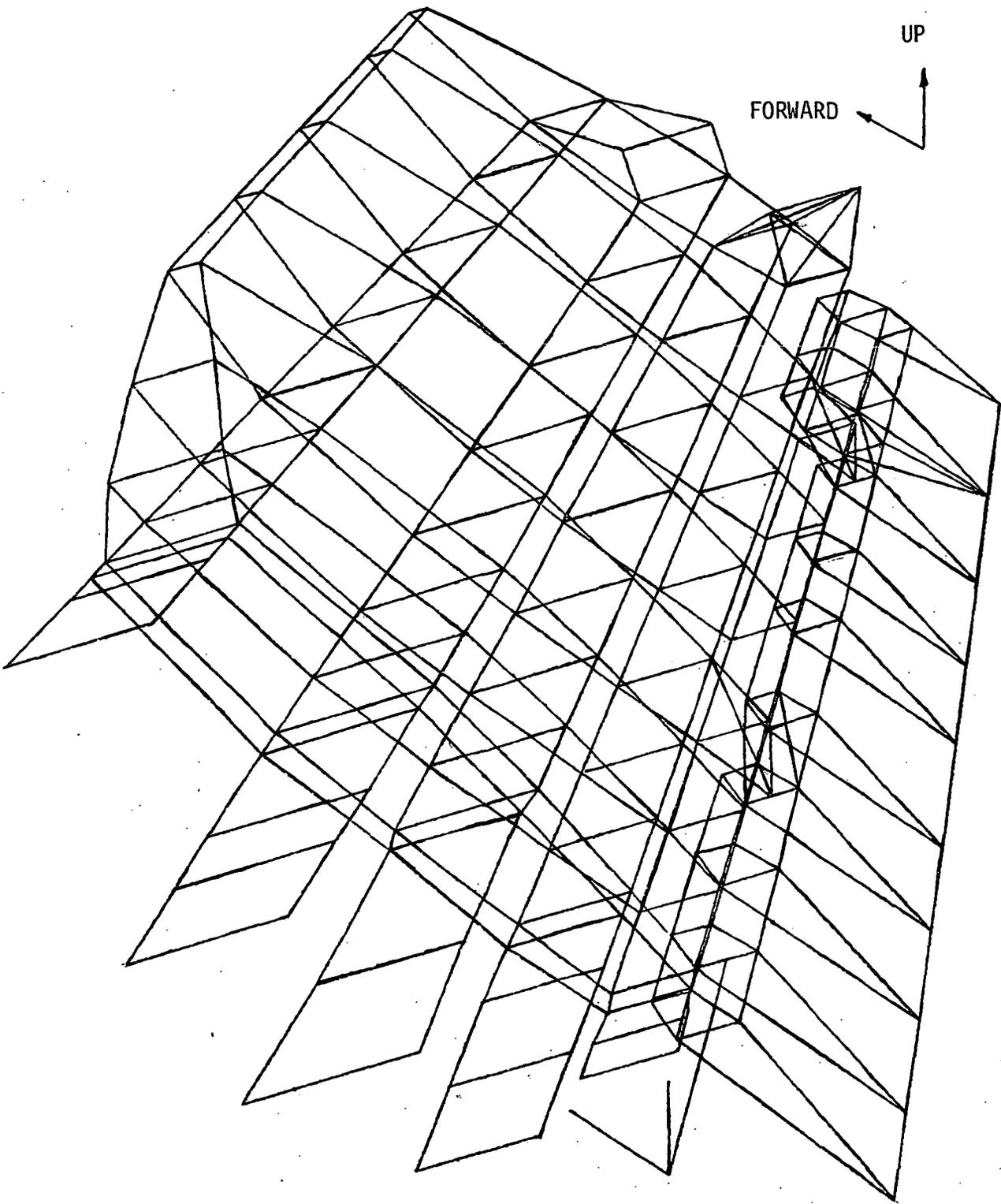


FIGURE 9 - VERTICAL FIN SUBSTRUCTURE

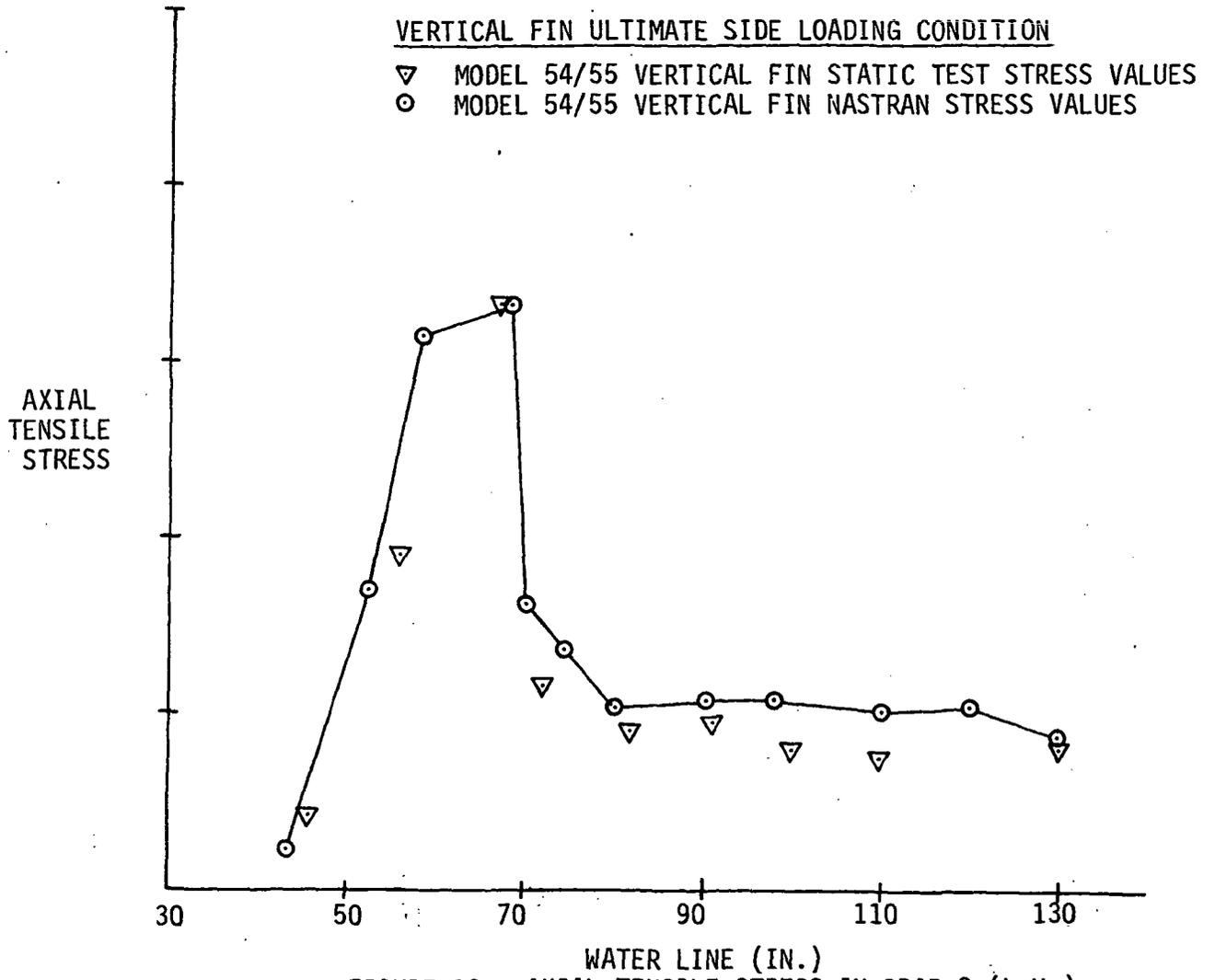


FIGURE 10 - AXIAL TENSILE STRESS IN SPAR 2 (L.H.)

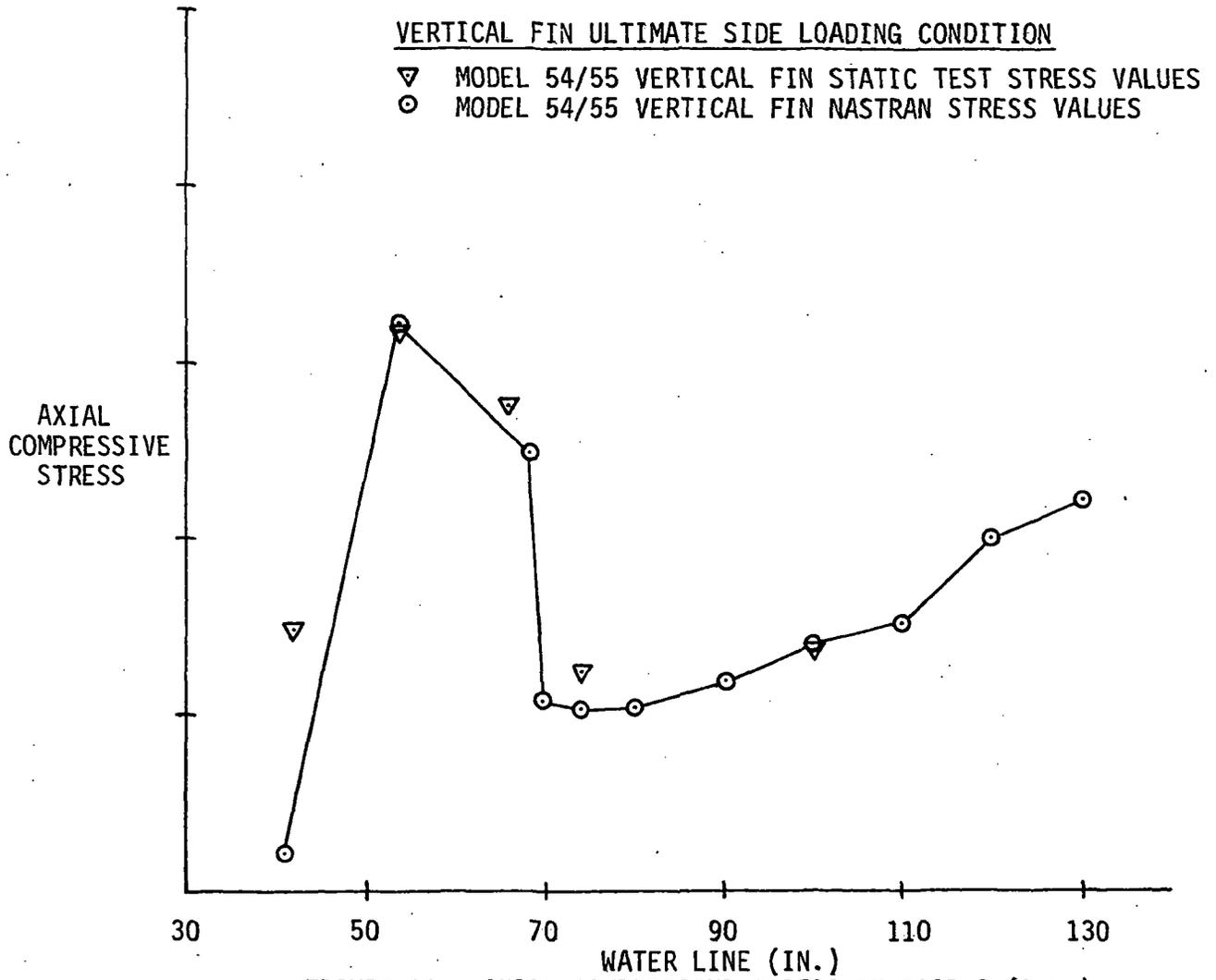


FIGURE 11 - AXIAL COMPRESSIVE STRESS IN SPAR 3 (R.H.)

$\frac{XXX}{YYY} = \frac{\text{NASTRAN STRESS (KSI)}}{\text{EXPERIMENTAL STRESS (KSI)}}$

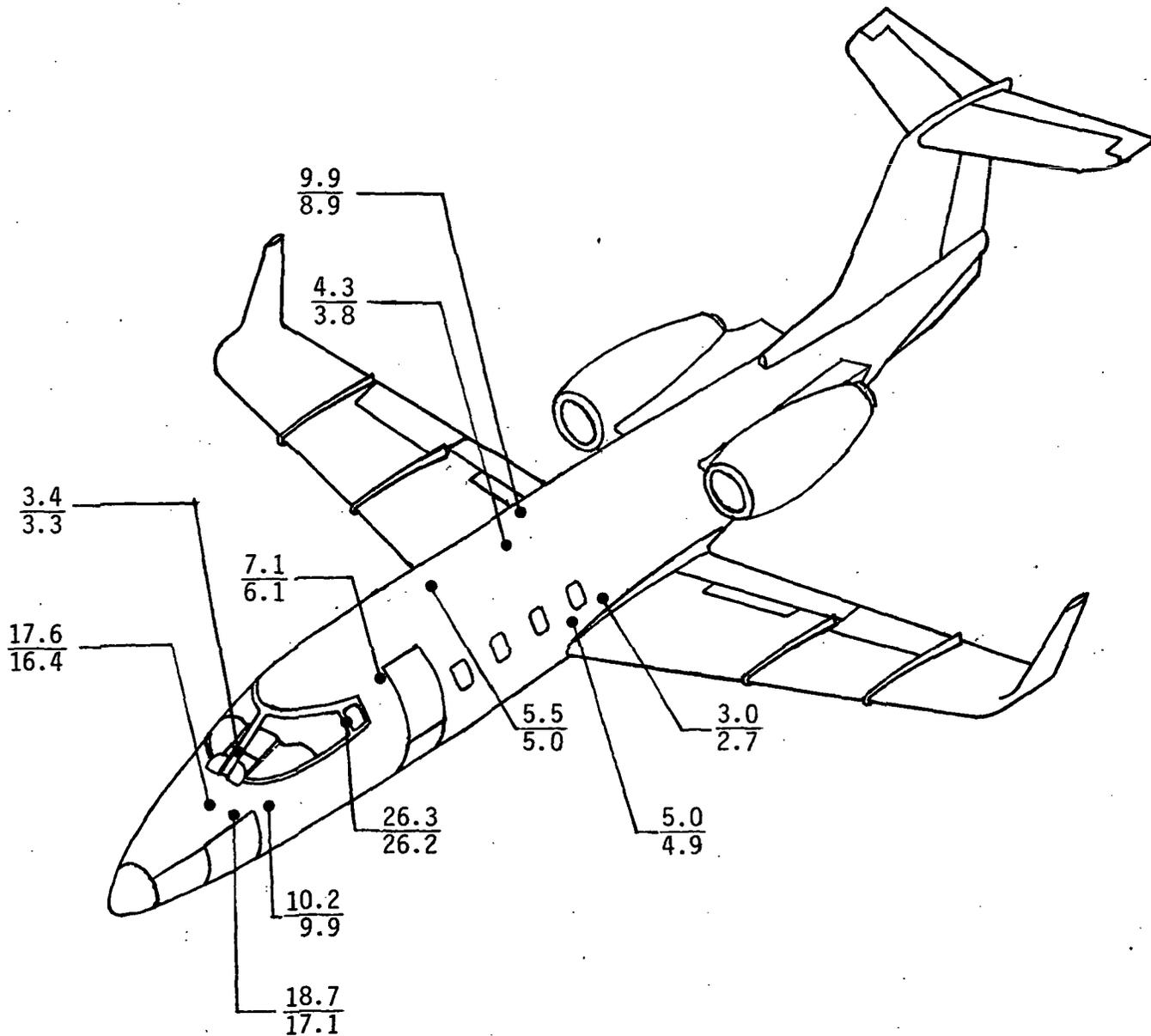


FIGURE 12 - LIMIT PRESSURE LOAD STATIC TEST VALUES AND NASTRAN VALUES