1982

NASA/ASEE SUMMER FACULTY RESEARCH FELLOWSHIP PROGRAM

MARSHALL SPACE FLIGHT CENTER
THE UNIVERSITY OF ALABAMA

FINITE ELEMENT ANALYSIS OF A DEPLOYABLE SPACE STRUCTURE

PREPARED BY:	DAVID V. HUTTON, Ph. D.
ACADEMIC RANK:	ASSISTANT PROFESSOR
UNIVERSITY AND DEPARTMENT:	WASHINGTON STATE UNIVERSITY MECHANICAL ENGINEERING
NASA/MSFC:	
(LABORATORY) (DIVISION) (BRANCH)	SYSTEMS DYNAMICS STRUCTURAL DYNAMICS SYSTEMS ANALYSIS
MSFC COUNTERPART:	R. S. RYAN
DATE:	AUGUST 6, 1982
CONTRACT NUMBER:	NGT-01-002-099 (UNIVERSITY OF ALABAMA)

FINITE ELEMENT ANALYSIS OF A DEPLOYABLE SPACE STRUCTURE

BY

David V. Hutton, Ph. D.
Assistant Professor of
Mechanical Engineering
Washington State University
Pullman, Washington 99164-2920

ABSTRACT

Development of a large-scale Space Station will require similarly large structural elements capable of assembly, fabrication, or deployment in space. Weight and volume constraints of the Space Shuttle orbiter payload bay make deployable structures with minimum on-orbit assembly requirements the favored alternative.

Current deployable structure concepts involve folding, three-dimensional trusses with automated deployment/retraction systems and having high deployed-to-stowed volume ratios. Such designs employ a large number of pin joints to allow the rotational motion required for deployability.

To assess the dynamic characteristics of a deployable space truss, a finite element model of the Scientific Applications Space Platform (SASP) truss has been formulated. The model incorporates all additional degrees of freedom associated with the pin-jointed members. Comparison of results with SPAR models of the truss show that the joints of the deployable truss significantly affect the vibrational modes of the structure only if the truss is relatively short.

LIST OF FIGURES

INTRODUCTION

The Scientific Applications Space Platform (SASP) Truss is a deployable three-dimensional structure designed to be a building-block element in large space structures. The basic design unit of the SASP truss is a folding cell composed of two bays. When fully deployed each bay has the overall dimensions of a 1.4 meter cube. Theoretically, any number of cells can be joined end to end to create a deployable structure of any length. Alternatively, several independent trusses could be joined to create a composite platform in a variety of configurations such as a "T" shape, for example.

The Structural Assembly Demonstration Experiment (SADE), tentatively assigned to STS 28, includes the objectives of demonstrating shuttle capacity to build a large space structure on-orbit and validating truss design including deployment, assembly, and connectors. As an integral part of three of the five options under consideration for SADE (Figure 1), detailed analysis and testing of the dynamic characteristics of the SASP truss are required. For testing deployment characteristics, the SASP ground test platform (Figure 2) has been fabricated and deployment testing is currently underway. In addition, a mathematical model of deployment was developed by Stoll [1].

As the relatively low natural frequencies of vibration expected of such large structures could affect shuttle control, analysis of the modal characteristics of the various SADE options is needed. Subsequent articles of this report describe such an analysis for the SASP truss for several configurations using two different mathematical models.

THE SASP STRUCTURE

When retracted, the SASP truss resembles a tightly folded accordion with the longitudinal (longeron), diagonal, and transverse members aligned as depicted in Figure 3a. Deployability of the truss is made possible by the telescoping design of the main diagonals and the free rotational motion of certain of the structural members at the joints. In the retraction positions, the length of the telescoping diagonal members is approximately 2.6 meters (104 inches). A deployment cable is strung through the diagonals in sequence and passes across pulleys located at the folding joints and is rigidly attached to the terminal bay of the truss. For depoyment, the cable is reeled in via a motor located at the fixed end of the truss. This action produces shortening and rotation of the telescoping diagonals as in Figure 3b. Simultaneously, the longerons rotate about pin connections (effectively) at each joint, and the transverse members execute pure translation in following the joint motion. In the fully deployed position, a locking mechanism on the telescoping diagonals is mechanically actuated, thus preventing further motion and locking each bay into its three dimensional truss configuration.

SADE OPTIONS

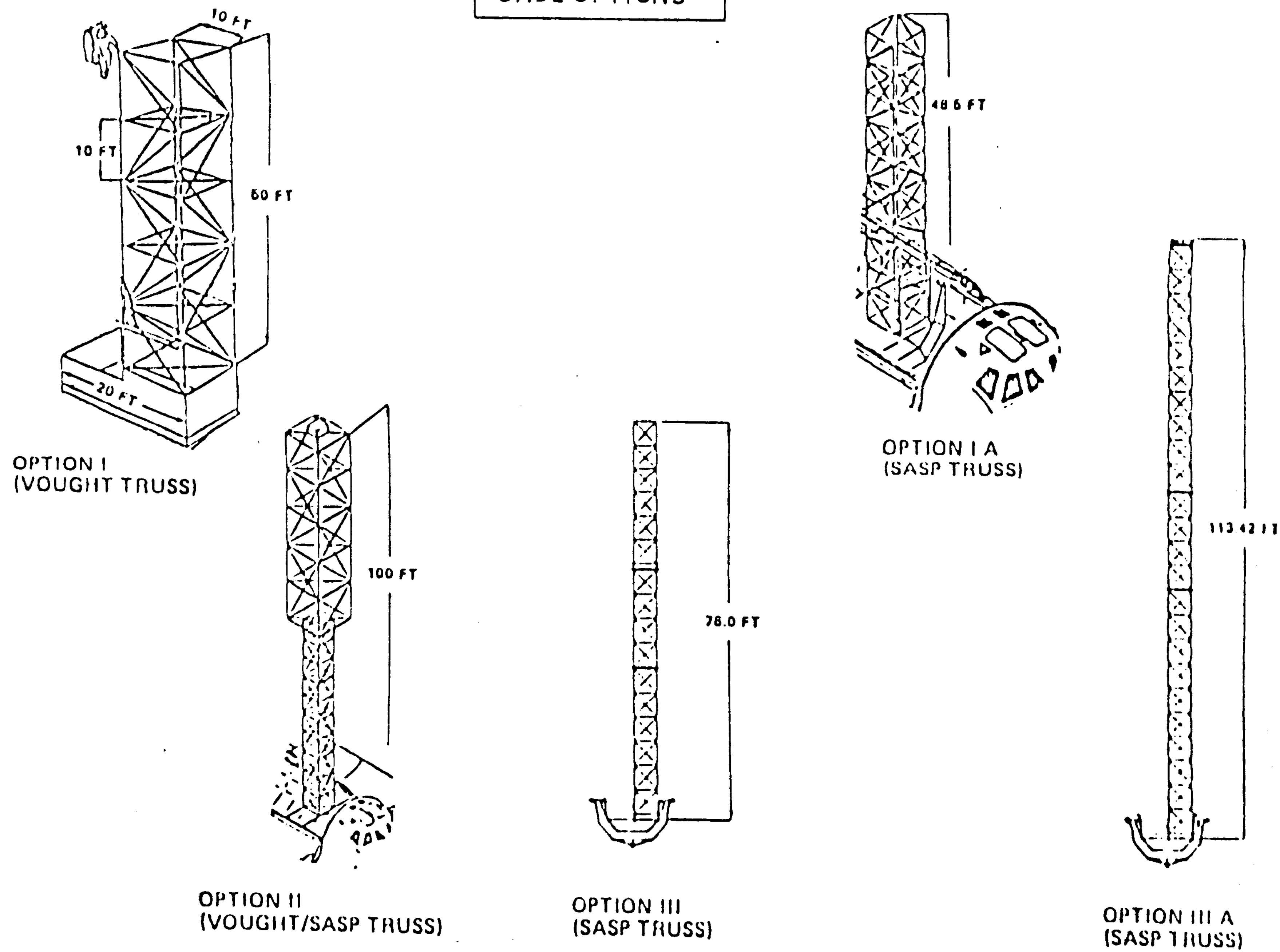
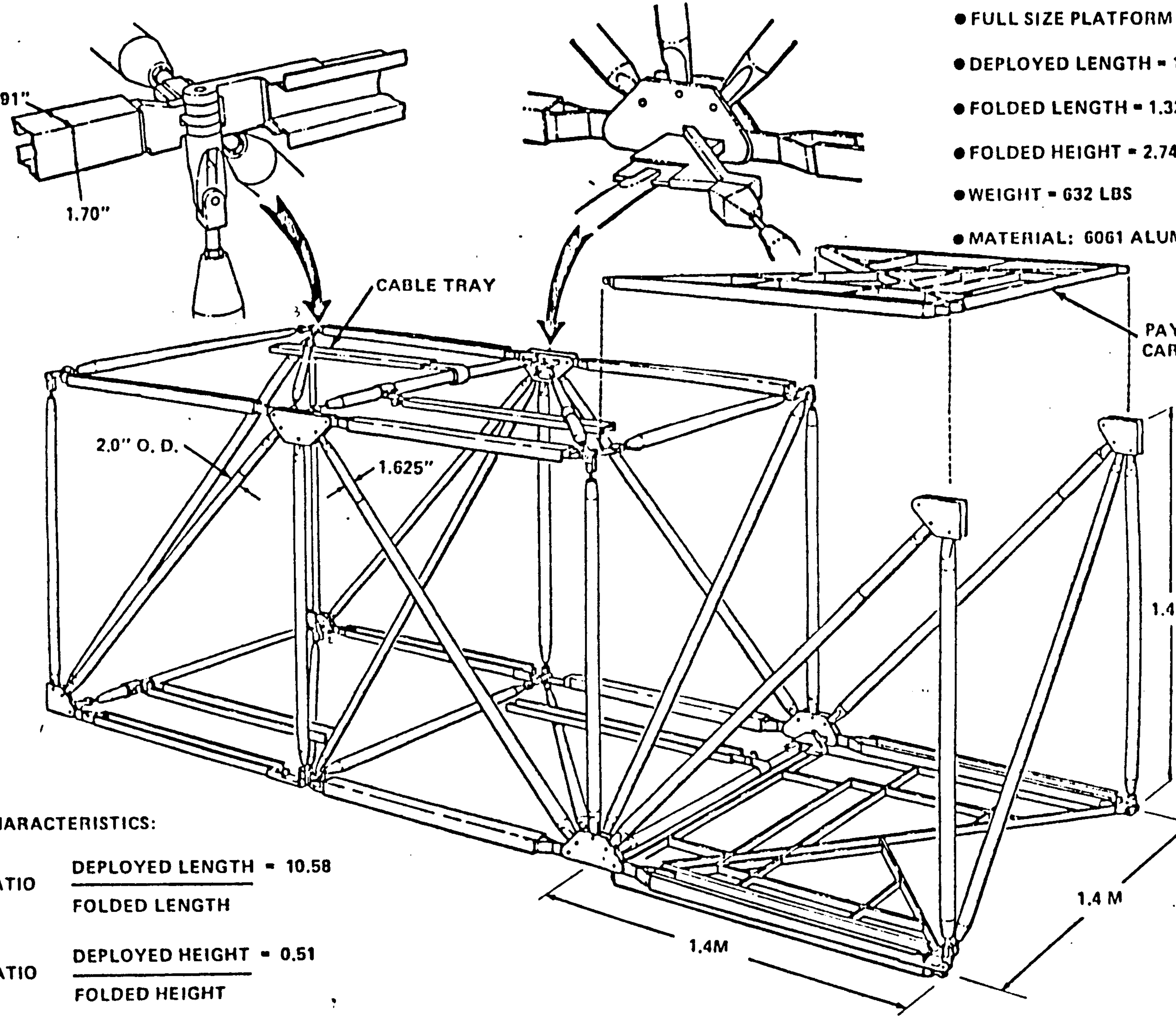


FIGURE 1

1876–81

SASP DEPLOYABLE STRUCTURE GROUND TEST PLATFORM



CHARACTERISTICS:

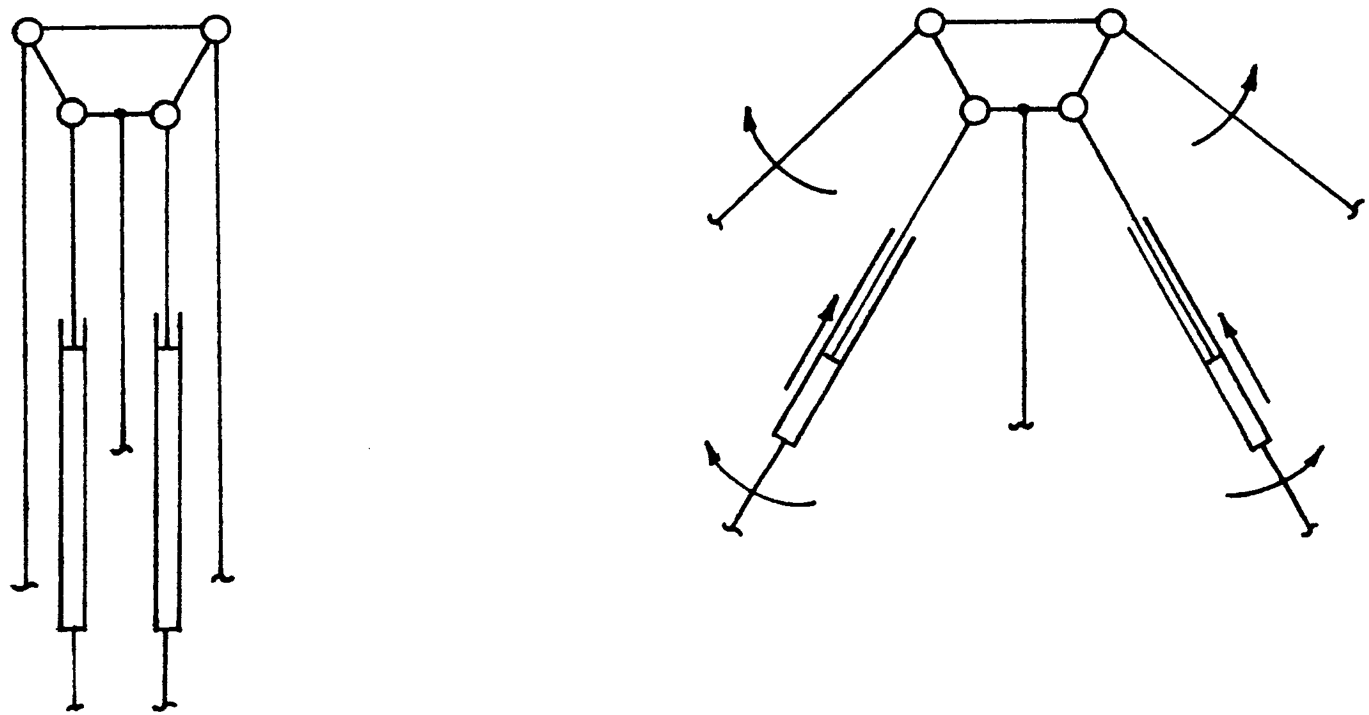
- FULL SIZE PLATFORM HAS 10 BAYS
- DEPLOYED LENGTH = 14.1 M (10 BAYS)
- FOLDED LENGTH = 1.32 M
- FOLDED HEIGHT = 2.74 M
- WEIGHT = 632 LBS
- MATERIAL: 6061 ALUMINUM

CHARACTERISTICS:

RATIO $\frac{\text{DEPLOYED LENGTH}}{\text{FOLDED LENGTH}} = 10.58$

RATIO $\frac{\text{DEPLOYED HEIGHT}}{\text{FOLDED HEIGHT}} = 0.51$

FIGURE 2



(a) RETRACTED

(b) DEPLOYING

FIGURE 3 SASP DEPLOYMENT

The SASP deployment mechanism is a single-fold concept. Telescoping and rotation of structural members occur in a single coordinate plane (actually in two parallel planes). In the two perpendicular planes, the truss configuration remains the same whether retracted or deployed. In these planes, frames designed for payload attachment can be substituted for truss members as suggested by Figure 2.

Excepting the longerons, which are open cross sections designed for nesting, the structural members are 2-inch outside diameter 6061 aluminum tubing having wall thickness of 0.072 inches. Joint fittings, payload carrier frames, and cable trays are of the same aluminum alloy. Each two-bay cell is composed of 26 members. Without payload carriers and accounting for end closure, the ten bay, five cell truss is composed of 135 structural members.

PRELIMINARY ANALYSIS

Initially, several models of the SASP truss were formulated and analyzed using the Structural Performance Analysis and Redesign (SPAR) system [2]. SPAR is a set of computer programs written for general structural analysis using the finite element method. The SPAR programs (referred to as processors) utilize sparse matrix techniques [3] which provide high computational speed and the capability of analyzing very large structures. Finite elements available include bar, beam, plate, shell, solid, and zero-length pure-stiffness elements. However, for analysis of a deployable structure such as the SASP truss, the SPAR system provides no reasonable means for directly modeling the many pin-jointed members. To illustrate this contention, the SPAR models of the SASP truss will be discussed briefly.

The most common approach in modeling planar truss structures is to treat the individual members as bar elements which have axial stiffness only. Effectively, this approach models all structural joints as pin connections since bending and torsion are not supported by the stiffness formulation for bar elements. Consequently, joint rotations are not allowed. For structures loaded symmetrically in the plane of the major load supporting framework, the bar formulation is generally adequate. In the case of the SASP truss, several shortcomings of bar-element modeling are apparent. In this analysis, we seek the natural frequencies (particularly the fundamental frequency) and mode shapes of forced vibrations of the truss in a cantilever configuration. It is not known a priori that the inertial loading associated with free vibration will occur in a plane or planes parallel to a major structural plane of the truss. In fact, considering the structural complexity of the truss, the likelihood of this occurrence is thought to be remote. The other objections to bar elements are rather obvious: all stiffness and inertia terms associated with bending, torsion and shear are ignored in the formulation.

Another possible approach is to model each structural member as a three-dimensional beam element. In this formulation, all stiffness and inertia terms arising from bending and shear in the principal planes are included as are those associated with torsion. For beam-element modeling, all elements connected to a common joint have identical displacements at the joint location. In the case of the SASP truss, the major joints connect eight structural members. Only two of the members are affixed so as to have displacement identical to those of the joint. Each of the other six has an additional, independent degree of freedom allowed by a pin connection at the joint. A similar situation exists at the smaller joints as well. Thus, straightforward beam-element modeling may result in overspecified stiffness and could give vibrational frequencies which are too large. It may be possible, using SPAR's zero-length, stiffness-only element capability, to produce joint models which more adequately simulate the actual joint construction of the deployable truss. This possibility was given a cursory examination but not pursued in-depth as it appeared to require an excessively large number of joints.

Although several objections to SPAR models of the SASP truss have been delineated above, computer runs using such models have been obtained. These results will be used for comparison with those of a SASP-unique model to be discussed in the next article.

SASP FINITE ELEMENT MODEL

In order to capture, as accurately as possible, the modal vibration characteristics of the SASP truss, a finite element model of the structure has been developed from scratch. Hereafter referred to as SFEM, this model formulation incorporates all additional degrees of freedom associated with the many pin joints in the structure.

In SFEM, each structural member of the SASP truss is treated as a three-dimensional beam element, as in Figure 4. With each element is associated an element reference frame xyz as shown, and the stiffness and inertia properties of the element are defined in terms of this frame. Generalized displacements u_1 through u_{12} represent the three displacements and three rotations of each end of the beam. The element reference frame is assumed to be oriented such that bending is referred to the principal planes of bending.

The stiffness and inertia properties of the beam element are obtained by considering the potential and kinetic energies of the element in conjunction with shape functions S(x) which describe the displacement of any point x along the beam in terms of the displacements at the end of the beam. For axial displacement, for example, we write

$$u(x,t) = S_1(x)u_1(t) + S_7(x)u_7(t) \tag{1}$$

where the shape functions (also known as assumed modes) must satisfy the boundary conditions

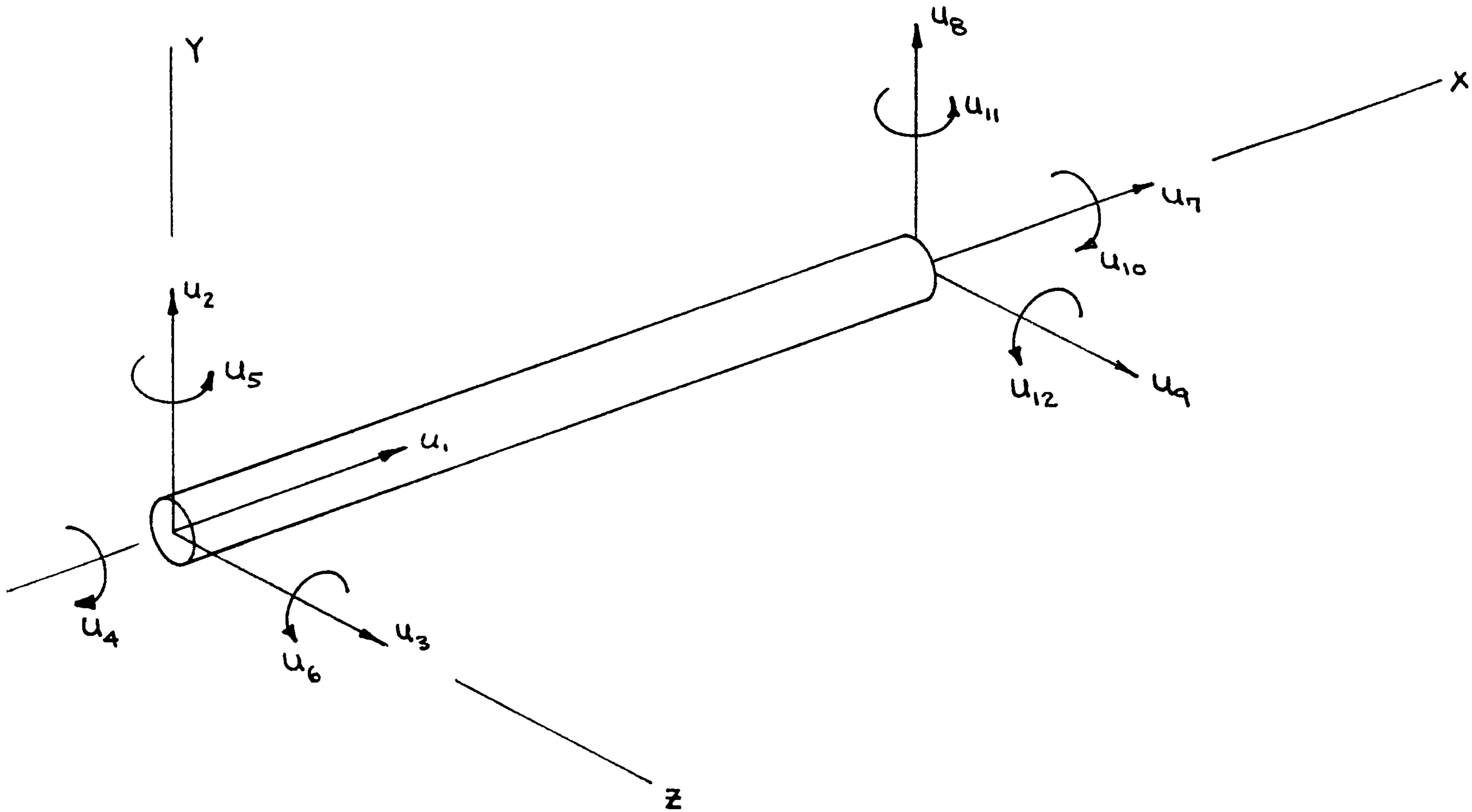


FIGURE 4 BEAM ELEMENT.

$$S_1(0) = S_7(L) = 1$$
$$S_1(L) = S_7(0) = 0 \tag{2}$$

in order to satisfy $u(0,t) = u_1(t)$ and $u(L,t) = u_7(t)$. The appropriate shape functions are $S_1(x) = 1 - x/L$ and $S_7(x) = x/L$ which are drawn from the solution for axial displacement under static loading. The elastic potential energy of the bar is

$$V = \frac{1}{2}\int_0^L EA\,(u')^2\,dx \tag{3}$$

Using $u(x,t)$ as in (1) the potential energy can be shown to be equivalent to

$$V = \frac{1}{2}\left(K_{11}u_1^2 + 2K_{17}u_1u_7 + K_{77}u_7^2\right) \tag{4}$$

where the stiffness coefficients K_{ij} are defined by

$$K_{ij} = K_{ji} = \int_0^L EA\,S_i'\,S_j'\,dx \tag{5}$$

Similar consideration of kinetic energy

$$T = \frac{1}{2}\int_0^L \rho A\,(\dot{u})^2\,dx \tag{6}$$

will lead to definition of the inertia coefficients as

$$m_{ij} = m_{ji} = \int_0^L \rho A\,S_i\,S_j\,dx \tag{7}$$

If we next consider transverse bending of the beam, we will obtain stiffness and inertia coefficients associated with u_2, u_6, u_8, u_{12} for bending in the xy plane and with u_3, u_5, u_9 and u_{11} for bending in the xz plane. Finally, considering torsional displacements will produce the coefficients associated with u_4 and u_{10}. The specific formulation used here is that of Craig [4] and is based on Bernoulli-Euler beam theory. The resulting element stiffness and mass matrices are as given in Figure 5. The mass matrix so developed is known as the consistent mass matrix since it is based on the same shape functions as the stiffness matrix.

Using the beam elements, the structure is defined by joining the ends of the beams at the appropriate joints and deriving system equations of motion which describe the joint displacements. As an intermediate step in this process, the element stiffness and mass matrices (described in the element reference frame) must be transformed to a common reference frame in which joint displacements will be measured. The latter is known as the Global Reference Frame. This is a straightforward procedure as it involves only a rotation of coordinates defined by a 3 by 3 matrix of direction cosines which will be denoted $[T_c]$. If the element displacements in the global system are represented by $\bar{u}_i$, $i = 1, 12$, the transformation is given by

$$[u] = [T][\bar{u}] \tag{8}$$

$$
\mathbf{k} = \begin{bmatrix}
\frac{EA}{L} & & & & & & -\frac{EA}{L} & & & & & \\
 & \frac{12EI_z}{L^3} & & & & \frac{6EI_z}{L^2} & & -\frac{12EI_z}{L^3} & & & & \frac{6EI_z}{L^2} \\
 & & \frac{12EI_y}{L^3} & & -\frac{6EI_y}{L^2} & & & & -\frac{12EI_y}{L^3} & & -\frac{6EI_y}{L^2} & \\
 & & & \frac{GJ}{L} & & & & & & -\frac{GJ}{L} & & \\
 & & -\frac{6EI_y}{L^2} & & \frac{4EI_y}{L} & & & & \frac{6EI_y}{L^2} & & \frac{2EI_y}{L} & \\
 & \frac{6EI_z}{L^2} & & & & \frac{4EI_z}{L} & & -\frac{6EI_z}{L^2} & & & & \frac{2EI_z}{L} \\
-\frac{EA}{L} & & & & & & \frac{EA}{L} & & & & & \\
 & -\frac{12EI_z}{L^3} & & & & -\frac{6EI_z}{L^2} & & \frac{12EI_z}{L^3} & & & & -\frac{6EI_z}{L^2} \\
 & & -\frac{12EI_y}{L^3} & & \frac{6EI_y}{L^2} & & & & \frac{12EI_y}{L^3} & & \frac{6EI_y}{L^2} & \\
 & & & -\frac{GJ}{L} & & & & & & \frac{GJ}{L} & & \\
 & & -\frac{6EI_y}{L^2} & & \frac{2EI_y}{L} & & & & \frac{6EI_y}{L^2} & & \frac{4EI_y}{L} & \\
 & \frac{6EI_z}{L^2} & & & & \frac{2EI_z}{L} & & -\frac{6EI_z}{L^2} & & & & \frac{4EI_z}{L}
\end{bmatrix}
$$

$$
\mathbf{m} = \frac{\rho A L}{420} \begin{bmatrix}
140 & & & & & & 70 & & & & & \\
 & 156 & & & & 22L & & 54 & & & & -13L \\
 & & 156 & & -22L & & & & 54 & & 13L & \\
 & & & \frac{140I_x}{A} & & & & & & \frac{70I_x}{A} & & \\
 & & -22L & & 4L^2 & & & & -13L & & -3L^2 & \\
 & 22L & & & & 4L^2 & & 13L & & & & -3L^2 \\
70 & & & & & & 140 & & & & & \\
 & 54 & & & & 13L & & 156 & & & & -22L \\
 & & 54 & & -13L & & & & 156 & & 22L & \\
 & & & \frac{70I_x}{A} & & & & & & \frac{140I_x}{A} & & \\
 & & 13L & & -3L^2 & & & & 22L & & 4L^2 & \\
 & -13L & & & & -3L^2 & & -22L & & & & 4L^2
\end{bmatrix}
$$

FIGURE 5 ELEMENT STIFFNESS AND MASS MATRICES.

where $[T]$ is the 12 by 12 transformation matrix given by

$$[T] = \begin{bmatrix} T_c & 0 & 0 & 0 \\ 0 & T_c & 0 & 0 \\ 0 & 0 & T_c & 0 \\ 0 & 0 & 0 & T_c \end{bmatrix} \tag{9}$$

The total elastic energy of a beam element can be written in matrix notation as

$$V = \frac{1}{2}[u]^T[K][u] \tag{10}$$

where $[K]$ is the 12 by 12 element stiffness matrix, $[u]$ is the row vector of element displacements and $[u]^T$ is the transpose of $[u]$. Formal substitution of eq. (8) into eq. (10) shows that the element stiffness matrix in the global reference system is given by

$$[\bar{K}] = [T]^T[K][T] \tag{11}$$

An identical procedure using kinetic energy gives the transformed mass matrix as

$$[\bar{m}] = [T]^T[m][T] \tag{12}$$

As a brief respite from the derivation, we note that the SASP truss is composed of members corresponding to five sets of structural and inertia properties. Thus five sets of stiffness and mass matrices are required. The truss includes eight different element orientations so that eight transformation matrices $[T_c]$ are used in the model.

Having transformed individual element matrices to a common frame of reference, displacement compatibility relations are applied at each joint in the structure to obtain the <u>system</u> stiffness and mass matrices. The procedure used here is known as <u>the direct stiffness method</u>. To obtain the displacement relations we let $\{U\}$ be the column vector of system displacement coordinates, and define, for each element, a <u>locator</u> (or label) matrix $[L_e]$ such that

$$\{\bar{u}\} = [L_e]\{U\} \tag{13}$$

where $\{\bar{u}\}$ is the vector of element displacements in the global frame. Equation (13) does nothing more than assign each of the twelve element displacements a particular system displacement. The locator matrix is composed strictly of zeroes and ones in twelve rows and N columns where N is the total number of system displacements (i.e. degrees of freedom).

Normally the vector of system displacements is composed of the six coordinate displacements (three translations, three rotations) at each joint. For the SFEM this is not the case as additional rotational degrees of freedom are allowed by the pin joints. To include these in the model, the method proposed by Winfrey [5] for analysis of elastic deformation in mechanisms is used. This will be discussed with reference to Figure 6 which is an X_G Y_G plane view of one of the major joints of the truss. In this plane, the joint displacements are U_1, U_2, and U_6 corresponding to translation along global axes X_G and Y_G, and rotation about Z_G, respectively. Of the structural members shown, only element 3 is attached such that its displacements are the same as those of the joints. Each of elements 1, 2, 4 and 5 are free to rotate about the pin connections at the joint although the translation displacements are the same as the joint. In this example joint then, there arise four additional degrees of freedom corresponding to U_7 through U_{10} as shown. Extending this procedure we find that a SASP truss with M bays has 24(M + 1) system coordinates associated with "standard" joint displacements and 40M + 4 system coordinates corresponding to the "extra" degrees of freedom.

Having defined the system coordinates as discussed above, final "assembly" of the system equations of motion is possible. The system potential energy can be expressed as

$$V = \frac{1}{2} \sum_{e=1}^{N} [\bar{u}]_e^T [\bar{k}]_e [\bar{u}]_e \tag{14}$$

where the summation is over the number of elements in the system. In terms of system coordinates, equation (13) is used to obtain

$$V = \frac{1}{2} \sum_{e=1}^{N} \{U\}^T [L_e]^T [\bar{k}]_e [L_e] \{U\} \tag{15}$$

which is equivalent to

$$V = \frac{1}{2} \{U\}^T \left(\sum_{e=1}^{N} [L_e]^T [\bar{k}_e] [L_e] \right) \{U\} \tag{16}$$

The summation term in eq. (16) is the <u>assembled system stiffness matrix</u> [K] such that

$$V = \frac{1}{2} \{U\}^T [K] \{U\} \tag{17}$$

Similarly it can be shown that the system consistent mass matrix is given by

$$[M] = \sum_{e=1}^{N} [L_e]^T [\bar{m}]_e [L_e] \tag{18}$$

in terms of which the kinetic energy is

$$T = \frac{1}{2} \{\dot{U}\}^T [M] \{\dot{U}\} \tag{19}$$

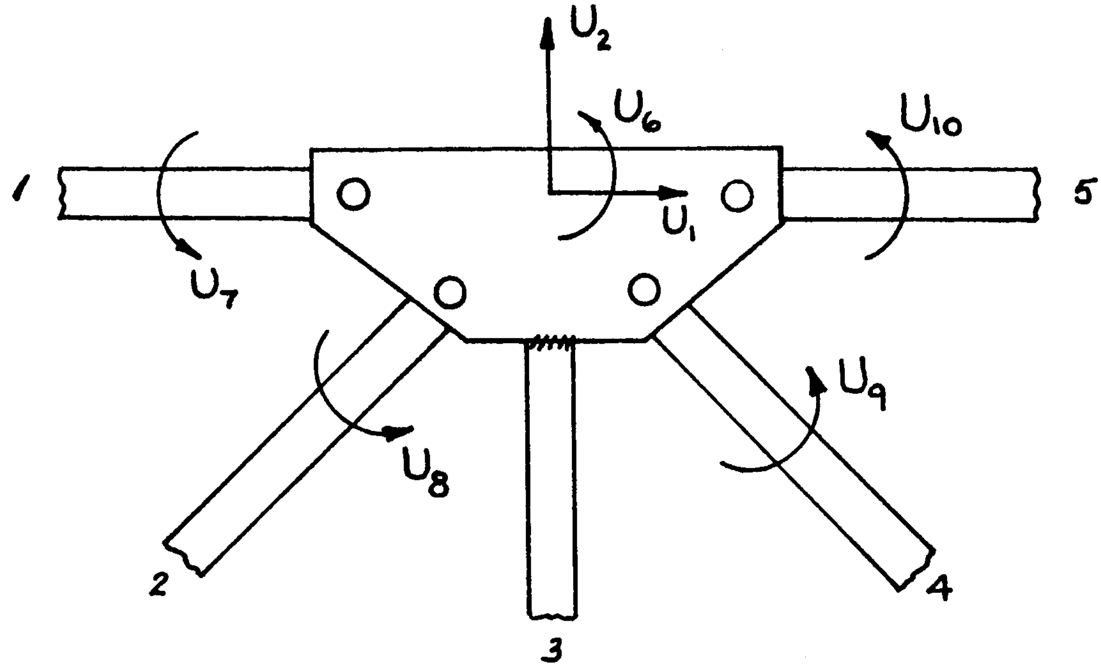


FIGURE 6 SASP JOINT DISPLACEMENTS.

By using eqs. (17) and (19) we obtain the system equations of motion in the matrix form

$$[M]\{\ddot{U}\} + [K]\{U\} = 0 \tag{20}$$

Equation (20) results from application of Hamilton's principle or by forming the Lagrangian and differentiating. Solution of the Eigenvalue problem represented by (20) will yield the natural frequencies and mode shapes of free vibrations of the truss structure.

DISCUSSION OF RESULTS

For simplicity, the SFEM was first applied to a single-cell, two-bay truss composed of 31 elements and constrained in a cantilever configuration. The resulting structural model has 92 active degrees of freedom after eliminating 24 displacements via the constraint conditions. Computer programs were written to sequentially calculate the element stiffness and mass matrices, transform the matrices to the global coordinate system, and assemble the system matrices. As pointed out by Craig, the matrix multiplications involving $[L_e]$ and $[L_e]^T$ do nothing more than locate terms from the element matrices into the proper row and column of the corresponding system matrix. To accomplish this and avoid a great number of matrix multiplications involving mostly zeroes, a locator vector was used for each element. This vector contains simply the row-column data relating element matrix to system matrix.

After assembly of the system matrices, the Eigenvalue problem was solved using the FORMA [6] matrix subroutine package. The results of the first SFEM run gave the fundamental frequency of the two-bay SASP truss as 27.9 Hz. For comparison, a SPAR model of the two-bay truss, using identical elements, resulted in a fundamental frequency of 50.6 Hz. This apparently significant difference in results was not accepted without considerable double checking of the model formulation and the computer program. By eliminating the "extra" degrees of freedom, the SFEM programs should produce the same results as SPAR. When this was done, the first six natural frequencies of the two models were found to differ by less than one percent. On this basis, the accuracy of SFEM was established.

Extending SFEM to the full size ten-bay truss required considerable rework of the computer programs. The ten-bay truss is composed of 135 structural members which leads to 444 degrees of freedom for the model. The huge matrices involved with a model of this magnitude are not amenable to routine manipulation as the computer time and storage requirements are astronomical. To assuage these difficulties, all matrix operations were converted to partition logic to eliminate storage and manipulation of vast number of zero terms. This conversion was readily accomplished since NASA's ZFORMA subroutine package could be used directly. These routines partition all matrices into 60 by 60 (maximum) submatrices for both storage and algebraic manipulation. Using subroutine ZMODE1 to solve the Eigenvalue problem, the two lowest natural frequencies of the ten-bay truss were determined in about eight minutes of actual computer time. The fundamental frequency so determined is 4.6 Hz while the second frequency is 5.4 Hz. Again for comparison, results from a ten-bay model using the SPAR system were obtained. To the surprise of the investigator, the frequencies from the SPAR model were substantially identical. Having learned years ago to believe and disbelieve simultaneously, the investigator again proceeded to check and double check. The partition-logic programs were reduced to the two-bay model to detect software errors which could have arisen in conversion. The results were identical to those previously obtained. On this basis, the ten-bay results were accepted as correct also.

Based on the contradictory comparisons of two-bay and ten-bay truss models using SFEM and SPAR, the programs for each model were run for four-, six-, and eight-bay truss configurations. The results of these runs, as shown by Figure 7, show that the fundamental frequencies given by the two models are convergent with respect to overall truss length. Since the major difference in SFEM and the SPAR model lies in the effective rotational stiffness of the joints, this phenomena can only be explained by surmising that the effects of joint rotations are reduced with increasing truss length not unlike the decreasing significance of transverse shear versus length of a beam.

Per the request of R. E. Jewell (MSFC/ED21), fundamental frequencies of the SASP structure were obtained for truss lengths up to 50 bays (230 feet) and for the ten-bay truss with concentrated mass at one end. In each of these additional cases the cantilever constraint was retained so the length cases apply to SADE Options III and IIIA while the tip mass cases are applicable to Option IA. Fundamental frequency versus truss length as given by SPAR models is shown in Figure 8a. The results of frequency as a function of tip mass are as in Figure 8b. The latter results were again in agreement from both SFEM and SPAR.

CONCLUSIONS AND RECOMMENDATIONS

A detailed finite element analysis of the SASP deployable truss has shown that the fundamental frequencies of vibration are not significantly affected by the extra degrees of freedom associated with the many pin-pointed members except for short overall truss lengths. This leads to the conclusion that simplified models utilizing the SPAR system can be used to adequately assess the dynamic characteristics of the structure for the configurations being considered for SADE.

As additional deployable truss designs evolve or composite platform configurations using SASP are considered, it is recommended that similar analyses be conducted to insure accuracy of simplified models. Detailed modeling similar to SFEM may be required if, in the latter case, a platform concept includes short truss sections to connect payloads for example.

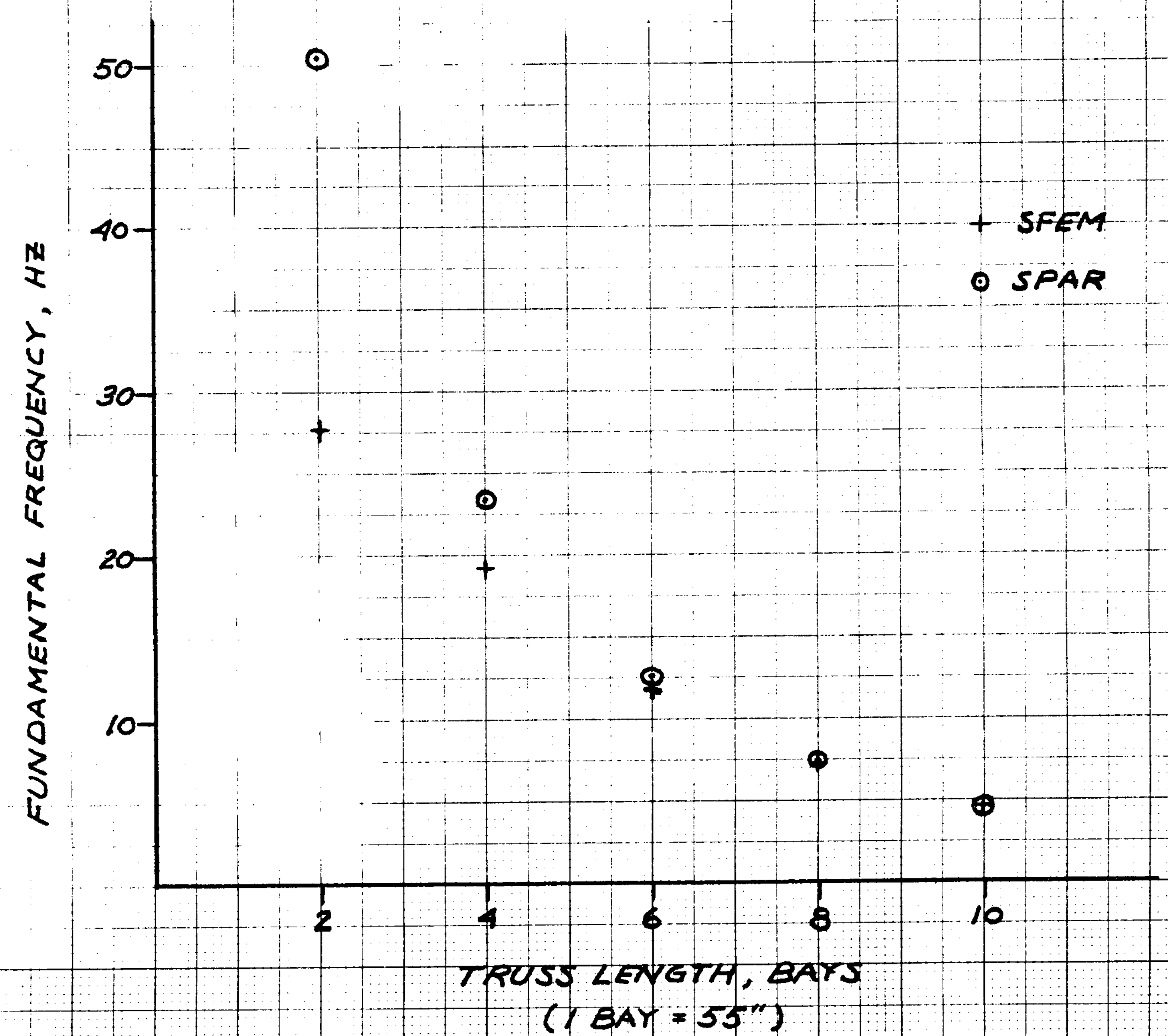


FIGURE 7 COMPARISON OF SFEM AND SPAR RESULTS.

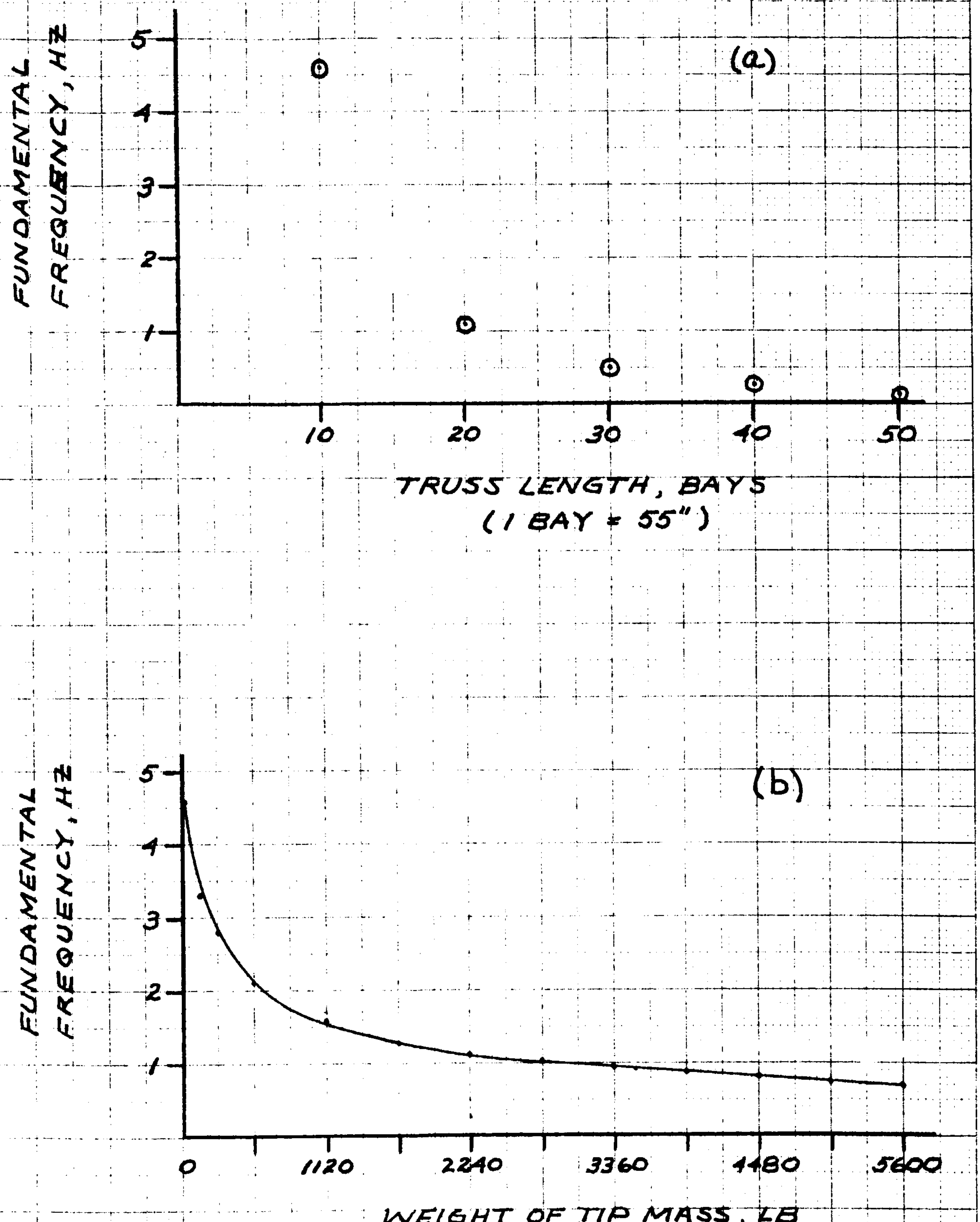


FIGURE 8 (a) FREQUENCY VERSUS TRUSS LENGTH (SPAR)
(b) TIP MASS EFFECTS ON FREQUENCY (SFEM AND SPAR)

REFERENCES

1. Stoll, H. W., "Computer Simulation for Dynamic Analysis of a Deployable Space Structure", Final Report, Contract No. NAS8-34506, September 1981.

2. SPAR Structural Analysis System Reference Manual, NASA CR158970-2, 1978.

3. Whetstone, W. D., "Computer Analysis of Large Linear Frames", Journal of the Structures Division, ASCE, November 1969.

4. Craig, Roy R., Jr., Structural Dynamics, John Wiley and Sons, New York, 1981.

5. Winfrey, Richard C., "The Finite Element Method as Applied to Mechanisms", Finite Element Applications in Vibration Problems, ASME, New York, 1977.

6. Expansion and Improvement of the FORMA System For Response and Load Analysis, MCR-76-217, Martin-Marietta, May 1976.