L·GARDE

SMART SPACE TECHNOLOGY

L • GARDE INC. CORPORATE PRESENTATION

Design Tool for Inflatable Space Structures

Arthur L Palisoc and Yuli Huang

DESIGN TOOL FOR INFLATABLE SPACE STRUCTURES

Arthur L. Palisoc and Yuli Huang*
L'Garde, Inc.
Tustin, California

Abstract

A finite element code was developed for the prediction of the on-orbit static and modal dynamic performance of inflatable antennas and inflatable solar concentrators. The computer program for the Finite Element Analysis of Inflatable Membranes (FAIM) is a geometric nonlinear finite element solver with nonlinear material capability. The code was interfaced to an RF antenna code, a ray-tracing code, and a commercially available graphical pre- and post-processor. The result was an integrated set of tools for the analysis and design of inflatable antennas and concentrators. This enables one to design an inflatable antenna and predict its surface accuracy and RF antenna parameters. For solar concentrators, the companion code, RAYTRK, calculates the solar collected intensity and concentration ratios. The code calculates the deformations and stresses due to the applied loads and outputs the stiffness and mass matrices for use by a companion code that calculates the natural frequencies and mode shapes.

Introduction

Inflatable structures have been shown to possess tremendous potential as antennas and reflective devices in aerospace applications. They are very lightweight and packaging volume reductions achievable are extremely impressive. Their low weight and small packaging volume permit even larger diameter inflatable antennas to be launched from only ATLAS-type boosters. NASA, JPL, and L'Garde, Inc. have teamed up and launched a 14-meter diameter inflatable antenna on May 20, 1996. The antenna was launched from the Space Shuttle Endeavour (STS-77) and was called the Inflatable Antenna Experiment (IAE).[1,2]

For these inflatable structures to be effective, their geometric shape must be accurately determined and controlled when subjected to pressure, thermal, and dynamic loads. Solar reflectors for example require the membrane to assume and maintain a parabolic shape of considerable accuracy. The most critical issues of concern are their long term survival, structural integrity, and their surface accuracy.

The success or failure of inflatable antennas and reflectors is directly tied to their ability to maintain a smooth parabolic shape. One of the major issues that must be solved is what is commonly referred to as the *inverse problem.* This may be stated in the form of a question: what must be the initial shape of an inflatable shell structure such that it attains a smooth parabolic surface after experiencing large structural deformations? Specifically, a parabolic surface that maintains a surface slope on the order of 0.001 radian must be achieved after an initially prescribed surface has undergone a large deformation due to the inflation pressure. Furthermore, the fact that these structures are constructed from specially-shaped flat gores, lead to structural models that defy closed-form solutions. L'Garde, Inc. fabricates these structures from precision-cut, flat, pre-shaped gores that are joined together by a doubler material at the seams. In the early 1980's L'Garde, Inc. developed the FLATE code to calculate the precise gore shapes.[3,4] It became obvious that an analytical tool able to simulate the deformation of initially flat gores upon inflation is needed. Do they form the desired parabolic shape?

It was in the mid-1980's that L'Garde, Inc. developed a finite element modeling program, FAIM (Finite Element Analysis of Inflatable Membranes), for the analysis of the deformations of inflatable membranes. From 1994 to 1996, through a Phase I and Phase II SBIR contracts from NASA-JPL, L'Garde further improved FAIM by (a) adding nonlinear material capability, (b) adding a modal dynamic capability, (c) interfacing with a commercially available graphical pre- and post-processor, (d) interfacing with an RF antenna code, and (e) interfacing with a ray-tracer.‘,’

The FAIM Code

FAIM Description

FAIM is a specialized, problem-dependent, material and geometrically nonlinear finite element program. It is iterative in nature; an initially prescribed, uninflated

*Arthur L. Palisoc, Member; Yuli Huang, Senior Member structural shape is input to be analyzed for stress and deformations. An 8-node isoparametric quadrilateral, a 6-node isoparametric triangle, and a 3-node tension-only cable are the basic elements in FAIM A conscious decision was made early on to discard the computational advantage of the 4-node quadrilateral and the 3-node triangle shell elements since these elements were deemed too crude to provide the requisite deformation information. Also, because the need to include the effect of the (thicker) seams was recognized early on, a third element was added to FAIM's library. It is a 3-node tension-only cable element. It is true that the thicker seams could be modeled by the quad and triangle elements but the seam width is much smaller than the gore width. Modeling the seam this way would result in a prohibitively large number of elements. We have modeled the seams as thicker quads and triangles for the much smaller F.E. model of an axisymmetric on-axis inflatable antenna by considering only half of the gore.' The results are in very good agreement with a similar model but using cable elements for the seams. The 3-node cable element allows one to simulate inexpensively, the local stiffening effect produced by the seams.

FAIM's basic elements are shown in Fig. 1. Figure 2 shows how the flat gores are joined together to form the initial unstressed antenna profile. A finite element model of the antenna in Fig. 2 is shown in Fig. 3. Because of azimuthal symmetry, half of a gore may be used in the modeling.

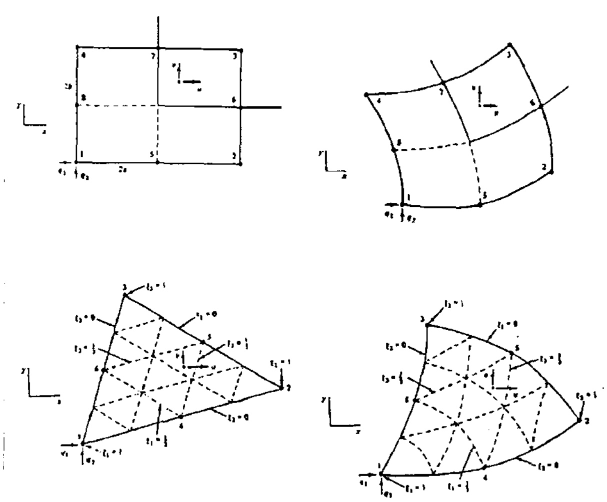


Fig. 1 (a) **8-node quadrilateral, (b) 6-node triangle**

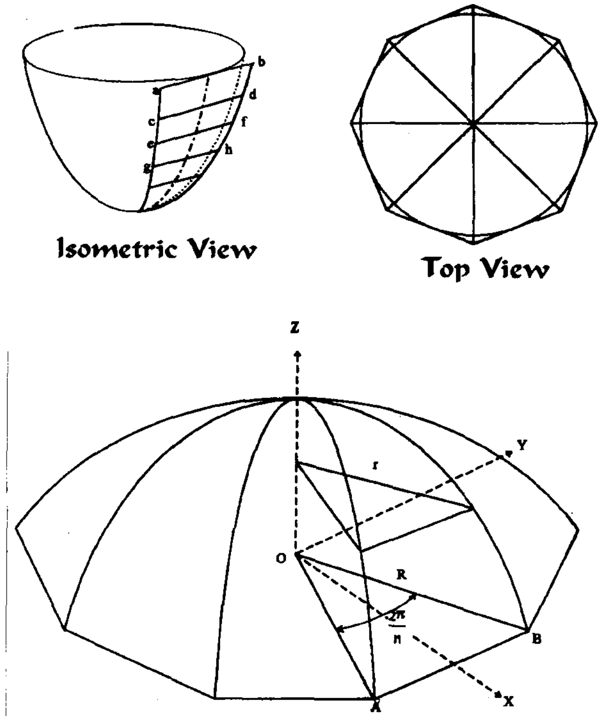


Fig. 2. Flat gores joined at the seams to form the initial unstressed antenna profile

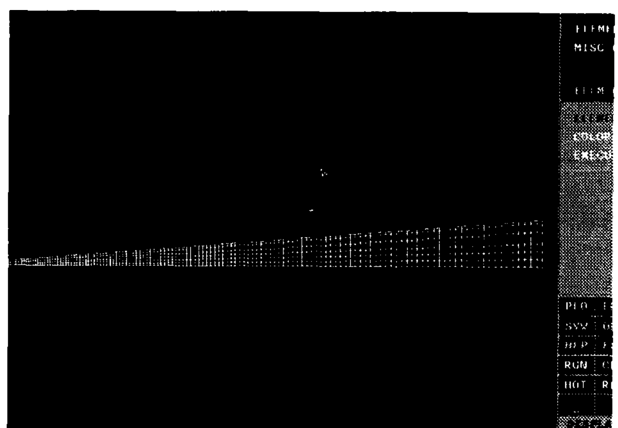


Fig. 3. Finite element model of an (axisymmetric) on-**axis parabolic antenna**

<u>FAIM Analysis Capabilities</u>

A number of enhancements have been made to FAIM over the years in order to increase its analysis capabilities as well as its computational efficiency. A "spring boundary condition" capability was coded in anticipation of the need to characterize the outer radius mounting surface of the parabolic shell to a flexible rather than an infinitely rigid support. Another feature available irFAIM is what is called a "skew" boundary condition. Referring to the axisymmetric finite element model depicted in Fig. 3 we note that this model required the x-y-z displacements at the outer radius to be zero as well as specifying that the circumferential displacements along the radial edges be zero due to symmetry conditions. The symmetry requirement on the circumferential displacement necessitated an implementation of the "skew" boundary condition of the finite element code. A skew boundary condition means a node is restrained to move along a plane not parallel to the global x-y, y-z, and x-z planes.

The significant computational efficiency enhancements to FAIM include a new solution strategy for solving the large number of simultaneous equations as well as a "restart capability". The iterative solution of the nonlinear equilibrium equations associated with large deformations of shell membranes leads to a set of simultaneous, algebraic equations. There are three significant contributions to the coefficient matrix of these equations: (1) elastic terms, (2) geometric terms, and (3) pressure terms. Whereas (1) and (2) lead to a symmetric coefficient matrix, the third leads to a skew symmetric. The combination of these three effects leads to an unsymmetric matrix. FAIM has two simultaneous equation algorithms for solving the equilibrium equation of a shell membrane model: (1) a symmetric, banded coefficient matrix and (2) an unsymmetric, banded, coefficient matrix solution. Both solution strategies are implemented in-core. This has an obvious advantage and disadvantage at the same time. Because it is an in-core solution, it is very fast. However, for the same reason, it can analyze only a problem "small enough" that will fit in memory.

The symmetric solution strategy takes some numerical shortcuts to achieve efficiency. Although the combination of the three effects mentioned above leads to an unsymmetric matrix, one may ignore the minor contributions of the pressure terms by assuming that the assembled matrix is symmetric. The computational shortcut significantly reduces the execution time of a typical analysis by a factor of 5 to 10 and requires only about half as much computer storage. Once a satisfactory equilibrium state has been achieved to a desired degree of accuracy with the symmetric albeit slightly inaccurate, coefficient matrix, one can repeatedly solve the simultaneous equations with the proper unsymmetric coefficient matrix. This is tantamount to using the symmetric solution strategy to obtain a good approximation to the deformed shell membrane followed by the second strategy in conjunction with the restart feature, to obtain the final deformed shell membrane. Experience to date has shown that in most cases, the skew-symmetric contribution from the pressure terms is negligible so that the symmetric and unsymmetric equations solvers give nearly identical results.

Because most materials of interest to inflatable structures technology exhibit nonlinear material behavior, it became imperative that the analytical tools used must also be able to model the same behavior. Furthermore, the natural frequency of inflatable systems must be known in order to determine whether or not it could interact undesirably with closed-loop control systems. These are the two most recent capabilities added to FAIM: (a) nonlinear material and (b) modal dynamic capability. The two significant material models in FAIM are (a) orthotropic material properties and (b) orthotropic plastic behavior.

The companion code to FAIM, SME (Shell Membrane Eigenvalue) uses the subspace iteration algorithm to calculate the eigenvalues (natural frequencies) and eigenvectors (mode shapes). Input is obtained from the initial FAIM run where the membrane undergoes loading. The input to SME consists of the stiffness and mass matrices.

The loadings available in FAIM are: (a) follower pressure, (b) concentrated nodal forces, (c) 3D arbitrary body force accelerations, and (d) nodal or element temperatures.

Bandwidth and Wavefront Solution Methods

The most efficient, in-core procedure for solving a set of simultaneous linear equations is the direct Gaussian elimination algorithm. The banded stiffness matrix of most finite element formulations, such as the one used in FAIM provides economy in both storage as well as equation solution (bandwidth). There are times, however, in which alternate methods must be considered when the banded stiffness matrix is too large to fit within core storage. The wavefront technique is one such method.[8] This technique has earned the reputation of being easy and inexpensive to use. This procedure, which uses the frontal method of equation assembly and reduction, is similar in scope to the Gaussian elimination algorithm. The interested reader may consult the references for a more detailed description of the method.

Bandwidth and Wavefront Minimization

When either the bandwidth or the wavefront method of solution is used, the user must construct the input geometry configuration in such a way that the bandwidth or the wavefront size is minimized in order to conserve disk storage, main memory usage, and reduce CPU execution time. In some representative membrane problems, the execution time has been reduced by 1 to 2 orders of magnitude by simply finding the optimum node numbering to minimize the bandwidth. One problem analyzed by the wavefront method was observed to be oscillating and not converging up to about 1000 iterations. When the input geometry was optimized for the wavefront solution, convergence was achieved in only 5 iterations! FAIM is equipped with a bandwidth and wavefront minimization routine. It uses the GPS algorithm (Gibbs, Poole, and Stockmeyer) for bandwidth minimization and the GK (Gibbs, King) algorithm for wavefront size minimization.[9]

A Pre- and Post-processor for FAIM

Individuals familiar with computer-aided finite element analysis will appreciate the fact that the data preparation (i.e., the modeling part) can take up to 60 to 80 percent of the entire finite element analysis task. And this is with the use of a computer graphical pre-processor. The input phase of a CAD-type data preparation for finite element analysis involves defining the geometry, determining the nodal coordinates, and forming the element connectivities. This is usually followed by defining the material and physical properties. The loadings and boundary conditions must likewise be specified. In a computer environment the whole process is usually done in a “windows-type” of environment with pull-down menus and a “mouse-driven” cursor for “clicking” on the desired parameter(s). The pre- and post-processor by Engineering Mechanics and Research Corporation (EMRC), called DISPLAY III was selected as the pre- and post-processor to use for FAIM.[10,11] The only reason for the choice is that we already have DISPLAY III in-house for use with the NISA finite element package by the same company.

While executing, FAIM displays the program status in real time on the computer monitor. Currently, this graphics capability is only for the PC, but it is not a difficult task to do the same for other CPU platforms. On the screen, the user sees (in graphic and numeric format) the convergence status at the current iteration, the maximum number of iterations allowed, and the convergence criterion, among others. Furthermore, the user may increase the number of iterations if the convergence is slow or tighten or relax the convergence criterion in real time.

FAIM Family of Programs

Figure 4 shows how a typical membrane analysis proceeds. Table 1 describes the functions of each of these utility programs. In the table, NECREF is an RF antenna code by Ohio State University.[12]

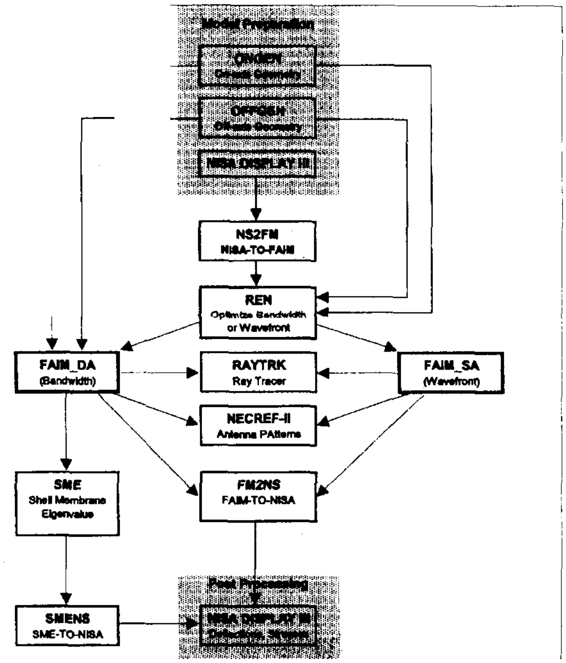


Fig. 4. FAIM Family of Programs - FAIM Analysis Procedure

Table 1. The FAIM Family of Utility Programs

FAIM_DA	FAIM code that uses the bandwidth technique.
FAIM_SA	FAIM code that uses wavefront strategy.
NISA-DISPLAY III	Pre- and post-processor for FEM by EMRC[10,11]
NS2FM	"NISA-to-FAIM". Code that translates DISPLAY III output to FAIM format.
FM2NS	Code that converts FAIM output into DISPLAY format for post-processing
ONGEN	Code that generates FAIM model of on-axis membrane antenna/reflector.
OFFGEN	Code that generates FAIM model of off-axis membrane antenna/reflector.
REN	Nodal/element renumbering program used to minimize bandwidth or wavefront. Input to REN must be in FAIM input format.
RAYTRK	A ray tracing code interface to FAIM.
NECREF	Antenna code by Ohio State University.
SME	“Shell Membrane Eigensolver". Calculates eigenvalues and eigenvectors of shell membrane problems.

FAIM Code Verification

Analytical Model for On-Axis Seamless Parabolic Antenna

An exact solution has been derived for the "inverse problem" described above.[3] This is depicted schematically in Fig. 5. Given the desired inflation pressure p_0, and final parabolic shape, what must be the initial uninflated shape so that on inflation to the pressure p_0, the desired parabolic shape is obtained?

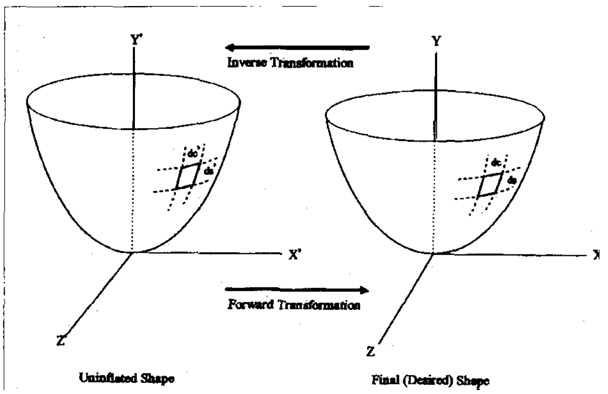


Fig. 5. Inverse problem solved by FLATE code.[3,4]

This analytical model was used to calculate the uninflated shape for a seamless paraboloid. The results are expressed in terms of the dimensionless parameters $(Z/2F)$ and $(r/2F)$ where F is the paraboloid focal length, z is the vertical displacement, and r is the radial distance from the vertex. The result for three different values of the Poisson's ratio is shown in Fig. 6.

Each of the three different uninflated membranes has been specified to FAIM as a seamless shell consisting of a given number of doubly-curved gores. The bottom perimeter of each of the uninflated membranes has been displaced prior to the run, to $r/2F=2.5$ to simulate the stretching of the membrane on the frame prior to inflation. It is evident from Fig. 6 that there is excellent agreement between the FAIM results and the analytical solution. In the figure, p is the inflation pressure, E is Young's Modulus and t is the membrane thickness.

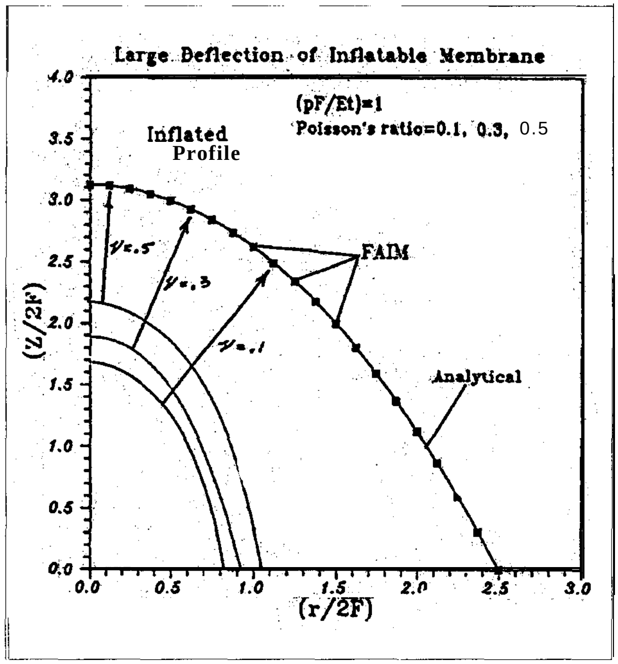


Fig. 6. Comparison of exact paraboloid with FAIM-calculated inflated shapes for $(pF/Et)=1$

Pressurized Circular Membrane

A classic series-solution for the stress and deformation of a pressurized circular membrane may be derived.[13] The finite element approximation to the large deformation of the circular membrane was obtained by discretizing a 15° wedge of the membrane with 42 quadrilateral and 3 triangle elements as shown in Fig. 7. Figure 8 shows the results for the circular membrane.

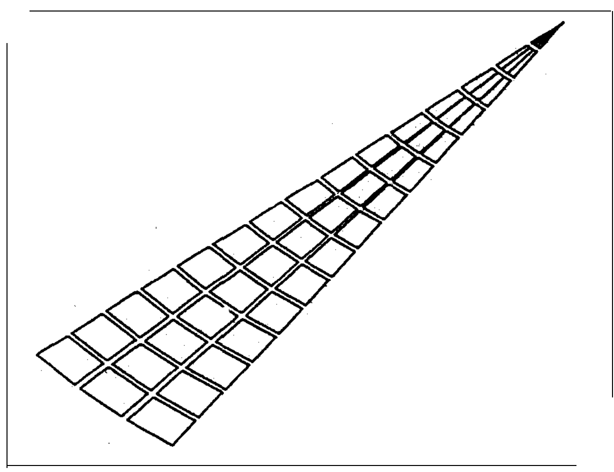
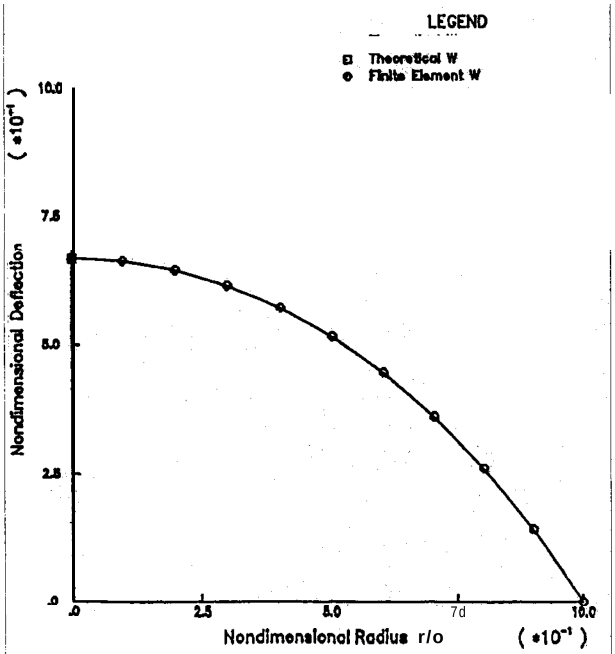
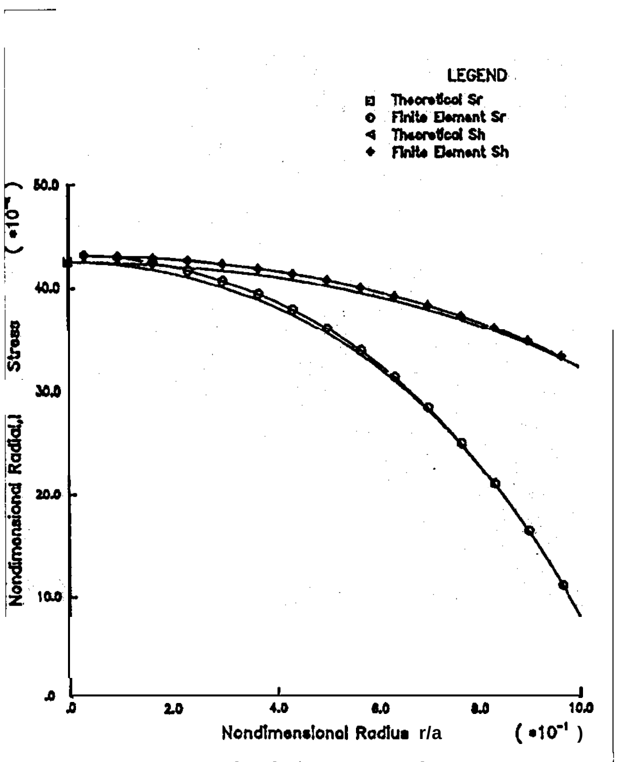


Fig. 7. FEM model of pressurized circular membrane



Pressurized Circular Membrane

Fig. 8a. Nondimensional deflections of circular membrane



Pressurized Circulor Membrane

Fig. 8b. Nondimensional radial and hoop stress

As a means of checking the modal analysis capability, the frequencies and mode shapes of a flat "drumhead" subjected to a uniform tension were determined for various finite element models. The theoretical results are determined from the zeros of the Bessel Function J_o, the membrane tension, and the mass density per unit area.[14]

The SME code computed the first 5 frequencies to less than 0.25 percent of the theoretical values whereas the FAIM and exact mode shapes were indistinguishable when plotted to a common scale. The first 5 radial mode shapes are shown in Fig. 9. The parameters of the circular membrane are given in Table 2.

Table 2. Parameters of Circular Membrane

Radius, a	100 inches
Tension	115 lb/inch
Area1 density	7.5 x 10-4 lb-sec^2/in^3

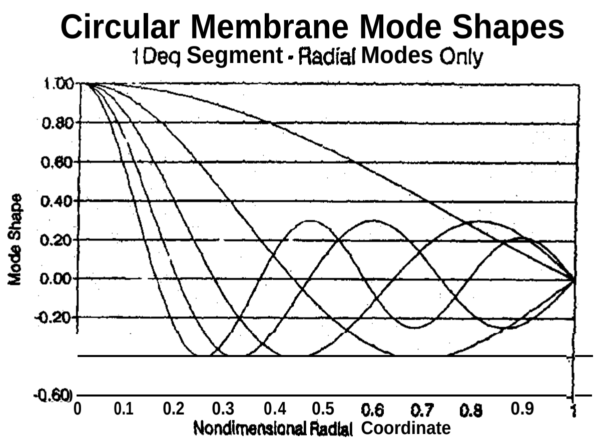


Fig. 9. First five radial modes of circular membrane

<u>The Rectangular Membrane</u>

The parameters of the rectangular membrane studied are listed in Table 3. Table 4 compares selected mode shapes calculated by FAIM-SME against theory.

Table 3. Parameters of Rectangular Membrane

Length, a	24 inches
Width. b	12 inches
Tension	1 lb/inch
Density	1.29 x 10^{-4} lb-sec^2/in^4

Table 4. Comparison to Theory

Mode No.	m	n	Theory	FAIM	% Diff
1	1	1	258.84 Hz	258.75 Hz	0.004%
3	2	1	328.13 Hz	327.30 Hz	0.25%
6	3	1	418.28 Hz	417.22 Hz	0.25%
15	3	3	778.22 Hz	778.77 Hz	0.07%
20	8	1	956.64 Hz	955.92 Hz	0.08%

FAIM Interface to NECREF and RAYTRK

Interface to NECREF

We have coded an interface computer program between FAIM and an RF antenna code. Specifically, we used the Ohio State University's NECREF antenna code.[12] The current version of the OSU Reflector Antenna code has the capability of analyzing reflector antennas with certain types of surface distortion from an ideal parabolic surface. However, the code does not have the capability of accepting an actual antenna surface profile which is obtained either from measurement of the surface or from a numerically generated surface such as that generated by FAIM. We have modified NECREF so that one can now input the nodal coordinates of the surface as it is output from FAIM. Figure 10 shows the interpolated surface for a 3m antenna formed from 36 *flat* gores. The FAIM-calculated profile has a surface accuracy of 0.3 mm rms. The NECREF-calculated H-Plane patterns for 8 and 25 GHz are shown in Figs. 11 and 12.

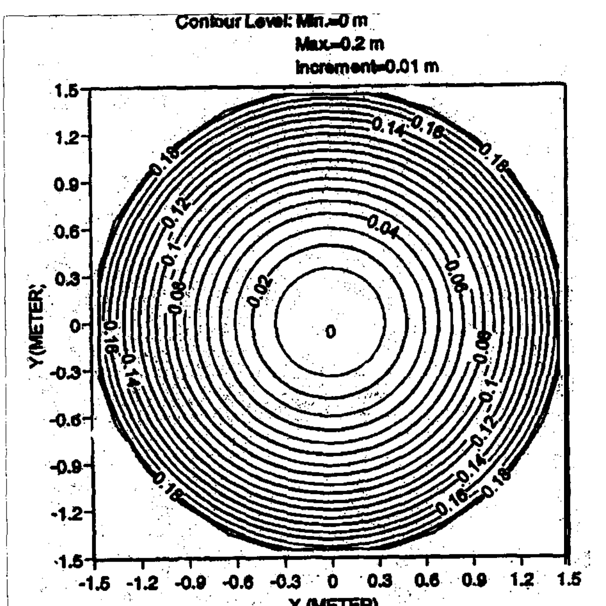


Fig. 10. Interpolated surface for 3-meter antenna

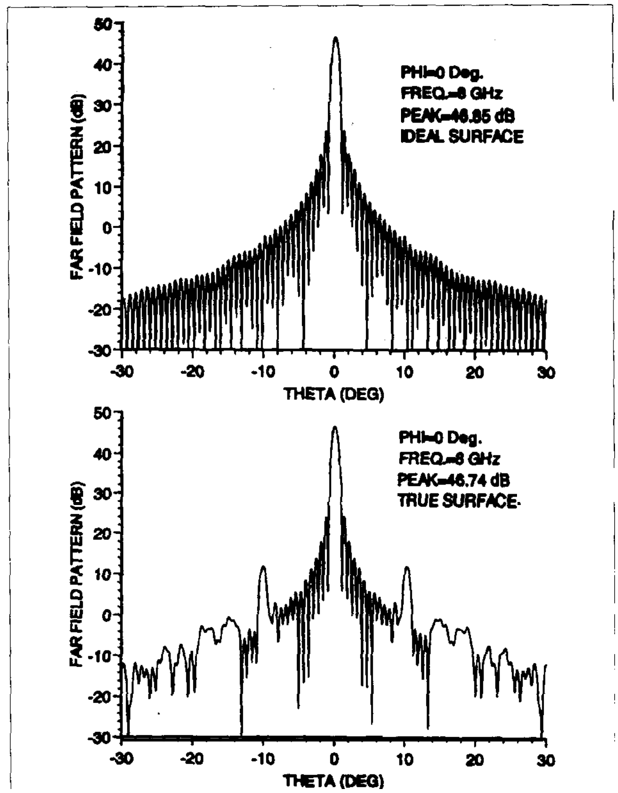


Fig. 11. Calculated H-plane patterns of 3m antenna at 8 GHz

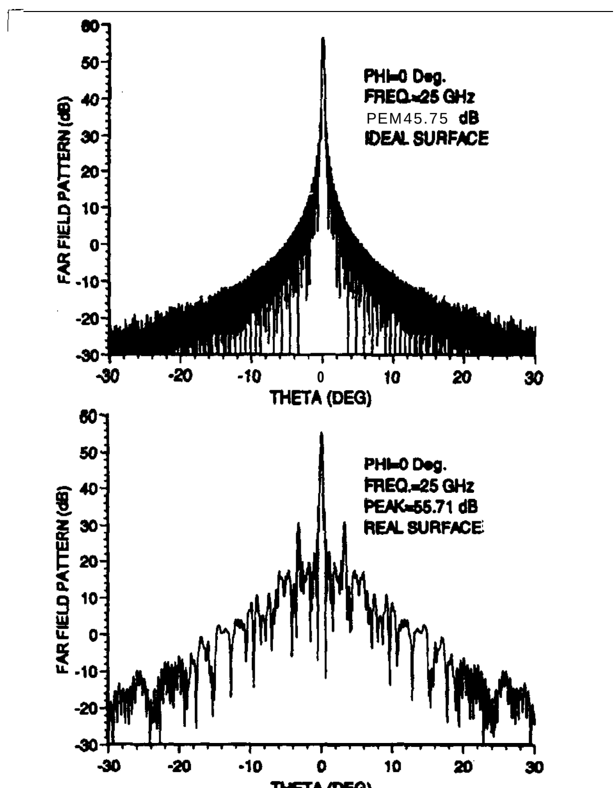


Fig. 12. Calculated H-plane patterns ot 3m antenna at 25 GHz

Interface to RAYTRK

RAYTRK, a ray-tracing computer code was developed to assess the performance of parabolic solar concentrators. It is designed to take as input, the updated surface generated by code FAIM. This updated surface profile is fed into RAYTRK in order to generate a uniform grid distribution over the reflector surface. Then, a ray of light originating from a point on the solar disk is traced towards a grid point (and every point on the reflector thereafter) and towards a focus plane. The intersection of this reflected ray with the focus plane is calculated. The results of one study using FAIM and RAYTRK are shown in Fig. 13

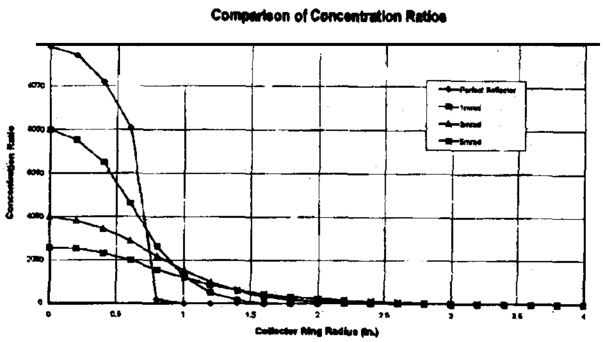


Fig. 13. Concentration ratios

Summary and Conclusions

We have developed a finite element analytical tool for the analysis and design of inflatable membrane structures. Its capabilities include (1) nonlinear material model and (2) modal dynamics capability. A pre- and post-processing interface program was coded and tested which renders the whole analysis/design process very streamlined. The loadings available in FAIM are (1) nodal or element thermal loads, (2) general 3D G-loads, (3) concentrated point loads, and (4) follower pressure. Specified nodal displacements as well as skew boundary conditions may be used. Furthermore, a nodal and element reordering capability was also coded to reduce computer storage and increase execution speed. This capability is a very important feature in that it significantly reduces computer run time.

In order to verify code performance, we compared the FAIM calculations to an analytical solution for a symmetric paraboloid. FAIM calculation results have also been compared to other verification cases that include the following, some of which were not presented in this paper:

* classical solution of the deflection and vibration of flat circular membranes.
* classical solution of the vibration of rectangular membranes
* analytical solution to the vibration of spherical membranes.
* experimental results on the natural frequencies of right circular and conical cylindrical tubes.

Finally, in order to increase the utility of the code, we have written interface programs between FAIM and an RF antenna code and between FAIM and a ray-tracer. Because of the FAIM code and the capabilities coded into it, we now have the capability of predicting the overall performance of membrane antennas and reflectors. The FAIM family of codes may be considered as a complete set of tools for the analysis of the surface precision and stresses within all types of precision inflatable space antennas and support structures. It makes the analysis and design of inflatable antennas and reflectors a very streamlined process.

7.0 Acknowledgments

The addition of nonlinear material and modal dynamics capability as well as interfacing FAIM to NECREF and RAYTRK was supported by a Phase I and II SBIR contract from NASA-JPL under contract NAS7-1219 and NAS7-1313, respectively. We also thank our consultant, Prof. Gerard C. Pardoen of the University of California at Irvine for his significant contributions.

References

1 Freeland, R. and Bilyeu, G., "In-Step Inflatable Antenna Experiment," paper presented at the 43rd Congress of the International Astronomical Federation, Aug. 28 - Sept 5, 1992, Washington, D.C.

2. Veal, G. and Freeland, R., "In-Step Inflatable Antenna Description," AIAA 1995 Space Programs and Technologies Conference, Huntsville, Alabama, Sept. 26-28 ,1995.

3. Thomas, M., "Effect of Inflation Pressure on Paraboloid Structure," L'Garde Memorandum, LM-79-MT-054, Aug. 10. 1979.

4. Thomas, M., "Changes in FLATE Code to Determine Flat Patterns," L'Garde Memorandum, LM-80-MT-082, Dec. 15, 1980.

5. Palisoc, A. and Huang, Y., "Finite Element Analysis of Inflatable Membranes (FAIM)," SBIR Phase II Final Report, L'Garde Technical Report, LTR-96-AP-012, May 1996.

6. Palisoc, A., "A User Interface and Finite Element Analysis of Off-Axis Inflatable Antennas," SBIR Phase I final Report, L'Garde Technical Report, LTR-93-AP-043, August 1993.

7. Huang, Y., "Test of Cable Elements in FAIM," L'Garde Technical Report, LTR-96-YH-023, October 1996.

8. Hinton, E. and Owen, D.R.J., "Finite Element Programming," Academic Press, London, 1977.

9. Lewis, J.G., "Implementation of the Gibbs-Poole-Stockmeyer and Gibbs-King Algorithms for Reordering Sparse Matrices," ACM Transactions on Mathematical Software, Vol. 8, No. 2, June 1982, pp. 190-194.

10. DISPLAY-III - A pre- and post-processor by Engineering Mechanics Research Corporation (EMRC) for its Finite Element Computer Package - NISA, Troy, Michigan.

11. NISA- Numerically Integrated Elements for System Analysis, A Finite Element Computer Package from Engineering Mechanics Research Corporation, Troy, Michigan.

12. NECREF - Ohio State University Reflector Antenna Code - Version 3.0, Ohio State University Electro Science Laboratory, Columbus, Ohio, Feb. 1994.

13. Hencky, H., "Uber den Spannunszustand in kreisrunden Platten mit vershwindender Biegungssteifigkeit," Zeits. Math. Phys., Vol. 63, pp. 311-316, 1915.

14. Meirovitch, L., "Analytical Methods in Vibration," McMillan Co., New York, 1971.

L•Garde Inc
15181 Woodlawn Avenue
Tustin, CA 92780-6487
www.LGarde.com