General Disclaimer

One or more of the Following Statements may affect this Document

- This document has been reproduced from the best copy furnished by the organizational source. It is being released in the interest of making available as much information as possible.

- This document may contain data, which exceeds the sheet parameters. It was furnished in this condition by the organizational source and is the best copy available.

- This document may contain tone-on-tone or color graphs, charts and/or pictures, which have been reproduced in black and white.

- This document is paginated as submitted by the original source.

- Portions of this document are not fully legible due to the historical nature of some of the material. However, it is the best reproduction available from the original submission.

FINITE ELEMENT ANALYSIS

OF

WRINKLING MEMBRANES

by

Richard K. Miller, John M. Hedgepeth*, Victor I. Weingarten
Prasanta Das, and Shahrzad Kahyai

Department of Civil Engineering
University of Southern California
Los Angeles, California 90089-1113

*   Consultant. Currently at Astro Research Corporation
                  Carpinteria, California

INTRODUCTION

The Department of Civil Engineering of the University of Southern California
received a research grant from the Langley Research Center of the National
Aeronautics and Space Administration to investigate the finite element analysis
of wrinkling membranes. The determination of stresses and deformations within
large partly-wrinkled membrane surfaces is a problem of significant technical
interest in such areas as conceptual design and analysis of ultralightweight
spacecraft structures.

When the research discussed herein was begun a three-phase project requiring
continued funding for approximately three years was envisioned. However,
shortly after funding for the first phase of the project was approved, plans
for funding the remaining two phases of the project were cancelled by NASA
due to a shortage of funds. Thus, what is reported herein is the result of
one year of effort toward the development and implementation of an algorithm
for the finite element analysis of quite general membrane surfaces. As
such, the effort was divided into two major tasks, which are discussed below.

AXISYMMETRIC DEFORMATION OF A SHALLOW INFLATED MEMBRANE

A closed-form solution to an axisymmetric problem involving partial wrinkling
of an inflated shallow membrane was obtained. In particular, a membrane in
the shape of a sperical annulus was considered. The outer edge of the annulus
was assumed to be fixed so that no displacements occur along the outer perimeter.
The inner edge is assumed to be clamped to a rigid movable plug. Solutions for

the complete stress, strain, and displacement fields under the assumption
of inextensional material behavior are presented for the case of pure
torsional loads applied to the plug, and for the case of pure axial loads
applied to the plug.

The analytical solution to this three-dimensional problem was intended to
serve as a "benchmark" against which comparisons could be made with future
numerical solutions obtained from the finite element procedure.

IMPLEMENTATION OF A FINITE ELEMENT ALGORITHM FOR FLAT MEMBRANES

A recently-developed algorithm [1] for finite element constitutive modeling
of wrinkling flat membranes based on the Stein-Hedgepeth [2] theory of
wrinkling was implemented on the SAP 7 [3] general purpose finite element
code at USC. The algorithm takes the form of a local strain-dependent
relationship between stresses and strains within an element, and currently
resides in the element library of nonlinear material behaviors accessible
by all users. The algorithm has been made compatible with the full library
of SAP 7 element types for two-dimensional stress analysis, including
isoparametric quadrilateral and higher order elements. Furthermore, the
algorithm has also been made compatible with the SAP 7 nonlinear analysis
capabilities for problems involving finite deformations and large displacements.

NUMERICAL EXAMPLE OF PURE BENDING OF A RECTANGULAR SHEET

As a first numerical example for evaluation of the finite element code just
described, the problem of pure bending of a stretched flat sheet was considered.
A closed-form analytical solution for this problem is available [2] for
comparison, and a finite element numerical model consisting of fifty isoparametric

8-node quadrilateral elements and 181 nodes was created. The edge conditions
on the finite element model were chosen to approximate as closely as possible.
those of the stress-resultant boundary conditions in the analytical formu-
lation, within the constraints that rigid body motion of the finite element
model is unacceptable.

Comparisons between the nondimensional numerical and analytical results for
the stress field within the membrane were made. A strip consisting of five
elements near the center of the sheet and spanning the complete range of
structural behavior from wrinkled to taut was chosen for comparison. It was
found that the numerical solutions for stresses agree very closely with the
analytical solutions with errors typically less than two or three percent.
Comparisons were made for three different load cases, and the numerically
obtained values for principal stresses in each case were interpolated to
obtain an estimate of the location of the boundary of the wrinkled region
within the sheet. These numerical estimates for boundary location were
compared with analytical results and again found to be very accurate, with
errors less than about one percent. Finally, numerical results for overall
moment-curvature behavior were compared with analytical results, and were
also found to be in very close agreement (less than about one percent error)
over a very large range of load magnitudes.

NUMERICAL EXAMPLE OF PURE TORSION OF RIGID HUB IN AN INFINITE SHEET

A second numerical example consisted of the problem of pure torsion applied
to a rigid circular hub bonded to a stretched infinite flat sheet. In order
to facilitate the finite element model, an infinite sheet was considered, and

the sheet was assumed to be pretensioned after the attachment of the hub.
The problem was then considered as a composite of two component problems.
The first component problem, which was modeled numerically with the finite
element code, consisted of an annular region near the hub. The hub was
considered to be fixed, and the outer edge of the annulus was subjected to
prescribed uniform tensile and shear stresses. The radius of the outer
edge was chosen to be large enough so that all wrinkling behavior occured
within the annulus.

The second problem consisted of the linear elasticity problem for the
exterior region. This problem was solved analytically. The behavior of
the original composite problem was then derived by enforcing continuity of
stresses and displacements at the interface between the annulus and exterior
region.

A closed-form analytical solution for the composite problem was developed as
a slight extension of the published solution in Ref. 4, Appendix b. Comparisons
between nondimensional numerical and analytical results for the stress field
within the membrane were made. A radial sector of the annulus, consisting of
three elements which span the range of structural behavior from wrinkled to
taut, was chosen for comparison. It was found that the numerical solutions
for the maximum principal stress as a function of radius agreed very well with
the analytical results, with errors typically less than about one percent.
Results for the minimum principal stress, which vanishes at the wrinkle boundary,
also compared favorably with the analytical results, but errors were typically
a few percent larger than those for the maximum stress. Similar accuracy was
obtained in comparisons of the location of the wrinkle boundary in both the
numerical and analytical solutions.

CONCLUSIONS

The research proposal upon which the approval of the supporting NASA grant was based, states the primary objective of the research as "to develop a general finite element analysis capability for predicting the mechanical behavior of membrane structures, including nonlinear geometric effects and wrinkling in an averaged sense." In particular, for Phase I, the only phase which was funded, the stated objective is "Develop the stated primary objective for the special case of flat membranes, and evaluate the results for accuracy and efficiency in benchmark numerical examples."

It is felt that these stated objectives for the research have been met, and that significant progress has been made toward the objectives of Phase II, which include generalizing the analysis capability to include curved and three-dimensional membranes.

REFERENCES

1. Miller, R.K., and Hedgepeth, J.M., "An Algorithm for Finite Element Analysis of Partly Wrinkled Membranes," AIAA Journal, Vol. 20, No. 12, pp. 1761-1763 (1982)

2. Stein, M., and Hedgepeth, J.M., "Analysis of Partly Wrinkled Membranes," NASA TN D-813, July 1961

3. "SAP 7 Theoretical Manual," Structural Mechanics Computer Laboratory, University of Southern California, Los Angeles, California (1982)

4. Mikulas, M.M. Jr., "Behavior of a Flat Stretched Membrane Wrinkled by the Rotation of an Attached Hub," NASA TN D-2456, Sept. 1964