

LEVEL II

12

DAVID W. TAYLOR NAVAL SHIP RESEARCH AND DEVELOPMENT CENTER



Bethesda, Maryland 20884

DTNSRDC-81/071

AD A109441

POSTPROCESSING OF ADINA STRAINS

by

Theodore G. Toridis
Peter N. Roth
Jana H. Judy

DTIC
SELECTED
JAN 08 1982

APPROVED FOR PUBLIC RELEASE: DISTRIBUTION UNLIMITED

RESEARCH AND DEVELOPMENT REPORT

December 1981

DTNSRDC-81

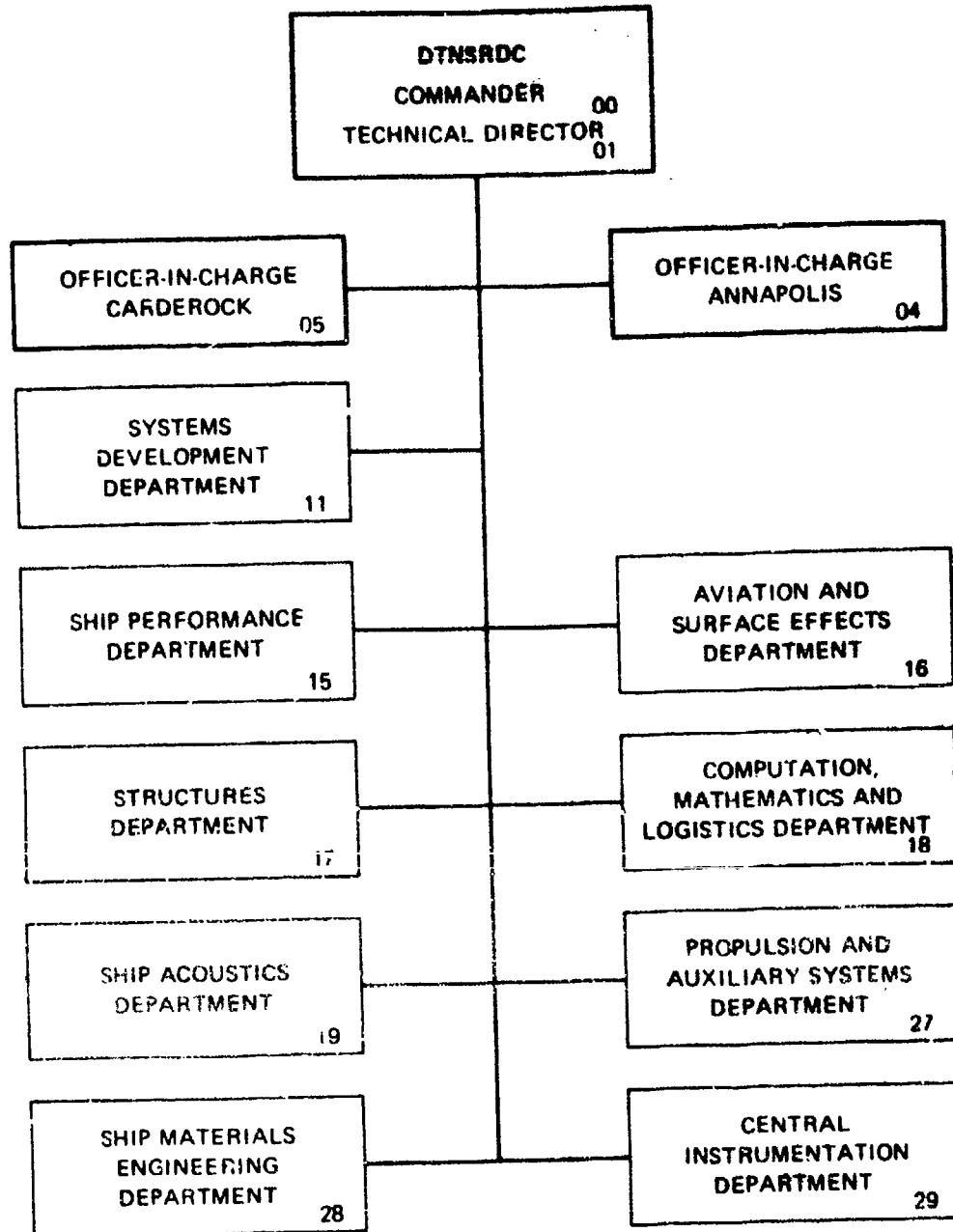
DTIC FILE COPY

DTIC FILE COPY

POSTPROCESSING OF ADINA STRAINS

387682 82 01 08140

MAJOR DTNSRDC ORGANIZATIONAL COMPONENTS



UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE		READ INSTRUCTIONS BEFORE COMPLETING FORM
1. REPORT NUMBER DTNSRDC-81/071	2. GOVT ACCESSION NO. AD-A109 444	3. RECIPIENT'S CATALOG NUMBER
4. TITLE (and Subtitle) POSTPROCESSING OF ADINA STRAINS		5. TYPE OF REPORT & PERIOD COVERED Final
		6. PERFORMING ORG. REPORT NUMBER
7. AUTHOR(s) Theodore G. Toridis Peter N. Roth Jana H. Judy		8. CONTRACT OR GRANT NUMBER(s)
9. PERFORMING ORGANIZATION NAME AND ADDRESS David W. Taylor Naval Ship Research and Development Center Bethesda, Maryland 20084		10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS Program Element 62543N Task Area SF43400001 Work Unit 1750-294
11. CONTROLLING OFFICE NAME AND ADDRESS		12. REPORT DATE December 1981
		13. NUMBER OF PAGES 26
14. MONITORING AGENCY NAME & ADDRESS (if different from Controlling Office)		15. SECURITY CLASS. (of this report) UNCLASSIFIED
		15a. DECLASSIFICATION/DOWNGRADING SCHEDULE
16. DISTRIBUTION STATEMENT (of this Report) APPROVED FOR PUBLIC RELEASE: DISTRIBUTION UNLIMITED		
17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)		
18. SUPPLEMENTARY NOTES Theodore G. Toridis is a Professor, School of Engineering and Applied Science, George Washington University, Washington, D.C. 20052.		
19. KEY WORDS (Continue on reverse side if necessary and identify by block number) Strain Output Linear Analysis Finite Element Analysis Nonlinear Analysis Static Response Fluid-Structure Interaction Dynamic Response		
20. ABSTRACT (Continue on reverse side if necessary and identify by block number) A procedure has been developed for processing the strains available with the computer program ADINA to provide strain output in any desired orientation and to calculate and output strains on the surface of a three-dimensional solid or shell element. To this end, proper additions and modifications have been made to ADINA to implement the procedure directly within the program to allow for the processing and output of strains during (Continued on reverse side)		

DD FORM 1473
1 JAN 73

EDITION OF 1 NOV 65 IS OBSOLETE
S/N 0102-LF-014-6601

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

(Block 20 continued)

the normal execution stage. This includes reading in additional input data (of minimum amount), transformation of strains from the global system to specified coordinate directions, the output of transformed strains, and the calculation and output of surface strains based on the interpolation functions available in ADINA. At the present time such capabilities are provided for a limited number of elements on a selective basis, in order to minimize the amount of computer storage required.

Some typical structures have been analyzed to compute and output strains both at interior points and on exterior surfaces of elements, for a set of elements specified by the user during the input stage. Results demonstrate the feasibility of the procedure for the transformation and extrapolation of strains.

Accession For	
NTIS GRA&I	<input checked="" type="checkbox"/>
DTIC TAB	<input type="checkbox"/>
Unannounced	<input type="checkbox"/>
Justification	
By	
Distribution	
Availability Codes	
Dist	
A	

TABLE OF CONTENTS

	Page
LIST OF FIGURES.	iii
TABLE.	iii
ABSTRACT	1
ADMINISTRATIVE INFORMATION	1
INTRODUCTION	1
INTERPOLATION AND EXTRAPOLATION OF STRAINS	4
USE OF ADINA INTERPOLATION FUNCTIONS.	6
BASIC PROGRAM MODIFICATIONS	9
EXAMPLE PROBLEMS	12
PURE SHEAR--NONLINEAR STRAIN CASE	12
LINEAR ANALYSIS OF A STIFFENED PLATE.	15
CLAMPED PLATE SUBJECTED TO HYDRODYNAMIC PRESSURE.	15
CONCLUDING REMARKS	19
ACKNOWLEDGMENT	20
REFERENCES	21

LIST OF FIGURES

1 - Variable Node Three-Dimensional Solid Element.	7
2 - Standard Meshes for Surface Strain Output.	10
3 - Nonlinear Shear Deformation of an Element.	14
4 - Stiffened Plate Structure.	16
5 - Time History of Strains at Midpoint.	19

Table 1 - Transformed Strains at Node Points of Elements	17
--	----

ABSTRACT

A procedure has been developed for processing the strains available within the computer program ADINA to provide strain output in any desired orientation and to calculate and output strains on the surface of a three-dimensional solid or shell element. To this end, proper additions and modifications have been made to ADINA to implement the procedure directly within the program to allow for the processing and output of strains during the normal execution stage. This includes reading in additional input data (of minimum amount), transformation of strains from the global system to specified coordinate directions, the output of transformed strains, and the calculation and output of surface strains based on the interpolation functions available in ADINA. At the present time such capabilities are provided for a limited number of elements on a selective basis, in order to minimize the amount of computer storage required.

Some typical structures have been analyzed to compute and output strains both at interior points and on exterior surfaces of elements, for a set of elements specified by the user during the input stage. Results demonstrate the feasibility of the procedure for the transformation and extrapolation of strains.

ADMINISTRATIVE INFORMATION

The work reported herein was funded by the Naval Sea Systems Command (NAVSEA 032R), under the Exploratory Development Program in Submarine Combat Protection. The work was performed at the David W. Taylor Naval Ship Research and Development Center under Program Element 62543N, Task Area SF43400001, and Work Unit 1750-294.

Dr. Theodore G. Toridis is a Professor, School of Engineering and Applied Science, George Washington University, Washington, D.C. 20052.

INTRODUCTION

The analytical solution of most practical structures culminates in the computation of displacements, stresses and strains, or other quantities of interest to the engineer. Whenever efforts are made to compare analytical results with test measurements, the comparison of analytical and experimental strains plays an important role in the process of verification of a structural model, because, in general, strain measurements are relatively easy to obtain and record during the testing process.

When a material behaves elastically, strain, in the engineering sense, is ordinarily defined as the change in length per unit initial length. This definition of strain was first introduced by Cauchy and is usually measured quantitatively by performing a uniaxial tension test and determining the changes in length as the specimen deforms. When finite strains and deformations take place in a body in a three-dimensional sense, the problem of characterizing strain becomes more complicated. In such cases, two different definitions of strain are commonly employed, one due to Green and the other due to Almansi. If the strains are referred to the initial (usually undeformed) configuration of the system, they correspond to the so-called Lagrangian formulation. If, on the other hand, strains are expressed as functions of the deformed configuration, they are normally identified with the Eulerian formulation. If large deformations are associated with material and geometric nonlinearities, coupled with large strains, the computation of strains in a complex structural system becomes a formidable task, and can be accomplished effectively only by the use of a general purpose finite element computer program, such as ADINA.

In ADINA, in the case of a linear elastic continuum, components of the strain tensor ${}^{t+\Delta t}e_{rs}$ at time $t + \Delta t$ are computed incrementally based on the usual infinitesimal theory and expressed as^{1*}

$${}^{t+\Delta t}e_{rs} = \frac{1}{2} \left({}^{t+\Delta t}u_{r,s} + {}^{t+\Delta t}u_{s,r} \right) \quad (1)$$

where ${}^{t+\Delta t}u_{r,s}$ are the displacement gradients at time $t + \Delta t$ measured with reference to the initial configuration of the body at time 0. In the case of large deformations, when the Total Lagrangian Formulation coupled with an incremental procedure is used, the strain relations are represented by

$${}^{t+\Delta t}e_{ij} = {}^t e_{ij} + {}^0 e_{ij} \quad (2)$$

*A complete listing of references is given on page 21.

where

$$0^{\epsilon_{ij}} = 0^e_{ij} + 0^n_{ij} \quad (3)$$

$$0^e_{ij} = \frac{1}{2} \left(0^{u_{i,j}} + 0^{u_{j,i}} + 0^{u_{k,i}} 0^{u_{k,j}} + 0^{u_{k,i}} 0^{u_{k,j}} \right) \quad (4)$$

$$0^n_{ij} = \frac{1}{2} \left(0^{u_{k,i}} 0^{u_{k,j}} \right) \quad (5)$$

When the updated Lagrangian Formulation is employed, the corresponding expressions for the strains are given as

$${}^{t+\Delta t} t^{\epsilon_{ij}} = t^{\epsilon_{ij}} \quad (6)$$

in which

$$t^{\epsilon_{ij}} = t^e_{ij} + t^n_{ij} \quad (7)$$

and

$$t^e_{ij} = \frac{1}{2} (t^{u_{i,j}} + t^{u_{j,i}}) \quad (8)$$

$$t^n_{ij} = \frac{1}{2} (t^{u_{k,i}} t^{u_{k,j}}) \quad (9)$$

Relations of this type can be employed to compute strains as part of the overall computational process. In the case of a thin shell element, complete strain-displacement transformation matrices for the ADINA shell element, for both the linear and nonlinear cases, have been derived and given in Reference 2.

The basic computational algorithm embedded in ADINA makes provision for the computation of strains, but no special effort is made to generate elaborate strain output at the option of the user of the program. More specifically, the strain output provided in ADINA allows for storing the strains in a porthole for subsequent access, but the strains are referred to the global Cartesian coordinate system. For a linear material model, the strains are available at certain nodes defined by the stress tables, or at the element integration points. For nonlinear material models, the strains are available at the integration points only.

In the present study, for a three-dimensional solid and shell element, the above capabilities were extended in order to provide for a more comprehensive and versatile strain output. At present, the options provided are summarized below:

1. For linear material models, strains are output at nodal points specified by stress tables. However, the strains can be referred to an arbitrarily oriented Cartesian coordinate system by inputting the corresponding direction cosine matrix for each element. Alternatively, the user can specify the output of strains with reference to the natural (r,s,t) coordinate system as defined by ADINA.

2. For linear or nonlinear material models, the strains are output at integration points with reference to any coordinate system as explained in Item 1, or to the natural (r,s,t) coordinate system.

3. For either linear or nonlinear material models, in addition to the above output, the option of providing printout of the strains on any two surfaces of a three-dimensional solid or shell element has been provided. For this purpose, the user can specify a variable mesh for output of strains at the mesh points (a 3x3 to 5x5 mesh), or input an array of coordinate locations for strain output. In computing these strains, the interpolation functions available within ADINA are employed. As pointed out previously, this option is significant in engineering research applications, because experimental strains are measured on the structure surface and must be correlated with analytically computed strains at the same locations.

INTERPOLATION AND EXTRAPOLATION OF STRAINS

As mentioned earlier, in the case of a linear material model of a structure made up of three-dimensional or shell elements, the output of stresses and strains in ADINA is provided at selected nodal points specified by means of stress tables. The computation of strains at any other point in the interior or at the surface of the element must invariably involve an interpolation process. Within the framework of a stress table employed in ADINA, a maximum of 16 points can be employed to perform such interpolation. On the other hand, for nonlinear material models, stresses and strains can be output only at Gaussian integration points. Because the maximum order of numerical integration in each of the three coordinate directions is 4, at most a total of 64 points can be used to perform extrapolation.

Although such approximations can be carried out based on a variety of procedures, for the purpose of accomplishing this with a moderate amount of effort and a reasonable number of modifications to ADINA, the use of the shape functions and associated derivatives inherent in ADINA appears to be the logical approach to follow, as was done in the present study. It should be pointed out, however, that, in the case of the three-dimensional solid element or shell element, the interpolation process involved in ADINA uses functions of order no higher than two (quadratic). Nevertheless, in the case of regular geometrical surfaces, matching the location of computed strains with measured strains resulting from an experiment would ordinarily pose no difficulty. On the other hand, for bodies of arbitrary shape, a more accurate description of the location of surface points may be desirable. Also, in view of the fact that up to four Gaussian integration points can be employed in each coordinate direction, the extrapolation process for the strains can be based on higher order functions.

In addition, as proposed by Hinton and Campbell³ concerning an improved extrapolation method for stresses, in a similar manner local least squares smoothing techniques can be used to perform an extrapolation of strains from sampling locations corresponding to the Gaussian integration points. Alternatively, the concept of conjugate approximations developed by Oden and Brauchli⁴ for obtaining consistent approximations of stress fields in the displacement formulation of the finite element method can be extended to the case of approximation of strain fields. It should be pointed out, however, that such approaches would require major modifications to ADINA leading to extended efforts beyond the normal level.

Therefore, from the viewpoint of computational efficiency, the use of the original shape functions employed in the description of the element geometry and displacements to perform interpolation or extrapolation of the strains, appears to be the most feasible approach to follow for achieving the immediate goals of this study. Such a procedure can be used to evaluate both linear and nonlinear strains at any point in an element. The distinct advantage of this approach is that the process of interpolation or extrapolation for the geometry, the displacements and strains, are all based on the same functions leading to a more consistent and efficient procedure as far as the computational effort is concerned.

USE OF ADINA INTERPOLATION FUNCTIONS

The shape functions used in ADINA for an 8 to 21 variable-node-number three-dimensional solid element are in the form of (see Figure 1):

$$h_1 = g_1 - \frac{(g_9 + g_{12} + g_{17})}{2} - \frac{g_{21}}{8} \quad (10)$$

$$h_2 = g_2 - \frac{(g_9 + g_{10} + g_{18})}{2} - \frac{g_{21}}{8} \quad (11)$$

$$h_3 = g_3 - \frac{(g_{10} + g_{11} + g_{19})}{2} - \frac{g_{21}}{8} \quad (12)$$

$$h_4 = g_4 - \frac{(g_{11} + g_{12} + g_{20})}{2} - \frac{g_{21}}{8} \quad (13)$$

$$h_5 = g_5 - \frac{(g_{13} + g_{16} + g_{17})}{2} - \frac{g_{21}}{8} \quad (14)$$

$$h_6 = g_6 - \frac{(g_{13} + g_{14} + g_{18})}{2} - \frac{g_{21}}{8} \quad (15)$$

$$h_7 = g_7 - \frac{(g_{14} + g_{15} + g_{19})}{2} - \frac{g_{21}}{8} \quad (16)$$

$$h_8 = g_8 - \frac{(g_{15} + g_{16} + g_{20})}{2} - \frac{g_{21}}{8} \quad (17)$$

$$h_j = g_j - \frac{g_{21}}{4} \text{ for } j = 9, \dots, 20 \quad (18)$$

$$h_{21} = g_{21} \quad (19)$$

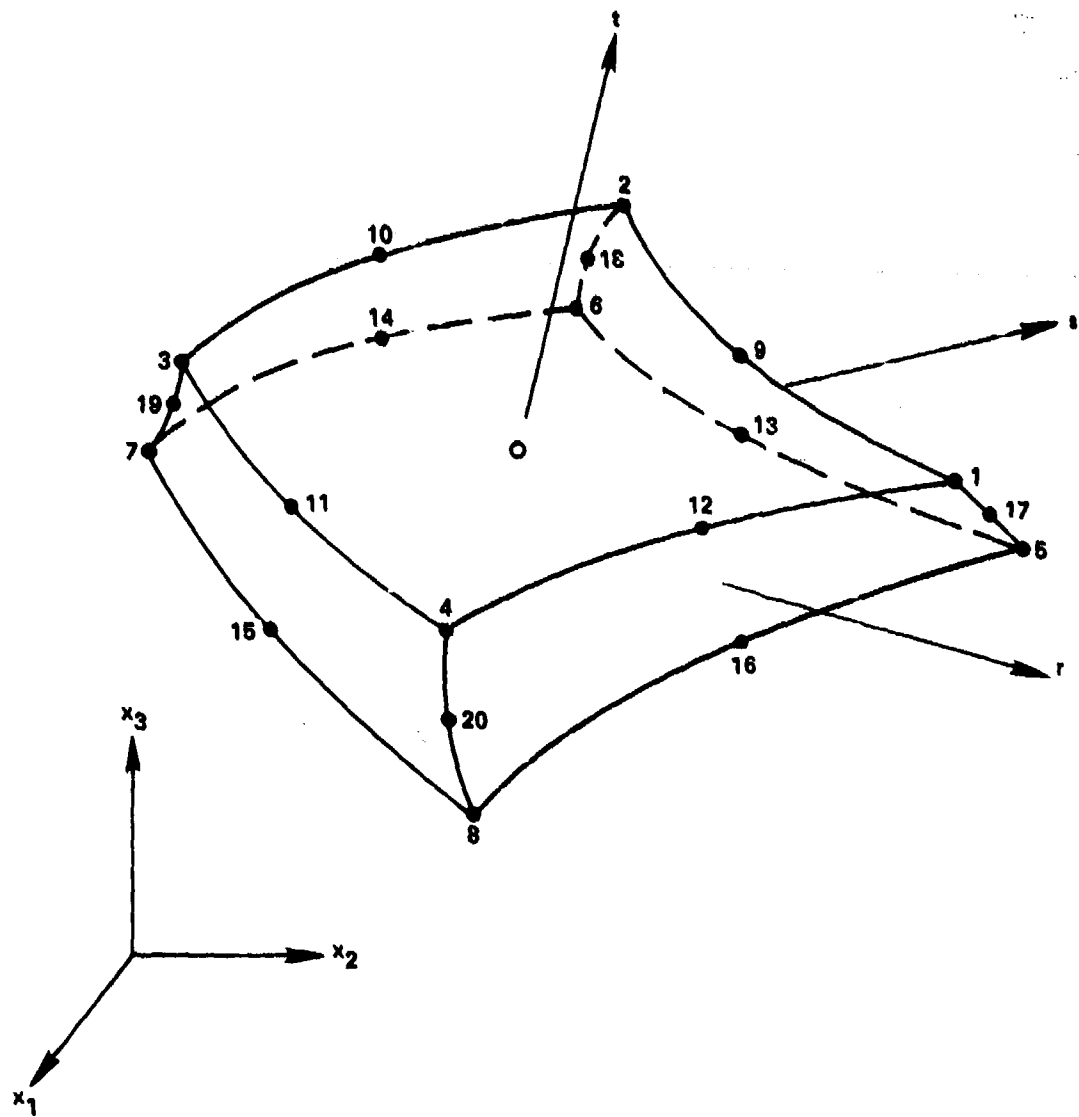


Figure 1 - Variable Node Three-Dimensional Solid Element

where $g_i = 0$ if node i is not included; otherwise

$$g_i = G(r, r_i) G(s, s_i) G(t, t_i)$$

$$G(\beta, \beta_i) = \frac{1}{2} (1 + \beta_i \beta) \text{ for } \beta_i = \pm 1; \beta = r, s, t \quad (20)$$

$$G(\beta, \beta_i) = (1 - \beta^2) \text{ for } \beta_i = 0; \beta = r, s, t$$

Similar shape functions are used for the ADINA shell element except that the more general concept of displacement and rotation isoparametric element is used in its development.²

These shape functions form the basis of evaluation of the element properties which are then synthesized to obtain the corresponding properties of the entire structure taking into account the boundary and constraint conditions. Once the displacements at the nodal points have been solved, the computation of the displacements and, consequently, the strains anywhere within an element can be accomplished using the shape functions and associated derivatives. As discussed previously, provisions were originally made in ADINA for the computation of strains--for both linear and nonlinear material models--at specific points, with reference to the global coordinate system. The procedure used in computing these strains at the nodal or Gaussian integration points has simply been repeated in this study to provide strain output at other points within the element or on its surface. For this purpose, it is only necessary to specify the desired values of the natural r, s, t coordinates at a point at which the strains are to be computed. However, in general, the $x, y,$ and z coordinates of the point will be known, and the determination of the corresponding natural coordinates for complicated geometries may involve an iterative process.

In the case of surface strain output, two different options are provided as follows:

1. Surface strains are output at points specified during the input stage by means of an array of natural (r, s, t) coordinates corresponding to the output locations.

2. In order to minimize the input, a variable mesh size is selected as part of the input, so that surface strains can be output automatically at locations corresponding to the grid points of the particular mesh. To this end, mesh sizes of 3×3 to 5×5 can be used as shown in Figure 2, in which the grid line locations are also indicated by specifying the constant coordinate values along grid lines. These locations have been selected to correspond to Gaussian integration points of corresponding order.

Once the strains are computed at each point of interest, they are transformed from the global coordinate directions to any desired direction using a standard rotation matrix $[\lambda]$. The entries of $[\lambda]$ are the direction cosines associated with the global (x_1, x_2, x_3) axes, and the inclined (x_1', x_2', x_3') axes. These entries are defined as part of the input for each element. Alternatively, in order to minimize the amount of input, the option has been provided for generating internally the direction cosines using the information available within ADINA. Then, denoting by $[e]$ and $[e']$ the strain tensors referred to the global and inclined axes, respectively, the transformation of strains is expressed in the usual form as

$$[e'] = [\lambda]^T [e] [\lambda] \quad (21)$$

BASIC PROGRAM MODIFICATIONS

The modifications made to the program are reflected in overlays ADINA and ADINI and to a greater extent in subroutines THDFE and SHELTH. Minor modifications have also been made in some other subroutines.

Modifications made to ADINA are minimal to allow for the addition of new common blocks associated with strain computations and output, and to perform dimensioning and initialization of certain new variables.

Overlay ADINI, which administers input and solution control data, was modified by allocating permanent memory locations at the beginning of the blank common area for variables required in strain computations and output. Also, certain input statements were modified to accommodate some key parameters that control the output of stresses and strains. In particular, two parameters were added to the input stream to serve as overall control flags for determining whether or not any output of stresses and/or strains is to take place. These represent overall control parameters in addition to the ones available at the element level.

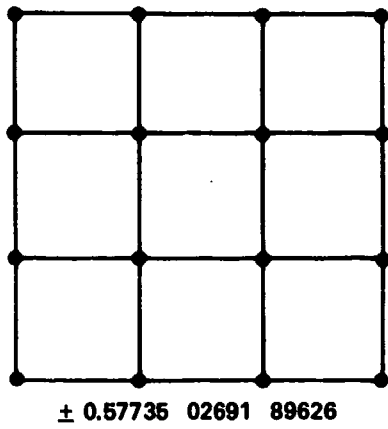


Figure 2a - 3 × 3 Mesh

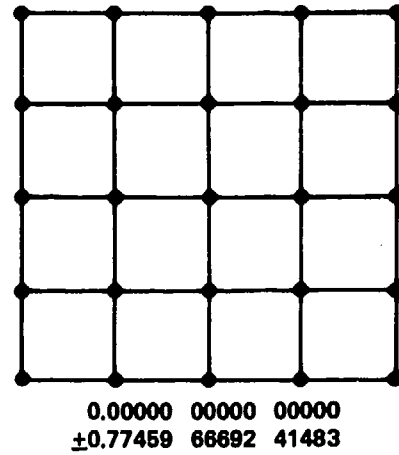


Figure 2b - 4 × 4 Mesh

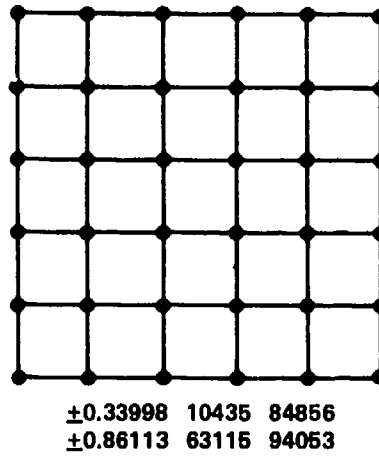


Figure 2c - 5 × 5 Mesh

Figure 2 - Standard Meshes for Surface Strain Output

Subroutines THDFE and SHELTH were modified extensively as outlined below:

1. New common blocks were added to allow for the flow of information between the various overlays. Also, dimension statements for new variables were incorporated, and key parameters initialized.

2. Input of information for the elements was expanded to include certain control parameters that monitor the input and output functions for the strains, the direction cosine matrices for strain transformations, and information for the surface strain output.

3. Because the strain output option is provided for a limited number of elements, arrays were generated near the end of the input phase in both subroutines to store the above information and other pertinent data to allow for the subsequent retrieval of the information during the execution (calculation) stage. Similarly, the necessary code was added to retrieve the information needed during the calculations of stresses and strains for both the branches dealing with the calculation of these quantities at specified nodal points and the branch pertaining to the Gaussian integration points.

4. Additions were made for the computation of the coordinate arrays for surface strain output locations, particularly for the case when these locations are defined as the grid points of standard meshes.

5. Changes were incorporated in both subroutines to compute the strains at points other than originally provided for in ADINA.

6. The computation or retrieval of the direction cosine matrix was added and the transformation of strains was accomplished based on standard operations.

7. Several control statements were added for monitoring output at the element level and changes were made to accomplish the additional output functions necessary for the transformed strains at nodal points and other specified locations.

As a result of these modifications, the following output options can be exercised in the output of stress and strains.

- a. The output of stresses and strains can be controlled at a global scale through the proper definition of the parameters ITHSE and ITHSA, respectively, that allow for the printout or suppression of the output of these quantities.

b. Control of stress and strain output at the element level is accomplished through the proper choice of the value of the control variable IPS, originally used to exercise the option of printing or suppressing the output of stresses. In this respect, the option of outputting stresses and strains can be exercised for any number of elements (belonging to various groups of elements), provided these quantities are output only at locations specified by means of stress tables or at Gaussian integration points. The strain output can be referred to any arbitrarily oriented orthogonal coordinate system through proper input.

c. In addition to the above option, for a limited number of elements (that may belong to different groups of elements), the option is provided for the output of strains at specified points in the element or at surface locations associated with grid points of 3×3 to 5×5 meshes. As in the above case, these strains can be referred to arbitrarily oriented coordinate axes through the input of direction cosine matrices or to the natural (r,s,t) coordinate system.

EXAMPLE PROBLEMS

Some simple problems were analyzed primarily for the purpose of testing the operation of the modified computer program. Brief descriptions of the example problems analyzed at present and the resulting strain output are given below.

PURE SHEAR--NONLINEAR STRAIN CASE

This problem represents a very simple example of a nonlinear strain case drawn out of Eringen.⁵ It consists of a cube in "simple shear," as shown in Figure 3a, where γ represents the angle of shear and SY the translational displacement in the X-direction at a distance Y measured along the Y-axis, so that

$$\tan \gamma = \frac{SY}{Y} = S \quad (22)$$

or

$$\gamma = \arctan S \quad (23)$$

The Lagrangian strain tensor \tilde{E} representing the state of deformation as given by Eringen⁵ is

$$\tilde{E} = \begin{bmatrix} 0 & S/2 & 0 \\ S/2 & S^2/2 & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

For the purpose of obtaining numerical results using the modified ADINA program, a single three-dimensional solid element with 21 nodes was used, as shown in Figure 3b. The dimensions of the cubical element were chosen to be 1.0 in all three coordinate directions. The structure was constrained against movement in the Y-direction so that no change in length takes place along the Y-axis. Also, constraints in the Z-direction were imposed and the movement in the X-direction of nodes 2, 3, 4, 9, 10, 11, 12, 17, 18, 19, 20, and 21 (see Figure 3b) was constrained to correspond to the movement of node 1. At node 1 a prescribed X-displacement of 0.01 inch was imposed to generate the desired state of simple shear. A nonlinear static analysis was then performed using a material model corresponding to an elastic-plastic material with isotropic hardening (Model 8) with the constants defined as:

Modulus of Elasticity E	$= 30 \times 10^6$ psi
Poisson's Ratio, ν	$= 0.30$
Yield Stress	$= 36 \times 10^3$ psi
Hardening Modulus	$= 0.0$ psi

A step size of 0.1 was used leading to a total of 10 solution steps. The results of the analysis were as expected so that, at the final solution step, the X-displacements of nodes 1 through 4 and 9 through 12 were found to be equal to 0.01 inches, of nodes 17 through 21 equal to 0.005 inches, and the remaining displacements equal to zero. With reference to the (r,s,t) coordinate system shown in Figure 3b, the diagonal and off-diagonal components of the Lagrangian strain components were found (at the end of the 10th solution step) to correspond to the exact values of

$$\begin{aligned} \tau_{rr} &= 0.00005 \\ \tau_{rt} &= -0.01000 \end{aligned}$$

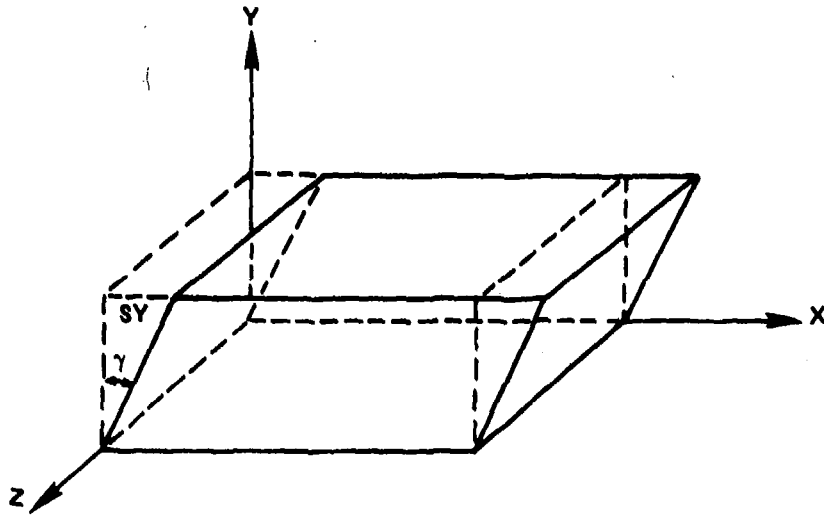


Figure 3a - Element in Simple Shear

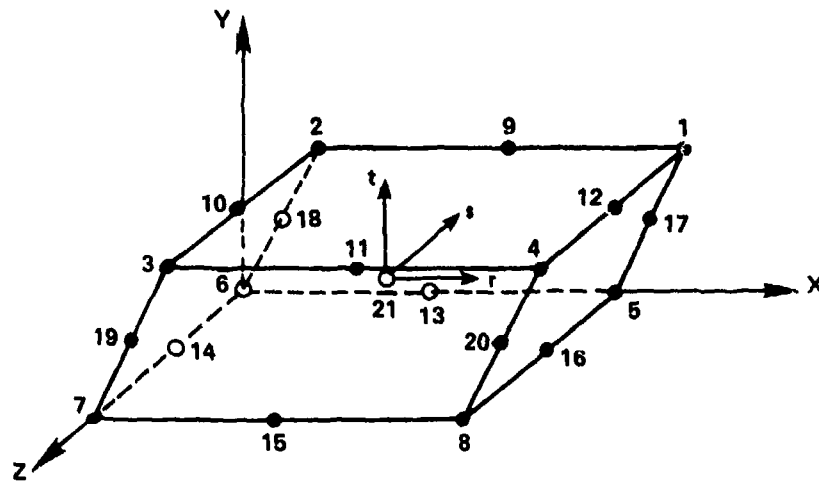


Figure 3b - Discrete Model Three-Dimensional Solid Element

Figure 3 - Nonlinear Shear Deformation of an Element

LINEAR ANALYSIS OF A STIFFENED PLATE

The structure considered in this case consists of a cantilever plate with vertical rectangular stiffeners along its longitudinal edges, as shown in Figure 4a. The plate has a width of 40 inches and a thickness of 1 inch. The stiffeners are 1 inch by 6 inches in cross section. This structure was previously analyzed by Bathe and Bolourchi² using ADINA SHELL elements. In their study they obtained favorable comparison with the solution calculated using elementary beam theory. In the present study, the same structure was analyzed using three-dimensional solid elements. The plate and stiffeners were each modeled using two three-dimensional elements, for a total of six elements, 66 node points and 108 degrees-of-freedom. This discrete model of the structure is shown in Figure 4b.

A linear static analysis was performed considering the material of the structure to be linear isotropic with a modulus of elasticity of 30×10^6 psi and Poisson's ratio of 0.30. The two concentrated loads acting on the tip of the cantilever were taken to be 5,000 lb each, so that $P = 10,000$ lb. The comparison of the vertical deflections of the tip of the cantilever obtained by means of beam theory and ADINA is given as:

Maximum Deflection - Beam Theory: 1.26 inch

Maximum Deflection - ADINA: 1.27 inch

Typical strain outputs for elements 1 and 2 are given in Table 1. These strains have been multiplied by a factor of 10^3 in order to retain a sufficient number of significant figures and are referred to the (r,s,t) local coordinate systems shown in Figure 4b. The strains are in agreement with the theoretical values based on the beam theory after the appropriate transformation of the strain from the global to the local system of axes, as verified by calculations independent of the computer program.

CLAMPED PLATE SUBJECTED TO HYDRODYNAMIC PRESSURE

In this case an experimental setup of a structure consisting of 4 inch steel plates of 1 inch thickness and clamped at both ends was analyzed using the modified ADINA program.

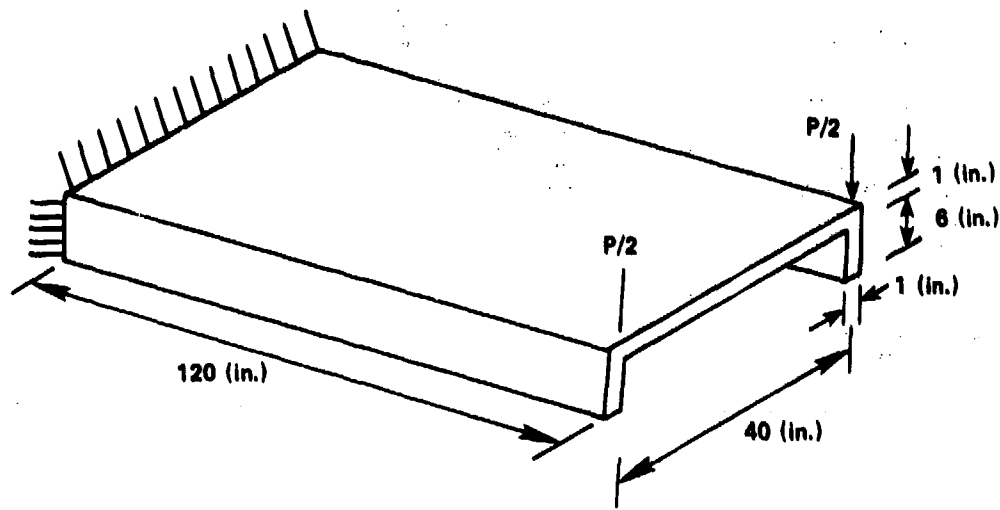


Figure 4a - Structure Considered

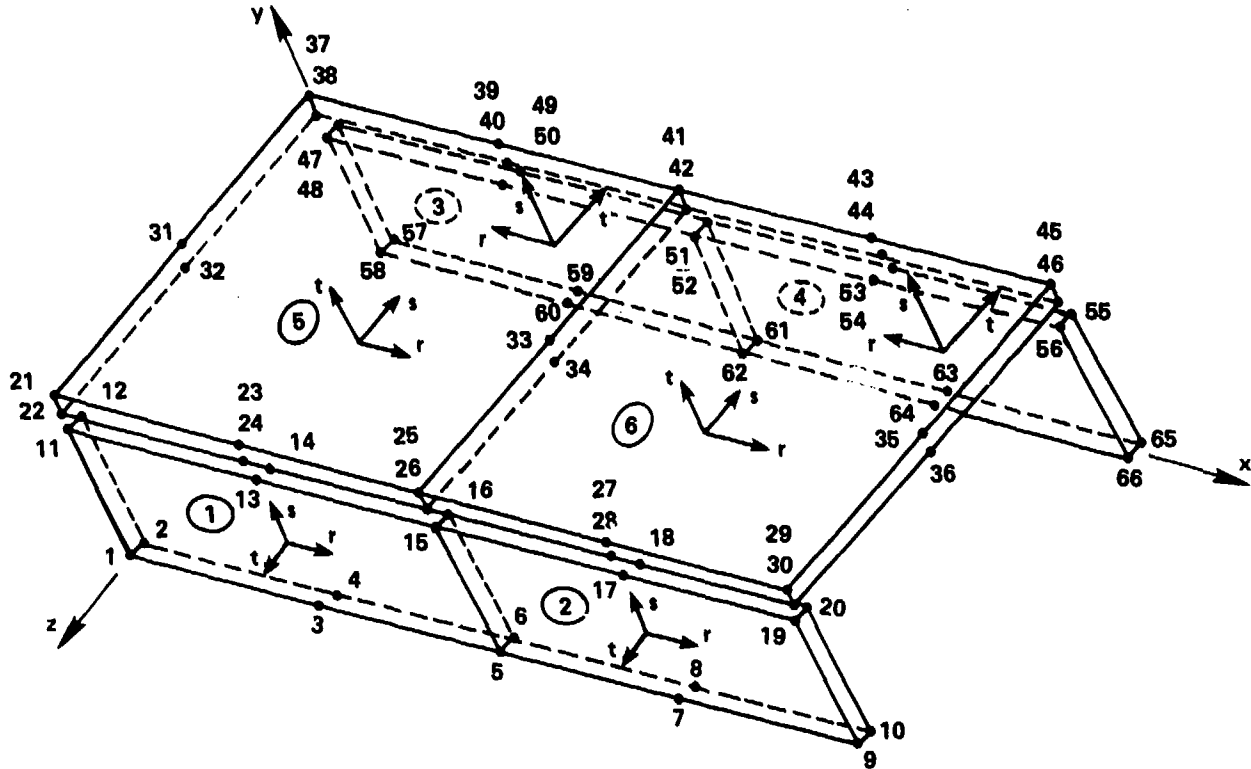


Figure 4b - Discrete Model of Structure

Figure 4 - Stiffened Plate Structure

TABLE 1 - TRANSFORMED STRAINS AT NODE POINTS OF ELEMENTS

Element 1							
Output Location	Strain 11x10 ³	Strain 22x10 ³	Strain 33x10 ³	Strain 12x10 ³	Strain 13x10 ³	Strain 23x10 ³	
1	-0.0874692	0.0923501	0.0000000	0.6769020	-0.0532644	-0.0303249	
2	-0.0955037	0.0000000	0.0000000	0.5903342	0.0418543	0.0000000	
3	1.4546710	0.0000000	0.0000000	0.6914583	-0.1498268	0.0000000	
4	0.7495472	0.0923501	0.2344822	0.5573078	-0.0594656	-0.0141154	
5	-0.0874692	0.0950517	0.0000000	0.7219976	-0.0532644	0.0087555	
6	-0.0955037	0.0000000	0.0000000	0.6181857	0.0418543	0.0000000	
7	1.4420149	0.0000000	0.0000000	0.7199353	-0.1057158	0.0000000	
8	0.7583076	0.0950517	0.2344822	0.6012378	-0.1113926	0.0249650	
Element 2							
Output Location	Strain 11x10 ³	Strain 22x10 ³	Strain 33x10 ³	Strain 12x10 ³	Strain 13x10 ³	Strain 23x10 ³	
1	0.0169166	0.1966553	0.0000000	0.8311109	0.0413899	-0.0437456	
2	-0.0451244	0.0923501	0.0000000	0.7208089	0.0080931	-0.0303249	
3	0.7155528	0.0923501	0.2344822	0.8254092	0.4109393	-0.0141154	
4	-0.0015520	0.1966553	0.4797637	0.7056495	0.4402443	-0.0381634	
5	0.0169166	0.1975856	0.0000000	0.7815843	0.0413899	0.0362150	
6	-0.0451244	0.0950517	0.0000000	0.6881250	0.0080931	0.0087555	
7	0.7599798	0.0950517	0.2344822	0.7940248	0.4545257	0.0249650	
8	-0.0432838	0.1975856	0.4797637	0.6551777	0.3884818	0.0417972	

Each plate basically behaves as a shallow beam of 4 inch width and 1 inch thickness in cross section fixed at both ends and having a span of 21 inches. Due to the nature of the dynamic loading, this problem corresponds to a nonlinear fluid-structure interaction problem. In order to carryout the analysis, the modifications for the strain output were incorporated in an augmented version of ADINA that provides the capability for hydrodynamic analysis due to underwater explosions.

In the finite element model of the structure, a total of 20 three-dimensional solid elements were used leading to a discrete model of 206 nodes and 583 degrees-of-freedom. A linear dynamic analysis was performed based on a linear isotropic material with the modulus of elasticity equal to 30×10^6 psi and Poisson's ratio of 0.30. A time step size of 0.5×10^{-5} was used in the analysis corresponding to a total of 100 time steps. The time history of strain at the midpoint on the top surface of the beam is shown in Figure 5, in which the horizontal scale is the longitudinal strain.

TABLE 1 - TRANSFORMED STRAINS AT NODE POINTS OF ELEMENTS

Element 1						
Output Location	Strain 11x10 ³	Strain 22x10 ³	Strain 33x10 ³	Strain 12x10 ³	Strain 13x10 ³	Strain 23x10 ³
1	-0.0874692	0.0923501	0.0000000	0.6769020	-0.0532644	-0.0303249
2	-0.0955037	0.0000000	0.0000000	0.5903342	0.0418543	0.0000000
3	1.4546710	0.0000000	0.0000000	0.6914583	-0.1498268	0.0000000
4	0.7495472	0.0923501	0.2344822	0.5573078	-0.0594656	-0.0141154
5	-0.0874692	0.0950517	0.0000000	0.7219976	-0.0532644	0.0087555
6	-0.0955037	0.0000000	0.0000000	0.6181857	0.0418543	0.0000000
7	1.4420149	0.0000000	0.0000000	0.7199353	-0.1057158	0.0000000
8	0.7583076	0.0950517	0.2344822	0.6012378	-0.1113926	0.0249650

Element 2						
Output Location	Strain 11x10 ³	Strain 22x10 ³	Strain 33x10 ³	Strain 12x10 ³	Strain 13x10 ³	Strain 23x10 ³
1	0.0169166	0.1966553	0.0000000	0.8311109	0.0413899	-0.0437456
2	-0.0451244	0.0923501	0.0000000	0.7208089	0.0080931	-0.0303249
3	0.7155528	0.0923501	0.2344822	0.8254092	0.4109393	-0.0141154
4	-0.0015520	0.1966553	0.4797637	0.7056495	0.4402443	-0.0381634
5	0.0169166	0.1975856	0.0000000	0.7815843	0.0413899	0.0362150
6	-0.0451244	0.0950517	0.0000000	0.6881250	0.0080931	0.0087555
7	0.7599798	0.0950517	0.2344822	0.7940248	0.4545257	0.0249650
8	-0.0432838	0.1975856	0.4797637	0.6551777	0.3884818	0.0417972

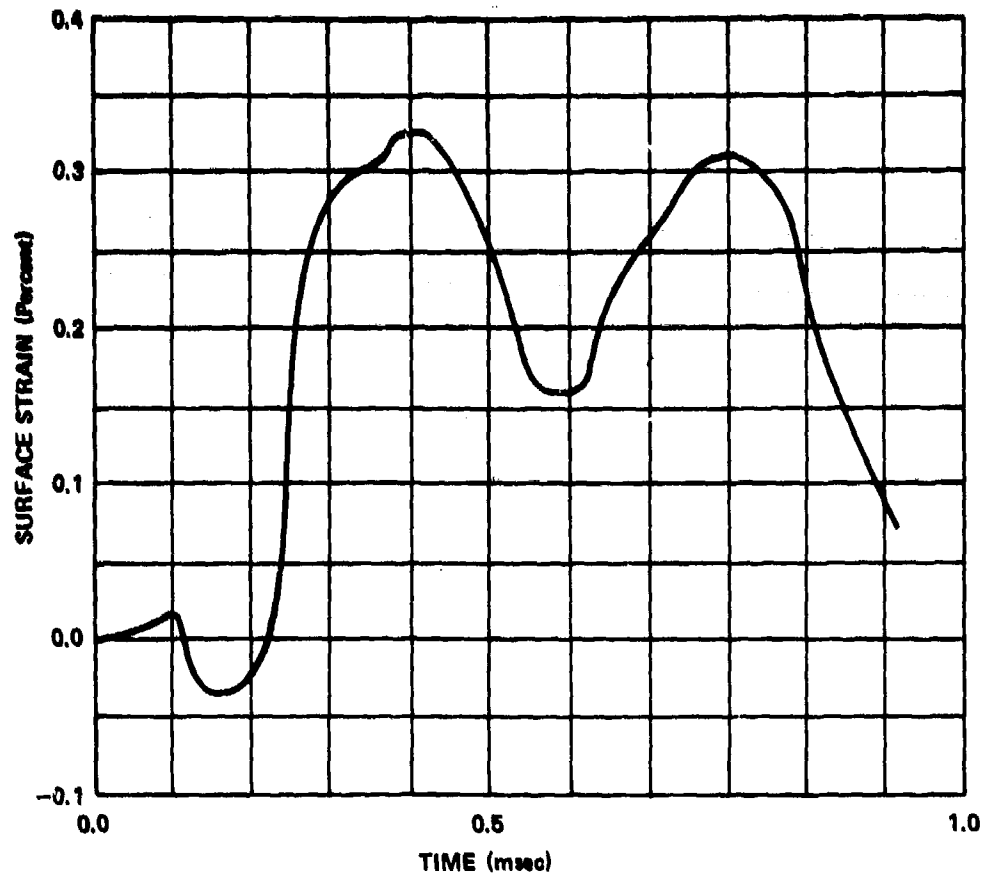


Figure 5 - Time History of Strains at Midpoint

CONCLUDING REMARKS

The computation and output of strains in the interior and on the surface of finite elements is of significance in engineering applications. The procedure outlined in this paper within the framework of ADINA allows for the computation and output of such strains in any arbitrary orientation. Results demonstrate the applicability and versatility of the method developed in this study for postprocessing of the strains.

ACKNOWLEDGMENT

The authors wish to thank Joan E. Roderick of the Center's Submarine Division for providing the results of the computer solution of the Clamped Plate corresponding to Example 3 of the numerical solutions.

REFERENCES

1. Bathe, K. J., "ADINA - A Finite Element Program for Automatic Dynamic Incremental Nonlinear Analysis," Report 82448-1, Acoustics and Vibration Laboratory, Department of Mechanical Engineering, Massachusetts Institute of Technology, (Sep 1975; revised Dec 1978).
2. Bathe, K. J. and S. Bolourchi, "A Geometric and Material Nonlinear Plate and Shell Element," Journal of Computers and Structures, Volume 11, No. 5 (Jun 1980).
3. Hinton, E. and J. S. Campbell, "Local and Global Smoothing of Discontinuous Finite Element Functions Using a Least Squares Method," International Journal for Numerical Methods in Engineering, Vol. 8, pp. 461-480 (1974).
4. Oden, J. T. and H. J. Brauchli, "On the Calculation of Consistent Stress Distributions in Finite Element Approximations," International Journal for Numerical Methods in Engineering, Vol. 3, pp. 317-325 (1971).
5. Eringen, A. C., "Nonlinear Theory of Continuous Media," pp. 54-55, McGraw-Hill (1962).

INITIAL DISTRIBUTION

Copies		CENTER DISTRIBUTION		
		Copies	Code	Name
1	DDR&E/Library			
1	DNA(SPSS)/D. Sabota	1	17	
1	ONR (474)	1	1706 (m)	
1	NAVMAT/Library	1	172	
1	USNA	1	172.3	J. E. Carlberg
1	USNA	1	173	
1	USNA	1	174	
3	NRL	1	175	
	1 8406/G. O'Hara	1	175.1	C. Stuber
	1 8422/H. Huang	1	175.2	B. Whang
	1 Library	1	175.2	W. Gilbert
1	NAVPGSCOL	1	175.2	T. Giacofci
1	NSWC/Library	1	175.2	S. Zilliacus
		1	175.2	P. N. Roth
		1	175.2	F. Fortin
7	NAVSEA	1	175.2	S. Walter
	1 SEA 08E	1	175.2	M. Hoffman
	1 SEA 32R	25	175.2	T. Toridis
	1 SEA 3221/H. Ward	1	175.3	W. Conley
	1 SEA 323			
	1 SEA 3232/D. Nichols	1	177	
	1 PMS 393	1	177.1	V. Bloodgood
	1 PMS 396	1	177.1	F. Cosinzo
12	DTIC	1	177.7 (m)	
1	General Dynamics/Electric Boat Div. ATTN: M. Pakstys, Dept. 443	10	5211.1	Reports Distribution
		1	522.1	Unclassified Lib (C)
1	PACIFICA TECHNOLOGY San Diego, CA ATTN: R. Dunham	1	522.2	Unclassified Lib (A)
1	PACIFICA TECHNOLOGY McLean, VA ATTN: A. Kushner			
1	WEIDLINGER ASSOCIATES New York Office ATTN: M. Baron			

DTNSRDC ISSUES THREE TYPES OF REPORTS

1. DTNSRDC REPORTS, A FORMAL SERIES, CONTAIN INFORMATION OF PERMANENT TECHNICAL VALUE. THEY CARRY A CONSECUTIVE NUMERICAL IDENTIFICATION REGARDLESS OF THEIR CLASSIFICATION OR THE ORIGINATING DEPARTMENT.

2. DEPARTMENTAL REPORTS, A SEMIFORMAL SERIES, CONTAIN INFORMATION OF A PRELIMINARY, TEMPORARY, OR PROPRIETARY NATURE OR OF LIMITED INTEREST OR SIGNIFICANCE. THEY CARRY A DEPARTMENTAL ALPHANUMERICAL IDENTIFICATION.

3. TECHNICAL MEMORANDA, AN INFORMAL SERIES, CONTAIN TECHNICAL DOCUMENTATION OF LIMITED USE AND INTEREST. THEY ARE PRIMARILY WORKING PAPERS INTENDED FOR INTERNAL USE. THEY CARRY AN IDENTIFYING NUMBER WHICH INDICATES THEIR TYPE AND THE NUMERICAL CODE OF THE ORIGINATING DEPARTMENT. ANY DISTRIBUTION OUTSIDE DTNSRDC MUST BE APPROVED BY THE HEAD OF THE ORIGINATING DEPARTMENT ON A CASE-BY-CASE BASIS.