COMPENDIUM

INTERACTIVE PREPROCESSORS FOR SAP IV, NASTRAN, AND ADINA IN THE TEACHING ENVIRONMENT

Movses J. Kaldjian

Departments of Naval Architecture and Marine Engineering, and Civil Engineering, University of Michigan, Ann Arbor, MI 48109-2145, U.S.A.

(Received 28 July 1988)

Abstract—Input data preparation in general is the main chore in finite element analysis. Interactive preprocessors can provide the engineer with an easy guide and efficient method of communicating the mathematical model to the computer. Essentially the same modeling information has to be supplied to all finite element programs. A preprocessor can be made to accommodate internally the input data variations peculiar to each finite element program. Learning to model on one finite element program may then be adequate to allow the engineer to prepare data for other finite element programs through a well planned preprocessor. Three such preprocessors, PREMSAP, PNASTRAN and PADINA, have been developed in Fortran for SAP IV, NASTRAN and ADINA respectively. Typical interactive conversations of these, and their output, are presented here for a 3D bracket, and may be compared. These preprocessors, coupled with a graphics package, proved very useful in training students in finite element analysis in class study, as well as with engineers and researchers in their work.

PREPROCESSORS

Input data preparation in general is the main chore in finite element modeling and analysis, and may account for 80% or more of the total analysis cost. Interactive preprocessors can provide the engineer with an easy guide and efficient method of communicating his mathematical model to the computer by avoiding unnecessary mistakes and reducing time consuming verifications.

Essentially the same modeling information has to be supplied to all finite element programs. Preprocessors can be made to accommodate internally the input data variations peculiar to each finite element program. Learning to model on one finite element program may then be adequate to allow the engineer to prepare data for other finite element programs through a well planned preprocessor.

Input data may be prepared with an interactive preprocessor on micro computers, which are readily available and inexpensive, and then this data may be verified graphically for accuracy with a graphics package before running it on micro or mainframe computers to obtain the finite element results.

Three such preprocessors, PREMSAP [1–3], PNASTRAN [3, 4] and PADINA, have been written in Fortran for SAP IV [5], NASTRAN [6] and ADINA [7] respectively for linear analysis. Typical interactive conversations of these programs and their output are shown below for a 3D bracket problem, and may be compared.

No attempt is made here to describe SAP IV, NASTRAN or ADINA’s program elements components nor their input data statements. For this the reader is referred to the respective manuals whose references appear at the end of this text. It is of course recommended, though not essential for simple problems, to read the manuals and get acquainted with finite element techniques prior to running the appropriate preprocessor on the computer.

The preprocessors prompt the engineer in very similar language. They all have nodal and element mesh generation, as well as 2D and 3D options with or without rotation constraints. The user is prompted each step of the way and needs to enter the requested information format freely.

After the user has defined his problem with proper sketches and dimensions, from a terminal he calls on the preprocessor he wants to use. This can be accessed on the Michigan Terminal System (MTS) by typing the appropriate command shown in the Appendix.

The name of the file to store the input data for the finite element program is created internally, and the data are saved within it according to the appropriate format of the particular finite element program being used.

Corrections to the input data file may be made by rerunning the preprocessors or by editing this file directly. Nonlinear parameters and/or bulk data may be added to the data file as desired.

The process of requesting information by the computer and the user’s response to it continues until the last bit of information necessary to complete the input data for MSAP is accomplished.

These preprocessors, when coupled with a graphics package [1, 2], have been shown to be very useful in training students in finite element analysis in class
study, as well as with engineers and researchers in their work. The computer knowledge required of them is thus reduced to a minimum.

It is easy to extend these preprocessors to accommodate other finite element programs. Use of these preprocessors by students, researchers and industry has proved to be very satisfying.

**ILLUSTRATED EXAMPLE**

Input data for finite element analysis of a 2 x 3 x 6 in. aluminum bracket, shown in Fig. 1, is to be prepared for: Y.M. = 10,000 ksi (68.95 GPa), v = 0.3, P = 8 kip (35.584 kN), and a = 3 in. (76.2 mm), b = 3 in. (76.2 mm), c = 2 in. (50.8 mm), and h = 1 in. (25.4 mm).

Using the preprocessors discussed above, appropriate input data for

(a) MSAP (SAP IV),
(b) NASTRAN and
(c) ADINA

have been prepared interactively, and are presented below for comparison.

**COMPUTER CONVERSATION**

The interactive computer conversation of the preprocessors PREMSAP, PNASTRAN, and PADINA to obtain input data for the bracket shown in Fig. 1 follows next, along with SAP IV, NASTRAN and ADINA analysis results of the same.
Interactive preprocessors for SAP IV, NASTRAN and ADINA

**AND NODE TEMPERATURE**
1: 0.6, 1.0, 0, 0, 0, 0
2: -0.6, 1.0
3: -3, 6.2
4: 0, 6.2
5: 0, 3, 0
6: -3, 3.0
7: -3, 3.2
8: 0, 3.2
9: 0.0, 0, 1, 1, 1
10: -3, 0.0, 1, 1, 1
11: -3, 0.2, 1, 1, 1
12: 0, 0.2, 1, 1, 1
13: -1.5, 6.1
14: -3.6, 1.3
15: -1.5, 6.2
16: 0.6, 1.5
17: -1.5, 0.0, 1, 1, 1
18: -3, 0.1, 1, 1, 1
19: -1.5, 0.2, 1, 1, 1
20: 0.6, 1.1

**ELEMENT DATA**

**ENTER ELEMENT TYPE NUMBER**

1 = TRUSS, 2 = BEAM,
3 = MEMBRANE (3D), 4 = PLANE STRESS, STRAIN, OR AXISYM.,
5 = 8-NODE-BRICK (3D), 6 = PLATE AND THIN SHELL,
7 = BOUNDARY ELEM, 8 = VARIABLE-NO. THICK SHELL,
9 = CONTACT ELM, 10 = OLD 16-NOD-BRICK,
12 = PIPE ELEMENT.

8

**ENTER THE FOLLOWING 6 ITEMS**
...NUMBER OF ELEMENTS, NUMBER OF DIFFERENT MATERIALS,
...MAX NUMBER OF TEMP. POINTS FOR ANY MATERIAL,
...NUMBER OF DIFFERENT DISTRIBUTED LOAD SETS,
...MAX NUMBER OF NODES TO DESCRIBE AN ELEMENT, AND
...NUMBER OF SETS OF DATA REQUESTING STRESS OUTPUT...
...AT VARIOUS LOCATIONS.
... (ON ONE LINE SEPARATED BY COMMA)
? 1, 1, 1, 0, 20, 1

...FOR EACH MATT ENTER NUMBER OF DIFFERENT TEMPERATURES
...AT WHICH MATT PROPERTIES ARE SPECIFIED,
...AND WEIGHT & MASS DENSITIES
1: 1

> FOR EACH DFF. TEMP FOR THIS MATERIAL, ENTER
> TEMPERATURE, MODULUS OF ELASTICITY, POISSON'S RATIO,
> AND THERMAL EXPANSION COEFFICIENT.
> 0.10000, .3

...SPECIFY UP TO SEVEN STRESS LOCATIONS
...FOR EACH OUTPUT SET (FROM 1 TO 27)
1: 3, 4, 7, 8, 10, 12, 14

...ARE THERE GRAVITY, THERMAL, OR DISTRIBUTED LOADS
...APPLIED? (ENTER Y OR N)
? N

**IS LINEAR ELEMENT MESH GENERATION**

**TO BE PERFORMED? (ENTER Y OR N)**
? N

**ENTER THE FOLLOWING 7 ITEMS PER ELEMENT**
...NUMBER OF NODES DESCRIBING THE ELEMENT DISPLACEMENTS,
...NUMBER OF NODES TO DESCRIBE ELEMENT GEOMETRY,
...MATERIAL ID NUMBER,
...STRESS OUTPUT SET NUMBER,
...STRESS FREE TEMPERATURE,
...STIFFNESS SAME AS PRECEDING ELEMENT? (0 = NO, 1 = YES),
...AND PRESSURE SET NUMBER
1: 20, 20, 1, 0, 0, 0

**LIST DESCRIBING NODE NUMBERS**
1: 3, 13, 15, 4, 2, 10, 9, 1, 7, 19, 6, 15, 6, 17, 5, 13, 14, 18, 20, 16

**CONCENTRATED LOAD DATA**

**ENTER NUMBER OF JOINTS WITH CONCENTRATED LOADS**

1

**FOR EACH LOADED JOINT ENTER FOLLOWING PER LINE**

**JOINT NO. (IN INCRASING SEQ.), AND LOADS IN X-, Y-, Z-DIR.
1: 4, 0.0, -8"
## Example 6

**Bracket 3/D Multi-Nodal Points**

### Node Displacements / Rotations

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>Load Case</th>
<th>X-Translation</th>
<th>Y-Translation</th>
<th>Z-Translation</th>
<th>X-Rotation</th>
<th>Y-Rotation</th>
<th>Z-Rotation</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>18</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>17</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>16</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>15</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

### Node Solid Element Stress

<table>
<thead>
<tr>
<th>Element</th>
<th>Load Location</th>
<th>SIG-XX</th>
<th>SIG-YY</th>
<th>SIG-ZZ</th>
<th>SIG-XY</th>
<th>SIG-YZ</th>
<th>SIG-ZX</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>0.852499E+01</td>
<td>0.198916E+02</td>
<td>0.852499E+01</td>
<td>0.371079E+01</td>
<td>-0.182331E+01</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>0.973166E+01</td>
<td>0.227072E+02</td>
<td>0.973166E+01</td>
<td>0.923149E+01</td>
<td>-0.453165E+01</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>-0.105657E+02</td>
<td>-0.245837E+02</td>
<td>-0.105657E+02</td>
<td>-0.164965E+01</td>
<td>-0.212358E+01</td>
<td>0.0</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>-0.145832E+02</td>
<td>-0.366024E+02</td>
<td>-0.145832E+02</td>
<td>-0.164965E+01</td>
<td>-0.212358E+01</td>
<td>0.0</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>-0.368208E+00</td>
<td>0.830203E+01</td>
<td>-0.368208E+00</td>
<td>0.369464E+01</td>
<td>0.173997E+01</td>
<td>0.0</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>0.164817E+01</td>
<td>0.131731E+02</td>
<td>0.164817E+01</td>
<td>0.358456E+01</td>
<td>0.312967E+01</td>
<td>0.123658E+01</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>-0.685221E+00</td>
<td>-0.702557E+00</td>
<td>-0.685221E+00</td>
<td>0.328581E+00</td>
<td>0.787253E+00</td>
<td>0.599738E+00</td>
</tr>
</tbody>
</table>
Interactive preprocessors for SAP IV, NASTRAN and ADINA

RUN CENA: NASTRAN

1. **EXECUTIVE CONTROL DECK:**
   - Enter Problem ID Name in two words (each 8 character or less) separated by a comma.
   - THICK, SHELL

2. **CASE CONTROL DECK:**
   - Enter a descriptive TITLE
   - TITLE = BRACKET 3/D (MULT-NODAL-POINT) NASTRAN
   - Is the problem being analyzed 2-dimensional?
   - Note: Plates with bending are considered 3-d.
   - Enter Y for yes, or N for no
   - N

3. **ARE THERE BENDING MOMENT RESISTING MEMBERS?**
   - (Such as beam, plate, etc.)? Enter Y or N.
   - N

4. **DEFINING THE GRID POINTS:**
   - Is auto grid point (node) mesh generation to be performed? Enter Y or N.
   - N

5. **FOR EVERY GRID POINT (JOINT):** Enter per line, grid ID, and the X, Y, and Z coordinates.
   - To correct an error: re-enter "GRID ID" and the COORD/S.
   - To delete a line: enter "-GRID ID", and ZEROS for the coordinates (any number will do).

   Example:
   ```
   1, 0, 6, 6
   2, -3, 6, 6
   3, -3, 6, 2
   4, 0, 6, 2
   5, 0, 3, 0
   6, -3, 3, 0
   7, -3, 3, 2
   8, 0, 3, 2
   9, 0, 0, 0
   10, -3, 0, 0
   11, -3, 0, 2
   12, 0, 0, 2
   13, -1, 5, 6, 1
   14, -3, 6, 1, 5
   15, -1, 5, 6, 2
   16, 0, 6, 1, 5
   17, -1, 5, 0, 0
   18, -3, 0, 1
   19, -1, 5, 0, 2
   20, 0, 6, 1
   ```

   **GRID POINT (JOINT) CONSTRAINTS:**
   - Enter grid ID and up to three digit constraint (1 for X-DIR, 2 for Y-DIR, and 3 for Z-DIR - without blanks or commas in between - i.e. 123).
   - Example:
     ```
     9, 123
     10, 123
     11, 123
     12, 123
     17, 123
     18, 123
     19, 123
     20, 123
     ```

6. **DEFINING THE ELEMENTS:**
   - **CONNECTIVITY**

   - Enter element type (number):
     1 = BEAM
     2 = PLANE STRESS
     3 = PLANE STRESS
     4 = PLATE
     5 = SOLID
     6 = SOLID
     7 = SOLID
     8 = SOLID
     9 = SOLID
     10 = SOLID
     11 = SOLID
     12 = SOLID
     13 = SOLID
     14 = SOLID
     15 = SOLID
     16 = SOLID
     17 = SOLID
     18 = SOLID
     19 = SOLID
     20 = SOLID

   - Example:
     ```
     1 = BEAM
     2 = PLANE STRESS
     3 = PLANE STRESS
     4 = PLATE
     5 = SOLID
     ```

   **IS AUTO ELEMENT (CONNECTIVITY) MESH GENERATION**
**TO BE PERFORMED FOR THIS ELEMENT TYPE? ENTER Y OR N ? N**

***ELEMENT DATA FOR: SOLID ELEMENT "CHEXA"***

*FOR EACH PROPERTY OF THIS ELEMENT TYPE, ENTER PROPERTY ID, AND MATERIAL ID.  
>>TO CORRECT AN ERROR: RE-ENTER "PROPERTY ID", AND MATERIAL ID.  
>>TO DELETE A LINE: ENTER "-PROPERTY ID", AND A ZERO (OR ANY OTHER NUMBER) FOR MATERIAL ID.  
*TERMINATE THIS SEQUENCE BY TYPING: $ENDFILE (OR CTRL-C) ? 21, 11  
? \ 

*ENTER THE YOUNG'S MODULUS, AND POISSON'S RATIO FOR MATERIAL ID:  
? 11= 10000, 0.3  

*FOR EACH CHEXA ELEMENT, ENTER ELEMENT ID, AND ITS 20 GRID ID'S (CONNECTIVITY POINTS - USE THE RIGHT HAND RULE).  
>>TO CORRECT AN ERROR: RE-ENTER "ELEMENT ID", AND THE 20 GRID ID'S.  
>>TO DELETE A LINE: ENTER "-ELEMENT ID", AND ZEROS (OR ANY OTHER NUMBER) FOR THE 20 GRID ID'S.  
*TERMINATE THIS SEQUENCE BY TYPING: $ENDFILE (OR CTRL-C) ? 1, 4,3,11,12, 1,2,10,9, 15,7,19,8, 16,14,18,20, 13,6,17,5  
? \ 

**ARE THERE OTHER ELEMENT TYPES YET TO BE ENTERED IN THIS PROBLEM? (Y OR N) ? N**

***LOAD SUBCASES***

*ENTER THE NUMBER OF LOAD SETS TO BE APPLIED.  
? 1  
*ENTER TITLE FOR SUBCASE 1  
? POINT FORCE  

***CONCENTRATED FORCES ON GRID POINTS:***

>>TO CORRECT AN ERROR: RE-ENTER "GRID ID", AND THE FORCE COMPONENTS.  
>>TO DELETE A LINE: ENTER "-GRID ID", AND ZEROS (OR ANY OTHER NUMBER) FOR ITS FORCE COMPONENTS.  
*TERMINATE THIS SEQUENCE BY TYPING: $ENDFILE (OR CTRL-C)  
FOR SUBCASE( 1)  
SUBTITLE: POINT FORCE  
? 4, 0,0,-8  
? \ 

#LIST EXPL. EN  (DATA FOR NASTRAN)  
ID THICK, SHELL  
TIME 1  
SOL 24  
$DIAG 14  
CEND  
TITLE=BRACKET 3/D  (MULT-NODAL-POINT)  NASTRAN  
ECK=SUBME  
DISPLACEMENT=ALL  
ELFORCE=ALL  
SPCLOADS=ALL  
SUBCASE 1  
LOAD= 61  
SUBTITLE= POINT FORCE  
OUTPUT(FIGURE) PLOTTER NASTRAN  
SET 53 INCLUDE ALL  
AXES Z,X,Y  
VIEW 0,0,0.  
FIND  
PLOT LABEL BOTH  
PLOT STATIC DEFORMATION 0 SET 53  
BEGIN BULK  
PARAM AUTOSFC YES  
GNDSET  
GRID 1 0.0 6.00 1.00  
GRID 2 -3.00 6.00 1.00  
GRID 3 -3.00 6.00 2.00  
456
Interactive preprocessors for SAP IV, NASTRAN and ADINA 891

GRID 4  0.0  6.00  2.00
GRID 5  0.0  3.00  0.0
GRID 6 -3.00  3.00  0.0
GRID 7 -3.00  0.00  2.00
GRID 8  0.0  3.00  2.00
GRID 9  0.0  0.00  0.0  123456
GRID 10 -3.00  0.00  2.00  123456
GRID 11 -3.00  0.00  2.00  123456
GRID 12 -3.00  0.00  2.00  123456
GRID 13  1.50  6.00  1.00
GRID 14  6.00  6.00  1.50
GRID 15  1.50  6.00  2.00
GRID 16  0.00  6.00  1.50
GRID 17 -3.00  0.00  1.00  123456
GRID 18 -3.00  0.00  1.00  123456
GRID 19 -1.50  0.00  2.00  123456
GRID 20  0.00  0.00  1.00  123456
CHEXA 1  2  4  3  11  12  1
+A1001
+BI001
+BI001

EXAMPLE SN
EXTRACT 9/D (MULTI-NODAL-POINT) (EXP 6 SOLVED WITH NASTRAN)

POINT ID. TYPE T1 T2 T3 R1 R2 R3
1 G -1.668535E-03 8.514361E-04 -2.769238E-02 0.0 0.0 0.0
2 G -7.478641E-04 5.756649E-03 -1.602456E-02 0.0 0.0 0.0
3 G 2.492231E-03 4.663334E-02 -1.609490E-02 0.0 0.0 0.0
4 G 4.233173E-03 6.719043E-03 -2.922228E-02 0.0 0.0 0.0
5 G -8.068213E-03 -3.449236E-03 9.643834E-03 0.0 0.0 0.0
6 G -1.735502E-03 -2.585230E-03 -5.102348E-03 0.0 0.0 0.0
7 G 2.116611E-03 3.491387E-03 -5.220585E-03 0.0 0.0 0.0
8 G 1.309823E-03 4.490137E-03 -9.778079E-03 0.0 0.0 0.0
9 G 0.00 0.00 0.00 0.0 0.0 0.0
10 G 0.00 0.00 0.00 0.0 0.0 0.0
11 G 0.00 0.00 0.00 0.0 0.0 0.0
12 G 0.00 0.00 0.00 0.0 0.0 0.0
13 G -1.270064E-03 4.632283E-04 -2.199817E-02 0.0 0.0 0.0
14 G 0.00 0.00 0.00 0.0 0.0 0.0
15 G 0.00 0.00 0.00 0.0 0.0 0.0
16 G 0.00 0.00 0.00 0.0 0.0 0.0
17 G 0.00 0.00 0.00 0.0 0.0 0.0
18 G 0.00 0.00 0.00 0.0 0.0 0.0
19 G 0.00 0.00 0.00 0.0 0.0 0.0
20 G 0.00 0.00 0.00 0.0 0.0 0.0

FORCES OF SINGLE-POINT CONSTRAINT
POINT ID. TYPE T1 T2 T3 R1 R2 R3
9 G -1.133974E+00 8.058518E+00 1.060886E+01 0.0 0.0 0.0
10 G 2.712208E+00 6.698411E+00 9.637033E+00 0.0 0.0 0.0
11 G -1.973005E+01 -1.232502E+00 6.225642E+00 0.0 0.0 0.0
12 G 3.722500E+00 -1.502029E+00 1.077604E+01 0.0 0.0 0.0
13 G -3.822497E+01 1.515567E+01 5.657792E+00 0.0 0.0 0.0
14 G -3.198428E+00 -5.691678E+00 -2.608670E+00 0.0 0.0 0.0
15 G -3.593664E+00 -5.130321E+01 -1.023325E+02 0.0 0.0 0.0
16 G -1.409033E+00 -6.533518E+00 -2.111522E+01 0.0 0.0 0.0

STRESSES IN HEXAHEDRON SOLID ELEMENTS (HEXA)

<table>
<thead>
<tr>
<th>ELEMENT-ID</th>
<th>GRID-ID</th>
<th>CORNER</th>
<th>NORMAL</th>
<th>PRESSURE</th>
<th>OCTAHEDRAL SHEAR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1.357754E+00</td>
<td>-8.297583E-03</td>
<td>A 4.113112E+00</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>1</td>
<td>2.893377E+00</td>
<td>1.494671E+00</td>
<td>B 9.804267E-01</td>
</tr>
<tr>
<td>3</td>
<td>1</td>
<td>1</td>
<td>4.600604E+00</td>
<td>5.231312E+00</td>
<td>A 4.590468E+00</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>2</td>
<td>5.231312E+00</td>
<td>3.610939E+00</td>
<td>B -9.303792E+01</td>
</tr>
<tr>
<td>5</td>
<td>2</td>
<td>2</td>
<td>6.407991E+00</td>
<td>1.634316E+00</td>
<td>C -6.543051E+00</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>3</td>
<td>4.963511E+00</td>
<td>3.539484E+00</td>
<td>A 6.903331E+00</td>
</tr>
<tr>
<td>7</td>
<td>3</td>
<td>3</td>
<td>2.894227E+00</td>
<td>1.246356E+00</td>
<td>B -3.482079E+00</td>
</tr>
<tr>
<td>8</td>
<td>3</td>
<td>3</td>
<td>9.228915E+00</td>
<td>3.966180E+00</td>
<td>C 8.497147E+00</td>
</tr>
<tr>
<td>9</td>
<td>4</td>
<td>4</td>
<td>9.228915E+00</td>
<td>3.966180E+00</td>
<td>A 2.238643E+01</td>
</tr>
<tr>
<td>10</td>
<td>4</td>
<td>4</td>
<td>2.117838E+00</td>
<td>5.383371E+00</td>
<td>B 5.662955E+00</td>
</tr>
<tr>
<td>11</td>
<td>5</td>
<td>5</td>
<td>9.587667E+00</td>
<td>3.070301E+00</td>
<td>C 1.193318E+01</td>
</tr>
<tr>
<td>12</td>
<td>5</td>
<td>5</td>
<td>1.046385E+00</td>
<td>4.559531E+00</td>
<td>A 2.493385E+01</td>
</tr>
<tr>
<td>13</td>
<td>5</td>
<td>5</td>
<td>2.414188E+00</td>
<td>3.066725E+00</td>
<td>B 3.086725E+00</td>
</tr>
<tr>
<td>14</td>
<td>6</td>
<td>6</td>
<td>1.070228E+00</td>
<td>5.423104E+01</td>
<td>C 1.066668E+01</td>
</tr>
</tbody>
</table>
**RUN-CNMA-PADNA**

*> ENTER FILE NAME TO STORE ADINA DATA (7 CHARACTERS OR LESS)
? EXPL. SA

*> ENTER PROBLEM TITLE ON ONE LINE.
? 3/D SOLID

*********** MASTER CONTROL CARDS ***********

* FOR STRUCTURE CONTROL CARD, ENTER THE FOLLOWING ON ONE LINE :
> (1) TOTAL NO. OF NODAL POINTS,
> (2) NO. OF LINEAR ELEMENT GROUPS,
> (3) PROBLEM TYPE (0= 2/DIMENSIONAL, 1= 3/DIMENSIONAL),
> (4) ARE THERE MOMENT-RESISTING MEMBERS (SUCH AS BEAM, SHELL, ETC) IN THIS PROBLEM ? (O= Y, 1= N), &
> (5) MODE ANALYSIS TYPE NO. (0= STATIC, 1= DYNAMIC)
? 20,1,1,1,0

* FOR LOAD CONTROL CARD, ENTER THE FOLLOWING ON ONE LINE :
> (1) NO. OF CARDS FOR CONCENTRATED LOADS,
> (2) NO. OF 2/D ELEMENT PRESSURE SURFACES,
> (3) NO. OF 3/D ELEMENT PRESSURE SURFACES,
> (4) NUMBER OF PRESCRIBED DISPLACEMENTS,
> (5) NO. OF CARDS USED TO PRESCRIBE NODAL TEMPERATURES.
? 1,0,0,0,0

*********** NODAL POINT DATA ***********

* FOR EACH NODAL POINT, ENTER THE FOLLOWING PER LINE :
> (1) NODE NO.,
> (2-4) X-, Y-, AND Z-COORD.,
> (5-7) X-, Y-, AND Z-TRANS. BOUNDARY CODE, ( 0= FREE, 1= CONSTRAINED )
> (8) NODE NO. INCREMENT FOR MESH GENERATION.
** TO CORRECT AN ERROR: RE-ENTER THE ENTIRE LINE.
** TO DELETE A LINE: ENTER "-NODE NO.", AND ZEROS.
** TERMINATE THIS SEQUENCE BY TYPING SENDFILE (OR CNTR-c).
?

*********** ELEMENT DATA ***********

* ENTER ELEMENT TYPE NO. FOR LINEAR ANALYSIS :
> (1) TRUSS,
> (2) 2/D PLANE/SOLID,
> (3) 3/D SOLID,
> (4) SHELL,
? 3

*********** 3/D SOLID ELEMENTS ***********

* FOR 3/D SOLID ELEMENTS, ENTER THE FOLLOWING ON ONE LINE :
> (1) NO. OF ELEMENTS,
> (2) MAX. NO. OF NODES USED TO DESCRIBE ANY ONE ELEMENT,
> (3) NO. OF STRESS OUTPUT TABLES (AT INTEG.PTS. = ZERO),

---

### Raw Text Representation

1. **X** -3.606999E+00 **XY** -5.17410E+00 **A** 5.57757E+00 **LX** 0.38-0.52-0.77 2.713829E+00 6.924195E+00
2. **X** -1.876137E+00 **Y** 6.141906E+00 **B** -1.137592E+01 **LY** 0.84-0.58-0.01
3. **X** 4.92548E-01 **Y** 5.92548E-01 **C** 2.184565E-01 **LY** 0.27-0.25-0.23
4. **X** -8.364594E+00 **Y** -1.983794E+00 **B** -2.108080E+01 **LY** 0.08 0.94 0.32
5. **X** -2.343137E+00 **Y** 1.593506E+00 **C** 2.184956E+00 **LY** 1.59 6.59 3.16
6. **X** 4.04066E+00 **Y** 1.452724E+00 **A** 4.040660E+00 **LY** 0.49-0.44-0.64 1.208219E+01
7. **X** 1.387618E+00 **Y** 1.387618E+00 **B** 5.032963E+00 **LY** 0.87 0.09 0.48

---

### Note

The text contains instructions and data entries for running an engineering simulation or analysis using ADINA software. The entries include control cards, nodal point data, element data, and other parameters necessary for the simulation process. The data is presented in a structured format with specific entries for coordinates, boundary conditions, and problem descriptions. The text is likely part of a larger report or document related to engineering analysis. The instructions guide the user through the setup and execution of the simulation, including how to input different types of data and control parameters. This type of documentation is common in engineering and scientific research, particularly in fields requiring complex simulations and analyses.
Interactive preprocessors for SAP IV, NASTRAN and ADINA

>> (4) NO. OF DIFFERENT SETS OF MATERIAL PROPERTIES.
? 1, 1,20, 1,1

* FOR EACH MATERIAL PROPERTY, ENTER THE FOLLOWING PER LINE :
>> (1) MATERIAL PROPERTY SET NO., (2) MASS DENSITY,
>> (3) YOUNG'S MODULUS, (4) POISSON'S RATIO.
** IF THE LINE INPUT IS ERRONEOUS, ENTER AGAIN.
** TERMINATE THIS SEQUENCE BY TYPING SENDFILE (OR CNTR-C).
? 1 , 0 10000 .3

* STRESS OUTPUT LOCATION POINTS ON ELEMENT:
>> ENTER TABLE NO. & NODE POINTS FOR DESIRED STRESSES.
>> NOTE: SEE ADINA FIG. XIII.7 FOR LOCATION NUMBERS.
** IF THE LINE INPUT IS ERRONEOUS, ENTER AGAIN.
** TERMINATE THIS SEQUENCE BY TYPING SENDFILE (OR CNTR-C).
? 1, 3 7 8 10 12 14 16

* FOR EACH 3/D SOLID ELEMENT, ENTER THE FOLLOWING PER LINE :
>> (1) ELEMENT NO., (2) NO. OF NODES,
>> (3) STRESS OUTPUT TABLE NO.,
>> (4) MATERIAL PROPERTY SET NO.,
>> (5-25) GLOBAL NODE NO. OF ELEMENT NODAL POINT 1 TO 21,
>> (26) NODE GENERATION INCREMENT.
** TO CORRECT AN ERROR: RE-ENTER THE ENTIRE LINE.
** TO DELETE A LINE: ENTER "-ELEMENT NO.=, AND ZEROS.
** TERMINATE THIS SEQUENCE BY TYPING SENDFILE (OR CNTR-C).
? 1, 20 1 1, 4 3 11 12, 1 4 10 9, 1 7 19 8, 13 1 17 5, 16 14 18 20 0, 0

********* APPLIED LOAD DATA *********
* FOR CONCENTRATED LOADS, ENTER THE FOLLOWING 5 ITEMS PER LINE:
>> (1) LOADING CARD NO.,
>> (2) NODE NO. TO WHICH THIS LOAD IS APPLIED,
>> (3) DOF NO. FOR THIS LOAD ~~~~~
>> (4) FUNCTION MULTIPLIER,
>> (5) NODE NO. INCREMENT FOR GENERATION.
** IF THE LINE INPUT IS ERRONEOUS, ENTER AGAIN.
** TERMINATE THIS SEQUENCE BY TYPING SENDFILE (OR CNTR-C).
? 1, 4,3,- 8,0

#Execution terminated

#LIST EXPL.6A (DATA FOR ADINA)
3/D SOLID
20000111 1 0 1 1 1. 0. 1 0 0 0
0 0 0 0
1 0 0 0 0 0 0 0 0
0 0 0
0 0 1 0 0 0 0 0 0 0.
0 0 0 0 0 0 0 .0 .0.
0 0 0 0 0 0 0 0 .0 .0.
0 0 0 0 0 0 0 0 0 0
1 2
1 2
0 0 1 1 1 1 .0. 1 0 0 0
0 0 0 0 1 1 1 1 -3.000 6.000 1.000 0 0 0
3 0 0 0 1 1 1 1 -3.000 6.000 1.000 0 0 0
4 0 0 0 1 1 1 1 0.0 6.000 2.000 0 0 0
5 0 0 0 1 1 1 1 0.0 3.000 0.0 0 0 0
6 0 0 0 1 1 1 1 -3.000 3.000 0.0 0 0 0
7 0 0 0 1 1 1 1 -3.000 3.000 2.000 0 0 0
8 0 0 0 1 1 1 1 0.0 3.000 2.000 0 0 0
9 1 1 1 1 1 1 0.0 0.0 0 .0 .0
10 1 1 1 1 1 1 -3.000 0.0 0 .0 .0
11 1 1 1 1 1 1 -3.000 0.0 0 .0 .0
12 1 1 1 1 1 1 0.0 0.0 2.000 0 0 0
13 0 0 0 1 1 1 1 -1.500 6.000 1.000 0 0 0
14 0 0 0 1 1 1 1 -3.000 6.000 1.500 0 0 0
15 0 0 0 1 1 1 1 -1.500 6.000 2.000 0 0 0
16 0 0 0 1 1 1 1 0.0 6.000 1.500 0 0 0
17 1 1 1 1 1 1 1 -1.500 0.0 0 .0 .0
18 1 1 1 1 1 1 -3.000 0.0 1.000 0 0 0
19 1 1 1 1 1 1 -1.500 0.0 2.000 0 0 0
20 1 1 1 1 1 1 0.0 0.0 1.000 0 0 0
3 1 0 0 0 0 0 20 3 3 1 1 1 0 0 0
1 0 0
0.100E+05 0.300E+00
3 4 7 8 10 12 14 16
Total computation time comparison (CPU, sec) for 3D Bracket (Mult.--Node): MSAP, 0.25; NASTRAN, 4.49; ADINA, 0.18.

REFERENCES


APPENDIX

How to run the following programs, PREMSAP, MSAP and MSAPLOT, also PNASTRAN/ NASTRAN and PADINA/ADINA on the University of Michigan Terminal System (MTS).

MSAP

To prepare finite element data interactively:

1. $RUN CENA:PREMSAP (and obtain input datafile, call it MDATA)

2. $RUN CENA:MSAP 5=MDATA 6=RESULT

3. $TRUNCATE -DEF (the deflection file -DEF is created by MSAP)

4. %RENAME -DEF MDATA

5. %CONTROL *PRINT* PRINTER = LINE ROUTE = CNTR COPIES = 2

6. %LIST MDATA *PRINT*

7. %COPY -RESULT *PRINT*

**MSAPLOT**

To obtain computer plots:
8. \$RUN CENA:MSAPLOT (on TEKTRONIX or Macintosh terminals) (You would need datafile MDATA, and deflection file MDATAD, and must follow the interactive instructions and menu list)

**NASTRAN**

To prepare finite element data interactively:
9. \$RUN CENA:PNASTRAN (and obtain input datafile, call it NDATA)

To obtain displacements and stresses (forces):
10. \$RUN NAST:NASTRAN SCARDS = NDATA SPRINT = RESULT

**ADINA**

To prepare finite element data interactively:
11. \$RUN CENA:PADINA (and obtain input datafile, call it ADATA)

To obtain displacements and stresses (forces):
12. \$RUN CENA:ADINA SCARDS = ADATA SPRINT = RESULT