Computers & Structures Vol. 33, No. 3, pp. 885–895, 1989
Printed in Great Britain.

0045-7949/89 $3.00 + 0.00

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 element 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 free.

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 processors, when coupled with a graphics package [1, 2], have been shown to be very useful in training students in finite element analysis in class

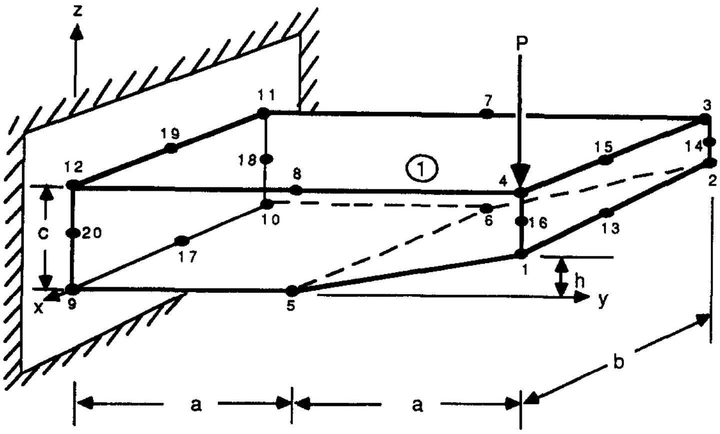

Fig. 1. Aluminum bracket.

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 $2 \times 3 \times 6$ in. aluminum bracket, shown in Fig. 1, is to be prepared for: $Y.M. = 10{,}000$ ksi (68.95 GPa), $\nu = 0.3$, $P = 8$ 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.

```
#RUN  CENA:PREMSAP

>> ENTER  ANALYSIS NUMBER (0= STATIC, 1= DYNAMIC)
   0

>> ENTER PROBLEM TITLE ON ONE LINE
   BRACKET 3/D   (MULT-NODAL-POINT)              EXPL.6

>> ENTER NUMBER OF JOINTS, NUMBER OF ELEMENT TYPES, AND
>> NUMBER OF LOAD CASES (ON ONE LINE SEPARATED BY COMMA)
   20,1,1

>> IS THE PROBLEM 2-DIMENSIONAL?
>> ENTER Y FOR YES OR N FOR NO
   N

>> ARE THERE BEAM, PIPE, THIN SHELL OR PLATE
>> ELEMENTS INVOLVED IN THE PROBLEM? (ENTER Y OR N)
   N

>> IS 2-D (QUADRILATERAL) NODAL POINT MESH
>> GENERATION TO BE PERFORMED? (ENTER Y OR N)
   N

>> IS LINEAR NODAL POINT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
   N

   *** JOINT DATA

>> FOR EACH JOINT ENTER FOLLOWING PER LINE
>> THE X-, Y-, Z-COORDS., BOUNDARY COND. CODES
>> (0 FOR FREE, 1 FOR CONSTRAINED) IN X- ,Y-, Z-DIR.,
>> COORD. TYPE (0 FOR RECTANGULAR, 1 FOR CYLINDRICAL),
```

```
>> AND NODE TEMPERATURE
    1:-   0,6,1, 0,0,0, 0
    2:-  -3,6,1
    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:-  -3,6,1.5
   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:-   0,0,1,   1,1,1

   *** 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-NODE THICKSHELL,
>>   9= CONTACT  ELEM,     10= OLD 16-NODE-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 MATL ENTER NUMBER OF DIFFERENT TEMPERATURES
...AT WHICH MATL PROPERTIES ARE SPECIFIED,
...AND WEIGHT & MASS DENSITIES
    1:  1

      > FOR EACH DIF. TEMP FOR THIS MATERIAL ENTER
      > TEMPERATURE, MODULUS OF ELASTICITY, POISSONS RATIO,
      > AND THERMAL EXPANSION COEFFICIENT.
         1>  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:  20,20,1,1,0,0,0
        *LIST DESCRIBING NODE NUMBERS
        ? 3,11,12,4, 2,10,9,1, 7,19,8,15, 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 INCREASING SEQ.), AND LOADS IN X-, Y-, Z-DIR.
    4, 0,0,-8
```

```
>> THE PROGRAM IS READY TO STORE THE DATA IN YOUR OWN FILE
>> ACCORDING TO THE INPUT FORMAT OF MSAP
>> ENTER YOUR OWN FILE NAME(8 CHARACTERS OR LESS)
   EXPL.6M

>> YOU ARE NOW READY TO OBTAIN YOUR RESULTS BY TYPING:
>> $LIST EXPL.6                                < LISTING
>> $RUN CENA:MSAP SCARDS=EXPL.6   SPRINT=-RESULT < ANALYSIS
>> $RUN CENA:MSAPLOT                           < DISPLAY
```

```
#LIST EXPL.6M      (DATA FOR MSAP)
BRACKET 3/D  MULT. NODAL POINTS          EXPL.6
  20    1     1    0    0    0
   1    0     0    0   -1   -1   -1     0.0       6.000     1.000     0      0.0
   2    0     0    0   -1   -1   -1    -3.000     6.000     1.000     0      0.0
   3    0     0    0   -1   -1   -1    -3.000     6.000     2.000     0      0.0
   4    0     0    0   -1   -1   -1     0.0       6.000     2.000     0      0.0
   5    0     0    0   -1   -1   -1     0.0       3.000     0.0       0      0.0
   6    0     0    0   -1   -1   -1    -3.000     3.000     0.0       0      0.0
   7    0     0    0   -1   -1   -1    -3.000     3.000     2.000     0      0.0
   8    0     0    0   -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      0.0
  10    1     1    1   -1   -1   -1    -3.000     0.0       0.0       0      0.0
  11    1     1    1   -1   -1   -1    -3.000     0.0       2.000     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    -1.500     6.000     1.000     0      0.0
  14    0     0    0   -1   -1   -1    -3.000     6.000     1.500     0      0.0
  15    0     0    0   -1   -1   -1    -1.500     6.000     2.000     0      0.0
  16    0     0    0   -1   -1   -1     0.0       6.000     1.500     0      0.0
  17    1     1    1   -1   -1   -1    -1.500     0.0       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
   8    1     1    1    0    0   20     1    3    3
   1    1 0.0          0.0
 0.0       1.000E+04 1.000E+04 1.000E+04 3.000E-01 3.000E-01 3.000E-01
 3.846E+03 3.846E+03 3.846E+03 0.0       0.0       0.0
   3    4     7    8   10   12   14
     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       0.0       0.0
   1   20    20    1    0    1 0.0         1    0    0    0 0 0 0 0
   4    3    11   12    1    2   10    9   15    7   19    8   13    6   17    5
  16   14    18   20    0
   4    1 0.0          0.0       -8.000E+00 0.0       0.0       0.0

     0.0       0.0       0.0       0.0
```

EXAMPLE 6 BRACKET 3-D MULT> NODAL POINTS

N O D E   D I S P L A C E M E N T S / R O T A T I O N S

NODE NUMBER	LOAD CASE	X- TRANSLATION	Y- TRANSLATION	Z- TRANSLATION	X- ROTATION	Y- ROTATION	Z- ROTATION
20	1	0.0	0.0	0.0	0.0	0.0	0.0
19	1	0.0	0.0	0.0	0.0	0.0	0.0
18	1	0.0	0.0	0.0	0.0	0.0	0.0
17	1	0.0	0.0	0.0	0.0	0.0	0.0
16	1	0.14281E-02	0.31968E-02	-0.30550E-01	0.0	0.0	0.0
15	1	0.35994E-02	0.63629E-02	-0.24155E-01	0.0	0.0	0.0
14	1	0.97794E-03	0.20974E-02	-0.17239E-01	0.0	0.0	0.0
13	1	-0.10236E-02	-0.56131E-03	-0.24065E-01	0.0	0.0	0.0
12	1	0.0	0.0	0.0	0.0	0.0	0.0
11	1	0.0	0.0	0.0	0.0	0.0	0.0
10	1	0.0	0.0	0.0	0.0	0.0	0.0
9	1	0.0	0.0	0.0	0.0	0.0	0.0
8	1	0.13852E-02	0.43833E-02	-0.96359E-02	0.0	0.0	0.0
7	1	0.22320E-02	0.34501E-02	-0.50260E-02	0.0	0.0	0.0
6	1	-0.16029E-02	-0.29156E-02	-0.48642E-02	0.0	0.0	0.0
5	1	-0.84729E-03	-0.40202E-02	-0.94254E-02	0.0	0.0	0.0
4	1	0.41312E-02	0.74123E-02	-0.31443E-01	0.0	0.0	0.0
3	1	0.31390E-02	0.49344E-02	-0.17259E-01	0.0	0.0	0.0
2	1	-0.11032E-02	-0.58570E-03	-0.17398E-01	0.0	0.0	0.0
1	1	-0.81531E-03	-0.56885E-03	-0.30128E-01	0.0	0.0	0.0

21 - N O D E   S O L I D   E L E M E N T   S T R E S S

ELEMENT	LOAD	LOCATION	SIG-XX	SIG-YY	SIG-ZZ	SIG-XY	SIG-YZ	SIG-ZX
1	1	1	0.852499E+01	0.198916E+02	0.852499E+01	0.371079E+01	-0.182331E+01	0.0
		2	0.973166E+01	0.227072E+02	0.973166E+01	0.903514E+00	-0.455165E+01	0.0
		3	-0.106507E+02	-0.248517E+02	-0.106507E+02	-0.340255E+01	-0.131951E+01	0.0
		4	-0.149153E+02	-0.348024E+02	-0.149153E+02	-0.164985E+01	-0.485444E+01	0.0
		5	-0.360208E+00	0.832034E+01	0.681567E+00	0.369457E+01	0.147388E+01	0.212200E+01
		6	0.168178E+01	0.131731E+02	0.104896E+01	0.335841E+01	-0.312963E+01	-0.123695E+01
		7	-0.685221E+00	-0.702557E+01	-0.222446E+01	-0.328581E+01	0.787253E+00	0.599737E+00

```
#RUN CENA:PNASTRAN

 <*> ENTER FILE NAME TO STORE NASTRAN DATA
    ? EXPL.6N

      ***EXECUTIVE CONTROL DECK:
 *ENTER PROBLEM ID NAME IN TWO WORDS
  (EACH 8 CHARACTER OR LESS) SEPARATED BY A COMMA.
    ?ID= THICK, SHELL

      ***CASE CONTROL DECK:
 *ENTER A DESCRIPTIVE TITLE
    ?TITLE= BRACKET 3/D   (MULT-NODAL-POINT)     NASTRAN

 <*> IS THE PROBLEM BEING ANALYSED 2-DIMENSIONAL?
 <*> NOTE: PLATES WITH BENDING ARE CONSIDERED 3-D.
 <*> (ENTER Y FOR YES, OR N FOR NO)
    ? N

 <*> ARE THERE BENDING MOMENT RESISTING MEMBERS
 <*> (SUCH AS BEAM, PLATE, ETC.)? ENTER Y OR N.
    ? N

      ***DEFINING THE GRID POINTS:     **NODES**


 <*> IS AUTO GRID POINT (NODE) MESH GENERATION
 <*> TO BE PERFORMED? ENTER Y OR N
    ? N

 *FOR EVERY GRID POINT (JOINT) ENTER PER LINE,
  GRID ID, AND THE X, Y, AND Z COORDIATES.
   >>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).
 *TERMINATE THIS SEQUENCE BY TYPING: $ENDFILE (OR CTRL-C)
    ?  1,  0,6,1
    ?  2, -3,6,1
    ?  3, -3,6,2
    ?  4   0,6,2
    ?  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,  0,0,1
    ? \

      ***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 ).
*TERMINATE THIS SEQUENCE BY TYPING: $ENDFILE (OR CTRL-C)
   ?  9,123
   ? 10,123
   ? 11,123
   ? 12,123
   ? 17,123
   ? 18,123
   ? 19,123
   ? 20,123
   ? \

     ***DEFINING THE ELEMENTS:    **CONNECTIVITY**


*ENTER THE ELEMENT TYPE (NUMBER):
     1 = BEAM . . . . . . . CBAR
     2 = PLANE STRESS . . . CQUAD4
     3 = PLANE STRESS . . . CQUAD8
     4 = PLATE. . . . . . . CQUAD4
     5 = SOLID. . . . . . . CHEXA20
   ? 5

<*> 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:

*FOR EACH FORCE LOAD ENTER PER LINE
 THE GRID ID WHERE THE LOAD IS APPLIED, AND ITS X, Y, AND Z
 COMPONENTS (I.E. GID, FX, FY, FZ).
  >>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.6N      (DATA FOR NASTRAN)
 ID THICK, SHELL
 TIME 1
 SOL 24
 $DIAG 14
 CEND
 TITLE=BRACKET 3/D   (MULT-NODAL-POINT)    NASTRAN
 ECHO=SORTED
 DISPLACEMENT=ALL
 ELFORCE=ALL
 SPCFORCES=ALL
 SUBCASE 1
 LOAD= 61
 SUBTITLE= POINT FORCE
 OUTPUT(PLOT)
 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   AUTOSPC   YES                                     456
 GRDSET
 GRID           1              0.0     6.00    1.00
 GRID           2            -3.00     6.00    1.00
 GRID           3            -3.00     6.00    2.00
```

```
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    3.00    2.00
GRID      8          0.0     3.00    2.00
GRID      9          0.0     0.0     0.0          123456
GRID     10         -3.00    0.0     0.0          123456
GRID     11         -3.00    0.0     2.00         123456
GRID     12          0.0     0.0     2.00         123456
GRID     13         -1.50    6.00    1.00
GRID     14         -3.00    6.00    1.50
GRID     15         -1.50    6.00    2.00
GRID     16          0.0     6.00    1.50
GRID     17         -1.50    0.0     0.0          123456
GRID     18         -3.00    0.0     1.00         123456
GRID     19         -1.50    0.0     2.00         123456
GRID     20          0.0     0.0     1.00         123456
CHEXA     1      21      4       3      11      12       1
+A1001
+A1001   10       9     15       7      19       8      16       +B1001
+B1001   18      20     13       6      17       5
PSOLID   21      11
MAT1     111.00E+04           0.300
FORCE    61       4      0      1.0     0.0     0.0    -8.00
ENDDATA
```

EXAMPLE 6N BRACKET 3/D (MULT-NODAL-POINT) (EXPL.6 SOLVED WITH NASTRAN)

D I S P L A C E M E N T   V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	-1.668535E-03	8.514381E-04	-2.769290E-02	0.0	0.0	0.0
2	G	-7.478641E-04	5.756649E-04	-1.602405E-02	0.0	0.0	0.0
3	G	2.494223E-03	4.465334E-03	-1.609402E-02	0.0	0.0	0.0
4	G	4.233170E-03	6.719045E-03	-2.922228E-02	0.0	0.0	0.0
5	G	-8.082183E-04	-3.449236E-03	-9.643834E-03	0.0	0.0	0.0
6	G	-1.735006E-03	-2.582530E-03	-5.102344E-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	0.0	0.0	0.0	0.0	0.0	0.0
10	G	0.0	0.0	0.0	0.0	0.0	0.0
11	G	0.0	0.0	0.0	0.0	0.0	0.0
12	G	0.0	0.0	0.0	0.0	0.0	0.0
13	G	-1.270064E-03	4.623283E-04	-2.199817E-02	0.0	0.0	0.0
14	G	8.520414E-04	2.422651E-03	-1.599720E-02	0.0	0.0	0.0
15	G	3.316960E-03	5.908825E-03	-2.222653E-02	0.0	0.0	0.0
16	G	1.035775E-03	3.516742E-03	-2.825741E-02	0.0	0.0	0.0
17	G	0.0	0.0	0.0	0.0	0.0	0.0
18	G	0.0	0.0	0.0	0.0	0.0	0.0
19	G	0.0	0.0	0.0	0.0	0.0	0.0
20	G	0.0	0.0	0.0	0.0	0.0	0.0

F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

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	6.898411E+00	9.637033E+00	0.0	0.0	0.0
11	G	-1.973585E-01	-1.231961E+00	8.229043E+00	0.0	0.0	0.0
12	G	3.722500E+00	-1.550229E+00	1.077604E+01	0.0	0.0	0.0
17	G	-3.822497E-01	1.515567E+01	5.657729E+00	0.0	0.0	0.0
18	G	-1.718428E+00	-5.691678E+00	-2.608670E+01	0.0	0.0	0.0
19	G	-1.593664E+00	-1.510521E+01	1.029323E+01	0.0	0.0	0.0
20	G	-1.409033E+00	-6.533518E+00	-2.111522E+01	0.0	0.0	0.0

S T R E S S E S   I N   H E X A H E D R O N   S O L I D   E L E M E N T S   (H E X A)

ELEMENT-ID	CORNER GRID-ID		------CENTER AND CORNER POINT STRESSES--------- NORMAL		SHEAR		PRINCIPAL	DIR. COSINES -A- -B- -C-	MEAN PRESSURE	OCTAHEDRAL SHEAR
1	-1GRID CS 20 GP									
	CENTER	X	1.357754E+00	XY	-8.287583E-03	A	4.113112E+00	LX 0.02-0.22-0.98	-2.156391E+00	1.392985E+00
		Y	2.983337E+00	YZ	1.496471E+00	B	9.804267E-01	LY 0.80-0.58 0.15		
		Z	2.128083E+00	ZX	9.943342E-02	C	1.375635E+00	LZ 0.60 0.78-0.16		
	4	X	9.600604E-02	XY	5.231612E+00	A	4.590969E+00	LX 0.77-0.03 0.63	6.994268E+00	9.648793E+00
		Y	-2.870858E+00	YZ	3.619038E+00	B	-1.903072E+01	LY 0.62-0.21-0.76		
		Z	-1.820795E+01	ZX	1.634315E+00	C	-6.543051E+00	LZ 0.15 0.98-0.15		
	3	X	4.961511E+00	XY	3.539494E+00	A	6.903331E+00	LX 0.87-0.33-0.37	-1.418489E+00	4.259912E+00
		Y	2.684227E-01	YZ	-2.148305E+00	B	-3.482097E+00	LY 0.49 0.68 0.54		
		Z	-9.744632E-01	ZX	5.254660E-01	C	8.342367E-01	LZ-0.08 0.65-0.75		
	11	X	9.229815E+00	XY	3.966180E+00	A	2.238643E+01	LX 0.30 0.76 0.58	-1.333195E+01	6.911533E+00
		Y	2.117838E+01	YZ	5.383371E-01	B	5.616253E+00	LY 0.95-0.21-0.20		
		Z	9.587667E+00	ZX	-3.070301E+00	C	1.199318E+01	LZ-0.03 0.62-0.79		
	12	X	1.044585E+01	XY	4.559531E-01	A	2.493385E+01	LX 0.04-0.49-0.87	-1.508845E+01	6.973473E+00
		Y	2.411661E+01	YZ	3.360752E+00	B	9.670828E+00	LY 0.97-0.18 0.15		
		Z	1.070288E+01	ZX	5.423104E-01	C	1.066066E+01	LZ 0.23 0.85-0.47		

```
 1 X  -3.604998E+00  XY  -5.174102E+00  A   5.577570E+00  LX-0.38-0.52-0.77   2.713829E+00   6.926195E+00
   Y  -1.876137E-01  YZ   6.141906E+00  B  -1.137592E+01  LY 0.81-0.58-0.01
   Z  -4.348876E+00  ZX   1.593506E+00  C  -2.343137E+00  LZ 0.44 0.63-0.64
 2 X  -2.917389E+00  XY  -4.076189E+00  A   4.040660E+00  LX-0.54 0.84-0.07  -1.123819E-01   4.310342E+00
   Y   1.452724E+00  YZ   5.925348E-01  B  -5.888470E+00  LY 0.79 0.49-0.37
   Z   1.801809E+00  ZX   1.996762E+00  C   2.184956E+00  LZ-0.27-0.25-0.93
10 X  -8.364594E+00  XY  -3.168418E+00  A  -5.032841E+00  LX 0.72 0.28-0.64   1.208219E+01   6.694973E+00
   Y  -1.983794E+01  YZ  -1.591482E+00  B  -2.108080E+01  LY-0.08 0.94 0.32
   Z  -8.044034E+00  ZX  -3.085908E+00  C  -1.013293E+01  LZ-0.69 0.18-0.70
 9 X  -1.160486E+01  XY  -8.408312E-01  A  -1.049852E+01  LX 0.48 0.05-0.87   1.676257E+01   7 903296E+00
   Y  -2.773615E+01  YZ   1.458991E+00  B  -2.791110E+01  LY 0.05 0.99 0.09
   Z  -1.094672E+01  ZX   6 593165E-01  C  -1.187810E+01  LZ 0.87-0.09 0 48
```

```
#RUN CENA:PADINA

*> ENTER FILE NAME TO STORE ADINA DATA (7 CHARACTERS OR LESS)
 ? EXPL.6A

*> 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 ? (0= 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 $ENDFILE (OR CNTR-C).
 ? 1,0,6,1,0,0,0,0
 ? 2,-3,6,1,0,0,0,0
 ? 3,-3,6,2,0,0,0,0
 ? 4,0,6,2,0,0,0,0
 ? 5,0,3,0,0,0,0,0
 ? 6,-3,3,0,0,0,0,0
 ? 7,-3,3,2,0,0,0,0
 ? 8,0,3,2,0,0,0,0
 ? 9,0,0,0,1,1,1,0
 ? 10,-3,0,0,1,1,1,0
 ? 11,-3,0,2,1,1,1,0
 ? 12,0,0,2,1,1,1,0
 ? 13,-1.5,6,1,0,0,0,0
 ? 14,-3,6,1.5,0,0,0,0
 ? 15,-1.5,6,2,0,0,0,0
 ? 16,0,6,1.5,0,0,0,0
 ? 17,-1.5,0,0,1,1,1,0
 ? 18,-3,0,1,1,1,1,0
 ? 19,-1.5,0,2,1,1,1,0
 ? 20,0,0,1,1,1,1,0
 ? /

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

 * ENTER ELEMENT TYPE NO. FOR LINEAR ANALYSIS :
 >>  1= TRUSS,              2= 2/D PLANE/SOLID,
 >>  3= 3/D SOLID,          4= BEAM,
 >>  7= 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),
```

```
  >> (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 $ENDFILE (OR CNTR-C).
  ? 1, 0 10000 .3

  * STRESS OUTPUT LOCATION POINTS ON ELEMENT:
  >> ENTER TABLE NO. & 8 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 $ENDFILE (OR CNTR-C).
  ? 1, 3 4 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 $ENDFILE (OR CNTR-C).
  ? 1, 20 1 1, 4 3 11 12, 1 2 10 9, 15 7 19 8, 13 6 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 COMPONENT,
   >>      (1,2,3= X-, Y-, Z-TRANS., 4,5,6= X-, Y-, Z-ROT.)
   >> (4) FUNCTION MULTIPLIER,
   >> (5) NODE NO. INCREMENT FOR GENERATION.
   * IF THE LINE INPUT IS ERRONEOUS, ENTER AGAIN.
  ** TERMINATE THIS SEQUENCE BY TYPING $ENDFILE (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    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    1    0    0
    1    2
    1    2
       0.        1.        1.        1.
    1    0    0    0    1    1    1    0.0        6.000      1.000     0    0    0
    2    0    0    0    1    1    1   -3.000      6.000      1.000     0    0    0
    3    0    0    0    1    1    1   -3.000      6.000      2.000     0    0    0
    4    0    0    0    1    1    1    0.0        6.000      2.000     0    0    0
    5    0    0    0    1    1    1    0.0        3.000      0.0       0    0    0
    6    0    0    0    1    1    1   -3.000      3.000      0.0       0    0    0
    7    0    0    0    1    1    1   -3.000      3.000      2.000     0    0    0
    8    0    0    0    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    0    0
   10    1    1    1    1    1    1   -3.000      0.0        0.0       0    0    0
   11    1    1    1    1    1    1   -3.000      0.0        2.000     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   -1.500      6.000      1.000     0    0    0
   14    0    0    0    1    1    1   -3.000      6.000      1.500     0    0    0
   15    0    0    0    1    1    1   -1.500      6.000      2.000     0    0    0
   16    0    0    0    1    1    1    0.0        6.000      1.500     0    0    0
   17    1    1    1    1    1    1   -1.500      0.0        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
    0    0    0    0
   3   1    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
```

1	20	20	1	1	0	0	0		0.	0	
4	3	11	12	1	2	10	9				
15	7	19	8	13	6	17	5	16	14	18	20
4	3	1	-8.000			0.	0				

EXAMPLE 6A   3-D SOLID (EXPL.6 SOLVED WITH ADINA)

D I S P L A C E M E N T S

NODE	X-DISPLACEMENT	Y-DISPLACEMENT	Z-DISPLACEMENT	X-ROTATION	Y-ROTATION	Z-ROTATION
1	-0.815261E-03	-0.568847E-03	-0.301279E-01	0.0	0.0	0.0
2	-0.110322E-02	-0.585713E-03	-0.173985E-01	0.0	0.0	0.0
3	0.313887E-02	0.493439E-02	-0.172596E-01	0.0	0.0	0.0
4	0.413101E-02	0.741231E-02	-0.314425E-01	0.0	0.0	0.0
5	-0.847244E-03	-0.402017E-02	-0.942524E-02	0.0	0.0	0.0
6	-0.160280E-02	-0.291563E-02	-0.486425E-02	0.0	0.0	0.0
7	0.223195E-02	0.345010E-02	-0.502602E-02	0.0	0.0	0.0
8	0.138511E-02	0.438329E-02	-0.963579E-02	0.0	0.0	0.0
9	0.0	0.0	0.0	0.0	0.0	0.0
10	0.0	0.0	0.0	0.0	0.0	0.0
11	0.0	0.0	0 0	0.0	0.0	0.0
12	0.0	0.0	0.0	0.0	0.0	0.0
13	-0.102355E-02	-0.561315E-03	-0.240650E-01	0.0	0.0	0.0
14	0.977896E-03	0.209739E-02	-0.172393E-01	0.0	0.0	0.0
15	0.359925E-02	0.636292E-02	-0.241545E-01	0.0	0.0	0.0
16	0.142806E-02	0.319675E-02	-0.305497E-01	0.0	0.0	0.0
17	0.0	0.0	0.0	0.0	0.0	0.0
18	0.0	0.0	0.0	0.0	0.0	0.0
19	0.0	0.0	0.0	0.0	0.0	0.0
20	0.0	0.0	0.0	0.0	0.0	0.0

S T R E S S   C A L C U L A T I O N S   F O R   E L E M E N T   G R O U P   1   (3/D CONTINUUM)

ELEMENT NUMBER	OUTPUT LOCATION	STRESSXX	STRESSYY	STRESSZZ	STRESSXY	STRESSXZ	STRESSYZ
1							
	3	8.5250	19.8917	8.5250	3.7108	0.0	-1.8234
	4	9.7316	22.7071	9.7316	0.9035	0.0	-4.5517
	7	-10.6508	-24.8518	-10.6508	-3.4026	0.0	-1.3196
	8	-14.9152	-34.8022	-14.9152	-1.6498	0.0	-4.8545
	10	-0.3602	8.3204	0.6816	3.6946	2.1220	1.4739
	12	1.6818	13.1731	1.0489	3.3584	-1.2370	-3.1296
	14	-0.6852	-7.0256	-2.2245	-3.2858	0.5997	0.7873
	16	0.3479	-7.7469	-0.9176	-2.5478	-1.6646	-4.1011

Total computation time comparison (CPU, sec) for 3D Bracket (Mult.–Node): MSAP, 0.25; NASTRAN, 4.49; ADINA, 0.18.

REFERENCES

1. M. J. Kaldjian, Interactive data preprocessor program for Michigan SAP (MSAP). *Comput. Struct.* **6** (4/5), 405–412 (1976).
2. M. J. Kaldjian, PREMSAP and MSAPLOT Manual. NAME Report No. 236, University of Michigan, Ann Arbor, MI (1981).
3. M. J. Kaldjian, MSAP/SAP IV MSC/NASTRAN ADINA, interactive and comparative training programs. Course pack. Department of Naval Architecture and Marine Engineering, University of Michigan, Ann Arbor, MI (1988).
4. W. J. Anderson, *MSC/NASTRAN Interactive Training Programs*. John Wiley, New York (1983).
5. K. J. Bathe, E. L. Wilson and F. E. Peterson, SAP IV—a structural analysis program. University of California, Berkeley, CA (1974).
6. H. G. Schaeffer, MSC/NASTRAN static and normal modes analysis (1979).
7. K. J. Bathe, ADINA—a finite element program for automatic dynamic incremental nonlinear analysis. Report AE 79, ADINA Engineering, Watertown, MA (1970).
8. M. J. Kaldjian, Three dimensional interactive graphic display program for Michigan SAP (MSAP). *Comput. Struct.* **7** (2), 183–187 (1977).

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)

To obtain displacements and stresses (forces):

2. $RUN CENA:MSAP 5 = MDATA 6 = -RESULT
3. $TRUNCATE -DEF (the deflection file -DEF is created by MSAP)
4. $RENAME -DEF MDATAD

To print out input data and results:

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