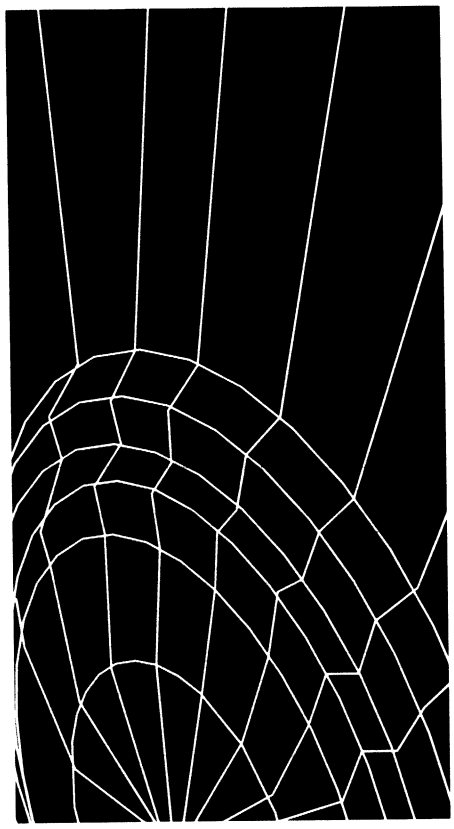
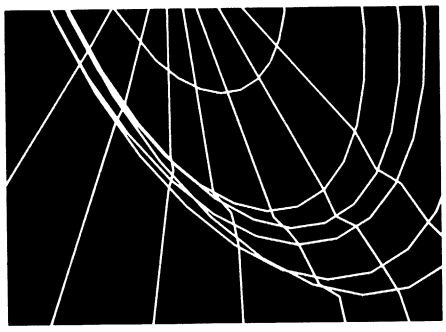


MARC



Volume C

Program Input

Version K7

MARC
Analysis
Research
Corporation

MARC Analysis Research Corporation
260 Sheridan Avenue, Suite 309
Palo Alto, CA 94306 USA

Phone: (650) 329-6800
FAX: (650) 323-5892

Document Title: **MARC Volume C: Program Input, Version K7**
Part Number: RF-3003-07
Revision Date: August, 1997

Proprietary Notice

Restricted Rights Notice

Trademarks

Preface

About this Manual

This document describes the file format of the MARC input file. Its chapters and sections roughly parallel the organization of that file. Appendices describe MARC program messages and provide an alphabetical list of parameters and options for easy reference. Plus, at the beginning of each chapter is a list of the parameters or options discussed in that particular chapter.

Who Should Read this Manual

This document is intended for current and new users of the MARC programs. It does not purport to teach the use of MARC programs, but is a reference to its specific functioning. For a list of MARC documents, see below.

Other MARC Manuals

The MARC Reference Library includes:

- *Volume A: Theory and User Information*
- *Volume B: Element Library*
- *Volume C. Program Input*
- *Volume D: User Subroutines and Special Routines*
- *Volume E: Demonstrations Problems*
- *MARC Primer*
- *Mentat Reference Manual*
- *Mentat User's Guide*

Chapter Contents

Contents

1 Introduction

2 List of Parameters

2 Parameters

3 List of Model Definition Options

3 Model Definition Options

4 List of History Definition Options

4 History Definition Options

5 List of Rezoning Options

5 Rezoning

Appendices

A Program Messages

B Workspace Definition and the Sizing Option

C Default File

D Control File

E Environment Variables

F Parameters

G Options

Appendices

Figures

Tables

1 Introduction

This chapter contains a brief outline of the various data input options and problem solution setups which are available to the user of the MARC program. It highlights only a small segment of the total problem solution capability available. You are allowed the freedom to select only those options required for the solution of your problem. In addition, you may further elect to use the many default and built-in conditions which have been provided in these options.

This user-selection feature forms the basis for the program and input data organization. The program then provides a solution capability based on your selection of options. Further details on the program organization can be found in *Volume A: Theory and User Information.*

Formats used by MARC are discussed in this chapter. A short description of the organization of the input data is given, followed by an illustrative example particularly useful for new users. Selected portions of the output generated by MARC are shown and discussed. The last section of this chapter summarizes the input requirements for different classes of analyses.

Formats in MARC Programs

The MARC programs are written in FORTRAN, but do their own data conversion to avoid system aborts due to user data errors. All input data files are read as alphanumeric and are converted to integer, floating point, or keywords, as necessary. The program issues error messages and displays the illegal image if it cannot interpret the data field according to the specifications given in the manual. When such errors occur, the program attempts to scan the remainder of the data files, and will end the run with an exit error message at the END OPTION (or end file). Two conventions are allowed for input format control – fixed and free format. Fixed and free format can be mixed within a data file; but on a single data line, only one type of format can be entered. The syntax rules for each format type are as follows:

Fixed Field

1. Integers must be right-justified (right blank fill) in their fields.
2. Floating point numbers can be given with or without exponent. In either case, the mantissa must contain a decimal point. If an exponent is given, it must be preceded by the character E or D and must be right justified. The size of the number must lie in the range 10^{-72} to 10^{72}.

Note that, in the this manual, integer fields are indicated as "I" and floating point numbers are indicated by "F". When fixed field is used, the size of the interger or precision is governed by the allowable column field.

Free Field

Data can be input in free field, under the following syntax rules:

1. Each data block must contain the same number of data items that it would contain under standard fixed format control, as documented in this manual. Thus, for example, the 3rd data block of the CONNECTIVITY option is given as (16I5); therefore, no more than 16 numbers can appear on a data line in this data block under fixed or free field format. This syntax rule allows mixing of fixed-field and free-field data in the data file, since the number of data blocks needed to input any data list is the same in both cases.
2. Data items on a data line must be separated by a comma. This separator can be surrounded by an arbitrary number of blanks. Within the data item itself, no embedded blanks can appear.
3. Floating point numbers can be given with or without exponent. In either case, the mantissa must contain a decimal point. If an exponent is given, it must be preceded by the character E or D and must immediately follow the mantissa (no embedded blanks). The size of the number must lie in the range 10^{-35} to 10^{35}.
4. Keywords must be typed exactly as written in the manual. Embedded blanks do not count as separators here (for example, BEAM SECT is one word only).
5. Note that you must distinguish between a real and integer zero when entering data; the floating point zero must contain a decimal point, as in rule 3, above.
6. If a data line contains only one free-field data item, that item must be followed by a comma. Thus, "1" must be entered as "1," if it is the only data item on a data line.

Input of List Items

The MARC program requests that you input a list of items in association with certain program functions. These items, as an example, can be a set of elements as in conjunction with the ISOTROPIC option, or a set of nodes as in conjunction with the POINT LOAD option. There are six types of items that can be requested:

- Element Numbers
- Node Numbers
- Degree of Freedom Numbers

- Integration Point Numbers
- Layer Numbers
- Increment Numbers

A set of items can be expressed as a combination of one or more subsets. These subsets can be specified in three different forms, depending on your convenience. The operations that can be performed between subsets are:

```
AND
INTERSECT
EXCEPT
```

In forming a set, subsets are combined in binary operations going from left to right. Hence, a set can be formed as:

1. SUBLIST1 AND SUBLIST2

   which implies all items in `SUBLIST1` and `SUBLIST2`. Duplicate items are eliminated and the resultant set are sorted.

2. SUBLIST1 INTERSECT SUBLIST2

   which implies only those items occurring both in `SUBLIST1` and `SUBLIST2`.

3. SUBLIST1 EXCEPT SUBLIST2

   which implies all items in `SUBLIST1` except those which occur in `SUBLIST2`.

4. SUBLIST1 AND SUBLIST2 EXCEPT SUBLIST3 INTERSECT SUBLIST4

   which implies take the items in `SUBLIST1` and `SUBLIST2` and remove those items that occur in `SUBLIST3`. Then, if these items also occur in `SUBLIST4`, include them in the set.

The `SUBLISTS` can have the form:

1. A range of items can be specified as:

   ```
   l TO m BY n
   ```

   or

   ```
   1 THROUGH M BY n
   ```

   which implies items `l through m by n`; if `BY n` is not included, it is assumed to be `BY 1`. Note that the range can be either increasing or decreasing.

2. A string of items can be specified as:

   a_1 a_2 a_3 ... a_n

   which implies that n items are to be included. If continuation data is necessary, then a "C" or CONTINUE should be the last item on the data line.

3. A setname can be specified as:

   MYSET

   which implies that all items previously specified to be in the set MYSET are to be used. The items in a set are specified using the DEFINE option.

Sorted vs. Unsorted Lists

In the MARC program, most lists are sorted lists. That is, regardless of the order of the list items on the list line, the MARC program returns these items sorted from lowest to highest. Unsorted lists are required in several places, however. These places are:

1. List of nodes in the TYING option.
2. List of nodes in the SUPERINPUT option.
3. List of degrees of freedom in the FIXED DISP option.

When defining unsorted lists, the sublist connectors EXCEPT and INTERSECT cannot be used. Setnames can be used as long as the sets themselves are unsorted. In the MARC program, degree of freedom sets are always unsorted. Unsorted node sets can be defined by using set type NDSQ (for "node sequence") rather than set type NODE (see DEFINE model definition option).

Examples

Define subsets FLOOR, NWALL, WWALL

```
DEFINE NODE SET FLOOR
            1 TO 5 (i.e. NODES 1,2,3,4,5)

DEFINE NODE SET NWALL
            5 TO 15 BY 5 AND 20 to 22 (i.e. NODES 5,10,15,20,21,22)

DEFINE NODE SET WWALL
            11 TO 20 (i.e. NODES 11,12,13,14,15,16,17,18,19,20)
```

Possible lists can be:

1. NWALL AND WWALL, which would contain nodes
   5 10 11 12 13 14 15
   16 17 18 19 20 21 22
2. NWALL INTERSECT WWALL, which would contain node
   15 20
3. NWALL AND WWALL EXCEPT FLOOR, which would contain nodes:
   10 11 12 13 14 15
   16 17 18 19 20 21 22

Guide To Organization Of Marc Input Data

The input data for MARC is organized into three basic groups. These groups form a natural subdivision of the data. Each group is then subdivided into various optional blocks of input data. The optional blocks of data within each group have been organized to minimize the input of unnecessary data. The main idea is to enable you to specify only the data for the optional blocks that you need to define your problem. The various blocks of input are referred to here as optional in the sense that many have built-in default values which can be used and does not imply that they are optional in all cases. The input data is divided into the following three groups:

Parameter Data

This group of data is used to allocate the necessary working space for the problem and to set up initial switches which control the flow of the program through the desired analysis. This set of input data is terminated with END parameter data.

Model Definition Data

This set of data is used to read in the initial loading, geometry and material data of the model. It also provides nodal point data such a boundary conditions. In general, the initial model data is provided in this group and control restart, and print options can also be specified here for further program processing. This data provides the program with the necessary information for determination of an initial elastic solution (zero increment solution in program terminology). This group of data is terminated with END OPTION data.

Load Incrementation Data

This group of data provide the load incrementation and control of the program after the initial elastic analysis. The group also includes blocks which allow changes in the initial model specifications. Each set of load incrementation data is terminated with CONTINUE data. This data sends the program back for another increment or series of increments if the auto incrementation features are requested.

Input data file organization for program MARC is shown in Figure 1-1.

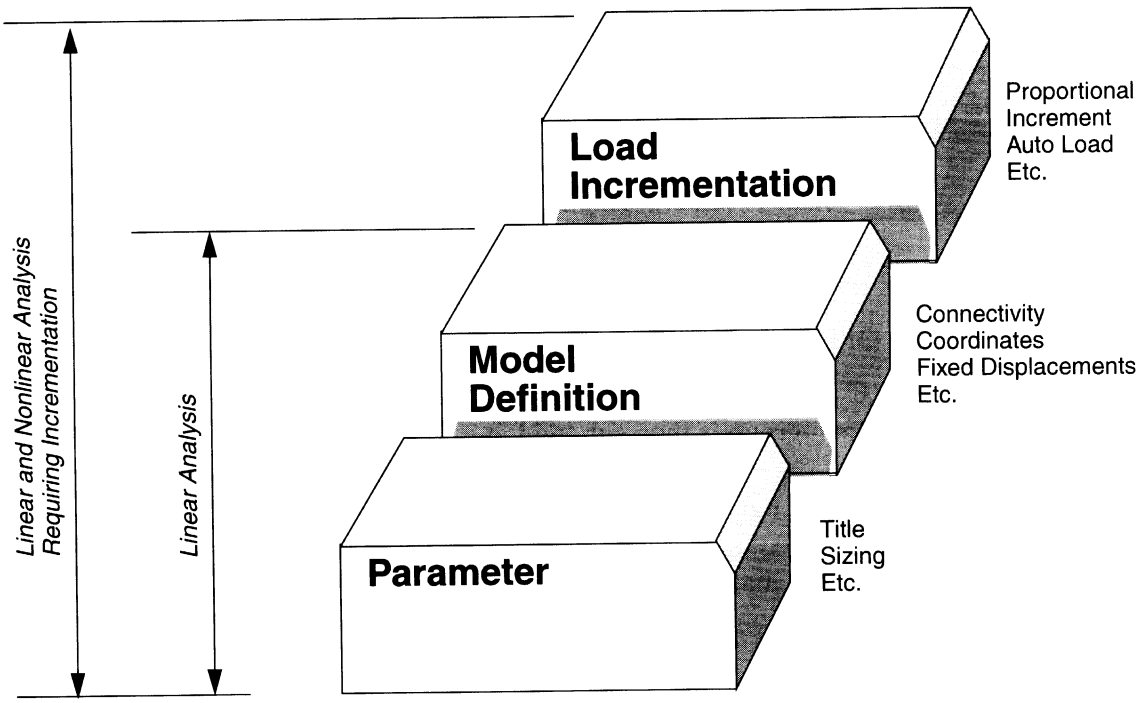


Figure 1-1 MARC Input Data File

Typical MARC Problem Data Files

MARC Parameter Data

END Data

MARC Model Definition Data
(Zero Increment)

END OPTION Data

Marc Load Incrementation Data
for the First Increment

CONTINUE Data

(Additional Load Incrementation Data
for the 2nd, 3rd, ..., Increments)

Discussion of MARC Input for New Users

The MARC input format is designed to allow the input of very complex problems. The new user is, however, faced with gaining familiarity with the system and its conventions. At the outset, therefore, you should adopt a systematic approach to the preparation of input data. One approach is to follow the construction of the program and adopt the procedure of preparing input for each of the data blocks (parameter, model definition and load incrementation) in turn.

In the following, we shall illustrate our discussion by preparing input for the analysis of a thin plate with hole subjected to pressure loading. The problem, as shown in Figure 1-2, is a well-known one so that the results can be compared to the exact solution (Timoshenko, *Theory of Elasticity*). The hole/plate size ratio is chosen to approximate an infinite plate. A procedure for preparing MARC input would take the following steps:

Finite Element Modeling

The plate has an outside dimension of 10 inches x 10 inches with a central hole of 1 inch radius. The thickness of the plate is assumed to be 0.1 inches and the material property is assumed to be isotropic and linear elastic. The Young's modulus is 30 x 10^6 psi with Poisson's ratio of 0.3. These quantities are sufficient to define the behavior of an isotropic, linear-elastic material.

As shown in Figure 1-2, due to symmetry conditions, only a quarter of the plate is analyzed. Prescribed displacement boundary conditions exist along the lines of symmetry (that is, u = 0 at line x = 0; v = 0 at line y = 0) and traction (pressure) boundary condition exits at the top of the plate.

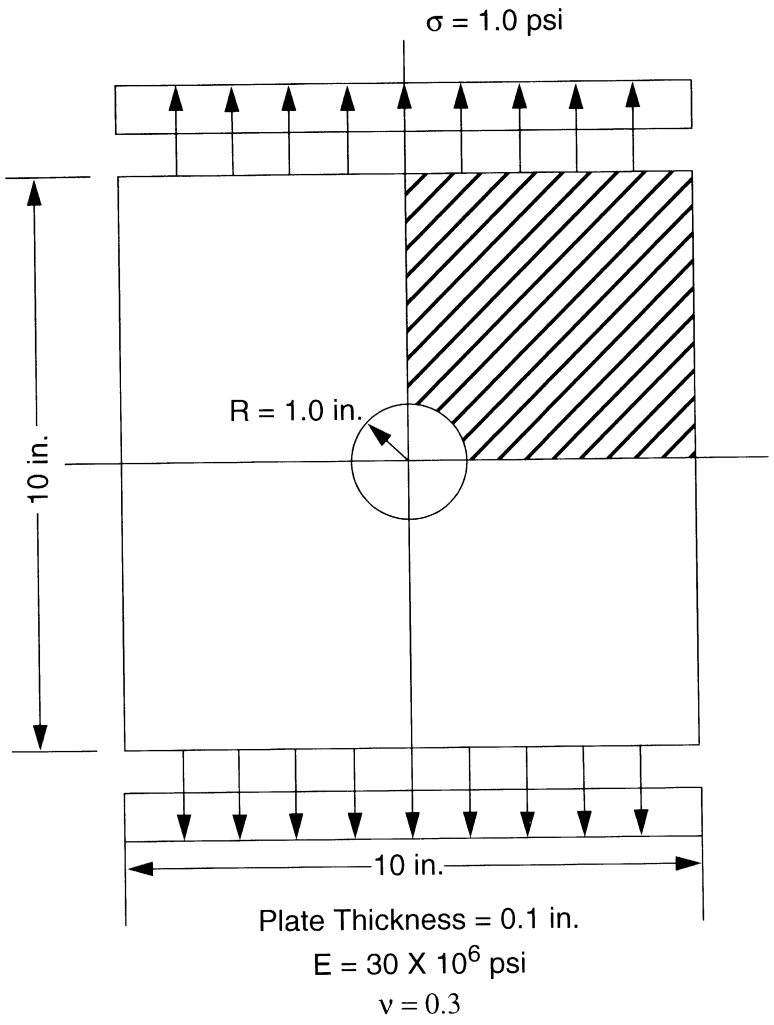


Figure 1-2 Plate with Hole

This quarter plate is approximated by a finite element mesh consisting of twenty 8-node plane stress elements with appropriate loading and boundary conditions. The element (MARC type 26) is a second-order, isoparametric two-dimensional element for plane stress. There are eight nodes with two translational degrees of freedom at each node. A description of element type 26 can be found in *Volume B: Element Library.*

This example uses a coarse mesh for demonstration purposes only. The analyst must anticipate the sharp stress gradients in this problem, and design the mesh accordingly. This is achieved in this problem by using progressively smaller elements as the hole is approached. If necessary, further mesh refinement can be achieved by adding elements to the mesh.

The preparation of parameter, model definition, and load incrementation data for this example is demonstrated below:

Parameter Data

The analysis to be carried out in this example is a linear elastic analysis with plots. Consequently, only five parameters are needed for the input data:

```
TITLE

                ELEMENTS
                SIZING
                END
```

In this example, the title, *Elastic Analysis of a Thin Plate with Hole*, is chosen for the problem and entered through the parameter TITLE.

The selected MARC element type 26 is entered through the parameter ELEMENTS.

The data on the SIZING parameter is selected as follows:

```
MAXALL = 50000 (core allocation)
```

Please note that the value of MAXALL should be checked with the in-house or data center system analyst for the maximum allowable core area on the system for running the MARC program. You should refer to Table 2-1 and Table 2-2 following the SIZING parameter definition in order to establish an estimate of the work space required in this problem. The estimate should only be approximate since the program adjusts the variables to use some out-of-core storage if necessary. You do not need to input maximum values on SIZING.

Finally, the parameters are completed with an END parameter.

At this stage, the input data is:

```
TITLE   ELASTIC ANALYSIS OF A THIN PLATE WITH HOLE
SIZING,50000,
ELEMENTS,26,
END
```

Model Definition Data

The model definition data contains the bulk data for the analysis. The data entered here concerns:

1. the topology of the model (finite element mesh in terms of element connectivity and nodal coordinates, as well as plate thickness),
2. material property (Young's modulus and Poisson's ratio),
3. pressure loading and prescribed displacement boundary conditions, and
4. plotting and output controls.

An index of the model definition data can be found in Chapter 3 of this document.

1. Topology of the Model

The topology of the plate model is numerically defined by the following model definition options:

```
CONNECTIVITY
COORDINATES
GEOMETRY
```

In this example, the mesh consists of 20 elements and 79 nodes. The data required for element connectivity and nodal coordinates are:

```
CONNECTIVITY
   20
    1   26    1    3   11    9    2    7   10    6
    2   26    3    5   13   11    4    8   12    7
    3   26    9   11   19   17   10   15   18   14
    4   26   11   13   21   19   12   16   20   15
    5   26    5    3   27   25    4   23   26   22
    6   26    3    1   29   27    2   24   28   23
    7   26   30   32   40   38   31   36   39   35
    8   26   32   34   42   40   33   37   41   36
    9   26   38   40   27   29   39   44   28   43
   10   26   40   42   25   27   41   45   26   44
   11   26    1    9   53   47    6   52   50   46
```

```
   12   26   47   53   55   49   50   54   51   48
   13   26    9   17   59   53   14   58   56   52
   14   26   53   59   61   55   56   60   57   54
   15   26   49   64   66   47   62   65   63   48
   16   26   47   66   29    1   63   67   24   46
   17   26   30   38   75   69   35   74   72   68
   18   26   69   75   77   71   72   76   73   70
   19   26   38   29   66   75   43   67   78   74
   20   26   75   66   64   77   78   65   79   76
COORDINATES
    0    0
    1    1.4000    1.4000
    2    1.5500    1.0500
    3    1.7000    0.7000
    .
    .
    .
   77    0.0000    1.2500
   78    0.4931    1.1910
   79    0.0000    1.3750
```

The data in the CONNECTIVITY block consists of element numbers (1,2,...,19,20); element type (26) and for each element, four corner node numbers and four mid-side node numbers.

The data in the coordinate block consists of the node number (1); and coordinates (x = 1.4, y = 1.4) of node 1 in the global coordinate system (x, y).

Finally, the plate thickness is entered through GEOMETRY as:

```
GEOMETRY
0,
0.1,
1 TO 20
```

A thickness of 0.1 inches is assumed for all twenty (1 to 20) elements.

2. Material Property

Material properties of the plate are entered through ISOTROPIC. For our problem, the only data required for a linear elastic analysis are Young's modulus and Poisson's ratio. The same material is used for the whole mesh (from Element No. 1 to Element No. 20). This is given a material id of 1. The data in ISOTROPIC is:

```
ISOTROPIC
1,
1 30.E6,0.3,
1 TO 20
```

3. Pressure Loading and Prescribed Displacement Boundary Conditions

As shown in Figure 1-3, the pressure loading is acted on two elements (elements 13 and 14), along the lines 61-60-59 and 59-58-17.

From CONNECTIVITY, we observe that these lines represent the 2-6-3 face of the elements. As a result, a distributed load type of 8 can be determined for the pressure loading from the **QUICK REFERENCE** of element 26 in *Volume B: Element Library.*

```
"LOAD TYPE (IBODY)=8 FOR UNIFORM PRESSURE ON 2-6-3 FACE"
```

In addition, as shown in Figure 1-4, the sign conversion of the pressure loading is that a negative magnitude represents a tensile distributed load. Consequently, the input for the 1 pound tensile distributed loading acting on elements 13 and 14 takes the following form:

```
DIST LOADS
0,
8,-1.,
13,14,
```

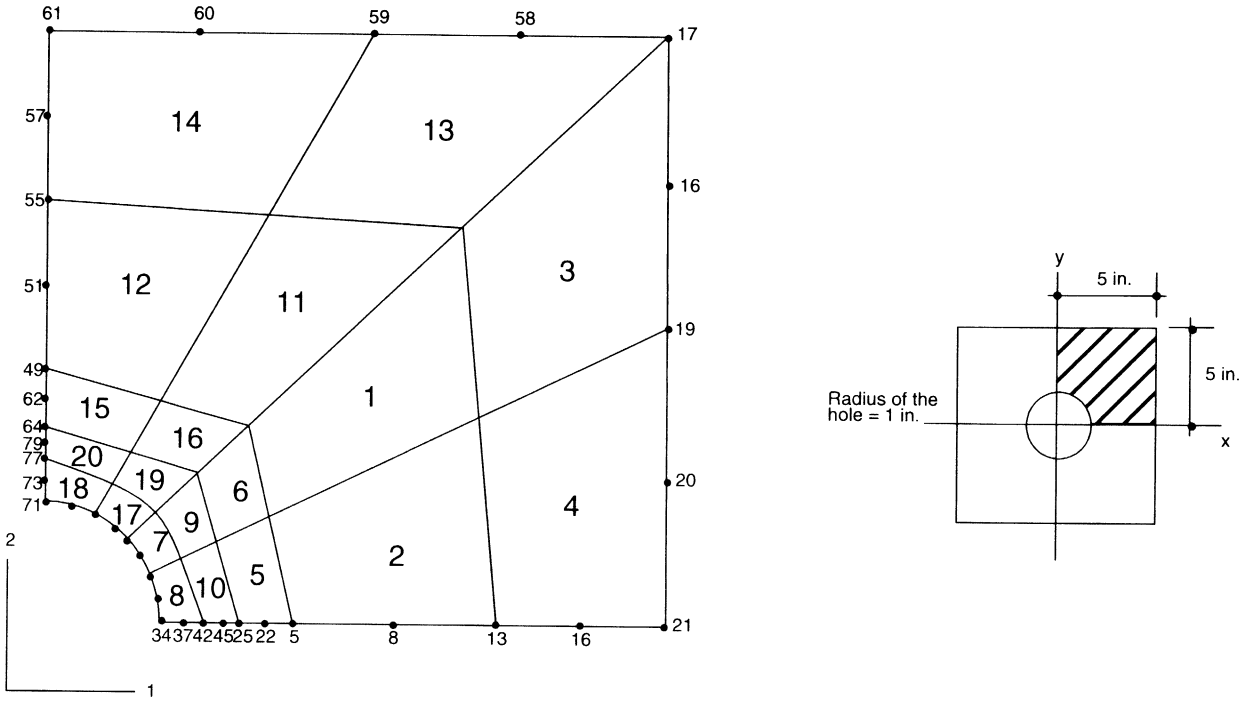


Figure 1-3 Mesh Layout for Plate with Hole

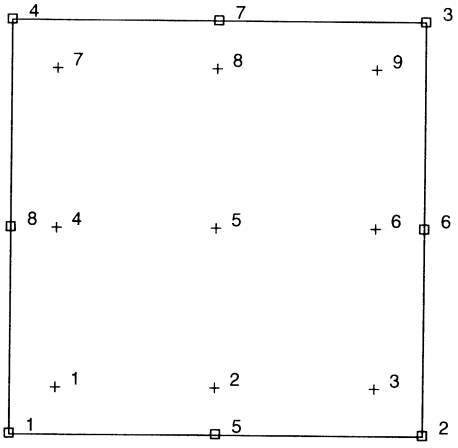


Figure 1-4 Integration Points of Eight-Node, 2D Element

The FIXED DISP block is used for the input of prescribed displacement boundary conditions at the lines of symmetry (x = 0, y = 0). As indicated in the **QUICK REFERENCE** of element 26, the nodal degrees-of-freedom are

```
dof 1 = u = global x-direction displacement
dof 2 = v = global y-direction displacement.
```

In this example, the symmetry conditions require that:

```
dof 1 = u = 0 for nodes (71, 73, 77, 79, 64, 62, 49, 51, 55, 57, 61)
            along the line x=0.
```

and

```
dof 2 = v = 0 for nodes (34, 37, 42, 45, 25, 22, 5, 8, 13, 16, 21)
            along the line y=0.
```

The input data takes the following format:

```
FIXED DISP
2,
0.,
2,
34,37,42,45,25,22,5,8,13,16,21,
0.,
1,
71,73,77,79,64,62,49,51,55,57,61,
```

4. Bandwidth Optimization and Output Controls

Although the bandwidth in this sample problem cannot be extremely large, the use of the OPTIMIZE model definition option demonstrates the bandwidth optimization capabilities in the MARC program. This option can reduce considerable computing costs in larger problems. The bandwidth optimization option creates an internal node numbering different from the your node numbering, but all data input and output is in the your node numbering system.

There are a number of options available to you for bandwidth optimization. The option number 2 (Cuthill-McKee algorithm) with a maximum of 10 iterations is selected for this example.

```
OPTIMIZE,2,0,0,1,
10,
```

In order to minimize the output quantity (number of printed pages), the PRINT ELEMENT option is used for printing out stresses and strains at a few integration points of a number of elements. The elements to be printed are:

From Element	to	Element
2		2
4		5
8		8
10		10

Only two integration points (numbers 4 and 6) where stresses and strains are to be printed. Nodal quantities (displacement, reactions, etc.) are printed for all nodes (from node 1 to node 79). The input data of PRINT ELEMENT is:

```
PRINT ELEMENT
1,
STRESS  STRAIN
2,4,5,8,10
4,6,
```

The SUMMARY option produces summary tables containing maximum and minimum values of stresses and strains.

The model definition data is completed with an END OPTION.

Load Incrementation Data (Plotting)

This following example is a linear-elastic analysis which requires no incrementation data.

```
TITLE          ELASTIC ANALYSIS OF A THIN PLATE WITH HOLE
SIZING         50000
ELEMENTS     26
END
CONNECTIVITY
   20
    1   26    1    3   11    9    2    7   10    6
    2   26    3    5   13   11    4    8   12    7
    3   26    9   11   19   17   10   15   18   14
    4   26   11   13   21   19   12   16   20   15
    5   26    5    3   27   25    4   23   26   22
    6   26    3    1   29   27    2   24   28   23
    7   26   30   32   40   38   31   36   39   35
    8   26   32   34   42   40   33   37   41   36
    9   26   38   40   27   29   39   44   28   43
   10   26   40   42   25   27   41   45   26   44
   11   26    1    9   53   47    6   52   50   46
   12   26   47   53   55   49   50   54   51   48
   13   26    9   17   59   53   14   58   56   52
   14   26   53   59   61   55   56   60   57   54
   15   26   49   64   66   47   62   65   63   48
   16   26   47   66   29    1   63   67   24   46
   17   26   30   38   75   69   35   74   72   68
   18   26   69   75   77   71   72   76   73   70
   19   26   38   29   66   75   43   67   78   74
   20   26   75   66   64   77   78   65   79   76
COORDINATES
    0    0
    1    1.4000    1.4000
    2    1.5500    1.0500
    3    1.7000    0.7000
    4    1.8500    0.3500
    5    2.0000    0.0000
    6    2.3000    2.3000
    7    2.5250    1.1500
    8    2.7500    0.0000
    9    3.2000    3.2000
   10    3.2750    2.4000
   11    3.3500    1.6000
   12    3.4250    0.8000
   13    3.5000    0.0000
   14    4.1000    4.1000
   15    4.1750    2.0500
   16    4.2500    0.0000
   17    5.0000    5.0000
```

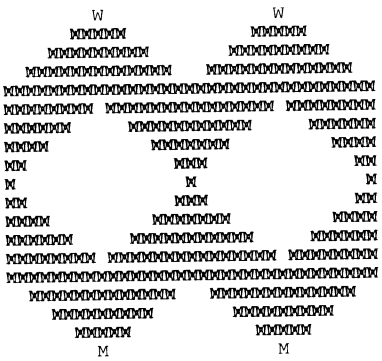
18	5.0000	3.7500
19	5.0000	2.5000
20	5.0000	1.2500
21	5.0000	0.0000
22	1.7500	0.0000
23	1.4900	0.6150
24	1.2300	1.2300
25	1.5000	0.0000
26	1.3900	0.2650
27	1.2800	0.5300
28	1.1700	0.7950
29	1.0600	1.0600
30	0.7070	0.7070
31	0.8315	0.5557
32	0.9238	0.3825
33	0.9810	0.1948
34	1.0000	0.0000
35	0.7953	0.7953
36	1.0129	0.4194
37	1.1250	0.0000
38	0.8835	0.8835
39	1.0008	0.6753
40	1.1019	0.4562
41	1.1855	0.2299
42	1.2500	0.0000
43	0.9718	0.9718
44	1.1910	0.4931
45	1.3750	0.0000
46	1.0500	1.5500
47	0.7000	1.7000
48	0.3500	1.8500
49	0.0000	2.0000
50	1.1500	2.5250
51	0.0000	2.7500
52	2.4000	3.2750
53	1.6000	3.3500
54	0.8000	3.4250
55	0.0000	3.5000
56	2.0500	4.1750
57	0.0000	4.2500
58	3.7500	5.0000
59	2.5000	5.0000
60	1.2500	5.0000
61	0.0000	5.0000
62	0.0000	1.7500

```
    63     0.6150     1.4900
    64     0.0000     1.5000
    65     0.2650     1.3900
    66     0.5300     1.2800
    67     0.7950     1.1700
    68     0.5557     0.8315
    69     0.3825     0.9238
    70     0.1948     0.9810
    71     0.0000     1.0000
    72     0.4194     1.0129
    73     0.0000     1.1250
    74     0.6753     1.0008
    75     0.4562     1.1019
    76     0.2299     1.1855
    77     0.0000     1.2500
    78     0.4931     1.1910
    79     0.0000     1.3750
GEOMETRY
     1
     0.1
     1 TO 20
ISOTROPIC
     1
     1
 30000000.    .3
     1 TO 20
DIST LOADS
     1
     8  -1.
    13   14
FIXED DISPLACEMENT
     2
  0.0000E+00
     2
    34   37   42   45   25   22    5    8   13   16   21
  0.0000E+00
     1
    71   73   77   79   64   62   49   51   55   57   61
OPTIMIZE,2,0,0,1,
10,
PRINT ELEMENT
     1
     STRESS STRAIN
     2    4    5    8   10
     4    6
END OPTION
```

Discussion of MARC Output for New Users

Selected portions of the output for this problem is shown in the following. The small type on the output are the author's comments and give a further explanation.

The program first gives a notes section which identifies the version of the program being used. This is followed by an echo of the input data and a summary of program sizing and options requested.



MARC REVISION K.7 1

MARC ANALYSIS RESEARCH CORPORATION

C A R D I N P U T D A T A

P A G E 1

```
          5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
-------------------------------------------------------------------------------------
TITLE          ELASTIC ANALYSIS OF A THIN PLATE WITH HOLE
SIZING          50000
ELEMENTS      26
MESH PLOT      2
CARD     5     END
CONNECTIVITY
   20
    1   26    1    3   11    9    2    7   10    6
    2   26    3    5   13   11    4    8   12    7
CARD    10      3   26    9   11   19   17   10   15   18   14
    4   26   11   13   21   19   12   16   20   15
    5   26    5    3   27   25    4   23   26   22
    6   26    3    1   29   27    2   24   28   23
    7   26   30   32   40   38   31   36   39   35
CARD    15      8   26   32   34   42   40   33   37   41   36
    9   26   38   40   27   29   39   44   28   43
   10   26   40   42   25   27   41   45   26   44
   11   26    1    9   53   47    6   52   50   46
   12   26   47   53   55   49   50   54   51   48
CARD    20     13   26    9   17   59   53   14   58   56   52
   14   26   53   59   61   55   56   60   57   54
   15   26   49   64   66   47   62   65   63   48
   16   26   47   66   29    1   63   67   24   46
   17   26   30   38   75   69   35   74   72   68
CARD    25     18   26   69   75   77   71   72   76   73   70
   19   26   38   29   66   75   43   67   78   74
   20   26   75   66   64   77   78   65   79   76
COORDINATES
    0    0
CARD    30      1   1.4000   1.4000
    2   1.5500   1.0500
    3   1.7000   0.7000
```

		4	1.8500	0.3500
		5	2.0000	0.0000
CARD	35	6	2.3000	2.3000
		7	2.5250	1.1500
		8	2.7500	0.0000
		9	3.2000	3.2000
		10	3.2750	2.4000
CARD	40	11	3.3500	1.6000
		12	3.4250	0.8000
		13	3.5000	0.0000
		14	4.1000	4.1000
		15	4.1750	2.0500
CARD	45	16	4.2500	0.0000
		17	5.0000	5.0000
		18	5.0000	3.7500
		19	5.0000	2.5000
		20	5.0000	1.2500
CARD	50	21	5.0000	0.0000
		22	1.7500	0.0000
		23	1.4900	0.6150
		24	1.2300	1.2300
		25	1.5000	0.0000
CARD	55	26	1.3900	0.2650
		27	1.2800	0.5300
		28	1.1700	0.7950
		29	1.0600	1.0600
		30	0.7070	0.7070
CARD	60	31	0.8315	0.5557
		32	0.9238	0.3825
		33	0.9810	0.1948
		34	1.0000	0.0000
		35	0.7953	0.7953
CARD	65	36	1.0129	0.4194
		37	1.1250	0.0000
		38	0.8835	0.8835
		39	1.0008	0.6753
		40	1.1019	0.4562
CARD	70	41	1.1855	0.2299
		42	1.2500	0.0000

		43	0.9718	0.9718
		44	1.1910	0.4931
		45	1.3750	0.0000
CARD	75	46	1.0500	1.5500
		47	0.7000	1.7000
		48	0.3500	1.8500
		49	0.0000	2.0000
		50	1.1500	2.5250
CARD	80	51	0.0000	2.7500
		52	2.4000	3.2750
		53	1.6000	3.3500
		54	0.8000	3.4250
		55	0.0000	3.5000
CARD	85	56	2.0500	4.1750
		57	0.0000	4.2500
		58	3.7500	5.0000
		59	2.5000	5.0000
		60	1.2500	5.0000
CARD	90	61	0.0000	5.0000
		62	0.0000	1.7500
		63	0.6150	1.4900
		64	0.0000	1.5000
		65	0.2650	1.3900
CARD	95	66	0.5300	1.2800
		67	0.7950	1.1700
		68	0.5557	0.8315
		69	0.3825	0.9238
		70	0.1948	0.9810
CARD	100	71	0.0000	1.0000
		72	0.4194	1.0129
		73	0.0000	1.1250
		74	0.6753	1.0008
		75	0.4562	1.1019
CARD	105	76	0.2299	1.1855
		77	0.0000	1.2500
		78	0.4931	1.1910
		79	0.0000	1.3750

```
                GEOMETRY
CARD   110          1
                    0.1
                    1 TO 20
                ISOTROPIC
                    1
CARD   115          1
                 30000000.    .3
                    1 TO 20
                DIST LOADS
                    1
CARD   120          8  -1.
                   13   14
                FIXED DISPLACEMENT
                    2
                  0.0000E+00
CARD   125          2
                   34   37   42   45   25   22    5    8   13   16   21
                  0.0000E+00
                    1
                   71   73   77   79   64   62   49   51   55   57   61
CARD   130      SUMMARY
                OPTIMIZE,2,0,0,1,
                10,
                PRINT ELEMENT
                    1
CARD   135          STRESS STRAIN
                    2    4    5    8   10
                    4    6
                END OPTION
-------------------------------------------------------------------------------
    5   10   15   20   25   30   35   40   45   50   55   60   65   70   75   80
-------------------------------------------------------------------------------
```

```
************************************************
************************************************

 PROGRAM SIZING AND OPTIONS REQUESTED AS FOLLOWS



 ELEMENT TYPE REQUESTED*************************   26
 NUMBER OF ELEMENTS IN MESH*********************   20
 NUMBER OF NODES IN MESH************************   79
 MAX NUMBER OF ELEMENTS IN ANY DIST LOAD LIST***    2
 MAXIMUM NUMBER OF BOUNDARY CONDITIONS**********   22
 LOAD CORRECTION FLAGGED OR SET*****************
 NUMBER OF LISTS OF DISTRIBUTED LOADS***********    3
 STRESSES STORED AT ALL INTEGRATION POINTS******
 TAPE NO.FOR INPUT OF COORDINATES + CONNECTIVITY    5
 NO.OF DIFFERENT MATERIALS     1 MAX.NO OF SLOPES    5
 MAXIMUM ELEMENTS VARIABLES PER POINT ON POST TP   33
 NUMBER OF POINTS ON SHELL SECTION *************   11
 OPTION FOR TERMINAL DEBUG**********************
 NEW STYLE INPUT FORMAT WILL BE USED************
 MAXIMUM NUMBER OF SET NAMES IS*****************   10
 NUMBER OF PROCESSORS USED **********************    1
 VECTOR LENGTH USED *****************************    1


 END OF PARAMETERS AND SIZING
************************************************
************************************************
```

At this stage, the program attempts to allocate core for input of the model definition data and assembly of the element stiffness matrix. The program first prints out the key to strain, stress and displacement output for each element type chosen. Column numbers identifying output quantities are referenced to the appropriate components of stress, strain or displacement. Then, the required number of words is printed out followed by a list of the internal core allocation parameters. They reflect the maximum requirements imposed by different elements. The internal element variables are different for each element type and are repeated for each element type used in a given analysis.

```
KEY TO STRESS, STRAIN AND DISPLACEMENT OUTPUT


          ELEMENT TYPE   26


8-NODE ISOPARAMETRIC PLANE STRESS QUADRILATERAL

STRESSES AND STRAINS IN GLOBAL DIRECTIONS
  1=XX
  2=YY
  3=XY

DISPLACEMENTS IN GLOBAL DIRECTIONS
  1=U GLOBAL X DIRECTION
  2=V GLOBAL Y DIRECTION


WORKSPACE NEEDED FOR INPUT AND STIFFNESS ASSEMBLY      48071


INTERNAL CORE ALLOCATION PARAMETERS
DEGREES OF FREEDOM PER NODE (NDEG)  2
COORDS PER NODE (NCRD)  2
STRAINS PER INTEGRATION POINT (NGENS)  3
MAX. NODES PER ELEMENT (NNODMX)  8
MAX.STRESS COMPONENTS PER INT. POINT (NSTRMX)   3
MAX. INVARIANTS PER INT. POINTS (NEQST)  1


FLAG FOR ELEMENT STORAGE (IELSTO)  0
ELEMENTS IN CORE, TOTAL SPACE REQUIRED (NELSTO)       1838
VECTORS IN CORE, TOTAL SPACE REQUIRED      2350


WORDS PER TRACK ON DISK SET TO 4096
```

```
 INTERNAL ELEMENT VARIABLES



 INTERNAL ELEMENT NUMBER  1  LIBRARY CODE TYPE 26
 NUMBER OF NODES=  8
 STRESSES STORED PER INTEGRATION POINT =  3
 DIRECT CONTINUUM COMPONENTS STORED =  2
 SHEAR CONTINUUM COMPONENTS STORED =  1
 SHELL/BEAM FLAG = 0
 CURVILINEAR  COORD. FLAG = 0
 INT.POINTS FOR ELEM. STIFFNESS  9
 NUMBER OF LOCAL INERTIA DIRECTIONS  2
 INT.POINT FOR PRINT IF ALL POINTS NOT FLAGGED  5
 INT. POINTS FOR DIST. SURFACE LOADS (PRESSURE)  3
 LIBRARY CODE TYPE = 26
 NO LOCAL ROTATION FLAG = 1
 GENERALISED DISPL. FLAG = 0
 LARGE DISP. ROW COUNTS     4    4    7



 RESIDUAL LOAD CORRECTION IS INVOKED
```

For nonlinear problems, it is important to note if the residual load correction was turned on. This is automatically done in the current version.

This is followed by the model definition data how it is read and interpreted by the program. The program then calculates the bandwidth of the stiffness matrix and optimizes it if the OPTIMIZE model definition option is included. The original bandwidth (try 0) and the optimized bandwidth (try 10), as well as a correspondence table for nodes are then printed out.

```
 MAXIMUM CONNECTIVITY IS   17  AT NODE   75


 WORKSPACE NEEDED FOR OUT OF CORE CUTHILL OPTIMIZER =      46177
 MAXIMUM SKY-LINE INCLUDING FILL-IN IS      1526 AT TRY    0 (FORWARD NUMBERING)
 MAXIMUM SKY-LINE INCLUDING FILL-IN IS      1128 AT TRY    0 (BACKWARD NUMBERING)
```

```
                .
                .
                .
 MAXIMUM SKY-LINE INCLUDING FILL-IN IS      1307 AT TRY   10 (FORWARD NUMBERING)
 MAXIMUM SKY-LINE INCLUDING FILL-IN IS       900 AT TRY   10 (BACKWARD NUMBERING)


                .

          C O R R E S P O N D E N C E   T A B L E   F O R   N O D E S

USER, INTERNAL,   USER, INTERNAL, ETC

    1...   43     2...   45     3...   46     4...   49     5...   48     6...   23     7...   27     8...   28
    9...   21    10...   22    11...   24    12...   26    13...   25    14...    8    15...   11    16...   14
   17...    7    18...    9    19...   10    20...   13    21...   12    22...   50    23...   47    24...   44
   25...   65    26...   64    27...   62    28...   61    29...   59    30...   70    31...   71    32...   76
   33...   79    34...   78    35...   69    36...   75    37...   77    38...   67    39...   68    40...   72
   41...   74    42...   73    43...   60    44...   63    45...   66    46...   42    47...   40    48...   30
   49...   29    50...   20    51...   17    52...   19    53...   18    54...   16    55...   15    56...    5
   57...    3    58...    6    59...    4    60...    2    61...    1    62...   31    63...   41    64...   33
   65...   32    66...   51    67...   53    68...   58    69...   57    70...   39    71...   38    72...   56
   73...   37    74...   55    75...   54    76...   36    77...   35    78...   52    79...   34
```

After the bandwidth calculation (and optimization), the program assigns the necessary workspace for the in-core solution of this matrix. If the work space allocated in SIZING was insufficient, it attempts to allocate workspace for an out-of-core solution. Information on workspace requirement is printed out.

```
 MAXIMUM CONNECTIVITY IS   14  AT NODE   40

  MAXIMUM HALF-BANDWIDTH IS   26  BETWEEN NODES   21  AND   46
   NUMBER OF PROFILE ENTRIES INCLUDING FILL-IN IS        900
   NUMBER OF PROFILE ENTRIES EXCLUDING FILL-IN IS        546

  TOTAL WORKSPACE NEEDED WITH IN-CORE MATRIX STORAGE =     56133
```

The program then calculates the loading and sums the load applied to each degree of freedom for distributed loads and point loads. This information provides for a valuable check on the total loads in the different degrees of freedom.

```
                    LOAD INCREMENTS ASSOCIATED WITH EACH DEGREE OF FREEDOM
                    SUMMED OVER THE WHOLE MODEL
                    DISTRIBUTED LOADS
0.000E+00 5.000E-01
                    POINT LOADS
0.000E+00 0.000E-00
```

It then prints out the time (system billing units) at the start of assembly measured from the start of the job. It then prints out the bandwidth which might have changed due to optimization of the nodal numbering (if specified by the user). This is followed by a printout of the time at the start of the matrix solution

```
START OF ASSEMBLY
 TIME =       0.18
START OF MATRIX SOLUTION
 TIME =       0.24
```

If the out-of-core solver is used, a figure representing the profile of the global stiffness matrix is shown.

It then prints out the following message which gives an estimate of the conditioning of the matrix. If the singularity is of the order of the accuracy of the machine (10 for 64 bits), the equations can be considered singular and the solution unreliable. For nonlinear problems, incremental changes in the singularity ratio reflects approaching instabilities. The program then prints the time at the end of the matrix solution. This is the time at the end of matrix triangularization.

```
SINGULARITY RATIO    1.8140E-01
END OF MATRIX SOLUTION
 TIME =       0.25
```

At this stage, the program enters a back substitution for the displacements. This is followed by calculation of element stress values. Default yield stress is set by the program for a linear elastic analysis.

```
MARC K7-1,  05/07/92, OUTPUT FOR INCREMENT    0.         ELASTIC ANALYSIS OF A THIN PLATE WITH HOLE
   ELEMENT WITH HIGHEST STRESS RELATIVE TO YIELD IS    8  WHERE EQUIVALENT STRESS IS 0.309E-19  OF YIELD
```

A heading is printed next. The Tresca Intensity is output for application in ASME code applications. The Mises intensity is the equivalent yield stress. Principal stress and strain values are output. This is followed by individual stress and strain components. The number of each column is to be used with the key printed at the beginning of the analysis.

```
          TRESCA     MISES     MEAN   P R I N C I P A L   V A L U E S          P H Y S I C A L   C O M P O N E N T S
        INTENSITY INTENSITY  NORMAL   MINIMUM INTERMEDIATE MAXIMUM     1         2         3         4         5         6

                          INTENSITY

 ELEMENT   20  POINT   4        INTEGRATION PT. COORDINATE=      0.234E+00     0.121E+01
 SECTION THICKNESS = 0.100E+00
 ENGSTS  5.802E-01 5.413E-01-1.342E-01-4.914E-01 0.000E+00 8.880E-02-4.531E-01 5.052E-02-1.440E-01
 ENGSTN  2.514E-08 1.550E-08-3.131E-09-1.727E-08 0.000E+00 7.874E-09-1.561E-08 6.215E-09-1.248E-08

 ELEMENT   20  POINT   6        INTEGRATION PT. COORDINATE=      0.261E+00     0.137E+01
 SECTION THICKNESS = 0.100E+00
 ENGSTS  6.055E-01 5.255E-01-2.275E-02-3.369E-01 0.000E+00 2.686E-01-2.677E-01 1.995E-01-1.926E-01
 ENGSTN  2.624E-08 1.518E-08-5.307E-10-1.391E-08 0.000E+00 1.232E-08-1.092E-08 9.326E-09-1.669E-08
```

The stress and strain results are followed by the increment of displacements and the total displacements for all the nodes. If it is requested to print and store all stress points, a printout of the reaction forces would follow the displacement output.

```
                          N O D A L   P O I N T   D A T A


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

     1 -2.17163E-08  7.15861E-08          2 -3.08177E-08  5.15029E-08          3 -4.07290E-08  3.20392E-08
     4 -4.76926E-08  1.49932E-08          5 -5.04297E-08        0.             6 -2.76616E-08  9.27126E-08
     7 -4.39062E-08  4.43055E-08          8 -5.45603E-08        0.             9 -3.22702E-08  1.16274E-07


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

     1 -2.17163E-08  7.15861E-08          2 -3.08177E-08  5.15029E-08          3 -4.07290E-08  3.20392E-08
     4 -4.76926E-08  1.49932E-08          5 -5.04297E-08        0.             6 -2.76616E-08  9.27126E-08
     7 -4.39062E-08  4.43055E-08          8 -5.45603E-08        0.             9 -3.22702E-08  1.16274E-07
```

```
                TOTAL EQUIVALENT NODAL FORCES (DISTRIBUTED PLUS POINT LOADS)

     1       0.          0.          2       0.          0.          3       0.          0.
     4       0.          0.          5       0.          0.          6       0.          0.
     7       0.          0.          8       0.          0.          9       0.          0.


          REACTION FORCES AT FIXED BOUNDARY CONDITIONS, RESIDUAL LOAD CORRECTION ELSEWHERE

     1  1.21431E-17 -3.61690E-16     2  1.24033E-16 -1.11022E-16     3 -1.86483E-16  9.54098E-17
     4  1.31839E-16  1.42247E-16     5 -4.68375E-17 -4.27307E-02     6 -7.19910E-17  1.66533E-16
     7 -5.20417E-18  1.11022E-16     8 -3.96005E-17 -0.11445         9 -1.72388E-17  1.04083E-16


                    SUMMARY OF EXTERNALLY APPLIED LOADS

 0.00000E+00    0.50000E+00

                    SUMMARY OF REACTION/RESIDUAL FORCES
-0.72045E-17   -0.50000E+00
```

The results are concluded with an indication of the magnitude of distributed loads.

```
DISTRIBUTED LOAD     TYPE     CURRENT
LIST NUMBER                   MAGNITUDE

          1            8      -1.000          0.          0.
```

The SUMMARY model definition option prompts the program to print out summary tables of stresses and strains as below:

```
*************************************************************************
*************************************************************************
*                                                                       *
*     ELASTIC ANALYSIS OF A THIN PLATE WITH HOLE                        *
*                                                                       *
*     INCREMENT     0                                      MARC K7      *
*                                                                       *
*************************************************************************
*                                   *              *      *     *      *
*              QUANTITY             *    VALUE     * ELEM.* INT.*LAYER*
*                                   *              *NUMBER*POINT*      *
*                                   *              *      *     *      *
*************************************************************************
*                                   *              *      *     *      *
* MAX FIRST  COMP. OF STRESS        *  0.52712E+00 *    7 *  2  *   1 *
* MIN FIRST  COMP. OF STRESS        * -0.11257E+01 *   18 *  7  *   1 *
*                                   *              *      *     *      *
*                                   *              *      *     *      *
* MAX SECOND COMP. OF STRESS        *  0.31370E+01 *    8 *  3  *   1 *
* MIN SECOND COMP. OF STRESS        * -0.75958E-01 *   18 *  4  *   1 *
*                                   *              *      *     *      *
*                                   *              *      *     *      *
* MAX THIRD  COMP. OF STRESS        *  0.15887E+00 *   18 *  1  *   1 *
* MIN THIRD  COMP. OF STRESS        * -0.84812E+00 *    7 *  3  *   1 *
*                                   *              *      *     *      *
*                                   *              *      *     *      *
* MAX   EQUIVALENT    STRESS        *  0.30910E+01 *    8 *  3  *   1 *
* MIN   EQUIVALENT    STRESS        *  0.26979E+00 *   17 *  4  *   1 *
*                                   *              *      *     *      *
*                                   *              *      *     *      *
* MAX      MEAN       STRESS        *  0.10821E+01 *    8 *  3  *   1 *
* MIN      MEAN       STRESS        * -0.38696E+00 *   18 *  7  *   1 *
*                                   *              *      *     *      *
*                                   *              *      *     *      *
* MAX     TRESCA      STRESS        *  0.31419E+01 *    8 *  3  *   1 *
```

```
* MIN      TRESCA       STRESS          *  0.29647E+00 *    17 *  4  *    1 *
*                                       *              *       *     *      *
*                                       *              *       *     *      *
* MAX FIRST  COMP. OF TOTAL STRAIN      *  0.58578E-08 *     7 *  1  *    1 *
* MIN FIRST  COMP. OF TOTAL STRAIN      * -0.37172E-07 *    18 *  7  *    1 *
*                                       *              *       *     *      *
*                                       *              *       *     *      *
* MAX SECOND COMP. OF TOTAL STRAIN      *  0.10347E-06 *     8 *  3  *    1 *
* MIN SECOND COMP. OF TOTAL STRAIN      *  0.34023E-08 *    17 *  7  *    1 *
*                                       *              *       *     *      *
*                                       *              *       *     *      *
* MAX THIRD  COMP. OF TOTAL STRAIN      *  0.13769E-07 *    18 *  1  *    1 *
* MIN THIRD  COMP. OF TOTAL STRAIN      * -0.73504E-07 *     7 *  3  *    1 *
*                                       *              *       *     *      *
*                                       *              *       *     *      *
* MAX    EQUIVALENT    TOTAL STRAIN     *  0.87678E-07 *     8 *  3  *    1 *
* MIN    EQUIVALENT    TOTAL STRAIN     *  0.77458E-08 *    17 *  4  *    1 *
*                                       *              *       *     *      *
*                                       *              *       *     *      *
* MAX       MEAN       TOTAL STRAIN     *  0.00000E+00 *     1 *  1  *    1 *
* MIN       MEAN       TOTAL STRAIN     *  0.00000E+00 *     1 *  1  *    1 *
*                                       *              *       *     *      *
*************************************************************************
```

The message `END OF INCREMENT 0` signifies the end of analysis for 0th increment. Additional output concerns only with post plottings. The output is finally concluded by plot messages, since plotting was requested.

```
*************************************************************************
*************************************************************************
*                                                                       *
*     ELASTIC ANALYSIS OF A THIN PLATE WITH HOLE                        *
*                                                                       *
*     INCREMENT     0                                     MARC K7       *
*                                                                       *
*************************************************************************
*                                    *              *      *     *      *
*              QUANTITY              *    VALUE     * ELEM.* INT.*LAYER*
*                                    *              *NUMBER*POINT*      *
*                                    *              *      *     *      *
*************************************************************************
*                                    *              *      *     *      *
* MAX      TRESCA       TOTAL STRAIN *  0.13162E-06 *    8 *  3  *   1 *
* MIN      TRESCA       TOTAL STRAIN *  0.12847E-07 *   17 *  4  *   1 *
*                                    *              *      *     *      *
*                                    *              *      *     *      *
* MAX TEMPERATURE                    *  0.00000E+00 *    1 *  1  *   1 *
* MIN TEMPERATURE                    *  0.00000E+00 *    1 *  1  *   1 *
*                                    *              *      *     *      *
*************************************************************************
*************************************************************************


*********************************************************************
*********************************************************************
*                                                                   *
*     ELASTIC ANALYSIS OF A THIN PLATE WITH HOLE                    *
*     INCREMENT     0                                 MARC K7       *
*                                                                   *
*********************************************************************
*                                    *              *        *
*              QUANTITY              *     VALUE    *  NODE  *
```

```
*                                    *                *  NUMBER  *
*                                    *                *          *
*********************************************************************
*                                    *                *          *
* MAX FIRST  COMP. OF INCREMENTAL DISP *  -0.19968E-08  *     48   *
* MIN FIRST  COMP. OF INCREMENTAL DISP *  -0.73223E-07  *     21   *
*                                    *                *          *
*                                    *                *          *
* MAX SECOND COMP. OF INCREMENTAL DISP *   0.20382E-06  *     61   *
* MIN SECOND COMP. OF INCREMENTAL DISP *   0.14872E-07  *     26   *
*                                    *                *          *
*                                    *                *          *
* MAX FIRST  COMP. OF TOTAL DISP.      *  -0.19968E-08  *     48   *
* MIN FIRST  COMP. OF TOTAL DISP.      *  -0.73223E-07  *     21   *
*                                    *                *          *
*                                    *                *          *
* MAX SECOND COMP. OF TOTAL DISP.      *   0.20382E-06  *     61   *
* MIN SECOND COMP. OF TOTAL DISP.      *   0.14872E-07  *     26   *
*                                    *                *          *
*                                    *                *          *
* MAX FIRST  COMP. OF REACTION FORCE   *   0.12293E-01  *     73   *
* MIN FIRST  COMP. OF REACTION FORCE   *  -0.13867E-01  *     57   *
*                                    *                *          *
*                                    *                *          *
* MAX SECOND COMP. OF REACTION FORCE   *  -0.13839E-01  *     34   *
* MIN SECOND COMP. OF REACTION FORCE   *  -0.11445E+00  *      8   *
*                                    *                *          *
*********************************************************************
*********************************************************************


E N D   O F   I N C R E M E N T   0
 TIME =       1.17
```

The MARC exit number 3004 indicates the problem is completed.

2 List of Parameters

Parameters

Parameters

2 *List of Parameters*

2 Parameters

This chapter describes the parameter section of the MARC input file. It is the first section of the file. The parameter section is used to specify the title of the file, the work space requirements, the elements to be used in the analysis, and the type of analysis to be performed.

Only the TITLE, SIZING, and END parameters are ***required***. Optional parameters flag the use of certain elements, analysis capabilities, or change the default values. The first ten columns of the parameter data are reserved for the key words which control the input of the parameters. These key words must be entered as left justified. Some options are set by the order in which data is input.

This chapter is organized according to loosely defined categories of parameter types, as shown in the list below.

- Basic Input Requirements
- Analysis Types
- Rezoning and Substructure Parameters
- Additional Flags for Various Analyses
- Program Function and I/O Controls
- Modifying Default Values
- Defining Cross-sections of Beam Elements

2

Basic Input Requirements

■ TITLE

Output Title Definition

This parameter is REQUIRED.

Description

This required parameter defines the output title. There is no limit to the number of the title data read in as long as the word TITLE appears in the first field. However, only the last TITLE data is used as an output header. Due to the free-format processor, do not place commas within the TITLE data (Columns 11-80).

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word TITLE.
11-80	2nd	A	Enter the title to be output with results.

SIZING

Working Space Definition

This parameter option is REQUIRED.

Description

This required parameter sets up the allocation of working space required for the problem. In the K7 release and later, the number entered as the value on the SIZING parameter is used for the initial memory allocation. The program uses additional memory if necessary and it is available. See Appendix B for more details. Values which are too large waste memory. Most entries here should be filled in after the other data has been completed.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word SIZING.
11-20	2nd	I	Size of work space vector, MAXALL.
21-25	3rd	I	Maximum number of elements.
26-30	4th	I	Maximum number of nodal points.
31-35	5th	I	Maximum number of degrees of freedom constrained (displacement or temperature boundary conditions).
36-40	6th	I	Maximum number of elements in the largest list of distributed loads or fluxes (see DIST LOADS or FLUXES box).
41-45	7th	I	Library code of the first type of element selected.
46-50	8th	I	Library code of the second type of element selected.
51-55	9th	I	Library code of the third type of element selected.
Etc.	Etc.	I	Etc. Up to 8 elements types in I5 format.

In version K and beyond, the third to last values set here are all optional values. If entered, they will be the maximum allowed values. These values should be set to an upper bound if a rezoning analysis will be performed.

The element types are described in the *MARC Reference Library, Volume B: Element Library.*

Note that the ALIAS parameter is available to change element library code descriptions on the CONNECTIVITY model definition block.

Note that Table 2-1 and Table 2-2 have been prepared to help with the estimation of the work space vector. The estimate is approximate and the user should increase the estimate by ten percent. The estimate for the work space vector is given by taking the appropriate quantities from Table 2-1 and Table 2-2 and using the following formula:

Work Space in decimal = Element Stiffness Generation + Number of elements x storage per element + nodal vectors + space for solution.

The space for solution in the out-of-core solution must be multiplied by the number of degrees of freedom transferred into core at any one time. (This consists of MXRD x DOF where MXRD is the number of nodes transferred at any one time.) Note that the number of transfer times in an out-of-core solution is inversely proportional to the square of the number of degrees of freedom transferred into core at any one time. Hence, it is advantageous to increase the number of degrees of freedom transferred into core at any one time.

Table 2-1 Estimated Core Storage for Elements in Words

	Overhead for Element Generation in Words per Element	Centroid Only in Words per Element	All Points in Words per Element
Linear Beams and Shells	700	150	600
2D Triangles	400	40	40
2D 4-Node Quadrilaterals	1200	40	160
2D 8-Node Quadrilaterals	1000	120	500
Shells Triangular	400	200	1100
Shells Quadrilateral	7000	400	1500
3D 8-Node	2500	60	250
3D 20-Node	9000	120	1100
Note: For machines requiring double precision, double the number of words required.			

Table 2-2 Estimated Core Storage for Matrix Solution and Displacement Vectors in Words

Total Degrees of Freedom for Problem	Nodal Vectors	Space for Stiffness Matrix Assume 0.2 Bandwidth	
		In Core	Out-of-core Per DOF in Block Transfer
50	500	500	20
100	1,000	2,000	40
500	5,000	50,000	200
1,000	10,000	200,000	400
2,500	25,000	1,250,000	1,000
5,000	50,000	5,000,000	2,000
7,500	75,000	11,250,000	3,000
10,000	100,000	20,000,000	4,000

■ ELEMENTS

Element Type Selection

This parameter is REQUIRED.

Description

This required parameter is used to identify the elements used in the analysis. Element codes for all the allowable element types are found in the *MARC Reference Library, Volume B: Element Library*. This is an alternative mechanism to the SIZING data. This data can be repeated as often as necessary.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word ELEMENTS.
11-15	2nd	I	Library code of the first type of element selected.
16-20	3rd	I	Library code of the second type of element selected.
21-25	4th	I	Library code of the third type of element selected.
Etc.	Etc.	I	Etc. up to 14 element types.

■ PROCESSOR

Vectorization and Parallelization Control

Description

This parameter controls the use of vectorization and parallelization in the element assembly phase.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word PROCESSOR.
11-15	2nd	I	Enter the number of CPUs to use.
16-20	3rd	I	Enter the optimal vector length. Defaults to 32 or 64, depending upon the machine.
21-25	4th	I	Enter 1 if beta matrices are to be formed in parallel.

$NO LIST

No Listing of Input Data

Description

Using this parameter results in the suppression of the printout of the remainder of the input file.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the words $NO LIST.

■ EXTENDED

Extended Precision of Reading in Data

Description

This parameter is used to indicate that models are to be in extended precision and/or a large number of elements or nodes exist in the model. If this option is included, then the width of all the data fields described in this manual must be doubled. For example, all I5 integer fields change to I10. If this parameter is included, all input lines must be in this format.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word EXTENDED.

■ END

End of Parameter Section

This parameter option is REQUIRED.

Description

This required parameter terminates the input of parameter data, signaling the end of the parameter section.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word END.

Analysis Types

■ ELASTIC

Elastic Analysis with Multi-loads

Description

This parameter allows repeated back substitution on a series of load vectors. This parameter uses the decomposed stiffness matrix. Total loads must be input with the POINT LOAD, DIST LOADS, or CHANGE STATE/THERMAL LOADS blocks after END OPTION.

The decomposed stiffness matrix is only written once to the restart tape, following increment 0. When using the RESTART option with this parameter, always restart at increment 0.

When the ADAPTIVE meshing option is used in conjunction with this parameter, only the loads before the END OPTION (increment zero) are considered. This load is then re-analyzed until the error criteria is satisfied.

Notes: This data should *never* be used with any data which flags nonlinear analysis or which change the stiffness matrix; for example, the LARGE DISP parameter or the DISP CHANGE option.

If temperature dependent material properties are included, then a new assembly is performed (if temperature loading is on).

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-10	1st	A	Enter the word ELASTIC.
11-15	2nd	I	Element storage parameter, to reduce storage in elastic analysis. Set to 1, so that creep, swelling, plastic, incremental strains, plastic strain rates, and incremental stresses are not stored. Set to 2, so that in addition to 1 above, strain energies, thermal strains, and elastic strains are not stored. **Note:** If you request these items on the post file and they are not stored, the information will be incorrect.

DESIGN SENSITIVITY

Perform Sensitivity Analysis Only

Description

This parameter invokes the design sensitivity capability in MARC. In this release, the capability is restricted to linear static structural analysis and eigenvalue analysis. This option requires the model definition options: DESIGN VARIABLES and at least one of DESIGN DISPLACEMENT CONSTRAINTS, DESIGN STRESS CONSTRAINTS, DESIGN STRAIN CONSTRAINTS, or DESIGN FREQUENCY CONSTRAINTS. If multiple load cases are to be evaluated, the ELASTIC parameter should be included.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-18	1st	A	Enter the words DESIGN SENSITIVITY. Fixed field can be truncated to 12 characters
N/A	2nd	A	Enter the word SORT if you desire to sort the constraints by degree of criticalness (optional), and also optionally to limit the number of constraints to be analyzed (see third field).
N/A	3rd	I	Only if SORT is invoked, enter the number of most critical constraints to be analyzed. If no number is entered, the default number (100) of most critical constraints are isolated by sorting and are subjected to sensitivity analysis.

■ DESIGN OPTIMIZATION

Perform Design Optimization

Description

This parameter invokes the design optimization capability in MARC. In this release, the capability is restricted to linear static structural analysis and eigenvalue analysis. This option requires the model definition options: DESIGN OBJECTIVE, DESIGN VARIABLES and at least one of DESIGN DISPLACEMENT CONSTRAINTS, DESIGN STRESS CONSTRAINTS, DESIGN STRAIN CONSTRAINTS, or DESIGN FREQUENCY CONSTRAINTS. The DESIGN OBJECTIVE option is used to define the objective function. If multiple load cases are to be evaluated, the ELASTIC parameter should be included.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-19	1st	A	Enter the words DESIGN OPTIMIZATION. Fixed field can be truncated to 12 characters.
N/A	2nd	A	Enter the word ACTIVESET (optional).
N/A	3rd	I	Only if ACTIVESET is invoked, enter the maximum number of constraints for the active set. This does not limit the number of constraints that can be prescribed by the user. Default is 100.
N/A	4th	A	Enter the word CYCLES (optional).
N/A	5th	I	Only if CYCLES is invoked, enter the maximum number of design optimization cycles (including analyses). Default is 20.

Note: The order of the ACTIVESET and CYCLES can be reversed; for example, Free Formats 4th and 5th become 2nd and 3rd while Free Formats 2nd and 3rd become 4th and 5th.

■ ADAPTIVE

Adaptive Mesh Refinement

Description

This parameter allows an adaptive meshing analysis to be performed. The error criteria used is specified through the ADAPTIVE model definition option.

In an elastic analysis, the program iterates based upon the excitation given to satisfy an error tolerance. The ELASTIC parameter must be included.

In a nonlinear incremental analysis, the program adapts the mesh at each increment, or user controlled frequency to improve the solution.

New elements are created as described in the *MARC Reference Library, Volume A: Theory and User Information.*

This capability can be used with the following lower order linear, triangular, quadrilateral, tetrahedral, and brick elements.

Format

Format		Data	
Fixed	Free	Type	Entry
1-8	1st	A	Enter the word ADAPTIVE.
11-15	2nd	I	Enter an upper bound to the number of elements in the mesh.
16-20	3rd	I	Enter an upper bound to the number of nodes in the mesh.
21-25	4th	I	Enter 1 to continue to perform an incremental analysis. If the number of nodes or elements created exceeds the maximums specified, the previous mesh is used.

■ LINEAR

Matrices Saved for Linear Analysis

Description

This parameter allows additional values to be stored rather than being recalculated during subsequent increments. This means an increase in the overall size of the workspace used for the problem, but may actually result in a reduced computation time. The efficiency of this parameter is highly dependent upon the analysis data and the machine on which the problem is computed. It has proven very effective in reducing computation time for linear elastic and small displacement dynamic problems. When set to 0, the parameter has also been used effectively on nonlinear problems such as rigid plastic flow.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-10	1st	A	Enter the word LINEAR.
11-15	2nd	I	Enter 0 (default) to save BETA matrix (Strain-Displacement). Enter 1 to save the BETA matrix and the stress-strain law.

■ FOURIER

Arbitrary Loading of Axisymmetric Structures

Description

This parameter governs the analysis of axisymmetric structures under arbitrary loading by means of the Fourier series expansion technique. See the *MARC Reference Library, Volume A: Theory and User Information* for a description of this analysis technique. To perform a modal Fourier analysis, you must include a DYNAMIC parameter. To perform a Fourier buckling analysis, you must include a BUCKLE parameter.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word FOURIER.
11-15	2nd	I	Total number of FOURIER series expansions needed for characterizing the circumferential variation of tractions, thermal loads and boundary conditions.
16-20	3rd	I	Maximum number of harmonics in any of the series. The number of series terms will be two times the number of harmonics plus one.
21-25	4th	I	Total number of nodal degrees of freedom that are loaded by concentrated forces or restrained by nonzero boundary conditions described by a FOURIER series expansion.
26-30	5th	I	If only symmetric (cos) terms are present in all expansions used, set this flag to 1. For strictly antisymmetric (sin) expansions, set this flag to 2. Default is 0 which allows for the full expansion containing sine and cosine terms. To skip increments 0 and 1 for symmetric terms only, set this flag to -1. To skip increments 0, 1 and 2 for antisymmetric terms only, set this flag to -2.

Format Fixed	Format Free	Data Type	Entry
			A negative value of this flag means that no constant loading or constant nonzero boundary condition around the circumference are present.
31-35	6th	I	Maximum number of stations around circumference used for printout during the superposition using CASE COMBIN option. Default is 24.
36-40	7th	I	Enter 1 if the initial stress stiffness is to be included in modal Fourier calculation.

■ DYNAMIC

Dynamic Analysis

Description

This parameter sets the flags for one of several possible dynamic analysis methods. Any of several optional data blocks can be required. See the *MARC Reference Library, Volume A: Theory and User Information*. The MODAL SHAPE history definition or the MODAL INCREMENT model definition options control the eigenvalue extraction. The DYNAMIC CHANGE or AUTO TIME history definition options control the time steps. The RECOVER history definition option allows for modal stress recovery or storing eigenvectors on the post file. Eigenvectors can also be stored on the post file with the MODAL INCREMENT option.

Note: 1) The central difference operator will not work with zero mass at any degrees of freedom.

2) The direct integration operators automatically use residual load correction, and this cannot be overridden. The use of ALL POINTS is recommended for these options.

3) The Houbolt and central difference (IDYN=4) operators can only be used with constant time step. If the time step is changed during analysis, results will be in error. The fast central difference operator (IDYN=5) may use a variable time step.

4) Rigid body modes can be handled by the inverse power sweep or Lanczos method. Use the flag in the CONTROL block for solving a singular equation.

5) The fast central difference operator may be used with element types (2, 3, 5, 6, 7, 9, 10, 11, 18, 19, 20, 52, 64, 75, 98, 114, 115, 116, 117, 118, 119, 120).

6) The Newmark-beta method is unconditionally stable for linear analysis, with $\beta = 0.25$, $\gamma = 0.50$. These parameters can be reset through the PARAMETERS option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word DYNAMIC.
11-15	2nd	I	Enter the dynamic operator type (IDYN).
			Set to 1 for modal superposition dynamic response.
			Set to 2 for Newmark direct integration.
			Set to 3 for Houbolt direct integration.
			Set to 4 for explicit direct time integration using central difference.
			Set to 5 for fast explicit direct integration.

Format Fixed	Format Free	Data Type	Entry
16-20	3rd	I	Maximum number of modes to be used in the modal superposition dynamic option. If the inverse power sweep method is used for eigenvalue analysis, it is also the number of mode shapes and frequencies to be extracted.
21-25	4th	I	Set to 0 for Inverse power sweep with double eigenvalue extraction. Set to 1 to for the Lanczos method. Set to 3 for Inverse power sweep with single eigenvalue extraction.
26-30	5th	I	Enter 1 if modal stress recovery or storing eigenvectors on post tape is to be performed in this analysis.

■ HARMONIC

Frequency Response Analysis

Description

The HARMONIC parameter allows the frequency response analysis to be superimposed upon the deformed configuration. This parameter can also be used in conjunction with the parameter EL-MA for electromagnetic (harmonic) analysis.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word HARMONIC.
11-15	2nd	I	Enter 1 if complex damping matrix is used. Default is no complex damping.
16-20	3rd	I	Maximum number of excitation boundary conditions.
21-25	4th	I	Maximum number of excitation distributed load lists.
26-30	5th	I	Maximum number of elements in any excitation distributed load list.
31-35	6th	I	Enter 1 to include inertia effects in the calculation of the harmonic reaction force.

■ RESPONSE

Spectrum Response Analysis

Description

This parameter allows the user to perform a spectrum response analysis. See the *MARC Reference Library, Volume A: Theory and User Information* for detailed directions. The modes used are specified in the SPECTRUM response load incrementation data. To perform a SPECTRUM response calculation, it is also necessary to include the DYNAMIC parameter and either the MODAL INCREMENT or MODAL SHAPE option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word RESPONSE.
11-15	2nd	I	Enter the number of data points used to specify spectral density function. Enter 0 to use user subroutine USSD.

■ R-P FLOW

Rigid-Plastic Flow

Description

This parameter is used to specify a rigid, perfectly-plastic flow analysis. See the *MARC Reference Library, Volume A: Theory and User Information* for an introduction to this technique. This parameter is used either with the Herrmann elements or with conventional elements. In the latter case, a penalty function is used to apply the incompressibility constraint. The penalty factor is defined through the PARAMETERS option.

This option has two modes. In the first mode, a steady state solution is obtained.

This option can also be used for the analysis of laminar fluid flow problems. See user subroutine UNEWTN in *Volume D: User Subroutines and Special Routines*. This method requires iteration on the velocity field for convergence; convergence controls are input in the CONTROL option.

In the second mode, a transient solution is obtained. This mode is always used in contact problems. This method required iteration on the incremental displacements. Increment 0 is suppressed.

In this formulation, if the strain rate falls below a certain value, the material is effectively rigid. This cutoff value is specified through the PARAMETERS or CONTROL option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-8	1st	A	Enter the words R-P FLOW.
11-15	2nd	I	Enter 1 for steady state procedure.
			Enter 2 for transient procedure.

■ LARGE DISP

Large Displacement or Buckling

Description

This parameter is used to specify large displacement or buckling analysis. It signals the program to calculate the geometric stiffness matrix and the initial stress stiffness matrix. This parameter automatically switches on the residual load correction option and switches off the scaling option. Default is no large displacement terms. See the *MARC Reference Library, Volume A: Theory and User Information* for more information about large displacement and buckling analysis.

Note: The CENTROID parameter should not be used in conjunction with this parameter.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-10	1st	A	Enter the words LARGE DISP.

■ UPDATE

Updated Lagrange Procedure

Description

This parameter flags the use of the classical updated Lagrange procedure for elastic-plastic materials, the elements for which such a formulation can be applied. The use of the procedure has two consequences. First, the element stiffnesses are assembled in the current configuration of the element. Second, the stress and strain output is given in the coordinate system which is applicable in the updated configuration of the element.

The procedure is useful for:

A. Analysis of shell and beam structures in which rotations are large and the nonlinear terms in the curvature expressions can no longer be neglected.

B. Large strain plasticity analysis, for calculations in which the plastic deformations cannot be assumed infinitesimal. For this type of analysis, the FINITE parameter must be included.

The updated Lagrange procedure can be used with or without the LARGE DISP parameter. With the LARGE DISP parameter invoked, the effect of the internal stresses on the stiffness is taken into account. Also, the strain increment is calculated to second order accuracy and, hence, large rotation increments might be allowed. Refer to the *MARC Reference Library, Volume B: Element Library* for a list of the elements that can be used in an updated Lagrangian analysis.

When the UPDATE parameter is used in conjunction with a coupled thermal-stress analysis, the element conductivity is assembled based on the current configuration of the element.

Note: For greater flexibility, the ELASTICITY and PLASTICITY parameters are an alternative to this parameter.

Format

Format		Data	
Fixed	Free	Type	Entry
1-6	1st	A	Enter the word UPDATE.

■ FINITE

Finite Strain Plasticity

Description

This parameter flags the use of the large strain plasticity option. With this option invoked, the effects of the change in metric due to large inelastic deformations is included. This results in a different stiffness of the structure as well as in a modified calculation of stresses and inelastic strains. This parameter is only used for the elements which are formulated in terms of the updated Lagrange procedure. When using this parameter, true stresses are printed out.

The UPDATE parameter must be included in all cases when this parameter is invoked. When the FINITE parameter is used, the work hardening slope for plasticity is defined as the rate of true stress versus the true plastic strain rate. Hence, the work hardening curve must be entered as the true stress versus logarithmic plastic strain curve in a uniaxial tension test. The anisotropic plasticity formulation cannot be used with this option. The finite strain option in MARC is written such that fairly large strain increments (up to 3%) can be allowed. However, large increments can result in many recycles as well as in decreased accuracy.

Format

Format		Data	
Fixed	Free	Type	Entry
1-6	1st	A	Enter the word FINITE.

■ CONSTANT DILATATION

Define That Elements Are to Use Constant Dilatation Formulation

Description

When performing nearly incompressible analysis with displacement based elements, the conventional isoparametric interpolation methods result in poor behavior for lower order elements. This results in overly stiff behavior when using element type 7 (3D brick), type 10 (axisymmetric), type 11 (plane strain), type 19 (generalized plane strain) or type 20 (axisymmetric with twist). When this option is included, all elements of these types are modified to use the constant dilatation formulation. This is recommended for elastic-plastic analysis and creep analysis because of the potentially nearly incompressible behavior.

Format

Format		Data	
Fixed	Free	Type	Entry
1-19	1st	A	Enter the word CONSTANT.

■ ASSUMED STRAIN

Improved Bending Behavior

Description

The bending behavior can be improved by using the assumed strain formulation for element type 3 (plane stress), type 11 (plane strain), and type 7 (3D brick). This procedure replaces the standard linear interpolation functions with an enriched group that is able to represent pure bending behavior. This formulation results in improved accuracy for isotropic behavior, but it should be noted that the computational costs increase.

Format

Format		Data	
Fixed	Free	Type	Entry
1-14	1st	A	Enter the words ASSUMED.

■ ELASTICITY

Elasticity Procedure

Description

This option can be used to define which formulation is used for large strain elasticity. The default is that the total Lagrange formulation is used. Note that when using Mooney or Ogden material model with the total Lagrange procedure, if the elements are plane strain, generalized plane strain, axisymmetric, axisymmetric with twist or three-dimensional solids, the Herrmann incompressible elements must be used. For the foam material model, conventional displacement elements are used. The alternative model is written in the updated Lagrange formulation. This model can be used in conjunction with the Mooney or Ogden material models. When using this formulation, conventional elements are used. For more details, see the *MARC Reference Library, Volume A: Theory and User Information.*

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word ELASTICITY.
11-15	2nd	I	Enter 1 for total Lagrange formulation. Enter 2 for updated Lagrange formulation.

■ PLASTICITY

Plasticity Procedure

Description

This option can be used to define the plasticity procedure that is used in MARC. The default is the mean normal procedure for satisfying the yield criteria and the additive decomposition of the incremental strains into elastic and plastic parts. For problems which have large elastic and plastic strains, the multiplicative decomposition is more accurate. The multiplicative decomposition implementation requires that the elasticity is isotropic. For more details, see the *MARC Reference Library, Volume A: Theory and User Information.*

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word PLASTICITY.
11-15	2nd	I	Enter 1 for additive decomposition using the mean normal method; small strain formulation. Enter 2 for additive decomposition using the radial return method; small strain formulation. Enter 3 for additive decomposition using the mean normal method; large strain formulation using the updated Lagrange procedure. Enter 4 for additive decomposition using the radial return method; large strain formulation using the updated Lagrange procedure. Enter 5 for multiplicative decomposition (F_eF_p) using the radial return method and the three field variational principle; large strain formulation using the updated Lagrange procedure.

■ FOLLOW FOR

Follower Forces: Distributed Load Application on Current Geometry – Application of Total Boundary Condition

Description

The FOLLOW FOR parameter is used for follower force (for example, pressure) problems. When this parameter is used, all distributed loads are formed on the basis of current geometry. This parameter requires the use of the residual load correction and, therefore, forces the use of that option regardless of other parameters (for example, the NO LOADCOR parameter is ignored).

Whenever FOLLOW FOR is used, the distributed load magnitude given in user subroutine FORCEM must be the total magnitude to be reached after the current increment, and not the incremental magnitude. If the loads are specified, incremental values are given as usual.

In a coupled thermal-stress analysis, the fluxes are based upon the current geometry.

Boundary conditions in structural analysis are normally entered as incremental values. To specify total values, use the third field of this option

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the words FOLLOW FOR.
11-15	2nd	I	Enter 1 if follower force stiffness is not required (default). Enter 2 if follower force stiffness is to be included. Enter 3 if the follower force is based upon the displacement at the beginning of the increment, as opposed to the last iteration. Enter -1 if the undeformed geometry is required but total values of load are to be used.

Format Fixed	Format Free	Data Type	Entry
16-20	3rd	I	Enter 1 if total values of boundary conditions are to be entered on DISP CHANGE, POINT LOADS, and DIST LOADS option as opposed to the default incremental loads. **Note:** If the follower force stiffness is included, the use of the SOLVER option can be used to specify a nonsymmetric formulation. This improves convergence, but results in longer solver times.

Follower force stiffness is available for element types (3, 7, 10, 11, 18, 72, 75, 80, 82, 84, 114, 115, 116, 117, 118, and 119).

BUCKLE

Buckling Load Estimation via Eigenvalue Analysis

Description

This parameter specifies the use of buckling load estimation by eigenvalue analysis, based on a perturbation of the tangent stiffness. Multiple eigenvectors are allowed for the case where the closest root to the current load set is not pertinent. Either the inverse power sweep method or the Lanczos method can be used. The BUCKLE history definition option or the BUCKLE INCREMENT modal definition option controls the eigenvalue extraction. The RECOVER history definition option allows for modal stress recovery or storing eigenvectors on a post file. The LARGE DISP parameter should be included to obtain the nonlinear collapse load estimate.

For additional information about this type of analysis, see *"Effective Use of the Incremental Stiffness Matrices in Nonlinear Geometric Analysis"* by Dupuis, Pfaffinger and Marcal in the *MARC Reference Library, Volume F: Background Information.*

Format

Format		Data	
Fixed	Free	Type	Entry
1-6	1st	A	Enter the word BUCKLE.
11-15	2nd	I	Maximum number of buckling modes to be estimated at any time.
16-20	3rd	I	Number of buckling modes with positive eigenvalues to be estimated at any time. In many buckling problems, collapse modes corresponding to loads of opposite magnitude to those of interest exist. By specifying a larger number of modes (say 5) in Columns 11-15 and one or two modes in this field, you can ensure getting the one or two modes you are interested in. The program stops the modal search when all these modes have been formed, or when all the modes requested in columns 11-15 have been formed, whichever occurs first. If this field is left blank, all modes asked for in columns 11-15 are formed regardless of sign.

Format		Data	
Fixed	Free	Type	Entry
21-25	4th	I	Enter 1 if modal stress recovery or storing eigenvectors on post tape is to be performed in this analysis.
26-30	5th	I	Enter 3 to perform non-axisymmetric Fourier buckling.
31-35	6th	I	Enter 1 to use inverse power sweep with single eigenvalue extraction.
36-40	7th	I	Enter 0 if inverse power sweep method is to be used (default). Enter 1 if Lanczos method is to be used.

■ CREEP

Creep Analysis

Description

This parameter specifies a creep analysis. For more information about creep analysis, see the Creep Constitutive Data block model definition block in the Material Properties section of this manual and *Volume A: Theory and User Information* of the *MARC Reference Library.* The MARC user subroutines CRPLAW and VSWELL, used with creep analysis, are explained in the *MARC Reference Library, Volume D: User Subroutines and Special Routines.*

Note: When using the implicit Maxwell creep model, the stress dependence must be in exponential form, user subroutine CRPLAW cannot be used.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word CREEP.
11-15	2nd	I	Enter the flag for type of explicit creep analysis. Default of 0, normal creep (Maxwell Model); 1, viscoplastic creep; 2, viscoplastic creep with nonassociative flow rule.
16-20	3rd	I	Enter 1 for explicit Kelvin Model. (This is identical to the VISCO ELAS parameter.)
21-25	4th	I	Enter 1 for implicit Maxwell creep or implicit viscoplastic model. (See note, above.)
26-30	5th	I	For the implicit Maxwell creep model or implicit viscoplastic model: Enter 0 for elastic tangent Enter 1 for secant tangent Enter 2 for radial return

■ VISCO ELAS

Visco Elastic Analysis (Kelvin Model)

Description

This parameter flags the use of user subroutine CRPVIS to model generalized Kelvin material behavior using an explicit procedure. See the *MARC Reference Library, Volume A: Theory and User Information* for details. This parameter automatically flags the CREEP option as well, so that Maxwell behavior (user subroutines VSWELL, CRPLAW) can be included with CRPVIS.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the words VISCO ELAS.

■ COUPLE

Coupled Thermal-Stress Analysis

Description

This parameter allows a coupled thermal-stress analysis. In these problems, the independent variables are displacements and temperatures. If the user defines displacement elements in the connectivity, heat transfer capabilities are included for these elements. To obtain the coupling between plastic work and internal heat generated, a DIST FLUXES model definition option with a flux type of 101 must be included. User subroutine CUPFLX can be used to define an alternative model for internal heat generation. Care must be taken in defining the factor used to convert inelastic mechanical energy to thermal energy (see model definition option CONVERT). If shell elements are present or latent heats are used, the HEAT parameter might also be required.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word COUPLE.

■ FLUID

Fluid, Fluid-Thermal, Fluid-Solid, and Fluid-Thermal-Solid Analysis

Description

This parameter controls the procedure when performing a fluid analysis. In this release, MARC provides the ability to solve the Navier Stokes equations, excluding turbulence for incompressible fluids. MARC offers either weakly (staggered method) or strongly (simultaneous method) procedures in multi-physics type problems. Using the weak formulation, more iterations might be necessary, but overall computation costs might be less. For fluid-thermal problems, the strongly coupled procedure is recommended; while for fluid-solid problems, the weakly coupled procedure should be used. Furthermore, you can select how the fluid incompressibility conditions are to be satisfied. Either a mixed method, with degrees of freedom of velocity and pressure or a penalty method with degrees of freedom of velocity can be used with the continuum elements. The penalty factor can be entered through the PARAMETERS option. For more details, see the *MARC Reference Library, Volume A: Theory and User Information.*

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word FLUID.
11-15	2nd	I	Enter one of the following codes: 10 – fluid analysis – mixed method 11 – fluid analysis – penalty method 12 – fluid-thermal – mixed method – strong coupling 13 – fluid-thermal – penalty method – strong coupling 2 – fluid-thermal – mixed method – weak coupling 3 – fluid-thermal – penalty method – weak coupling 40 – fluid-solid – mixed method – weak coupling 41 – fluid-solid – penalty method – weak coupling

Format Fixed	Format Free	Data Type	Entry
			42 – fluid-thermal-solid – mixed method – strong – weak coupling 43 – fluid-thermal-solid – penalty method – strong – weak coupling 44 – fluid-thermal-solid – mixed method – weak – weak coupling 45 – fluid-thermal-solid – penalty method – weak – weak coupling
16-20	3rd	I	Enter 1 to obtain gradients and fluxes at integration points. Enter 2 to obtain in addition external flux values at nodal points. If this field is left blank, only temperatures at integration points and nodal temperatures are printed.

■ PORE

Soil Analysis

Description

This parameter sets the flags for one of several possible soil analysis. It is possible to perform either a pore pressure calculation (transient or steady state), a soil analysis (including the effects of previously obtained pore pressures), or a coupled pore-soil plasticity analysis. For information about soil analyses, see the *MARC Reference Library, Volume A: Theory and User Information.*

Notes: If only a pore pressure calculation is be performed, use element types 41, 42 or 44.

If an uncoupled soil calculation is to be performed, use element types 27, 28 or 21.

If a coupled fluid-soil analysis is to be performed, use element types 32,33 or 35.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word PORE.
11-15	2nd	I	Enter 0 if pore pressure data is to be entered. Enter 1 if a steady state pore pressure calculation is to be performed. Enter 2 if a transient pore pressure calculation is to be performed.
16-20	3rd	I	Enter 0 if only the pore pressure is to be calculated; for example, no stress analysis. Enter 1 if a stress analysis is to be performed.

Example

If a fully coupled analysis is required, enter:

```
PORE, 2, 1
```

■ T-T-T

Time-Temperature-Transformation

Description

This parameter allocates storage for the time-temperature-phase dependent properties. The properties themselves are defined using the TIME-TEMP model definition option.

Most materials, when quenched or air cooled from a sufficiently high temperature, exhibit a change in mechanical or thermal properties. At any stage during the cooling process, these properties are usually dependent not only on the current temperature but also on the previous thermal history. This is due to the fact that the properties are influenced by the internal microstructure of the material and this in turn usually depends on the rate at which the temperature changes. Only in instances where the temperature is changed very gradually does the material respond in an equilibrium manner where properties are simply a function of the current temperature. In addition, during the cooling process, certain solid-solid phase transformations can occur. These represent another form of change in the material microstructure which can influence the mechanical or thermal properties. This parameter allows you to account for the time-temperature-transformation interrelationships of what are generally termed thermomechanical effects. For more information about this type of analysis, see the *MARC Reference Library, Volume A: Theory and User Information.*

Format

Format		Data	
Fixed	Free	Type	Entry
1-5	1st	A	Enter the expression T-T-T.
11-15	2nd	I	Enter the maximum number of material groups with time-temperature-transformation dependent material properties (default is 1).
16-20	3rd	I	Enter the maximum number of cooling rates used to define any one property of any material group (see *Volume A: Theory and User Information* for details). Default is 3.

Format Fixed	Format Free	Data Type	Entry
21-25	4th	I	Enter the maximum number of temperature points at which a property value is specified for any cooling rate. Default is 5.
25-30	5th	I	If the thermal coefficient of expansion for any material group is to be expressed in terms of polynomial expansions in temperature, enter the maximum number of temperature points at which an expansion is defined for any cooling rate.

HEAT

Heat Transfer (Conduction) Analysis

Description

This parameter specifies a heat transfer (conduction) analysis instead of displacement/stress analysis. Convection can be included if the velocities are known or in a steady-state rigid plastic analysis. For the solution of the coupled thermal/flow problem, the FLUID parameter should be used. For more information about heat transfer capabilities in MARC, see the *MARC Reference Library, Volume A: Theory and User Information.*

Format

Format		Data	
Fixed	Free	Type	Entry
1-4	1st	A	Enter the word HEAT.
11-15	2nd	I	Temperature distribution in thickness direction of heat transfer shell elements 50, 85, 86, 87, and 88. Enter 0 for linear temperature distribution in thickness direction. Enter 1 for quadratic temperature distribution in thickness direction. Default is 0.
16-20	3rd	I	Maximum number of latent heats associated with any material type. Default is 0.
21-25	4th	I	Enter 1 to obtain gradients and fluxes at integration points. Enter 2 to obtain in addition external flux values at nodal points. If this field is left blank, only temperatures at integration points and nodal temperatures are printed.
26-30	5th	I	Enter 2 to include convective terms. This automatically initiates the nonsymmetric solver. The velocity must be prescribed.

■ JOULE

Joule Heating (Coupled Thermo-Electrical) Analysis

Description

This parameter allows you to perform a coupled thermoelectrical (Joule heating) problem. The coupling between the electrical problem and the thermal problem is because: (1) the resistance in the electric problem is dependent on temperatures and (2) the internal heat generation in the thermal problem is a function of the electric flow. For more information about the finite element formulation of Joule heating problems, see the *MARC Reference Library, Volume A: Theory and User Information.*

In the analysis of Joule heating, the model definition options JOULE, VOLTAGE, DIST CURRENT and POINT CURRENT must be used for the definition of electric problems. However, options for the heat transfer analysis remain unchanged.

Format

Format		Data	
Fixed	Free	Type	Entry
1-5	1st	A	Enter the word JOULE.
11-15	2nd	I	Enter 1 for conventional model
			Enter 2 if electrical properties are a strong function of temperature.

■ BEARING

Bearing Analysis

Description

This parameter activates the bearing analysis facility for the analysis of lubrication problems. For more information about MARC's bearing analysis capabilities, see the *MARC Reference Library, Volume A: Theory and User Information.*

In a bearing analysis, the model definition options VELOCITY, THICKNESS, RESTRICTOR, ISOTROPIC, TEMPERATURE EFFECTS, FIXED PRESSURE, DAMPING COMPONENTS, STIFFNS COMPONENTS, and THICKNS CHANGE can be used to define the problem.

Format

Format		Data	
Fixed	Free	Type	Entry
1-7	1st	A	Enter the word BEARING.
11-15	2nd	I	Enter the maximum number of stiffness and/or damping constants to be calculated at subincrements. Default is 4.

■ ELECTRO

Electrostatic Analysis

Description

This parameter allows an electrostatic analysis to be performed. Model definition options ISOTROPIC and ORTHOTROPIC are used to define the material properties. The FIXED POTENTIAL, DIST CHARGE, and POINT CHARGE options are used to prescribe the boundary conditions while the load incrementation option STEADY STATE is used for the steady state solution. For more information about the electrostatic analysis capability in MARC, see the *MARC Reference Library, Volume A: Theory and User Information.* For information about elements used in electrostatic analysis, see *Volume B: Element Library.*

Format

Format		Data	
Fixed	Free	Type	Entry
1	1st	A	Enter the word ELECTRO.
11-15	2nd	I	Potential distribution in thickness direction of shell elements 50, 85, 86, 87, and 88. Enter 0 for linear potential distribution in thickness direction. Enter 1 for quadratic potential distribution in thickness direction. Default is 0.

■ MAGNETO

Magnetostatic Analysis

Description

This parameter specifies a magnetostatic analysis. The model definition options ISOTROPIC and ORTHOTROPIC are used for the input of isotropic or orthotropic magnetic permeabilities. The model definition options FIXED POTENTIAL, POINT CURRENT, and DIST CURRENTS are used for prescribed potential and current boundary conditions; B-H RELATION is used for the input of the variation of magnetic permeability with either the magnetic field density or the magnetic field vector. Permanent magnets can be introduced by using the PERMANENT model definition option. The load incrementation option STEADY STATE is used for the steady state option.

For more information about the magnetostatic analysis capability in MARC, see the *MARC Reference Library, Volume A: Theory and User Information*. For information about elements used in magnetostatic analysis, see the *MARC Reference Library, Volume B: Element Library.*

Format

Format		Data	
Fixed	Free	Type	Entry
1-6	1st	A	Enter the word MAGNETO.

EL-MA

Perform Electromagnetic Analysis

Description

This parameter activates the capability in the program to perform an electromagnetic analysis. The electromagnetic analysis can be either a harmonic or a transient analysis. Model definition options ISOTROPIC and ORTHOTROPIC are used to define the material properties. The FIXED POTENTIAL, DIST CURRENT, and POINT CURRENT-CHARGE options are used to prescribe the boundary conditions while the load incrementation options HARMONIC and DYNAMIC CHANGE are used for the harmonic and transient solutions, respectively. Refer to *MARC Reference Library, Volume A: Theory and User Information* for a description of the electromagnetic analysis capability in MARC. An electromagnetic analysis can be performed with element types 111, 112, or 113. See *MARC Reference Library, Volume B: Element Library* for further details.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word EL-MA.
11-15	2nd	I	Set to 0 for transient electromagnetic analysis. Set to 1 for harmonic electromagnetic analysis. Default is 0.

■ ACOUSTIC

Acoustic Analysis

Description

This parameter activates the capability to perform an acoustic analysis in a cavity with rigid reflecting surfaces. Model definition option ISOTROPIC is used to prescribe the material behavior of the fluid. Model definition options FIXED PRESSURE, DIST SOURCES, and POINT SOURCE are used to prescribe boundary conditions. The program calculates the fundamental frequencies when the MODAL SHAPE option is encountered. A transient analysis can be performed using the DYNAMIC CHANGE option.

For more information about the acoustic analysis capability in MARC, see the *MARC Reference Library, Volume A: Theory and User Information*. For information about elements used in acoustic analysis, see the *MARC Reference Library, Volume B: Element Library*.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word ACOUSTIC.
11-15	2nd	I	Maximum number of modes to be used in the modal superposition dynamic option. If the inverse power sweep method is used for eigenvalue analysis, it is also the number of mode shapes and frequencies to be extracted.
16-20	3rd	I	Flag to indicate the Lanczos method will be used. Set to 1 to force the Lanczos method.
21-25	4th	I	Enter 1 if modal stress recovery or storing eigenvectors on post file is to be performed in this analysis.

■ RADIATION

Radiation Analysis

Description

This parameter activates the radiation analysis capability in the heat transfer option for the analysis of thermal radiation. This option is used in conjunction with either the RADIATING CAVITY option or the VIEW FACTOR option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-9	1st	A	Enter the word RADIATION.
11-15	2nd	I	View factor calculation flag (IRADFL). Set to 0 to calculate view factors during analysis using RADIATING CAVITY input. Set to 1 to read view factors from a file created by the RADIATING CAVITY option. Set to 2 if radiation view factors calculated by Mentat. They are read from `jid.vfs`. See the VIEW FACTOR option. Default is 0.
16-20	3rd	I	File number for view factors. View factors will be written if IRADFL = 0, and will be read if IRADFL = 1. When IRADFL = 0 and IFILVF = 0, the view factors are not saved. Default is 0.
21-25	4th	I	Temperature unit flag. Set to 1 if user input is in degrees Celsius. Set to 2 if user input is in degrees Kelvin. Set to 3 if user input is in degrees Fahrenheit. **CAUTION:** Do not enter temperatures in degrees Rankine.
26-35	5th	F	Enter the Stefan-Boltzmann constant in the correct units.

Format Fixed	Format Free	Data Type	Entry
Fields 6 though 8 are only necessary if using RADIATING CAVITY input.			
36-40	6th	I	Number of divisions used in view factor calculation. Default is 3. Only necessary if using input.
41-45	7th	I	Number of Gauss points used in each subdivision. Default is 3.
46-50	8th	I	Enter unit number for debug printout.
51-55	9th	I	Enter 4 for alternative (K4) method for calculation of view factors.

■ RESTRICTOR

Restrictor Input in Lubrication Analysis

Description

This parameter must be included to allow the use of restrictors in lubrication analysis. See the RESTRICTOR model definition option section of this document for more information.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word RESTRICTOR.
11-15	2nd	I	Number of element surfaces for which restrictor conditions will be imposed.

Rezoning and Substructure Parameters

■ REZONING

Allow Rezoning

Description

This parameter is used to indicate that rezoning can occur during this run. During rezoning, it is possible to add and/or delete elements and/or nodal points. If elements and/or nodal points are added, there should be enough elements and nodes allocated with the SIZING parameter in the initial run. The REZONING parameter can be used with all continuum displacement elements, shell elements 22, 75, 138, 139, and 140 and Herrmann elements 80 through 84.

Note: If rezoning is to be performed in this analysis, this parameter *must* be included from the very beginning. It cannot be added upon restart.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-8	1st	A	Enter the word REZONING.

■ SUBSTRUC

Substructure Definition

Description

This parameter allows the definition of a substructure in this analysis. The nodes and degrees of freedom belonging to the substructure are defined with the SUBSTRUCTURE model definition option. To use the substructure in a later run, the SUPER parameter must be used in combination with the SUPERINPUT model definition option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-8	1st	A	Enter the word SUBSTRUC.
11-15	2nd	I	Enter Fortran unit number for the database containing the substructure information.
16-20	3rd	I	Enter Fortran unit for the optional sequential file to write the data for this substructure. If zero, data is written to the data base.
21-25	4th	I	Enter level number for this substructure.
26-30	5th	I	Enter substructure number within above level for this substructure.
31-35	6th	I	Enter 1 if this substructure is a copy of a previous substructure.

The following four entries are only used if, in the sixth field, a 1 is entered to flag the duplication of a previously defined substructure.

36-40	7th	I	Enter load flag for substructure copy: 0 Generate new load for substructure 1 Copy load for substructure 2 No load on this substructure

Format		Data	
Fixed	Free	Type	Entry
41-45	8th	I	Enter level number of the substructure to be copied.
46-50	9th	I	Substructure number within above level of the substructure to be copied.
51-55	10th	I	Enter Fortran unit for the optional sequential file containing the data for the substructure that is copied. If 0, data is supposed to be on the data base.

■ SUPER

Super Element Input

Description

This parameter allows for the input of stiffness and load terms generated by the SUBSTRUCTURE commands in a previous analysis. It can be used in a multilevel substructure definition in combination with the SUBSTRUC parameter or on the main level of a substructure analysis. If this parameter is used, no other elements are necessary; in which case, the ELEMENTS parameter is not used.

Format

Format		Data	
Fixed	Free	Type	Entry
1-5	1st	A	Enter the word SUPER.
11-15	2nd	I	Enter Fortran unit number for the data base containing the super element information.
16-20	3rd	I	Enter the maximum number of nodes in any super element.
21-25	4th	I	Enter the maximum number of degrees of freedom per node number in any super element.
26-30	5th	I	Enter the number of super elements in this analysis.

■ NEWDB

Initiate New Data Base for Substructure Analysis

Description

This parameter initiates a new data base for substructure analysis. If this parameter is not used when substructures are being formed, the program assumes that the substructures are to be added to an already existing data base.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word NEWDB.

■ USER

Create User-defined Element

Description

You can define your own stiffness or mass matrix using this parameter and user subroutine USELEM to specify equivalent nodal loads, stiffness matrix, mass matrix, stress recovery, and internal force. When using this capability, the element type given on the CONNECTIVITY model definition option and the ELEMENTS parameter is a negative number. This parameter can be used repeatedly to define different element types.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word USER.
11-15	2nd	I	Enter the element type; must be a negative number.
16-20	3rd	I	Number of degrees of freedom per node.
21-25	4th	I	Maximum number of stress quantities to be stored per "integration" point; can be 0.
26-30	5th	I	Number of nodes per element; must be less than 21.
31-35	6th	I	Number of generalized strains; may be 0.
36-40	7th	I	Number of coordinates per node.
41-45	8th	I	Number of integration points per element; can be 0.
46-50	9th	I	Number of direct components of stress; can be 0.
51-55	10th	I	Number of shear components of stress; can be 0.

Format Fixed	Format Free	Data Type	Entry
56-60	11th	I	Element class; can be 0.

Conventional MARC Element Classes	
1 = Truss	9 = Fourier
2 = Shell	10 = Axisymmetric solid with twist
3 = Plate	11 = Axisymmetric shell
4 = Plane stress	12 = Open section beam
5 = Plane strain	13 = Closed section beam
6 = Generalized plane strain	14 = Membrane
7 = Axisymmetric solid	15 = Gap
8 = 3D solid (brick, tet)	

61-65	12th	I	Heat transfer flag; 0 = Stress element 1 = Heat transfer element

Additional Flags for Various Analyses

■ CENTROID

State Storage at Centroid Only

Description

This parameter is used for calculation and storage of stress and strain (or, for heat transfer, temperature) at the centroid of each element only.

The CENTROID parameter reduces the storage requirements, and the computational costs. However, it is not recommended for nonlinear analysis because it reduces the accuracy of the solution. If this parameter is used, the residual load correction should be switched off by using the NO LOADCOR parameter.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word CENTROID.

■ ALL POINTS

State Storage at All Points

Description

This parameter is used for calculation and storage of stress and strain (or, for heat transfer, temperature) at all integration points of all elements. Output is obtained for each integration point of each element. For the integration point locations of MARC elements, see the *MARC Reference Library, Volume B: Element Library.*

If this parameter is switched off by the use of the CENTROID parameter, the state at each integration point of the element is set equal to the value at the centroid of the element. This is not important in small displacement elastic solutions and might not be significant where the mesh is very fine. However, the utility of the sophisticated elements lies in the use of integration point data with relatively few elements.

Use of this parameter is recommended for any nonlinear analysis, particularly nonlinear shell and large displacement analysis. If this parameter is turned off, the residual load correction should be switched off (using the NO LOADCOR parameter) since an accurate stress distribution is necessary for this correction to be effective. In general, use of this feature increases the run time; however, this parameter allows the use of a coarser mesh, which can result in a lower overall cost for the analysis. Storage requirements are also higher. This also can affect THERMAL LOADS or CHANGE STATE input requirements.

Note: This parameter has the default value of on in the K2 and subsequent versions. This parameter has the default value of off in all versions previous to K2.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the words ALL POINTS.

■ LOAD COR

Residual Load Correction

Description

This parameter is used to ensure that the nonlinear solution is always in equilibrium. In versions subsequent to K2, this was the default option. It is recommended that the ALL POINTS parameter always be used in conjunction with residual load correction. The residual correction depends on integrating stress over the elements, and this can only be accurate if stresses are stored at all points.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the words LOAD COR.

■ NO LOADCOR

Suppression of Load Correction

Description

Residual load correction is automatically included for any analysis. This parameter is used to override any automatic setting. This parameter should be the last parameter given before the END parameter.

Note: Certain parameters (DYNAMICS and LARGE DISP) override this parameter and always turn the load correction on. The use of this parameter should be limited to linear elastic problems.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-10	1st	A	Enter the words NO LOADCOR.

■ SCALE

Scaling to First Yield

Description

This parameter causes scaling of the linear-elastic solution to first yield in the highest stressed element, for small displacement elastic-plastic analysis where element properties are not temperature dependent. Using this parameter causes all aspects of the initial solution (displacements, strains, stresses, temperature changes, loads) to be scaled. Thus, subsequent incrementation is built onto the scaled solution; for example, the PROPORTIONAL INCREMENT history definition set proportions the scaled load.

Note: This parameter cannot be used in conjunction with the following parameters: DYNAMIC, LARGE DISP, COUPLE.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-10	1st	A	Enter the word SCALE. This entry automatically switches on the load correction. This flag is ignored if large displacement or dynamic analysis is flagged.

■ THERMAL

Thermal Stress Analysis

Description

This parameter specifies the use of thermal loading or temperature-dependent material properties in the analysis. See THERMAL LOADS, CHANGE STATE, INITIAL TEMP, POINT TEMPERATURE, and TEMPERATURE EFFECTS model definitions in this document for more information.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word THERMAL.

■ ISTRESS

Define Initial Stress

Description

This parameter allows you to input an initial set of stresses. It is your responsibility to input a self-equilibrating set of stresses. These stresses should not produce any material nonlinearity. The stresses are input through the user subroutine UINSTR or through the INIT STRESS model definition option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word ISTRESS.

■ LUMP

Lumped Mass or Specific-Heat Matrix

Description

This parameter lumps the mass matrix (for dynamics) or specific heat matrix (for heat transfer) onto a diagonal matrix.

Note: Use of this parameter is not recommended for second order elements (8-node quadrilateral or 20-node brick elements) or for shell type elements.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-4	1st	A	Enter the word LUMP.

■ APPBC

Application of Boundary Conditions

Description

The APPBC parameter specifies that the application of boundary conditions is performed by row-column elimination, forcing re-assembly if there are any nonzero applied displacements. If this option is not included, boundary conditions are applied using the penalty method. The penalty factor is entered through the PARAMETERS option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word APPBC.

■ ACCUMULATE

Accumulation of Strain and Displacements

Description

This parameter reserves workspace for the storage of accumulated total strains, plastic strains, creep strains and total displacements. Such accumulated values can be used for purposes of extrapolation in nonlinear creep and/or plasticity analysis. In particular, it can be used in analysis of cyclic loading problems, where from one complete cycle the accumulated strains and displacements can be extrapolated to cover multiple loading cycles.

Note: This parameter must be used with extreme care. Because of the nature of extrapolation, the results can only be considered to be an estimate of the values that would have otherwise been obtained with complete analysis. After an extrapolation, the analysis can be continued in the usual way. See the ACCUMULATE and EXTRAPOLATE options in the history definition section of this manual for more information.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word ACCUMULATE.

■ ALIAS

Define Aliases

Description

In many cases, you might wish to enter a different element type identification to the library element type given on the SIZING data when the mesh is read from the CONNECTIVITY model definition set. A common example is the use of the same mesh for heat transfer and stress analysis. The library element code on the SIZING parameter must be changed, but you might not wish to change the library code on the CONNECTIVITY option. This parameter defines the aliases corresponding to the library element types in this analysis. For example, if a heat transfer analysis is to be done with 4-node, axisymmetric quadrilateral (library code 40) but the mesh has been generated with element code type 10 (the corresponding stress analysis element) the alias is set up as 10 for library code type 40.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-5	1st	A	Enter the word ALIAS.
11-15	2nd	I	Number of aliases to be entered. More than one alias can be used for any one element library code.
16-20	3rd	I	Alias for element library code (the type given on the CONNECTIVITY option).
21-25	4th	I	Actual library code for the above alias (the type given on the ELEMENTS parameter and the one to be used in the analysis).
			Etc.

Note: Alias correspondence pairs are continued in fields of I5 to column 75. Continuation blocks, if needed, are given in 16I5 format.

Program Function and I/O Controls

■ NEW

Use New Format

Description

This parameter allows the input of data in the new Version K style format. Input is interpreted to be in this format until an OLD parameter is encountered. This parameter must not appear embedded inside a model definition or history definition option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word NEW.
11-15	2nd	I	Enter 1 if the default width of the data fields is used for input. Enter 2 if the double width of the data fields is used for input.

■ OLD

Use Old Format

Description

This parameter allows the input of data in the old (Rev. G, H, J) style format. Input is interpreted to be in this format until a NEW parameter is encountered. This parameter must not appear embedded inside a model definition or history definition option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word OLD.

■ COMMENT

Define Comment

Description

The COMMENT parameter is used to enter informative comments. This parameter can be used as often as desired within the model definition and history definition options. Use of this parameter does not affect the analysis.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word COMMENT.
11-80	2nd	A	User-entered comment.

Alternate Format

1-1	1st	A	Enter the $ character.
3-80	2nd	A	User-entered comment.

■ PRINT

Debug Printout

Description

This parameter allows printout of various items for debugging; however, the amount of output is increased accordingly. Default is no check printout. Multiple print flags can be set using columns 11 to 80.

Format

Format		Data	
Fixed	Free	Type	Entry
1-5	1st	A	Enter the word PRINT.
11-80	2nd	14I5	Enter as many print codes as required.
			Enter 1 for output of element stiffness matrices (this will also print out the shell surface metric for doubly curved shells 4, 8 and 24), consistent mass matrix and equivalent nodal loads.
			CAUTION: This produces significant output.
			Enter 2 for output of the matrices used in tying. (See TYING, SERVO LINK, UFORMS.)
			Enter 3 to force the solution of a nonpositive definite matrix. This is only recommended for the AUTO INCREMENT option to pass collapse points in the collapse analysis. This can be entered on the CONTROL option.
			Enter 5 to obtain additional information concerning gap convergence. In coupled analysis, set to 5 to print internal heat generated. In contact analysis, set to 5 to obtain information concerning nodes touching or separating from surfaces.
			Enter 6 to obtain output of nodal value array during rezoning.

Format Fixed	Format Free	Data Type	Entry
			Enter 7 to obtain tying information in CONRAD GAP option and fluid element numbers in CHANNEL option which is used to define FLUID CHANNEL input data in heat transfer analysis. Enter 8 to obtain incremental displacements in local system in contact problems. Enter 9 to obtain latent heat output. Enter 10 to obtain the stress-strain relation in the local coordinate system. Enter 11 to obtain additional information on the interlaminar stress calculation. Enter 12 to output the right hand side and solution vector. **CAUTION:** This produces significant output. Enter 13 to obtain additional information regarding CPU resources used. Enter 14 to obtain information regarding the mesh adapting process.

■ STOP

Exit following Workspace Allocation

Description

For large problems, you might often desire to see the exact sizing requirements for running a job without actually executing the analysis. The insertion of this parameter causes the program to exit normally following workspace allocation. The solution space allocated is based on the optimized bandwidth if you request the OPTIMIZE option in the model definition section.

Format

Format		Data	
Fixed	Free	Type	Entry
1-4	1st	A	Enter the word STOP.

■ NOTES

Print Notes and Updates

Description

This parameter provides detailed, updated information about the program (manual update, new program features, etc.)

Format

Format		Data	
Fixed	Free	Type	Entry
1-5	1st	A	Enter the word NOTES.

■ INPUT TAPE

Specify Device for Model Definition Data

Description

This parameter allows specification of a storage device which contains previously generated CONNECTIVITY and COORDINATES model definition data.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the words INPUT TAPE.
11-15	2nd	I	Unit number for main input of coordinates and connectivity. Default is unit 5. For larger problems, involving out-of-core options, you should avoid using unit 2, 3, 11, 12, 13, 14, or 15 for mesh input.

■ ELSTO

Out-of-Core Storage of Elements

Description

This parameter is used to save core storage for large problems. All element quantities, strains, stresses, etc. are stored on an auxiliary storage device. If the number of words actually required is less than the buffer specified below, this option is turned off by MARC.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word ELSTO. This stores element arrays on unit 3.
11-15	2nd	I	Buffer size for out-of-core element storage. A default of 40960 words has been set for UNIX machines.

Modifying Default Values

■ STATE VARS

Define Number of State Variables

Description

This parameter allows consideration of state variables in addition to that of temperature. The number of predefined state variables stored at each point of the structure can be increased from the default of one (temperature for heat transfer and lubricant pressure for bearing analysis) by the user of this parameter. In addition, additional storage can be allocated for user-defined global scalar quantities.

For more information, see THERMAL LOADS, INITIAL STATE, or CHANGE STATE model definition data in this document and user subroutine CREDE or NEWSV, in *Volume D: User Subroutines and Special Routines* of the *MARC Reference Library.*

Note: In bearing analysis, the first state variable equals the lubricant pressure. For this reason, the number of state variables must be set to 2 if viscosity varies with temperature in this type of analysis.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the words STATE VARS.
11-15	2nd	I	Number of state variables to be stored per point.
16-20	3rd	I	Number of global variables to be stored.

■ DIST LOADS

Distributed Loads or Point Loads

Description

This parameter allows for the input of the maximum number of different lists of distributed loads, the maximum number of elements in any particular distributed load list, and the maximum number of nodes with point loads applied.

This parameter is not necessary unless the number of different lists is greater than three, or the maximum number of elements per list is increased in the history definition section or the number of point fluxes is increased in the history definition section.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the words DIST LOADS.
11-15	2nd	I	Maximum number of different lists of distributed loads. The default is 3.
16-20	3rd	I	Maximum number of elements in any particular distributed load list.
21-25	4th	I	Enter the maximum number of nodes with point loads applied.

■ FLUXES

Distributed Fluxes or Point Fluxes

Description

This parameter allows for the input of the maximum number of different lists of distributed fluxes, the maximum number of elements in a particular distributed flux list, and the maximum number of nodes with point fluxes applied.

This parameter is not necessary unless the number of different lists is greater than three, or the maximum number of elements per list is increased in the history definition section or the number of point fluxes is increased in the history definition section.

Format

Format		Data	
Fixed	Free	Type	Entry
1-6	1st	A	Enter the word FLUXES.
11-15	2nd	I	Maximum number of different lists of distributed fluxes. Default is 3.
16-20	3rd	I	Maximum number of elements in any particular distributed flux list.
21-25	4th	I	Enter the maximum number of nodes with point fluxes applied.

■ SHELL SECT

Define Number of Layer Through Shell Thickness

Description

All the shell and beam-in-a-plane elements in MARC use a Simpson rule for integration through the cross section. This numerical integration allows any material behavior at each layer; for example, the yielding of a nonlinear elastic-plastic shell can be followed through the section, from a fully elastic to a fully plastic section. The density of integration points through the thickness is chosen with this parameter. For purely linear material behavior, 1 point (the minimum) is required for exact integration across the section. For most nonlinear problems, 7 points are sufficient to describe the nonlinear material response exactly. For extremely nonlinear response – such as elastic-plastic dynamic problems – 11 points might be needed. The default if this parameter is not used is 11 points.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the words SHELL SECT.
11-15	2nd	I	Number of points across the section for Simpson rule integration of stresses. The default is 11; minimum is 1. Must be an odd number.
16-20	3rd	I	Enter 1 if you are going to perform your own integration through the shell/beam thickness. This requires you to input a generalized stress-strain law through user subroutine GENSTR.

■ TSHEAR

Transverse Shear for Elements 22, 45, 75, and 140

Description

The default distribution of transverse shear strain through the thickness for thick shell element types 22, 75, and 140, and for thick beam 45, is a constant shear-strain distribution. With the inclusion of the TSHEAR parameter, a more parabolic beam-like distribution derived from a strength-of-materials approach is used. This distribution is exact for beam 45 but is only approximate for shells 22, 75, or 140 since it is based on the assumption that the stresses in perpendicular directions are independent of each other. The interlaminar shear is printed only if the PRINT ELEM option is used.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word TSHEAR.

■ SETNAME

Set Maximum Number of Set Names

Description

This parameter sets the maximum number of set names to be encountered in the input deck. The default is 10. The total number of items in all sets must be less than 50 times the maximum number of sets. If sets require more items, increase the value of the maximum number of sets.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word SETNAME.
11-15	2nd	I	Enter upper bound to the number of set names used. Default is 10.

TIE

Define Tying Data

Description

This allocates storage for tying data. See TYING and SERVO LINK model definition blocks in Chapter 3 of this manual. See also subroutine UFORMS in *MARC Reference Library, Volume D: User Subroutines and Special Routines.*

This parameter is necessary only if TYING CHANGE is used to increase the number of constraints.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word TIE.
11-15	2nd	I	Number of tying type constraints; i.e., number of constraint equations.
16-20	3rd	I	Number of different types of tying constraints.
21-25	4th	I	Maximum number of retained nodes plus one involved in any tying type or servo link constraint.
26-30	5th	I	Maximum number of servo links.

Defining Cross-sections of Beam Elements

■ BEAM SECT

Beam Section Definition

Description

This parameter is used to define the sectional properties of beam sections used in the analysis. The format and exact data entered depends upon which elements are used. Formats for all available MARC beam elements are shown below. This parameter must be included if element 13, 77 or 79 is used, if 14, 25, 31, 76, or 78 is used with a noncircular section, or if element 52 or 98 is used with a nonprismatic section. See the *MARC Reference Library, Volume B: Element Library* for more information about individual elements.

If used, the BEAM SECT parameter must directly follow the SIZING and ELEMENTS parameters. Each beam section used in the analysis must then be described. A LAST statement must follow the last beam section description to complete the BEAM SECT parameter definition.

Format

Format		Data	
Fixed	Free	Type	Entry

For all beam elements, use the 1st and last data blocks described below.

1st card series

1-10	1st	A	Enter the words BEAM SECT.
.			
.	(Include all beam section definitions here.)		
.			

Last card series

1-10	1st	A	Enter the word LAST.

For elements 13, 14, 25, and 76 to 79, use the following format to define each beam section:

2nd card series

1-10	1st	A	Descriptive title of beam section.

Format Fixed	Format Free	Data Type	Entry
3rd card series for elements 13, 14, 25, and 76 - 79			
1-5	1st	I	Number of branches used to input section.
6-10	2nd	I	Number of divisions in first branch. Must be an even number.
11-15	3rd	I	Number of divisions in second branch. Must be an even number.
Etc.	Etc.	I	Etc.

Data blocks 4a and 5a are given one pair per branch. (X, Y, and S are coordinates on the cross-section face.) See *Volume B: Element Library.*

4a card series			
1-10	1st	F	X-Coordinate of beginning of branch.
11-20	2nd	F	Y-Coordinate of beginning of branch.
21-30	3rd	F	DX/DS at beginning of branch.
31-40	4th	F	DY/DS at beginning of branch.
41-50	5th	F	X-Coordinate of end of branch.
51-60	6th	F	Y-Coordinate of end of branch.
61-70	7th	F	DX/DS at end of branch.
71-80	8th	F	DY/DS at end of branch.
5a card series			
1-10	1st	F	Length of branch.
11-20	2nd	F	Thickness of beginning branch.
21-30	3rd	F	Thickness of end branch. Default to thickness at beginning if left zero.

Note that a beam section consisting of one straight branch has no stiffness against rotation along the branch direction.

Format

Format Fixed	Format Free	Data Type	Entry
For elements 31, 52 or 98, use the following format to define each beam section:			
2nd card series			
1-10	1st	A	Descriptive title of section.
3rd card series for element type 31, 52 or 98			
1-5	1st	I	Enter 0.
6-15	2nd	F	Area of cross section A.
16-25	3rd	F	I_{xx} Moment of inertia about local x-axis.
26-35	4th	F	I_{yy} Moment of inertia about local y-axis.
36-45	5th	F	K Torsional stiffness factor. The torsional stiffness is calculated as $\frac{EK}{2(1+\nu)}$.
46-55	6th	F	A_x^s Effective transverse shear area in x-direction (only applies to element 31 and 98). Default A_x^s = A.
56-65	7th	F	A_y^s Effective transverse shear area in y-direction (only applies to element 31 and 98). Default A_y^s = A.

The cross-section properties can be specified directly in the GEOMETRY fields 1-3. It is then assumed that the torsional stiffness factor K is equal to the polar moment of inertia $J = I_{xx} + I_{yy}$, and for element 98, it is assumed that the effective transverse shear areas A_x^s and A_y^s are equal to the area *A*. If a different torsional moment is desired, the BEAM SECT parameter can be involved by leaving the first GEOMETRY field blank or zero.

3 List of Model Definition Options

Option	Page

3 Model Definition Options

The model definition input consists of a series of optional blocks of data. These blocks define the geometry of the mesh, material properties, boundary conditions, and analysis controls.

These options are read in by activating the respective option with an alphanumeric code word (a keyword), followed by sets of data. In this document, both fixed format and free input are described.

This code word is given in capital letters at the top of each block of data in the following section. An END OPTION is used to signify the end of all the model definition input data. Note that each option can be exercised more than once.

In general, there is no specific order required in reading the options; however, you should be aware that the same option flags can appear in different blocks and that the last data read controls that flag. The exceptions are as follows:

1. If the FXORD or UFXORD option is used, it must come after the COORDINATES option when it uses data read in the COORDINATES option;
2. If post-processor and restart files are being used, the results are order dependent. See the POST option for more details.
3. If the MESH2D option is used, it must follow the END parameter. If the MESH2D option is used to write a mesh file containing connectivity, coordinates and optional boundary conditions, the file needs to be read in sequential order. The CONNECTIVITY block must appear before the COORDINATES block followed by BOUNDARY conditions if stored on mesh file. Other options can be in any sequence and the above blocks can be repeated in any sequence to read data from other files or data input.

Information given in the last option overwrites any previous information, thus facilitating any minor corrections to the data. Any block not needed should be left out.

3

MESH2D

Two-Dimensional Mesh Generator

This two dimensional mesh generator generates a mesh composed of either triangular or quadrilateral elements. The results of the mesh generation are output on a specified file. This file can then be used as input for MARC. A detailed description of the capabilities of the MESH2D generation is contained in *Volume A: Theory and User Information.*

The call to the mesh generation feature is initialized by a block with MESH2D in the first six columns. This is followed by a series of optional sets of blocks; each of which has an alphanumeric keyword. The keywords are:

- BLOCKS (Required as first set)
- DEFINE
- MANY TYPES
- START NUMBER
- BOUNDARY
- SPECIFIED NODES
- MAPPER
- CONSTRAINT
- MERGE
- MERGE SELECTIVE
- CONNECT
- PRTCONNECT
- SYMMETRY
- GENERATE

BLOCKS must be the first set input. GENERATE must terminate the mesh generation. BLOCKS defines the parameters and the working space for the mesh generation. GENERATE tells the program to start generating the mesh, and then returns control to the MARC program. You should note that the generated mesh is written out on the specified file and must be read in from this file using the appropriate model definition data blocks (CONNECTIVITY and COORDINATES) described in a following section before plotting or optimization of the mesh can proceed.

When using the MESH2D option, divide the geometry into simpler regions called blocks. The program meshes each block into nodes and elements, and finally combines the blocks to form the complete mesh.

The MESH2D option can be used more than once in an analysis. Simply place the second group to be included after the previous GENERATE.

■ MESH2D

Define a Two-Dimensional Mesh

Description

This option starts the call to the two-dimensional mesh generation feature. This data must follow the END of the parameters.

Format

Format		Data	
Fixed	Free	Type	Entry
1-6	1st	A	Enter the word MESH2D.

■ BLOCKS

Define Working Size

Description

This option defines the parameters and the sizes of the working space for mesh generation.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-6	1st	A	Enter the word BLOCKS. ***This is required and must follow*** MESH2D***.***
2nd data block			
1-5	1st	I	Number of blocks.
6-10	2nd	I	Number of principal boundary nodes defining the geometry of all blocks. Note that principal nodes should be continuous in their numbering.
11-15	3rd	I	Code number of element type for use with MARC series of programs.
16-20	4th	I	Set to 1 for generation of 4-node quadrilateral elements. Set to 0 for triangles. Set to 2 for 8-node quads.
21-25	5th	I	Local file on which the output is written; to be used by the MARC series of programs.
26-30	6th		Not used. Enter 0.
31-35	7th	I	Number of times the mesh will be repeated due to use of the SYMMETRY block.
36-40	8th	I	Maximum number of degrees of freedom constrained during mesh generation. Default is 100.

Format		Data	
Fixed	Free	Type	Entry
41-45	9th	I	Number of degrees of freedom for each node. The default is two degrees of freedom.
46-50	10th	I	Maximum number of nodes on symmetry axis. Default is 50.
51-55	11th	I	Maximum number of connections of any blocks. Default is 10.

■ DEFINE

Define Block Type

Description

This option allows the block type and the nodal number of the boundary points to be specified.

Note: The program always generates node connections in a counterclockwise direction. See *Volume A: Theory and User Information* for correct specification of boundary node number order when a distributed load has to be applied to any surface of the block.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-6	1st	A	Enter the word DEFINE.
2nd data block			
One per block.			
1-5	1st	I	Type of block: 1, 2, 3, or 4.
6-10	2nd	I	Number of intervals in the first direction (P_1 - P_2). Number of increments between the first and second boundary nodes.
11-15	3rd	I	Number of intervals in the second direction (P_2 - P_3). M=N for a type 3 triangular block. Number of increments between second and third boundary nodes.
16-20	4th	I	First boundary node number defining the block.
21-25	5th	I	Second boundary node number defining the block.
26-30	6th	I	Third boundary node number defining the block.
31-35	7th	I	Fourth boundary node number defining the block.
Etc.		I	Continue until necessary boundary nodes have been defined. A maximum of 12 is possible and these must follow the order defined in *Volume A: Theory and User Information.*

■ MANY TYPES

Define Multiple Elements

Description

This option allows you to specify different element types per block. Default is that all elements are of the same type specified in the BLOCKS option.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words MANY TYPES.
2nd data block			
1-5	1st	I	Element type block 1.
6-10	2nd	I	Element type block 2.
			Etc. for all blocks.

Continuations (more than 16 blocks) are given in Format 16I5.

■ START NUMBER

Specify Starting Element

Description

This option allows you to give a lowest element number and node number for this MESH2D sequence. This option can be used more than once; for example, if MESH2D is used more than one time in a single run. Default is that the program starts generation with element 1 and node 1.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-12	1st	A	Enter the words START NUMBER.
2nd data block			
1-5	1st	I	Starting node number.
6-10	2nd	I	Starting element number.

■ BOUNDARY

Define Boundary Nodes

Description

This allows the coordinates of the boundary nodes to be read in.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-8	1st	A	Enter the word BOUNDARY.
2nd data block			
Boundary node coordinates, one per node; NNO series.			
1-5	1st	I	Boundary node number.
6-15	2nd	F	First (X or Z) coordinate.
16-25	3rd	F	Second (Y or R) coordinate.

■ SPECIFIED NODES

Specify Node Coordinates

Description

This option allows the coordinates of certain nodes of the generated mesh to be specified.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-15	1st	A	Enter the words SPECIFIED NODES.
2nd data block			
1-5	1st	I	Number of sets of nodal points coordinates to be specified, (maximum 100).
3rd data block			
One per specified node.			
1-10	1st	F	First coordinate of a specified nodal point.
11-20	2nd	F	Second coordinate of a specified nodal point.

■ MAPPER

Invoke User Subroutine MAP2D

Description

This option invokes the MAP2D user subroutine (see *Volume D: User Subroutines and Special Routines*) for boundary node coordinate generation or modification. It is used when it is more convenient to program the boundary node coordinates rather than reading them in.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-6	1st	A	Enter the word MAPPER. This invokes subroutine MAP2D. If the coordinates input on the BOUNDARY option are to be modified, this option must follow the BOUNDARY option.

■ CONSTRAINT

Generate Boundary Condition Constraints

Description

This feature allows boundary conditions to be generated for a particular degree of freedom for all the nodal points on one side of a block. At the present time, there is no method available for setting boundary conditions on those nodes generated via the SYMMETRY option.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word CONSTRAINT.
2nd data block			
1-5	1st	I	Number of sides to be constrained.
3rd data block			
1-5	1st	I	Number of the block to be constrained.
6-10	2nd	I	Number of the side to be constrained.
11-15	3rd	I	Degree of freedom to be constrained.
16-25	4th	F	Displacement value to be given to the constrained degree of freedom.

■ MERGE

Specify Minimum Distance Between Nodes

Description

This allows a minimum distance between nodes to be specified. Any nodes separated by less than the minimum distance is automatically merged into a single node.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-5	1st	A	Enter the word MERGE.
2nd data block			
1-10	1st	F	Separation distance below which nodes will be merged together.

■ MERGE SELECTIVE

Specify Minimum Distance Between Nodes by Block

Description

This option, used in conjunction with the CONNECT option, allows you to define which nodes are to be merged in mesh blocks that border each other. This is especially useful if gaps are to be defined between the blocks.

As with the MERGE option, you specify a minimum distance and a list of block numbers in which nodes are to be merged. Nodes which are separated by less than the minimum distance specified are considered duplicates, and merged into a single node if they lie within the same or connected blocks. Nodes located within the specified minimum distance on unconnected blocks (those disconnected using the CONNECT option) are not merged.

For more information about connecting blocks, see the description of the CONNECT option in this document.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-15	1st	A	Enter the words MERGE SELECTIVE.
2nd data block			
1-5	1st	I	Enter the number of sets of data to be used to enter merge distance and block numbers.

Data blocks 3 and 4 are entered as pairs, one for each data set.

3rd data block			
1-10	1st	F	Enter the minimum separation distance.
4th data block			
			Enter a list of block numbers.

■ CONNECT

Connect or Disconnect Mesh Blocks

Description

This option is used to connect or disconnect two blocks during the generation of the final mesh. Default is that two blocks are connected if they join the same boundary points in the DEFINE option. It is especially useful to disconnect two blocks if gaps are to be defined in between the blocks. The MERGE SELECTIVE option must be used in combination with this option.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-7	1st	A	Enter the word CONNECT.
2nd data block			
1-5	1st	I	Number of block connections to specify.
6-10	2nd	I	Unit number for input of 3rd data block; defaults to data input.
3rd data block			
One per continuation.			
1-5	1st	I	First connected block.
6-10	2nd	I	Second connected block.
11-15	3rd	I	Set to 0 to connect the two blocks; set to 1 to disconnect the two blocks.

■ PRTCONNECT

Print Out Block Connections

Description

This option gives a printout of the current BLOCKS connection information.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word PRTCONNECT.

■ SYMMETRY

Define Axis of Symmetry

Description

This allows symmetry axes to be defined so that further mesh blocks can be generated by reflection about the axes. Be sure that enough space is allocated in the BLOCKS option.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-8	1st	A	Enter the word SYMMETRY.
2nd data block			
1-5	21st	I	Number of symmetric axes.
3rd data block			
One per symmetry axis.			
1-10	1st	F	First coordinate of a point on the axis of symmetry.
11-20	2nd	F	Second coordinate of a point on the axis of symmetry.
21-30	3rd	F	First component of a vector along the axis of symmetry.
31-40	4th	F	Second component of a vector along the axis of symmetry.

■ GENERATE

End of Mesh Generation Data

Description

This signifies the end of the mesh generation data and instructs the program to proceed with the mesh generation. When the mesh has been generated, the program proceeds to the next option found in the model definition options. If you wish to stop without proceeding to plotting or analysis, a blank block should immediately follow the generate block. This causes the program to stop on an illegal data exit. Provisions should be made to save the mesh on permanent file by appropriate control blocks if only mesh generation is desired.

Format

Format		Data	
Fixed	Free	Type	Entry
1-8	1st	A	Enter the word GENERATE.

Mesh Definition

This section describes the geometry input required to describe the object to be analyzed. The finite element mesh can be generated using MESH2D, Mentat II, or some other pre-processor. The element topology and coordinates can be processed using a variety of options. This final set of connectivity and coordinates can be written to an auxiliary file through the WRITE option. Additional geometric quantities can be input through the GEOMETRY or NODAL THICKNESS options; see *Volume B: Element Library* for the data required for particular element types. The ROTATION A option is used to give the axis for the calculation of centrifugal loads. The degrees of freedom associated with nodes can be transformed from their natural system (see *Volume B: Element Library* for the definition for each particular element) to a user-defined local system. Kinematic constraints can be imposed between degrees of freedom using either the TYING or SERVO LINK option. Finally, springs can be defined using the SPRINGS option.

NEW and OLD parameters can be used to specify a change in format.

■ DEFINE

Define Sets

Description

This option allows you to define a setname and to associate members to the set. These sets can then be used wherever a list of items is requested. Multiple numbers of sets can be used by repeating this model definition block. In defining the members of a set, any of the conventions in "Input of List Items" in Chapter 1 can be used. A previously defined set can be used to describe a set. The name of a set is limited to 12 characters.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-6	1st	A	Enter the word DEFINE.
The rest of this data block is free format beginning with the third field.			
11-20	2nd	A	Enter the type of set: ELEMENT – set of elements NODE – set of nodes NDSQ – set of nodes (unsorted) INT – set of integration points LAYER – set of beam or shell layers DOF – set of degrees of freedom (unsorted) INCS – set of increment numbers
21-30	3d	A	Enter the word SET. (This is optional.)
31-38	4th	A	Enter the name of the set.

Format Fixed	Format Free	Data Type	Entry
2nd data block			
1-80	1st		Enter a list of items that are of the type defined to be members of the set whose name is given. **Note:** A setname cannot be used in a list unless it has been previously defined. For unsorted set types DOF and NDSQ, the verbs EXCEPT and INTERSECT cannot be used in the list of items.

Example

```
DEFINE NODE SET WALL
1 TO 20 BY 2
```

The above example defines a set to be called WALL consisting of nodes 1, 3, 5, 7, . . . 19.

■ CONNECTIVITY

Specify Element Connectivity

Description

This series gives the element connectivity; for example, the nodal points for each element. Data can be input from data or an external file by exercising the appropriate option. For two-dimensional elements, the nodal points must be given in a counterclockwise sequence. Several blocks of connectivity data can be input. For example, one block can be read in from file while additional ones are read from data blocks, each block starting with the word CONNECTIVITY.

In a coupled thermal-stress analysis, the element type (second field, 3rd data block) should be a stress type element. If a heat transfer element type is given, the element is considered rigid in the stress analysis.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-12	1st	A	Enter the word CONNECTIVITY.

In many cases, when the whole mesh is in a file, just CONNECTIVITY and a blank should be included here and the INPUT TAPE parameter must be used.

2nd data block			
1-5	1st	I	Number of elements to be read in this option (optional); defaults to total number of elements in the mesh.
6-10	2nd	I	Enter the unit number for input of connectivity; defaults to unit number specified on the INPUT TAPE parameter.
11-15	3rd	I	Set to 1 to suppress printing of element connectivity list during this option.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
Element connectivity array. This data block is repeated once for each element given in data block 2.			
1-5	1st	I	Element number.
6-10	2nd	I	Element type or alias (see ALIAS parameter).
11-15	3rd	I	Nodal point.
16-20	4th	I	Nodal point.
21-25		I	Repeat until all nodes of the element have been defined. The required ordering of the nodes is given in *Volume B: Element Library.*
			Note: Continuation for elements with more than 14 nodes/ element (for example, library element 21, 35, 57, etc.) is in format 16I5.

■ CONN FILL

Specify Element Connectivity Interpolator

Description

This option completes the filling of connectivity lists by generating midside nodes in between the corner nodes provided. At the same time, it generates coordinates for the new node created. The coordinates are formed by averaging the coordinates of the end nodes of the respective side on the connectivity list. It is used for converting linear displacement elements to quadratic displacement elements. The option works for 4-node quadrilaterals and 8-node bricks. You must remember to turn on the bandwidth optimization option after using this option and before an analysis.

Note: This does not calculate the coordinates correctly if going from element 4 to element 22 or 24.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words CONN FILL.
2nd data block			
1-5	1st	I	Enter the starting node number for the midside nodes.
6-10	2nd	I	Enter 1 to set the node count to the maximum node number used in this option.
11-15	3rd	I	Give the start of the element list for this option. Default is 1.
16-20	4th	I	Give the end of the element list for this option. Default is number of elements specified in the analysis.

■ CONN GENER

Copy Element Connectivity Data

Description

This input performs the function of an incremental mesh generator by copying the pattern of the connectivity data for previously defined elements. If the new elements are to be connected to the master elements, a common node A needs to be given. The position of node A in the connectivity list of the new element and its position in the connectivity list of the master element needs to be given. The program then numbers all the other nodes in the new element by making the algebraic difference between the numbers of all the nodes in the new element the same as that of the corresponding nodes in the element being copied.

This option copies the connectivity from a series of elements defined by a starting and end element number and uses it to calculate the connectivity for a new series of elements. The new series of elements is defined by the input of a starting and end element number. When the list of the elements being copied from is ended, the recently generated elements will take its place as the elements to be copied from. This is repeated until the list defined for the new elements is exhausted.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words CONN GENER.
2nd data block			
1-5	1st	I	Number of the first element for which the connectivity is being generated.
6-10	2nd	I	Number of the last element for which the connectivity is being generated. All the elements between the first and last element will be generated.
11-15	3rd	I	Number of the first element used as a master.
16-20	4th	I	Number of the last element used as a master.

Format Fixed	Format Free	Data Type	Entry
21-25	5th	I	Give the position of node A in the connectivity list of the generated element. Node A can belong to a master element. If there is no common node between the generated and the master elements, enter 1.
26-30	6th	I	Give the position of node A in the connectivity list of the master element. If there is no common node, enter 1.
31-35	7th	I	Enter 1 for decrease of 1 element per series of master elements. Enter 2 for decrease of 2 elements per series of master elements. The two element decrease is restricted to use with three-node triangular elements.
36-40	8th	I	This entry is only required if there is no common node between the generated and master elements. This defines an increment to each of the nodes in the master element which then gives the connectivity list of the generated element.

■ RENUMBER

Renumber Connectivity Data

Description

This option allows you to shift the node numbering on selected elements. This is intended primarily for putting distributed loads on the correct face of elements in an automatically generated mesh. The shift is defined by giving the current position (in the list of nodes for such elements) of the node which becomes first in the list. For each such element, the numbering scheme is shifted so that node is first and the other nodes appear in the same cyclic order. This option can be called as many times as needed, and can be over-ridden by any subsequent use of the CONNECTIVITY option.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-8	1st	A	Insert the word RENUMBER.
2nd data block			
1-5	1st	I	Position in connectivity list of node of element which will become first node.
6-10	2nd	I	Number of lists of elements given below; defaults to one.
3rd data block			
This data block is repeated once for each list of elements.			
			Enter a list of elements to be renumbered.

■ UFCONN

Invoke User Subroutine UFCONN

Description

This block calls the user subroutine UFCONN to generate or modify element connectivity (see *Volume D: User Subroutines and Special Routines.*) The block can be repeated as often as necessary. This option must follow the connectivity option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-6	1st	A	Enter the word UFCONN.
2nd data block			
			Enter a list of elements for which user subroutine UFCONN will be called.

■ COORDINATES

Enter Node Coordinates

Description

This option defines the coordinates of each nodal point. The nodal data can be input in several blocks. The latest data input for a particular nodal point is used. Like the element connectivity data, this data can be input from an external file since this coordinate data can be automatically generated by a mesh generator. Local corrections can be made to the generated mesh by input of the modified nodal coordinates from data blocks.

Usually for the general shell elements (4,8 and 24), the option FXORD and the user subroutine UFXORD can help with input of coordinates. In the MARC program, the nodes need not be numbered sequentially.

In most cases, when all the coordinates are input by file, just the coordinates and a blank are required here and the INPUT TAPE parameter must be used.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-11	1st	A	Enter the word COORDINATES.
2nd data block			
1-5	1st	I	Enter the maximum number of coordinate directions to be read in per node; defaults to the number of coordinates per node. Repeated COORDINATES blocks need not have the same value in this field.
6-10	2nd	I	Enter the number of nodal points read-in in this option; (optional) default to the number of nodes in the mesh.
11-15	3rd	I	Enter the unit number for input of coordinates; default to the file number specified on the INPUT TAPE parameter.
16-20	4th	I	Set to 1 to suppress print-out of nodal coordinate list during this option input.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
One data line per nodal point.			
1-5	1st	I	Nodal point number.
6-15	2nd	F	Coordinate 1.
16-25	3rd	F	Coordinate 2.
26-35	4th	F	Coordinate 3.
36-45	5th	F	Coordinate 4.
Etc.		Etc.	See library element description in *Volume B, Element Library*, for the definition of coordinates for a particular element. Input 6 coordinates per data line; continuation data lines in format 6E10.5

■ FXORD

Coordinate Generation and Transformation Coordinates

SHELL COORDINATE GENERATION OPTION

Description

This option is used to generate coordinates for Elements 4, 8, or 24. (Refer to *Volume B: Element Library* and *Volume A: Theory and User Information* for further information on the use of this block). This can be used for mapping of certain types of standard shell geometries such as cylinders, spheres, etc. It can also be used to transform cylindrical coordinates into Cartesian coordinates.

Note that when a continuous surface has a line of discontinuity in ϕ^1 or ϕ^2 (the surface coordinate) such as a complete cylinder has at $\phi = 0^\circ$ and 360°, two nodes must be placed at each nodal location on the line to allow the distinct coordinate input, and tying type 100 used to join degrees of freedom. In general, different surfaces coming together must also use the intersecting shell tyings.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-5	1st	A	Enter the word FXORD.
11-12	2nd	A	Enter the word NO if coordinates after FXORD are not to be printed out.
2nd data block			
1-5	1st	I	Number of sets of shell geometry specifications to be input.

Format Fixed	Format Free	Data Type	Entry

Data blocks 3, 4, 5, and 6 are provided once for each consecutive series of nodes with a different shell geometry specification.

3rd data block

1-5	1st	I	Identification number of surface type. See Table 3-1 and *Volume A: Theory and User Information.*
6-10	2nd	I	First node of this series.
11-15	3rd	I	Last node of this series.
16-20	4th	I	Set to 1 if local (x,y,z) set in which surface is defined must be transformed to the global (x,y,z) set. If so, data blocks 4, 5 and 6 must be input to define the transformation. If not, data blocks 4, 5 and 6 are omitted.

4th data block

Enter the global (x,y,z) coordinates of the origin of the local (x,y,z) system in which the shell surface is generated.

1-10	1st	F	Global x-coordinate origin.
11-20	2nd	F	Global y-coordinate origin.
21-30	3rd	F	Global z-coordinate origin.

5th data block

Enter the global (x,y,z) coordinates of a point on the positive x-axis of the local system.

1-10	1st	F	Global x-coordinate of the point.
11-20	2nd	F	Global y-coordinate of the point.
21-30	3rd	F	Global z-coordinate of the point.

6th data block

Enter the global (x,y,z) coordinates of a point on, or near to, the positive y-axis of the local system. This point defines the local (x,y) plane in the global system.

1-10	1st	F	Global x-coordinate of the point.
11-20	2nd	F	Global y-coordinate of the point.
21-30	3rd	F	Global z-coordinate of the point.

Table 3-1 describes the φ_1, φ_2 directions for the FXORD subroutine contained in the MARC program.

Table 3-1 Input to FXORD

Input Code Type	Surface Description	Nodal Data Input (See *Volume A: User Information*, Section 6) All units are length measure, unless specified otherwise	Surface Coordinates used in program (θ_1, θ_2)	Non-Cartesian Coordinates
0	General Surface	All 11 Coords. for El. 8 All 14 Coords. for El. 4	θ_1, θ_2, first two input	YES
1	Surface x3=x3(x1, x2)	$x_1, x_2, x_3, \frac{\partial x_3}{\partial x_1}, \frac{\partial x_3}{\partial x_2}, \frac{\partial^2 x_3}{\partial x_1 \partial x_2}$ (last coordinate only needed for element type 4)	x_1, x_2	YES
2	Axisymmetric shell (about x_3 axis)	$\theta, \phi, R, \frac{\partial R}{\partial \phi}$: $\frac{dR}{d\phi}$ in $\frac{\text{length}}{\text{radians}}$ units	θ, ϕ (in radians)	YES
3	General Cylinder	$s, x_3, x_1, x_2, \frac{dx_1}{ds}, \frac{dx_2}{ds}$	s, x_3	NO
4	Circular Cylinder (about x_3 axis)	q, x_3, R: θ in degrees R at 1st node only	R, θ, x_3 (R, θ in length measure)	NO
5	Flat Plate x_3=0	x_1, x_2	x_1, x_2	NO
6	Curved Pipe (elbow)	θ, ϕ, r, R: θ, ϕ in degrees r and R at 1st node only	r, θ, R, ϕ (both in length measure)	YES
7	Cylindrical	r, θ, x_3	x_1, x_2, x_3	NO
8	Spherical	r, θ, ϕ	x_1, x_2, x_3	NO

■ NODE CIRCLE

Generate Coordinates for Circular Arcs

Description

This option generates the coordinates of a series of nodes which lie on a circular arc. The coordinates of the first node on the arc must be previously given. The circle must lie in the x-y plane.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words NODE CIRCLE.
2nd data block			
1-5	1st	I	Node number of the first node on the arc.
6-10	2nd	I	Total number of nodes on arc.
11-15	3rd	I	First increment in series of node numbers to be generated.
16-20	4th	I	Second increment in series of node numbers to be generated. Entry of the second increment alternates the increment in node numbers and is used for 8 node quadrilaterals. It is only used if a nonzero number is entered in this field.
21-25	5th	I	Scaling of angle between each pair of nodes as a percentage. A default of 100 percent is used.
26-35	6th	F	Number of degrees between the first pair of nodes.
3rd data block			
1-10	1st	F	First coordinate of the center of the circle.
11-20	2nd	F	Second coordinate of the center of the circle.

■ NODE FILL

Coordinate Interpolation for Incremental Mesh Generation

Description

This option performs the function of an incremental mesh generator for nodes. It achieves this by interpolation. In its simplest form, it takes the coordinates defined by two end nodes and divides the line between them into a specified number of equally spaced nodes. Additional data can be input to vary the distances between the generated nodes in a geometric ratio. This option is often used with user subroutine UFXORD to obtain a warped curve in space.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words NODE FILL.
2nd data block			
1-5	1st	I	Number of the first node in the series. The coordinate data for this node must have been previously defined.
6-10	2nd	I	Number of the last node in the series. The coordinate data for this node must have been previously defined.
11-15	3rd	I	Node number increments to be taken in the above list.
16-20	4th	I	Second increment in series of node numbers to be generated. Entry of the second increment alternates the increment in node numbers and is used for 8-node quadrilaterals. It is only used if a nonzero number is entered in this field.
21-25	5th	I	Scaling of the distances between successive nodes as a percentage. A default of 100 percent is used.
26-30	6th	I	Number of times that the series will be repeated. You should ensure that all starting and ending nodes in the series have been defined previously. This repeat feature defaults to 1 series.

Format		Data	
Fixed	Free	Type	Entry
31-35	7th	I	Print flag is set equal to 1, nodal coordinate printout is omitted. If set equal to 2, only the generated coordinates are printed. If set equal to 0 or left blank, all coordinates are printed.
36-40	8th	I	Increment in first and last nodes in the series if the series is repeated more than once. Defaults so that the next series will start from the node after the last node in the preceding series.

■ NODE GENER

Generate Node Coordinates

Description

This option performs the function of an incremental mesh generator for nodes. It is used when elements such as the quadratic 8-node elements require different spacing in successive nodal rows. It achieves this by using a list of nodes as the master pattern. It then creates a new set of nodes by giving the new set of nodes the same coordinate spacing as the list of nodes. Additional optional data allows the spacing to be changed as a percentage of the spacing between the nodes. When the list of nodes is used up, the newly generated nodes takes its place on the list and the process is repeated until the number of nodes to be generated has been completed.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words NODE GENER.
2nd data block			
1-5	1st	I	Number of the first node for which coordinates are being generated. This coordinate data for this node must have been previously defined).
6-10	2nd	I	Number of the last node in the series. (This coordinate data for this node must have been previously defined.)
11-15	3rd	I	Number of the first node used as a master.
16-20	4th	I	Number of the last node used as a master.
21-25	5th	I	Increment in the node numbers of the two node series above. A default value of 1 is used.
26-30	6th	I	Second increment in series of node numbers to be generated. Entry of the second increment alternates the increment in node numbers and is used for 8-node quadrilaterals. It is only used if a nonzero number is entered in this field.

Format		Data	
Fixed	Free	Type	Entry
31-35	7th	I	Scaling of distance between each pair of nodes as a percentage. A default of 100 percent is used.
36-40	8th	I	Enter 1 for decrease of 1 node per series of master nodes. Default is 0. Enter 2 for a decrease of 2 nodes per series.
41-45	9th	I	Print flag if set equal to 1, nodal coordinate printout is omitted. If set equal to 2, only the generated coordinates are printed. If set equal to 0 or left blank, all coordinates are printed.

■ NODE MERGE

Merge Duplicate Nodes

Description

This option searches through all the nodes and merges all nodes which are closer to each other than a minimum-specified distance. The default minimum distance is 0.001

The merge provision only alters the node numbers defined by the COORDINATES and CONNECTIVITY options. Loading and boundary conditions must be applied to the new node numbers after nodal merge. The node merge command cannot be used with shells or beam elements.

The WRITE option can be used to save the new mesh.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words NODE MERGE.
2nd data block			
1-5	1st	I	Set to 1 to set the node count to the minimum number found after the use of this option.
6-10	2nd	I	Set to starting node number or nodal merge. Default is 1.
11-15	3rd	I	Set to last node number for nodal merge. Default to total number of nodes specified in SIZING parameter.
16-25	4th	F	Separation distance below which nodes will be merged together.
26-30	5th	I	Set to 1 to suppress printout of new connectivity.

■ UFXORD

Invoke User Subroutine UFXORD

Description

This block invokes the call to user subroutine UFXORD to generate or modify nodal coordinates (see *Volume D: User Subroutines and Special Routines*). The block can be repeated as often as necessary. This option ***must*** follow the COORDINATES option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-6	1st	A	Enter the word UFXORD.
2nd data block			
			Enter a list of nodes for which user subroutine UFXORD will be called.

■ CYLINDRICAL

Define Cylindrical Coordinate System

Description

This option allows for the input of a cylindrical coordinate system such that both the nodal input and output of a group of nodal points are treated in this cylindrical coordinate system. For nodes listed in this option, nodal input (that is, COORDINATES, POINT LOAD, FIXED DISP, etc.) and nodal output (that is, incremental and total displacements, etc.) are to be given in the cylindrical coordinate system defined here.

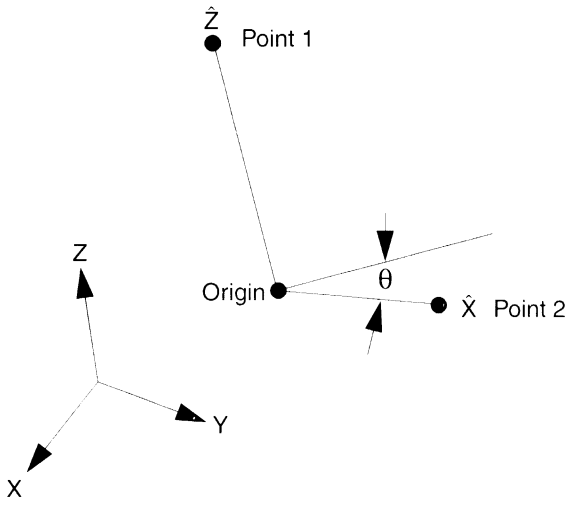
Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word CYLINDRICAL.
2nd data block			
1-5	1st	I	Number of sets of cylindrical coordinate data (required for this option).
6-10	2nd	I	Unit number for reading data. (Defaults to input deck.)
11-15	3rd	I	Enter nonzero integer to suppress printing of generated rectangular coordinates and nodal transformations after the END OPTION option.

Repeat data blocks 3, 4, 5, and 6 once for each data set.

3rd data block – coordinates of origin			
1-10	1st	F	Enter x coordinate (with respect to global Cartesian coordinates) of point defining origin of cylindrical coordinate system.
11-20	2nd	F	Enter y coordinate of origin.
21-30	3rd	F	Enter z coordinate of origin.

Format Fixed	Format Free	Data Type	Entry
4th data block – coordinates of point 1			
1-10	1st	F	Enter x coordinate (with respect to global Cartesian coordinates) of point such that vector from origin to this point defines the z axis of cylindrical coordinate system.
11-20	2nd	F	Enter y coordinate of z axis point.
21-30	3rd	F	Enter z coordinate of z axis point.
			Note: If the calculated direction cosines of the local Z-axis are zero, they are reset to (0., 0., 1.). This is the default for two-dimensional cylindrical coordinates
5th data block – coordinates of point 2			
1-10	1st	F	Enter x-coordinate (with respect to global Cartesian coordinates) of a point such that a vector from the origin to this point defines the axis from which θ is measured in planes perpendicular to the z-axis.
11-20	2nd	F	Enter y-coordinate of this point.
21-30	3rd	F	Enter z-coordinate of this point.
			Note: If the calculated direction cosines of the local X-axis are zero, they are reset to (1., 0., 0.).



Format Fixed	Format Free	Data Type	Entry
6th data block			
			Enter a list of nodes using this cylindrical coordinate system. MARC assumes that the coordinates of these nodes are given in cylindrical coordinates with respect to the cylindrical axes defined in data blocks 3, 4 and 5. Coordinates are entered as r, theta, z, where theta is in degrees. MARC also calculates nodal transformations for these nodes such that all nodal input and output is given with respect to the cylindrical coordinate system attached to these nodes.

■ WRITE

Write Connectivity and Coordinates

Description

This option allows you to write the final connectivity and coordinates to an auxiliary file. The values written are those after all internal mesh generations (MESH2D, FXORD or incremental generators) and all external (UFXORD, UFCONN) transformations have been performed. All node numbers are in the user system; that is, nonoptimized.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word WRITE.
11-15	2nd	I	Enter the unit number to write to. Default is unit 1.

■ ADAPTIVE

Define Error Criteria Used in Adaptive Analysis

Description

This model definition set controls the error criteria for adaptive meshing. The ADAPTIVE parameter must also be included.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-8	1st	A	Enter the word ADAPTIVE.
2nd data block			
1-5	1st	I	Enter the number of criteria to use.
6-10	2nd	I	Enter the unit number to read data, defaults to input.
11-15	3rd	I	Enter the frequency to perform adaptive meshing, default is every increment.
16-20	4th	I	Enter the unit number to which the adaptive mesh data will be written.

Data blocks 3 and 4 are repeated in pairs for each criteria selected.

3rd data block			
1-5	1st	I	Enter the criteria type: 1: Mean Strain Energy subdivide element if: element strain energy > f_1 * total strain energy/ NUMEL f_2 to f_6 is not used

Format Fixed	Format Free	Data Type	Entry

2: Zienkiewicz-Zhu Error Criteria

The error norm is defined as

$$\pi = \frac{\int(\sigma^* - \sigma)^2 dV}{\int\sigma^2 dV + \int(\sigma^* - \sigma)^2 dV} \quad \text{or}$$

$$\Upsilon^2 = \frac{\int(E^* - E)^2 dV}{\int E^2 dV + \int(E - E^*)^2 dV}$$

The stress error and strain energy errors are

$$X = \int(\sigma^* - \sigma)^2 dV \text{ and } Y = \int(E^* - E)^2 dV$$

The allowable element stress error is
AES = f_2 * X/NUMEL + f_3 * X * f_1/π/NUMEL
The allowable element strain energy error is
AEE = f_4 * Y/NUMEL + f_5 * X * f_1/γ/NUMEL
where
NUMEL is the number of elements in the mesh.
If f_2, f_3, f_4, f_5 are input as zero, f_2 = 1.0.
If stress error testing is to be performed, $f_1 \neq 0$ and f_2 and/or $f_3 \neq 0$, $f_4 = 0$, $f_5 = 0$.
The element will be subdivided when:
$\pi > f_1$ and X_{el} > AES.
If strain energy error testing is to be performed, $f_1 \neq 0$ and $f_2 = 0$, $f_3 = 0$, $f_4 \neq 0$, and/or $f_5 \neq 0$.
The element will be subdivided when:
$\gamma > f_1$ and Y_{el} > AEE
The default is f_2 = 1.0 if f_2, f_3, f_4, f_5 are input as 0.0.
It is advisable that $f_2 + f_3 \approx 1$ or $f_4 + f_5 \approx 1.0$.

3: Stress Discontinuity
(not yet implemented)

Format Fixed	Format Free	Data Type	Entry	
			4:	Location within Box subdivide element if at least one of the nodes: $f_1 < x < f_2$ and $f_3 < y < f_4$ and $f_5 < z < f_6$
			-4:	Location within box subdivide element if at least one of the nodes $f_1 < x < f_2$ and $f_3 < y < f_4$ and $f_5 < z < f_6$ If all nodes of element leave the box, the subdivided elements are merged together.
			5:	Node in Contact subdivide element if at least one of the nodes is in contact or belongs to a segment which is contacted f_1 to f_6 is not used, enter 0 or blank
			6:	Aspect Ratio (not yet implemented)
			7:	Skewness Ratio (not yet implemented)
			8:	Solution Gradient (used only for heat transfer) subdivide element if: gradient/maximum gradient > f_1 typical value of f_1 = 0.75 (f_1 must be < 1.0) f_2 to f_6 is not used, enter 0 or blank
			9:	Equivalent stress, strain subdivide element if: von Mises stress > f_1 * maximum von Mises stress or von Mises stress > f_2 or Equivalent strain > f_3 * maximum equivalent strain or Equivalent strain > f_4 Equivalent plastic strain > f_5 maximum equivalent plastic strain or Equivalent plastic strain > f_6

Format Fixed	Format Free	Data Type	Entry
			10: User subroutine UADAP subdivide element if: user/maximum user > f_1 or user > f_2 f_3 to f_6 is not used, enter 0 or blank user is defined in user subroutine UADAP 11: Previously Defined Mesh Use the refined mesh from a previous analysis as the starting point to this analysis. (see 4th data block).
6-10	2nd	I	Enter the maximum number of levels to adapt an element.
11-22	3rd	A	Enter the name of the element set that will use this criteria. Default is to apply the adaptive criteria to all elements.

4th data block (except for criteria type 11)

1-10	1st	E	First parameter f_1
11-20	2nd	E	Second parameter f_2
21-30	3rd	E	Third parameter f_3
31-40	4th	E	Fourth parameter f_4
41-50	5th	E	Fifth parameter f_5
51-60	6th	E	Sixth parameter f_6

4th data block (criteria type 11)

Include the data file written by the previous analysis (the unit number was specified on the second data block).

■ ATTACH NODE

Define the Nodes Which are Attached to Surfaces for Adaptive Mesh Capability

Description

This option allows you to attach nodes to a curve or surface. This option is used in conjunction with the SURFACE option in an adaptive mesh analysis. If two points on an edge of an element are attached to a curve or surface, any new points created by the adaptive procedure are placed on the curve. This improves the geometric modeling.

Note: In the case of updated Lagrange or if no surface is defined, the new nodes are placed midway between the previous nodes.

A node can be attached to as many as three surfaces; any additional surfaces are ignored.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word ATTACH NODE.
2nd data block			
1-5	1st	I	Enter the number of data sets.
6-10	2nd	I	Enter the unit number to read data, defaults to input.
3rd data block			
1-5	1st	I	Enter curve/surface identifier.

4th data block

Enter a list of nodes which are attached to this curve or surface.

■ SURFACE

Define the Curves and Surfaces used to Specify Coordinate Data

Description

This option allows the definition of a curve for one or two dimensional problems and a surface for three dimensional problems. These geometrical entities are used with the ADAPTIVE option for elastic analyses. The original nodal points can be associated (ATTACH NODE option) with these geometric entities. In such cases, when new nodes are created during the adaptive process, they are automatically placed on the curve or surface. This improves the geometric modeling.

Note: If no surface is defined, the new nodes are placed midway between the master nodes.

A node can be attached to as many as three surfaces, any additional surfaces are ignored.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-6	1st	A	Enter the word SURFACE.
2nd data block			
1-5	1st	I	Enter the number of curves to be defined.
6-10	2nd	I	Enter the unit number to read geometric information, defaults to input.
3rd data block			
1-5	1st	I	Curve/Surface identifier
6-10	2nd	I	Enter the curve or surface type: 1: 2D Polyline 2: 2D Circular Arc 3: 2D Circle 4: 2D NURB Curve (full description) 5: 2D NURB Curve (internally generate)

Format Fixed	Format Free	Data Type	Entry
			6: Plane 7: Sphere 8: Cylinder 9: NURB Surface (full description) 10: NURB Surface (internally generate)

A. FOR 2D POLYLINE

4th data block

1-5	1st	I	Number of points on polyline.

5th data block

Enter the coordinate points, one per line.

1-10	1st	I	X-coordinate.
11-20	2nd	I	Y-coordinate.

B. FOR CURVE TYPE CIRCULAR ARC

4th data block

1-10	1st	E	X-coordinate of starting point.
11-20	2nd	E	Y-coordinate of starting point.
21-30	3rd	E	X-coordinate of ending point.
31-30	4th	E	Y-coordinate of ending point.
41-50	5th	E	X-coordinate of center.
51-60	6th	E	Y-coordinate of center.
61-70	7th	E	Radius.

C. FOR CURVE TYPE CIRCLE

4th data block

1-10	1st	E	X-coordinate of center.
11-20	2nd	E	Y-coordinate of center.
21-30	3rd	E	Radius.

Format Fixed	Format Free	Data Type	Entry

D. FOR CURVE TYPE 2D NURB - FULL DESCRIPTION

4th data block

1-5	1st	I	Number of control points (NPU).
6-10	2nd	I	Order of NURB (NOU).

5th data block

Enter NPU homogeneous coordinates in E10 format (8 per line); use as many lines as necessary. The homogeneous coordinates are between 0 and 1.

6th data block

Enter (NPU plus NOU) knot vectors in E10 format (8 per line); use as many lines as necessary. The components of the knot vector are between 0 and 1.

7th data block

Enter the physical coordinates of control points - 1 control point per line (2 coordinates). There should be NPU lines.

1-10	1st	E	X-coordinate of control point.
11-20	2nd	E	Y-coordinate of control point.

E. FOR CURVE TYPE 2D NURB - INTERNALLY GENERATED

4th data block

1-5	1st	I	Number of control points (NPU). Minimum is 4.

5th data block

Enter the physical coordinates of control points - 1 control point per line (2 coordinates). There should be NPU lines.

1-10	1st	E	X-coordinate of control point.
11-20	2nd	E	Y-coordinate of control point.

Format Fixed	Format Free	Data Type	Entry

F. FOR PLANE

4th data block

Enter coordinates at four points, one point per line.

1-10	1st	E	X-coordinate.
11-20	2nd	E	Y-coordinate.
21-30	3rd	E	Z-coordinate.

G. FOR CURVE TYPE SPHERE

4th data block

1-10	1st	E	X-coordinate of center.
11-20	2nd	E	Y-coordinate of center.
21-30	3rd	E	Z-coordinate of center.
31-40	4th	E	Radius.

H. FOR CURVE TYPE CYLINDER/CONE

4th data block

1-10	1st	E	X-coordinate of starting point on axis.
11-20	2nd	E	Y-coordinate of starting point on axis.
21-30	3rd	E	Z-coordinate of starting point on axis.
31-40	4th	E	Radius at starting point.
41-50	5th	E	X-coordinate of ending point on axis.
51-60	6th	E	Y-coordinate of ending point on axis.
61-70	7th	E	Z-coordinate of ending point on axis.
71-80	8th	E	Radius at ending point.

Format		Data	
Fixed	Free	Type	Entry

I. FOR CURVE TYPE 3D NURB - FULL DESCRIPTION

4th data block

1-5	1st	I	Number of control points in first direction (NPU).
6-10	2nd	I	Order of NURB in first direction (NOU).
11-15	3rd	I	Number of control points in second direction (NPV).
16-20	4th	I	Order of NURB in second direction (NOV).

5th data block

Enter NPU times NPV homogeneous coordinates in E10 format (8 per line); use as many lines as necessary. The homogeneous coordinates are between 0 and 1.

6th data block

Enter (NPU plus NOU) plus (NPV plus NOV) knot vectors in E10 format (8 per line); use as many lines as necessary. The components of the knot vector are between 0 and 1.

7th data block

Enter the physical coordinates of NPU times NPV control points 1 control point per line (3 coordinates). There should be NPU times NPV lines.

1-10	1st	E	X-coordinate of control point.
11-20	2nd	E	Y-coordinate of control point.
21-30	3rd	E	Z-coordinate of control point.

Format Fixed	Format Free	Data Type	Entry

J. FOR CURVE TYPE 3D NURB – INTERNALLY GENERATED

4th data block

1-5	1st	I	Number of control points in first direction (NPU). Minimum is 4.
6-10	2nd	I	Not used; enter 0.
11-15	3rd	I	Number of control points in second direction (NPV). Minimum is 4.

5th data block

Enter the physical coordinates of NPU times NPV control points 1 control point per line (3 coordinates). There should be NPU times NPV lines.

1-10	1st	E	X-coordinate of control point.
11-20	2nd	E	Y-coordinate of control point.
21-30	3rd	E	Z-coordinate of control point.

■ GEOMETRY

Specify Geometrical Data

Description

The element geometry is specified in distinct sets. The information required varies from one element type to another. As a rule, the thickness of shell elements is given in the first defined variable (EGEOM1). The geometry for a particular element can be defined repeatedly and only the last data is used. This feature is designed for local variations of geometric data.

The geometry block is unnecessary if the element description does not require either EGEOM1, EGEOM2, or EGEOM3. (See *Volume B: Element Library*).

Note that the NODAL THICKNESS model definition block can also be used for the input of beam/shell thickness.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-8	1st	A	Enter the word GEOMETRY.
2nd data block			
1-5	1st	I	Number of distinct sets of element geometries to be input (optional).
6-10	2nd	I	Enter unit number for input of geometry defaults to input.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
Element geometries. The 3rd and 4th data blocks are entered as pairs, one for each distinct data set.			
1-10	1st	F	EGEOM1
11-20	2nd	F	EGEOM2
21-30	3rd	F	EGEOM3
31-40	4th	F	EGEOM4
41-50	5th	F	EGEOM5
51-60	6th	F	EGEOM6
			See library element descriptions in "Quick Reference" of *Volume B: Element Library* for the meaning of EGEOM1, etc. for each element type.
4th data block			
			Enter a list of elements to which the above geometry is applied.

Note: For elements 7, 10, 11 and 19, enter 1 in the EGEOM2 field to activate the constant dilatation option. This improves the behavior of the element for nearly incompressible analysis. See *Volume B: Element Library* for further details. This is an alternative to the CONSTANT DILATATION parameter.

For elements 3, 7 and 11, enter 1 in the EGEOM3 field to activate the assumed strain formulation. This improves the element bending behavior. This is an alternative to the ASSUMED STRAIN parameter.

■ NODAL THICKNESS

Define Nodal Thickness

Description

This option allows you to specify beam or shell thicknesses on a nodal basis. Interpolation to the element integration points is automatically taken care of using the element displacement shape functions as discussed for each element in *Volume B: Element Library* of the *MARC Reference Library.*

Note: If you specify element thicknesses for an element using the GEOMETRY model definition option, that data is used instead of the NODAL THICKNESS data input here. Also note that for composite elements, if you give the actual layer thicknesses, the sum of these layer thicknesses overrides both GEOMETRY data and NODAL THICKNESS data. If you input percentages of total thickness in the COMPOSITE data, then GEOMETRY data (or, if no GEOMETRY, then NODAL THICKNESS data from this option) is used.

Since the NODAL THICKNESS option allows input of only one thickness per node, thickness discontinuities must be input using GEOMETRY.

See *Volume B: Element Library* for elements which use nodal thicknesses.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word NODAL THICKNESS.
2nd data block			
1-5	1st	I	Number of data sets used to input nodal thickness values. The user subroutine UTHICK can be used for modifying NODAL THICKNESS values.
6-10	2nd	I	Enter the unit number for input of nodal thicknesses. Defaults to input deck.

Data blocks 3 and 4 are repeated as pairs, once for each data set.

3rd data block			
1-10	1st	F	Enter nodal thickness value
4th data block			
			Enter a list of nodes associated with the nodal thickness given above.

■ TRANSFORMATION

Define Nodal Coordinates for Transformation

Description

This option defines nodal coordinates for calculation of a direction cosine matrix, which is then used for transforming the global degrees of freedom of a specified node to a new local coordinate system.

This block can be used to set up local coordinate systems at a number of flagged nodes, for application of boundary conditions in a transformed system, or for printout purposes. Six points should be noted:

1. The displacements and loads or reactions are output in the transformed system at such nodes.
2. The transformation is done on all Cartesian displacements. Thus, for the shell elements, the derivative degrees of freedom become the derivative of the transformed displacements with respect to the original surface coordinate system.
3. Transformations are assumed to be orthogonal.
4. All kinematic conditions such as boundary conditions and ties at that node must be input in the transformed system.
5. All concentrated nodal loads must be applied in the transformed system.
6. Transformations should not be applied to nodes that can come into contact with either a rigid or deformable body,

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-14	1st	A	Enter the word TRANSFORMATION.

Format Fixed	Format Free	Data Type	Entry
2nd data block			
1-5	1st	I	Number of distinct sets of transformations data to be entered (optional).
6-10	2nd	I	Enter unit number for input of transformation data, defaults to input.
11-15	3rd	I	Enter 1 to suppress printout of transformation data.

The 3rd and 4th data blocks are entered as pairs, one for each distinct data set.

3rd data block			
1-5	1st	I	Node number. Enter 0 to read a list of nodes. See 4th data block.
6-15	2nd	F	Global coordinates of a first point A such that the vector from this point to the node is direction 1 of the local coordinate system. (See Figure 3-1a.)
16-25	3rd	F	
26-35	4th	F	
36-45	5th	F	Global coordinates of a second point, such that this point, the first point, and the node define the 1-2 plane of the local coordinate system.
46-55	6th	F	
56-65	7th	F	

Note: Direction 2 of the local coordinate system will be constructed perpendicular to direction 1 such that this second point has a positive 2 coordinate in the local 1-2 plane. See Figure 3-1 "(b) Three-dimensional Analysis)".

Direction 3 of the local coordinate system is given by a cross product of direction 1 with direction 2.

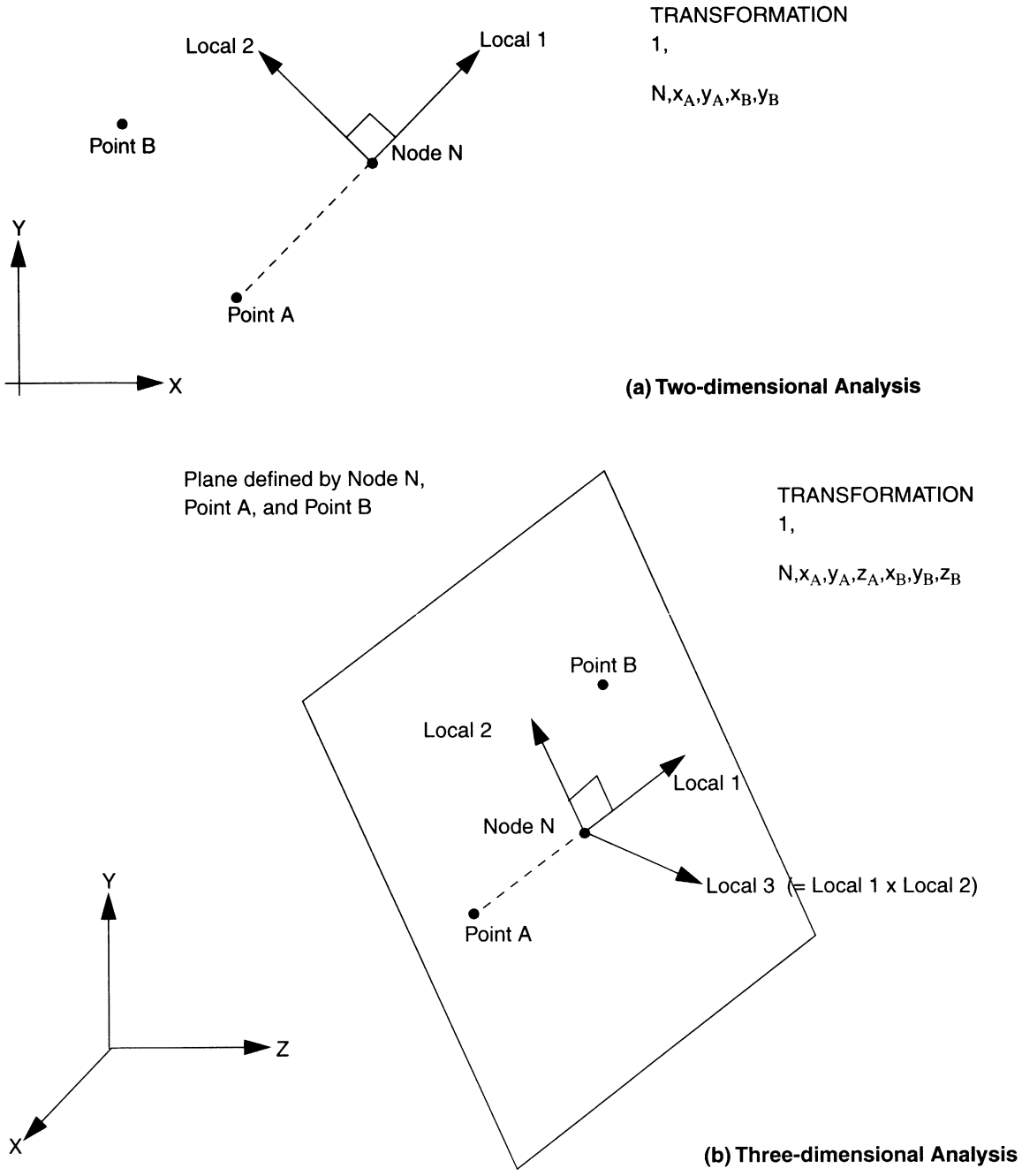


Figure 3-1 Transformation Option

Format Fixed	Format Free	Data Type	Entry
4th data block			
Include only if the first field in the 3rd data block is 0.			
			Enter a list of nodes for which the above transformation is applied. Note that for elements in a plane (for example,; 1, 2, 3, 5, 6, 10, 11, 12, 15, 16, 17, 19, etc.) only the first two coordinates of the first point (cols. 6-15 and 16-25) need be supplied. See Figure 3-1.

■ SHELL TRANSFORMATION

Define Shell Transformation

Description

This option allows you to transform the global degrees of freedom of (doubly curved) shells or beams to local degrees of freedom. It facilitates the input of boundary conditions, point loads and bending moments. A more detailed description of this capability is given in *Volume A: User Information.*

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-20	1st	A	Enter the words SHELL TRANSFORMATION.
2nd data block			
1-5	1st	I	Number of data sets to be input (optional).
6-10		I	Unit number from which input is to be read (defaults to input).
The 3rd and 4th data blocks are entered as pairs, one for each data set.			
3rd data block			
1-5	1st	I	Transformation type (see *Volume A: User Information*). Transformation type 1 is used for beam elements 15, 16, and 17. Transformation types 2 to 4 are used for shell elements 4, 8 and 24.
6-15	2nd	F	First component of $\hat{t}$ vector in surface θ^1 - θ^2 coordinate system. Only needed for transformation types 2 and 4.
16-25	3rd	F	Second component of $\hat{t}$ vector in surface θ^1 - θ^2 coordinate system. Only needed for transformation types 2 and 4.
4th data block			
			Enter a list of nodes to which the above displacements are applied.

■ UTRANFORM

Invoke User Subroutine UTRANS

Description

This option allows you to transfer the global degrees of freedom to local degrees of freedom. This is done through user subroutine UTRANS (see *Volume D: User Subroutines and Special Routines*).

Note: This option should not be used on boundary nodes which can come into contact with rigid surfaces in a contact analysis.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-9	1st	A	Enter the word UTRANFORM.
2nd data block			
1-5	1st	I	Number of data sets to be input.
6-10	2nd	I	Unit number from which input is to be read. Defaults to input.

The 3rd data block is entered once for each data set.

3rd data block			
			Enter list of node numbers to which user transformations are applied.

■ TYING

Define Tying Constraints

Description

This option is used to define homogeneous constraints. Constraints are defined by specifying a tied node and one or more associated retained nodes. Further details are provided in *Volume A: User Information*. Special types of tying can be obtained using user subroutine UFORMS (see *Volume D: User Subroutines and Special Routines*).

A rigid link for either small deformation or large deformation can be implemented by using tying type 80.

To obtain tying constraint based on updated current coordinates, add 1000 to tying type code. For tying type associated with user derived tying (UFORMS), subtract 1000 to tying type code.

In a coupled thermal-mechanical analysis during the heat transfer subincrements, tying type 1 is used for all tying types except 31, 32, 33, and 34.

If TYING CHANGE is subsequently used to increase the number of constraints, the TIE parameter must also be included.

A tying constraint always consists of a tied node (removed from the system) and one or more retained nodes (which remain in the system). Each tying constraint is specified by a series of two data blocks (data blocks 3 and 3a).

If a sequence of similar tying types must be specified, a list of nodes for tied nodes (3b) and corresponding retained nodes (3c - 3d) must be given.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-5	1st	A	Enter the word TYING.
2nd data block			
1-5	1st	I	Number of constraint equations to be read (optional).
6-10	2nd	I	Unit number for input of tying data. Defaults to input.
11-15	3rd	I	Enter 1 to suppress printout of tying data.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
Data blocks 3 and 3a or 3 and 3b, 3c, 3d, are given once for each constraint equation set.			
1-5	1st	I	Enter the code for tying type. SeeTable 3-2 and *Volume A: User Information* for definition of default types and user-defined routines.
6-10	2nd	I	Enter 0 to indicate a list of nodes to be tied will be defined in data block 3b. Enter a node number to indicate an individual node to be tied.
11-15	3rd	I	Number of retained nodes for this tying type. If a standard MARC tying type is used this does not need to be entered.
(3a) data block			
If the number of a node is entered in the second field of the 3rd data block (above), use data block 3a to list the retained nodes.			
1-80	1st	I	Sequence of retained nodes for constraint in (16I5) format, etc.
(3b) data block			
If no tied node is entered in the second field of the 3rd data block (that is, 0 is entered), use data blocks 3b, 3c, and 3d to enter a list of nodes to be tied.			
			Enter an unsorted list of nodes to be tied.
(3c) data block			
			Enter an unsorted list of nodes which will be the first retained nodes associated with tied nodes given in data block 3b.
(3d) data block			
			Same as 3c except second retained nodes, etc. **Note:** List verbs EXCEPT, INTERSECT and sorted node sets are illegal in these lists.

Table 3-2 Summary of Standard Tying Types

Tying Code	Number of Retained Nodes	Purpose	Remarks
I≤NDEG	1	Tie the Ith degree of freedom at the tied node to the Ith degree of freedom at the retained node	
100	1	Tie all degrees of freedom at the tied node to the corresponding degrees of freedom at the retained node	
23	2	Tie axisymmetric solid node to axisymmetric-shell (element type 1) node	Both tied and retained nodes must be transformed to local system. TRANSFORMATION option must be invoked. (See *Volume A: User Information*, Table 9-17)
15	Number of retained nodes is 1 less than the number of shell nodes in the z-r plane of the section	Special tying types for pipe bend element 17 to remove rigid body modes (see *Volume B: Element Library*)	
16	Number of shell nodes in the z-r plane of the section	Special tying types for pipe bend element 17 to remove rigid body modes (see *Volume B: Element Library*)	
17	2	Special tying types for pipe bend element 17 to couple bend section into pipe line (see *Volume B: Element Library*)	
18	2	Joining together the boundaries of intersecting shell, element type 4, 8 or 24. Fully moment carrying joint.	Tied node is also second retained node. Neither node can be transformed (see *Volume A: User Information*, Table 9-15)
28	2	Joining intersecting shells, element type 4, 8, or 24. Pinned joint.	Tied node is also second retained node (see *Volume A: User Information*, Table 9-15)
19	2	Use beam element 13 as a stiffener on shell elements 4 or 8. Tied node is beam node: First retained node is shell node, second is beam node again. Beam node should be on, or close to, the normal to the shell at the shell node.	

Table 3-2 Summary of Standard Tying Types (Continued)

Tying Code	Number of Retained Nodes	Purpose	Remarks
20	3	Create an extra node in a shell type 8 element tied to the interpolation function of the shell. Use in conjunction with tying type 21 to tie a beam element 13 or a stiffener across a shell element.	Always use after tying tying type 21.
21	2	Same as type 19, but tying beam to an interpolated shell node not as a vertex of an element – element type 8 only. Must be followed by type 20 to tie the interpolated shell node into the shell mesh.	Must be followed by a tying type 20.
24	2	Join intersecting shells or beams, element type 15-17.	Tied node is also second retained node. Neither node can be transformed. Tying is necessary only when there is a large angle between the two plates.
25	2	Join solid mesh to shell or beam (type 15 or 16).	Similar to 23, but no transformation needed. Tied node is also second retained node.
26	2	Join solid mesh to axisymmetric shell (type 1 or 89).	Similar to 5. Tied node is also second to retained node.
27	2	Join Fourier solid to Fourier shell (type 90).	Tied node is also second retained node.
31	2	Refine mesh of first order (linear displacement) elements in 2D.	Tie interior nodes on refined side to corner nodes on coarse side.
32	2	Refine mesh of second order (quadratic displacement) elements in 2D.	Tie interior nodes on refined side to the edge of an element on the coarse side.
33	4	Refine mesh of 8-node bricks	Tie interior node on the refined side to the 4 corner nodes of an element face on the coarse side.
34	8	Refine mesh of 20-node bricks	Tie interior nodes on refined side to the 8 (4 corner, 4 midside) nodes of an element on the coarse side.
52	1	Pin joint for beam types 14, 25, or 52.	
53	1	Fully moment carrying joint for beam types 14, 25, or 52.	

Table 3-2 Summary of Standard Tying Types (Continued)

Tying Code	Number of Retained Nodes	Purpose	Remarks
13	2	Joining two elements type 13 under an arbitrary angle. Fully moment carrying joint.	Tied node is also the second retained node.
>100	1	Generate several tyings of type ≤ NDEG.	Tying code is first d.o.f. multiplied by 100 added to last d.o.f.; that is, 209 means tie 2nd to 9th d.o.f. at tied node to resp. 2nd and 9th d.o.f. at retained node.
WARNING: TRANSFORMATION MUST NOT BE USED AT NODES INVOLVED IN TYING TYPES 13, 18, 19, 20, 21, 22, 24, 25, 49, 50, 51, 52, OR 53.			
80	2	Form a rigid link between tied node and retained node. This works for either small or large deformation. If a rigid region is to be modeled, use multiple ties of type 80, with the tied node of each link being a different node, and use the same common node as the retained node.	The second tied node is an extra node not connected to the structure which contains the rigid body rotation.
85	2	Tying of temperatures between shell and solid elements in heat transfer analysis (linear distribution of temperatures in the thickness direction of shell elements).	Tied node is the shell node and two retained nodes are nodes of the solid element. Order of the retained nodes follows the shell node degrees of freedom.
85	3	Tying of temperatures between shell and solid element in heat transfer analysis (quadratic distribution of temperatures in the thickness direction of shell element).	Tied node is the shell node and three retained nodes are nodes of the solid element. Order of the retained nodes follows shell node degrees of freedom.

■ SERVO LINK

Input Homogeneous Linear Constraints

Description

This option uses homogeneous linear constraint capability (TYING) to input simple constraints of the form:

$$u_t = a_1 u_{r1} + a_2 u_{r2} + \ldots$$

where u_t is a degree of freedom to be constrained.

u_{r1}, u_{r2} etc., are the other retained degrees of freedom in this structure.

a_1, a_2 etc., are constants provided in this option.

For more information, see *Volume A: Theory and User Information*. Note that more complex constraints can be entered via model definition set TYING and user subroutine UFORMS.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words SERVO LINK.
11-15	2nd	I	Number of servo links to be entered below; defaults to number given on TIE parameter.
16-20	3rd	I	Enter unit number for input of servo links. Defaults to input.

Data blocks 2 and 3 are entered as pairs, one for each servo link.

2nd data block			
1-5	1st	I	Number of retained nodes (must not exceed the value given in TIE parameter, fourth field).
6-10	2nd	I	Tied degree of freedom, at tied node.
11-15	3rd	I	Tied node.

Format Fixed	Format Free	Data Type	Entry
16-20	4th	I	First retained degree of freedom at first retained node.
21-25	5th	I	First retained node.
26-30	6th	I	Second retained degree of freedom at second retained node.
31-35	7th	I	Second retained node.
			Etc. (Continuation data in 16I5 format).

3rd data block

One set of this data block goes with each set from data block 2.

1-10	1st	F	Numerical constant a_1 joining tied and first retained variables.
11-20	2nd	F	Numerical constant a_2 joining tied and second retained variables
			Etc.

■ SPRINGS

Input Simple Linear Spring (Dashpot)

Description

This data set is used to input any simple linear springs. For dynamic analysis, a dashpot capability is offered as well.

The force in the spring/dashpot is:

$$F = K (u_2 - u_1) + C(\dot{u}_2 - \dot{u}_1)$$

where K is the spring stiffness, C is the damping coefficient, $\dot{u}_2$ is the displacement of the degree of freedom at the second end of the spring (third and fourth fields), and $\dot{u}_2$ is the displacement of the degree of freedom at the first end of the spring (first and second fields).

For a nonlinear spring/dashpot, the force can be specified with the user subroutine USPRNG with the general relation:

$$F = F(u_2 - u_1, \dot{u}_2 - \dot{u}_1)$$

See *Volume D: User Subroutines and Special Routines* for details.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-7	1st	A	Enter the word SPRINGS.
2nd data block			
Enter one data block per spring/dashpot.			
1-5	1st	I	Node to which first end of spring/dashpot will be attached.
6-10	2nd	I	Degree of freedom at above node to which spring/dashpot will be attached.
11-15	3rd	I	Node to which other end of spring/dashpot will be attached.

Format		Data	
Fixed	Free	Type	Entry
16-20	4th	I	Degree of freedom at above node to which spring will be attached.
21-30	5th	F	Stiffness of spring.
31-40	6th	F	Damping coefficient of dashpot (for dynamic analysis only).

■ SUBSTRUCTURE

Define External Degrees of Freedom for a Substructure

Description

This option defines which degrees of freedom are declared external in the definition of the current substructure. It can only be used if the parameter SUBSTRUC is used to indicate that the current analysis is creating a substructure.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-12	1st	A	Enter the word SUBSTRUCTURE.
2nd data block			
1-5	2nd	I	Enter number of sets of substructure data to be read.
6-10	3rd	I	Enter the unit number from which the following sets are read.

The 3rd and 4th data blocks are entered as pairs, once for each data set.

3rd data block			
			Enter a list of degrees of freedom which are declared external at the nodes in the 4th data block.
4th data block			
			Enter an unsorted list of nodes to which the above external declaration is applied.

■ SUPERINPUT

Define Superelement Input

Description

This data defines the input of superelements. The connectivity of the superelements is specified on nodal basis in the same order as the external nodes of the substructure. The order of the external nodes in the substructure is ascending and is printed in the substructure generation output.

This data can only be used if the SUPER parameter is used.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word SUPERINPUT.
2nd data block			
1-5	1st	I	Enter the number of superelements to be read.
6-10	2nd	I	Enter the unit number from which the following data is read.

The 3rd and 4th data blocks are entered as pairs, once for each superelement.

3rd data block			
1-5	1st	I	Level of the substructure.
6-10	2nd	I	Substructure number in the above level.
11-15	3rd	I	Fortran unit for the above substructure data base (defaults to unit number mentioned on SUPER parameter).
4th data block			
			Enter the list of unsorted nodes to which the external nodes of the substructure are connected in the current model in the correct order (ascending external node numbers of the substructure).
			Note: List verbs `EXCEPT`, `INTERSECT` and sorted node sets are illegal here.

■ BACKTOSUBS

Recover Substructure Output

Description

This option allows you to perform a displacement and stress calculation for a given substructure at the current main level increment. It can be followed by output control options, such as PRINT ELEM, PRINT CHOICE, etc.

To continue the analysis of the main level structure, the fourth field should be set to 1.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word BACKTOSUBS.
2nd data block			
1-5	1st	I	Level of substructure.
6-10	2nd	I	Substructure number in the above level.
11-15	3rd	I	Set to skip the stress analysis of the substructure.
16-20	4th	I	File unit for the optional sequential file containing the substructure data. If zero all data must be a data base unit given with the SUPER parameter.

■ STIFSCALE

Define Stiffness Scaling Factor

Description

This option allows the contributions of an element stiffness and mass matrix to be scaled before including them into the global stiffness matrix. The distributed loads associated with the element are also scaled. Note that this is a scalar multiple; no transformation occurs.

WARNING: If you use this option, you must define the scale factor for all elements; the default is zero.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word STIFSCALE.
2nd data block			
1-5	1st	I	Enter number of sets to be entered (optional).
6-10	2nd	I	Enter unit number from which the following data is read. Defaults to input.
3rd data block			
1-10	1st	F	Enter scaling factor.
4th data block			
			Enter a list of elements for which the above scaling is applied.

■ DEACTIVATE

Deactivate Elements

Description

This option allows you to deactivate elements during the course of an analysis, which can be useful to model ablation or excavation. After the elements are deactivated, they retain the stress state in effect at the time of deactivation and this state can be postprocessed or printed at any time. At a later stage in the analysis, the elements can again be activated with the ACTIVATE history definition option. As an alternative, you can use the user subroutine UACTIVE.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word DEACTIVATE.
2nd data block			
1-80	1st	I	Enter the list of elements to be deactivated at this time.

■ ERROR ESTIMATE

Create Error Estimation

Description

You can request that the program give information regarding the error associated with the finite element discretization. There are two measures; the first evaluates the stress discontinuity between elements. A large value implies that the stresses gradients are not accurately represented in the finite element mesh. In a classical linear elastic solution, this could be resolved by choosing quadratic elements over linear elements or refining the mesh.

The second error measure examines geometric distortion in the model. It first examines the aspect ratios and warpage of the elements and in subsequent increments measures how much these ratios change. This measure can be used to indicate if the original mesh is good and whether at a later time rezoning is required.

The evaluation of the stress error measure is moderately expensive. The evaluation of the geometric error measure is very inexpensive.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words ERROR ESTIMATE.
2nd data block			
1-5	1st	I	Enter 1 if the stress measure is to be evaluated.
6-10	2nd	I	Enter 1 if the geometric measure is to be evaluated.

■ USDATA

Invoke USDATA User Subroutine for Initialization

Description

This option invokes the call to user subroutine USDATA for the initialization of user variables. These variables (data) is stored in a common block USDACM that can be used in other user subroutines. This option requires the definition of the amount of data in the common block in REAL*4 words. The data is automatically saved on the restart file for use in subsequent analysis.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-6	1st	A	Enter the word USDATA.
11-15	2nd	I	Enter the number of real*4 words to be defined in common block USDACM via user subroutine USDATA.

Program Control

This section of the document describes various program control options. The information in this section is relevant to all types of analyses (mechanical, heat, Joule, bearing, acoustic, electrostatic, magnetostatic, and electromagnetic). In particular, the CASE COMBIN option allows you to combine separate load cases obtained from elastic analyses. The SOLVER option is used to control the solution procedure of the linearized equations. The default is the direct solvers; as an alternative, the iterative solver can be chosen. The OPTIMIZE option is very important to minimize the computational cost of the analysis, as the cost of analysis is proportional to the square of the size of the bandwidth. (Options POST, PRINT CHOICE, PRINT ELEM, and UDUMP all control the amount and method that you can obtain the resultant quantities.) The RESTART option is important for all nonlinear analysis or for postprocessing with MARC. As the solution to nonlinear problems is obtained using the incremental technique, the RESTART option is used to stop the analysis (checkpoint) and then continue it at some later time. The REAUTO option is used to overwrite previously defined control values upon restarting an analysis. The POST option is used to control the database that is used by Mentat II and Patran for postprocessing.

■ CASE COMBIN

Combine Load Cases

Description

This option allows you to combine different load cases for an elastic analysis. Each load case must be stored on a RESTART file and then combined with other cases as a scalar multiple (LAMBDA) of itself. All output element variables and nodal variables are combined.

This option can be used only in conjunction with the ELASTIC parameter. A new restart file of the resulting combination is written as increment 0 if it is requested.

The use of the CASE COMBIN option precludes the addition of any further load cases in the same run. Cases can only be combined from restart files.

This option can be used to perform the superposition of the results of a Fourier analysis at certain locations around the circumference. The positions for which superposition is requested can be either equally spaced or specified by you.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-11	1st	A	Enter the words CASE COMBIN.
2nd data block			
1-5	1st	I	Number of load cases to be read in and combined.
6-10	2nd	I	Number of stations for superposition of Fourier analysis. If input as a positive number the stations are equally spaced around the circumference starting at $\theta = 0$. If preceded by a minus sign, the θ - values are read from data lines.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
Only used for Fourier result superposition and if the second integer on the 2nd data block is negative.			
1-10	1st	F	Value of θ in degrees for first printout station.
11-20	2nd	F	Value of θ in degrees for second printout station.
			Continuation data is in Format 8E10.0.
4th data block			
The 4th data block is repeated for each load case.			
1-5	1st	I	Increment number on restart file to be read for this load case.
6-10	2nd	I	Input file number for restart file to be read. Default is Unit 9.
11-20	3rd	F	LAMBDA – Scalar multiplication factor to be used with this load case. Default is 1.0.

■ SOLVER

Specify Direct or Iterative Solver

Description

This option defines the solver to be used in the analysis. You can specify either the direct or iterative solver. The choice of whether the in-core or out-of-core procedure is used is automatically determined by the program, based upon the amount of workspace required and the number given on the SIZING parameter. You can also select whether the symmetric or nonsymmetric solver is used. At this time, the only nonsymmetric solver available is the direct profile one. Additionally, you can specify if the solution of a nonpositive definite system is to be obtained.

When the iterative solver is chosen, additional parameters must be defined which are used to control the accuracy.

Notes: It is not recommended to use the iterative solvers for beam or shell models, because these problems are ill-conditioned, resulting in a large-number of iterations. For a well-conditioned system, the number of iterations should be less than (and possible much less than) the square root of the total number of degrees of freedom in the system.

You control the maximum number of iterations allowed. If this is a positive number, the program stops if this is exceeded. If this is a negative number, the program prints a warning and continues to the next Newton-Raphson iteration or increment.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word SOLVER.
2nd data block			
1-5	1st	I	0 = Profile Direct Solver 1 = EBE Iterative 2 = Sparse Iterative 3 = Hardware Provided Profile Direct Solver 4 = Sparse Direct Solver 5 = Laypack Solver 6 = Hardware Provided Sparse Solver

Format Fixed	Format Free	Data Type	Entry
6-10	2nd	I	Enter 1 if the nonsymmetric solver is to be used.
11-15	3rd	I	Enter 1 if the solution of nonpositive definite system is to be obtained.

3rd data block

Only necessary if EBE iterative solver is used.

1-5	1st	I	Enter maximum number of groups; defaults to 30 times the number of element types.
6-10	2nd	I	Enter 1 if the previous solution is to be used as the initial trial solution.
11-15	3rd	I	Enter maximum number of conjugate-gradient iterations. Default is 1000, see note on previous page.
16-20	4th	I	Enter the type of preconditioner: Enter 0 for no preconditioner. Enter 1 for incomplete Cholesky preconditioner.

4th data block

Only necessary if EBE iterative solver is to be used.

1-10	1st	F	Enter tolerance on conjugate gradient convergence for stress analysis. Default is 0.01.
11-20	2nd	F	Enter tolerance on conjugate gradient convergence for heat transfer analysis.
21-30	3rd	F	Condition number cut-off. If calculated condition number is less than this value, the analysis is stopped.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
Only necessary if the sparse iterative solver is used.			
1-5	1st	I	Enter maximum number of conjugate-gradient iterations. Default is 1000; see note on previous page.
6-10	2nd	I	Enter 1 if the previous solution is to be used as the initial trial solution.
11-15	3rd	I	Enter 3 for diagonal preconditioner.
			Enter 4 for scaled-diagonal preconditioner.
			Enter 5 for incomplete Cholesky preconditioner.
4th data block			
Only necessary if the sparse iterative solver is to be used.			
1-10	1st	F	Enter tolerance on conjugate gradient convergence for stress analysis. Default is 0.001.

■ OPTIMIZE

Invoke Bandwidth Optimizers

Description

This option allows a choice of bandwidth optimizers to be invoked and is used to reduce computer costs in larger problems. Note that this option creates an internal node numbering different from your node numbering, but that all data input and output is in your node numbering system.

In addition, you can output the obtained correspondence table for later use. This correspondence table can then be read in subsequent analyses.

In a deformable contact analysis, the bandwidth is re-optimized when the contact conditions change.

Note: Gap elements can change the internal node numbers. This can result in a nonoptimal node numbering system.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-8	1st	A	Enter the word OPTIMIZE.
11-15	2nd	I	Enter: 2 Cuthill-McKee algorithm. 5 Read externally supplied correspondence table from unit specified in the fifth field. 9 Sloan Algorithm (default). 10 Minimum Degree Algorithm (only available for sparse direct solver). **Note:** If sparse direct solver is used, the minimum degree Algorithm is always used.
16-20	3rd	I	Enter 1 to have optimized mesh (elements then nodes) written to a file.
21-25	4th	I	Unit number of optimized mesh file. Only used if the third field is set to 1. Default is 18 if left blank.

Format Fixed	Format Free	Data Type	Entry
26-30	5th	I	Unit number of correspondence table. If the second field is not equal to 5 the correspondence table is written to this unit. If the second field is 5 then the correspondence table is read from this unit.
31-35	6th	I	Print flag for correspondence table. Set to 1 to suppress print out (default). Set to 2 to print table.

Option 2 – Cuthill-McKee

2nd data block

1-5	1st	I	Number of different numbering schemes to be tried. Usually less than 20.

Option 5 – User-specified Correspondence Table

2nd data block

	1st	I	Internal node numbers, continuation in 16I5 format on logical unit number given in the fifth field of data block 1.

Option 9 – Sloan Algorithm

No continuation data.

Option 10 – Minimum Degree Algorithms

No continuation data.

■ POST

Create File for Postprocessing

Description

This option creates a postprocessor file for time-history or variable versus variable plots using Mentat II or your own postprocessing. In the latter case, the file is accessed via the utility PLDUMP given in *Volume D: User Subroutines and Special Routines*.

You have two possibilities for the post file in association with restarted runs:

A. If the POST option follows the RESTART option, the program first copies the previous post file onto the new post file, thus providing a continuous post file from the beginning of the analysis. The old post file is closed after it has been read.

B. If the POST option precedes the RESTART option, the new post file contains only those increments analyzed in the current run.

One or the other options should be chosen – if (b) is used, a continuous post file is not created, so that (a) cannot subsequently be used for this analysis unless you combines the files with your own program.

Note: In a modal or buckling analysis in addition to POST option, the history definition RECOVER must be used for storing eigenvectors on post file.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-4	1st	A	Enter the word POST.
2nd data block			
1-5	1st	I	Number of element variables to be written on the file (at each integration point if the ALL POINTS parameter is present; otherwise, at the centroid of each element) at each increment (optional).
6-10	2nd	I	Unit number. Defaults to file16 if left blank.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Unit number of previous file, for a restarted run. Defaults to file 17 if left blank. Note that all data from this file (up to the restart increment) is copied to the new file upon restart, so that the post file is continuous from the start of the analysis, provided the POST option follows the RESTART option.
16-20	4th	I	Set to 0 for binary post file. Set to 1 for formatted post file. Set to 2 for both binary and formatted post file.
21-25	5th	I	**For Cray Only:** Set to 0 for internal binary. Set to 1 for IEEE binary compatible file. Set to 2 for VAX binary compatible file. Set to 3 for IBM (MVS/CMS) binary compatible file.
26-30	6th	I	Unit number for formatted post file. Defaults to 19.
31-35	7th	I	Unit number for previous formatted file, for a restart run. Defaults to 20.
36-40	8th	I	Set to 1 to convert restart file to post file with no analysis. Increments to be converted are given in the third and ninth field of RESTART model definition section.
41-45	9th	I	Number of increments between writing of post data. Defaults to write post file every increment.
46-50	10th	I	Number of user-defined post vectors. Vector is defined in user subroutine UPOSTV.
51-55	11th	I	Enter 1 to generate K2-style post file. Enter 3 to generate K3-style post file. Enter 4 to generate K4-style post file. Enter 5 to generate K5-style post file. Enter 6 to generate K6-style post file. Enter 7 to generate K7-style post file (default).

Format Fixed	Format Free	Data Type	Entry

3rd data block

This data block is used for input of post codes for selecting the strains and stresses to be written to the post file. This data block is repeated as many times as specified in the first field of the 2nd data block.

1-5	1st	I	Enter a post code.
6-10	2nd	I	Enter the layer number for shell elements.
11-35	3rd	A	Enter a 24 character label associated with this post code for use in post-processing. The code numbers are described in Table 3-3.

Table 3-3 Post Codes

Codes	Description
1-6	Components of strain. For rigid-perfectly plastic flow problems, components of strain rate
7	Equivalent plastic strain (integral of equivalent plastic strain rate). For rigid-perfectly plastic flow problems, equivalent plastic strain rate
8	Equivalent creep strain (integral of equivalent creep strain rate)
9	Total temperature
10	Increment of temperature
11-16	Components of stress
17	Equivalent Mises stress
18	Mean normal stress (tensile positive) for Mohr-Coulomb
19	User selected variable based on stress and temperatures defined in PLOTV. See *Volume D: User Subroutines and Special Routines.*
20	Thickness of element
21-26	Components of total plastic strain
27	Equivalent plastic strain. $\bar{\varepsilon}^{p} = \sqrt{\frac{2}{3}\Sigma\Delta\varepsilon_{ij}^{p}\Sigma\Delta\varepsilon_{ij}^{p}}$
28	Plastic strain rate
29	Total value of second state variable

Table 3-3 Post Codes (Continued)

Codes	Description
31-36	Components of creep strain
37	Equivalent creep strain. $\bar{\varepsilon}^{c} = \sqrt{\frac{2}{3}\Sigma\Delta\varepsilon^{c}_{ij}\Sigma\Delta\varepsilon^{c}_{ij}}$
38	Total swelling strain (from subroutine VSWELL)
39	Total value of third state variable
41-46	Components of Cauchy stress
47	Equivalent Cauchy stress
48	Strain energy density
49	Thickness strain for plane stress: Mooney or Ogden material
51-56	Real components of harmonic stress
57	Equivalent real harmonic stress
58	Elastic strain energy density
59	Equivalent stress/yield stress
60	Equivalent stress/yield stress (at current temperatures)
61-66	Imaginary components of harmonic stress
67	Equivalent imaginary harmonic stress
68	Plastic strain energy density
71-76	Components of thermal strain
78	Volume
81-86	Components of cracking strain (only for stress analysis)
91-103	Failure indices
108-109	Interlaminar shear for thick composite elements (TSHEAR parameter must be present)
111-116	Components of stress in preferred coordinate system defined by ORIENTATION option
121-126	Elastic strain
127	Equivalent elastic strain
175	Equivalent viscoplastic strain rate (powder material)
176	Relative density (powder material)
177	Void volume fraction (damage model)

Table 3-3 Post Codes (Continued)

Codes	Description
<0	Any negative user-selected variable based on stress and number temperatures defined in PLOTV
301	Total strains tensor
311	Stress tensor
321	Plastic strain tensor
331	Creep strain tensor
341	Cauchy stress tensor
351	Real harmonic stress tensor
361	Imaginary harmonic stress tensor
371	Thermal strain tensor
381	Cracking strain tensor
391	Stresses in preferred direction tensor
401	Elastic strain tensor
411	Stress in global coordinate system tensor
421	Elastic strain in global coordinate system tensor
431	Plastic strain in global coordinate system tensor
441	Creep strain in global coordinate system tensor
451	Velocity strains (for fluids)
461	Total strain in preferred direction tensor
Post Codes for Heat Transfer Analysis	
9 or 180	Total temperature
181-183	Components of temperature gradient T
184-186	Components of flux
Post Codes for Bearing Analysis	
190	Pressure
191-193	Components of pressure gradient
194-196	Mass flux vector

Table 3-3 Post Codes (Continued)

Codes	Description	
Post Codes for Joule Heating Analysis		
87	Voltage	
88	Current	
89	Heat generated	
Post Codes for Acoustic Analysis		
190	Pressure	
191-193	Components of pressure gradient	
Post Codes for Electrostatic Analysis		
130	Electric potential(V)	
131-133	Components of electric field intensity	(E)
134-136	Components of electric displacement	(D)
Post Codes for Magnetostatic Analysis		
140	Magnetic potential (2D analysis only)	(A_z)
141-143	Components of magnetic induction	(B)
144-146	Components of magnetic field intensity	(H)
Post Codes for Transient Electromagnetic Analysis		
131-133	Components of electric intensity	(E)
134-136	Components of electric displacement	(D)
137-139	Components of Lorentz force	
141-143	Components of magnetic induction	(B)
144-146	Components of magnetic field intensity	(H)
147-149	Components of current density	(J)
Post Codes for Harmonic Electromagnetic Analysis		
131-133	Real components of electric field intensity	(E)
134-136	Real components of electric displacement	(D)
137-139	Real components of Lorentz force	
141-143	Real components of magnetic induction	(B)
144-146	Real components of magnetic field intensity	(H)
147-149	Real components of current density	(J)

Table 3-3 Post Codes (Continued)

Codes	Description
151-153	Imaginary components of electric field intensity (E)
154-156	Imaginary components of electric displacement (D)
157-159	Imaginary components of Lorentz force
161-163	Imaginary components of magnetic induction (B)
164-166	Imaginary components of magnetic field intensity (H)
167-169	Imaginary components of current density (J)
Post Codes for Soil Analysis	
171	Porosity
172	Void ratio
173	Pore pressure
174	Preconsolidation pressure

Note: Codes 1-6 are the generalized strains and are section (not layer) quantities, so that the layer number need not be given.

Codes 11-16 give generalized stress quantities if no layer number is given for shell analysis. If a layer number is given, these are physical layer quantities.

For heat transfer, code 9 is used for all heat transfer elements.

When using shells in heat transfer, it is important to enter a code for each layer in chronological order if post file is to be correctly read by the INITIAL STATE or CHANGE STATE options.

Note that you do not need to select nodal values (that is, displacement, velocities and accelerations, and temperature for a heat transfer run) as these are automatically written to the post file.

Eigenmodes (dynamic analysis) and eigenvectors (buckling analysis) are written to the post file only if indicated by the RECOVER or MODAL INCREMENT/BUCKLE INCREMENT option.

For post codes 411, 421, 431, and 441 quantities can be obtained at either the midsurface, top surface, or bottom surface when shells are present. Layer 1 is, in this case, the midsurface, while layer 2 is the top surface, and layer 3 is the bottom surface.

■ IRM

Intergraph Interface

Description

This option allows you to generate an IRM file which is compatible with Intergraph. Two types of quantities can be on this file, element and nodal quantities. The element quantities stresses, strains, etc., can either be component values or invariant values or both. They are an average value within the element. The nodal quantities are values extrapolated from the integration points and a weighted average is calculated. Extreme care should be used interpreting the results using beam and/or shell elements.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-4	1st	A	Enter the word IRM.
11-15	2nd	I	Enter the unit number to which to write file, default is 39.
16-20	3rd	I	Frequency to write out file.

If no element date is required, skip to the 4th data block.

2nd data block

1-10	1st	A	Enter the word ELEMENT.

Repeat 3rd data block as often as required.

3rd data block

1-5	1st	I	Enter: 1 for stresses. 2 for total strains. 3 for creep strains. 4 for thermal strains. 5 for plastic strains. 6 for strain energy. 7 for stress/yield stress. 8 for failure indexes.

Format Fixed	Format Free	Data Type	Entry
6-10	2nd	I	Enter a layer number if shell elements. IL If the value in the first field is a 2, no layer number is required.
11-15	3rd	I	Enter 0 for component values. Enter 1 for invariants. Enter 2 for component and invariant values.

If no nodal data is required, skip to the next model definition point.

4th data block

1-8	1st	A	Enter the word NODAL.

Repeat 5th data block as often as required

5th data block

1-5	1st	I	Enter: 1 for displacements. 2 for velocities. 3 for acceleration. 4 for reaction forces. 5 for temperatures. 6 for generalized stresses. 7 for generalized strains. 8 for top/middle/bottom layer stresses. 9 for top/middle/bottom elastic strain. 10 for top/middle/bottom plastic strain. 11 for top/middle/bottom creep strain. 13 for eigenmode. 14 for harmonic displacement and reaction. **Note:** If shell elements are not included in the model, specifying 8, 9, 10, or 11 results in only the top or actual component

Internals of IRM files

Element Data

CAT = ELEM

TYPE - A.B.C.D where

A = S for stress
E for total strain
C for creep strain
T for thermal strain
P for plastic strain

B = C for component
I for invariant

C = 1 for 1st component
2 for 2nd component
3 for 3rd component
4 for 4th component
5 for 5th component
6 for 6th component

or

C = 1 for lowest principal
2 for intermediate principal
3 for highest principal
4 for Von Mises Intensity
5 for mean normal (hydrostatic)
6 for Tresca

D= layer number

If strain energy is requested, then:

TYPE A.D where

A = ETT for total strain energy density
ETI for incremental strain energy density
EET for total elastic strain energy density
EEI for incremental elastic strain energy density
EPT for total plastic strain energy density
EPI for incremental plastic strain energy density

D = layer number

If stress/yield stress is requested, then:

TYPE A.D where

A = SYI for stress/original yield stress
SYT for stress/yield stress at current temperature
D = layer number

If failure indexes are requested, then:

TYPE A.B.D where
A = FL
B = 1 to 6
D = layer number

Nodal Data

CAT = NODE
TYPE = A.B.C.

A =	D	for displacement
	V	for velocity
	A	for acceleration
	R	for reactions
	E	for eigenmode
	M	for magnitude of harmonic displacement
	P	for phase of harmonic displacement
	N	for magnitude of harmonic reaction
	Q	for phase of harmonic reaction
	TEMP	for temperatures
	GSC	for generalized stress components
	GSI	for generalized stress invariants
	GNC	for generalized strain components
	GNI	for generalized strain invariants
	TSC	for stress components, top layer
	TSI	for stress invariants, top layer
	MSC	for stress components, middle layer
	MSI	for stress invariants, middle layer
	BSC	for stress components, bottom layer
	BSI	for stress invariants, bottom layer
	TEC	for elastic strain components, top layer
	TEI	for elastic strain invariants, top layer
	MEC	for elastic strain components, middle layer
	MEI	for elastic strain invariants, middle layer
	BEC	for elastic strain components, bottom layer
	BEI	for elastic strain invariants, bottom layer
	TPC	for plastic strain components, top layer

	TPI	for plastic strain invariants, top layer
	MPC	for plastic strain components, mid layer
	MPI	for plastic strain invariants, mid layer
	BPC	for plastic strain components, bottom layer
	BPI	for plastic strain invariants, bottom layer
	TCC	for creep strain components, top layer
	TCI	for creep strain invariants, top layer
	MCC	for creep strain components, mid layer
	MCI	for creep strain invariants, mid layer
	BCC	for creep strain components, bottom layer
	BCI	for creep strain invariants, bottom layer
B =	X	for the X direction
	Y	for the Y direction
	Z	for the Z direction
	THX	for rotation about X
	THY	for rotation about Y
	THZ	for rotation about Z

Skipped if A is not D, V, A or R

C = 1 for 1st component
2 for 2nd component
3 for 3rd component
4 for 4th component
5 for 5th component
6 for 6th component

Skipped if A is D, V, A, or R

■ SDRC

SDRC Interface

Description

This option allows you to generate a Universal file which is compatible with the SDRC I-DEAS™ program. Two types of quantities can be on this file, element and nodal quantities. The element quantities stresses, strains, etc., can either be component values or invariant values or both. They are an average value within the element. The nodal quantities are values extrapolated from the integration points and a weighted average is calculated. Extreme care should be used interpreting the results using beam and/or shell elements. The SDRC file is written to a Universal file named `jid.unv`.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-4	1st	A	Enter the word SDRC.
11-15	2nd	I	Enter the unit number to which to write file; default is 40.
16-20	3rd	I	Frequency to write out file.

If no element date is required, skip to the 4th data block.

2nd data block

1-10	1st	A	Enter the word ELEMENT.

Repeat 3rd data block as often as required.

3d data block

1-5	1st	I	Enter: 1 for stresses. 2 for total strains. 3 for creep strains. 4 for thermal strains. 5 for plastic strains. 6 for strain energy. 7 for stress/yield stress. 8 for failure indexes.

Format Fixed	Format Free	Data Type	Entry
6-10	2nd	I	Enter a layer number if shell elements. IL If the value in the first field is a 2, no layer number is required.

If no nodal data is required, skip to the next model definition point.

4th data block

1-8	1st	A	Enter the word NODAL.

Repeat 5th data block as often as required

5th data block

1-5	1st	I	Enter: 1 for displacements. 2 for velocities. 3 for acceleration. 4 for reaction forces. 5 for temperatures. 6 for generalized stresses. 7 for generalized strains. 8 for top/middle/bottom layer stresses. 9 for top/middle/bottom elastic strain. 10 for top/middle/bottom plastic strain. 11 for top/middle/bottom creep strain. 13 for eigenmode. 14 for harmonic displacement and reaction. **Note:** If shell elements are not included in the model, specifying 8, 9, 10, or 11 results in only the top or actual component.

■ PRINT CHOICE

Specify Output

Description

This option allows you the control of the output from the program. The data given here remains in control until a subsequent PRINT CHOICE set is inserted – such a set can be included with either the model definition or with history definition data set. See also PRINT ELEMENT and PRINT NODE.

The default values print all elements and all nodes. Element quantities are printed at each integration point or at the centroid only, depending on whether CENTROID parameter is used. For shells, only the extreme fibers are output, plus layers where the inelastic strains or state variables are nonzero. For beams, fibers with inelastic strains or nonzero state variables are printed. In addition, section forces are given for these elements. This option also allows debug output of certain items. The default is no debug printout.

All of the above defaults are reset by the PRINT CHOICE option.

The PRINT CHOICE option has no effect on the post processor file.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-12	1st	A	Enter the words PRINT CHOICE.
2nd data block			
1-5	1st	I	Number of sets of first and last element numbers to be printed (maximum 10).
6-10	2nd	I	Number of sets of first and last node numbers to be printed (maximum 10).
11-15	3rd	I	Number of integration points to be printed in each element (not used if CENTROID is flagged).
16-20	4th	I	Number of layers to be printed. This is for beams and shells only and overrides the default described above.

Format Fixed	Format Free	Data Type	Entry
21-25	5th	I	Increments between printout. Default is print every increment.
26-30	6th	I	Enter 1 for complex nodal quantities to be output as magnitude and phase, otherwise real and imaginary components are given.
31-35	7th	I	Debug print flag. Enter a nonzero value and use data block 7.
36-40	8th	I	Log file flag. Enter unit number to which log file is to be written.

3rd data block

Include only if the first field of 2nd data block is not zero.

1-5	1st	I	First element in first set.
6-10	2nd	I	Last element in first set.
11-15	3rd	I	First element in second set.
16-20	4th	I	Last element in second set.
Etc.	5th		Etc. in I5 format.

4th data block

Include only if the second field of 2nd data block is not zero.

1-5	1st	I	First node in first set.
6-10	2nd	I	Last node in first set.
11-15	3rd	I	First node in second set.
16-20	4th	I	Last node in second set.
Etc.	5th		Etc. in I5 format.

5th data block

Include only if the third field of 2nd data block is not zero.

		I	Enter the list of integration points to be printed in (16I5) format (number of entries given in third field of data block 2). This is only used if CENTROID is not flagged. Be careful with analyses with several different element types.

Format Fixed	Format Free	Data Type	Entry
6th data block			
Include only if the fourth field of 2nd data block is not zero.			
		I	Enter the list of shell or beam fibers to be printed in (16I5) format. This overrides the program default, so you should be careful to not unintentionally suppress plasticity or creep printout.
7th data block			
Include only if the seventh field of 2nd data block is not zero			
1-5	1st	16I5	Enter debug print flags. See parameter PRINT.

■ PRINT ELEMENT

Specify Elements to be Included in Output

Description

This option allows you to choose which elements, and what quantities associated with an element are to be printed. If you do not specify NODE on the first data line, these values are at the integration points. If you specify the word NODE, these values are the extrapolated nodal values.

Note: This option revokes any NO PRINT that precedes it. Therefore, NO PRINT followed by PRINT ELEM and not followed by PRINT NODE results in the selected element printout and full nodal printout. Use PRINT NODE with a blank node list to suppress node output.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words PRINT ELEMENT.
11-20	2nd	A	Enter the word NODE (optional).
2nd data block			
1-5	1st	I	Enter the number of sets to be given below (optional).
6-10	2nd	I	Increment between printout. Default is print every increment.
11-15	3rd	I	File unit to which output is to be written. Defaults to standard output, unit 6.

Data blocks 3, 4, and, if necessary, 5 and 6 are given once for each data set.

3rd data block			
1-80	1st	A	Enter one or more of the following:
			STRAIN output total strain.
			STRESS output total stress.
			PLASTIC output plastic strain.

Format Fixed	Format Free	Data Type	Entry	
			CREEP	output creep, swelling and viscoelastic strain.
			THERMAL	output thermal strain
			ENERGY	output of strain energy densities: • total strain energy • incremental total strain energy • total elastic strain energy • incremental elastic strain energy • plastic strain energy • incremental plastic strain energy
			CRACK	output of cracking strain
			CAUCHY	output Cauchy stress.
			STATE	output state variables.
			PREFER	output stresses in preferred system.
			ELECTRIC	output electric field and electric flux
			MAGNETIC	output magnetic field and magnetic flux
			CURRENT	output current
			ALL	output of all of the above.

4th data block

Enter a list of elements to be printed.

Note: To suppress all element print-out, enter a blank list for the list of elements.

5th data block

Enter a list of integration points to be printed, or a list of nodes to be printed if the NODE option is specified on the 1st data block. This node list is a list of internal element node numbers (ranging from 1 to the maximum number of nodes per element).

6th data block

Enter a list of layers to be printed. This is only necessary if there are either thin walled beam or shell elements in the mesh, (that is, element types 1, 4, 5, 8, 13, 14, 15, 16, 17, 22, 24, 25, 45, 49, 50, 72, 75, 76, 77, 78, 79, 89, 90, 96, 97,138, 139, 140).

■ PRINT NODE

Specify Nodes to be Included in Output

Description

This option allows you to choose which nodes and what nodal quantities are to be printed.

Note: This option revokes any NO PRINT that precedes it. Therefore, NO PRINT followed by PRINT NODE and not followed by PRINT ELEM results in the selected nodal output and full element output. Use PRINT ELEM with a blank element list to suppress element printout.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words PRINT NODE.
2nd data block			
1-5	1st	I	Enter the number of sets to be given below (optional).
6-10	2nd	I	Increment between printout. Default is print every increment.
11-15	3rd	I	File unit to which output is to be written, default to standard output, unit 6.

Data blocks 3 and 4 are entered as pairs, one for each data set.

3rd data block			
1-80	1st	A	Enter one or more of the following:
			INCR output incremental displacement or potentials
			TOTA output total displacement or potentials
			VELO output velocity
			ACCE output acceleration
			LOAD output total applied load
			REAC output reaction/residual force

Format Fixed	Format Free	Data Type	Entry
			TEMP output temperature
			FLUX output flux
			Note: Fluxes are only available if the parameter HEAT, 0, 0, 2 is used.
			MODE output eigenvector (modal or buckle)
			STRESS output average generalized stresses at nodes
			VOLT output voltage (Joule analysis)
			PRES output pressure (bearing analysis)
			COOR output coordinates (for rezoning)
			ALL output all relevant quantities

4th data block

Enter a list of nodes to be printed.

Note: To suppress all nodal printout, enter a blank list for the list of nodes.

To suppress all nodal printout, enter a blank list for the list of nodes. The average nodal generalized stresses are obtained via an extrapolation and averaging procedure. If there is a geometric or material discontinuity at a node, this value is not correct unless either double nodes are used with kinematic tying, or you control which elements are to be averaged using the PRINT ELEMENT feature.

■ NO PRINT

Suppress Elements and Nodes in Output

Description

This option suppresses element and nodal output.

Note: This option is revoked by using either the PRINT CHOICE, PRINT ELEMENT, or PRINT NODE options. Therefore, NO PRINT followed by a PRINT ELEMENT, for example, results in element and full nodal printout. Use PRINT NODE or PRINT ELEMENT with blank node or element lists to suppress all node or element output.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words NO PRINT.

■ PRINT VMASS

Print Element Volumes, Masses, Costs, and Strain Energies

Description

This option allows you to obtain printed output of element volumes, masses, costs and strain energies. Options are provided for you to print the total quantities for each group of elements and the quantities for each element in the group or the total quantities for each group of elements only.

In order to have correct mass computations, mass density for each element must be entered through the ISOTROPIC/ORTHOTROPIC option. In order to have the correct cost, the cost per unit mass or the cost per unit volume must be defined through the ISOTROPIC/ORTHOTROPIC option. In order to have the correct cost computation, the cost per unit mass or the cost per unit volume must be defined. The total strain energy and the plastic strain energy, if applicable, are printed. Note that volumes and masses for some special elements (for example, gap element, semi-infinite element, etc.) is not be computed. These quantities can be written on either standard output file unit 6, or your specified unit.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-11	1st	A	Enter the words PRINT VMASS.
2nd data block			
1-5	1st	I	Enter the number of sets to be given below.
6-10	2nd	I	Enter 1 for option to print only total volumes, masses, costs, and strain energy for groups of elements. Default is 0.
11-15	3rd	I	File unit to which output is to be written; default to standard output, unit 6.

Data block 3 is repeated for each set.

3rd data block			
			Enter a list of elements to be printed.

■ REAUTO

Interrupt/Modify Load Sequence from Previous Analysis

Description

Used for changing conditions on restart of a problem in an autoloading sequence, dynamics, creep or heat transfer.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word REAUTO. This entry allows a reset of various parameters during restart. It can be used to override previously set values in the middle of automatic load incrementation. These values are originally set in the AUTO CREEP, AUTO INCREMENT, AUTO LOAD, AUTO STEP, DYNAMIC CHANGE, or TRANSIENT. Only the nonzero values set here are used. For AUTO LOAD, only the 3rd, 4th, and 5th fields are used; set other fields to 0.
2nd data block			
1-10	1st	F	Time step size. The value should only be set in dynamic problems.
11-20	2nd	F	End value of time for this set of boundary conditions.
21-25	3rd	I	Total number of time steps in this set of boundary conditions or, for AUTO LOAD, number of equal load increments. To immediately complete previous set of load history data, set to 1.
26-30	4th	I	Not used, enter 0.
31-35	5th	I	Reassembly interval for mass and stiffness matrices; for linear problems, set equal to the value given in the third field.

Format Fixed	Format Free	Data Type	Entry
36-40	6th	I	Desired number of recycles for the AUTO INCREMENT option.
41-50	7th	F	Maximum step size in AUTO INCREMENT option.
51-60	8th	F	Current percentage of total load to be applied (AUTO TIME or AUTO INCREMENT).

■ RESTART

Set Flags for Restart

Description

This option sets up the flags for the restart files; both for the input of a previous restart file and for output of a restart file from the current analysis. When the ELASTIC parameter is included, always restart at increment 0.

The following points should be noted concerning the RESTART option.

- A restart write frequency must be specified when a restart file is to be output. The analysis can then be restarted from any increment at which restart has been written.
- The restart file contains only those increments written during the current part of the analysis. The restart file is not continuous because of the large volume of data that can be involved. If it were written on the same file, the input/output time would be increased and also you might overflow the file storage in large problems.
- At restart, the data governing the increments (or increment set) next to be analyzed must follow the END OPTION as incremental input data. Any file input, such as a file of temperature increments describing a thermal history, must be skipped forward by you to the appropriate point; that is, to the beginning of the increment of the new part of the analysis.
- During any option set for a series of increments (AUTO CREEP, DYNAMIC CHANGE, AUTO LOAD, AUTO STEP, AUTO TIME, AUTO THERM CREEP, TRANSIENT), restart can be effected and control parameters changed. The program then continues to the end of the part of the analysis specified by the option. You have the option to terminate such a part of the analysis prematurely through the use of restart with the REAUTO option.
- The RESTART INCREMENT history definition option can be used to modify parameters defined in this option, or terminate the writing of a restart file.
- The RESTART LAST option can be used to save only the last converged increment. Even though it saves substantial file space, it is generally not recommended.
- The old restart file is closed after it has been read.

The input data describing the problem is not saved, and therefore must be read in with each restart. This option specifies restart parameters; for example, input/output files, restart increment, and intervals at which restarts are to be written.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-7	1st	A	Enter the word RESTART.
2nd data block			
1-5	1st	I	Set to 1 to write out restart data on a file. Set to 2 to read restart data from a file (that is, to restart a problem). Set to 3 to restart a problem and continue writing restart data for subsequent restart.
6-10	2nd	I	Number of increments between writing of restart data. For example, to write every three increments, set the second field to 3. This data is only used if the first field is set to 1 or 3. Defaults to 1 if left blank.
11-15	3rd	I	Enter the increment at which the restarted problem run begins. Only used if the first field of this data block is set to 2 or 3. The number here should be the number given in the message: `RESTART DATA AT INCREMENT i on TAPE j` which appears on the output of the previous run of the problem at the point where the restart is desired. **Note:** The problem can only be restarted at such points. The frequency of such points is determined by the data in columns 6 through 10 of this data block in the previous run of the problem. A restarted run should, in principle, have the same parameters as the original run. Only those parameters can be changed which do not affect the storage allocation within the program.
16-20	4th	I	Logical unit number for output of restart data; default logical unit number is 8 if nothing is given here and the RESTART option is specified in the parameters. Note that this file must be specified in the main program.
21-25	5th	I	Logical unit number for input of restart data from previous run; default is 9 if nothing is given here. Specify file in main program.
26-40	6th	I	Not used; enter 0.

Format Fixed	Format Free	Data Type	Entry
41-45	7th	I	Set to 1 to preserve triangularity stiffness matrix on restart file. With this flag set, reassembly and resolution at restart is avoided during creep, linear dynamics, linear elastic re-analysis, etc., if possible. For large problems, this saves considerable CPU. time at restart, but increases the size of the restart file substantially. Note that if this is set to 1, it must remain 1 on all restarted runs. If set to 0 on original run, it cannot be changed to 1 later. Note that this flag retains the value that it was given when the first increment was written on the restart file.
46-50	8th	I	Set to 1 to print out increment specified in the third field through increment specified in the ninth field. The job does not do any analysis. This is to allow you to recover increments suppressed by PRINT CHOICE.
51-55	9th	I	Set to last increment on restart file to be read. This is used in conjunction with the eighth field above or the eighth field of the POST option.
56-60	10th	I	Enter the subincrement at which the restart problem begins. Defaults to zero. This can be used to post-process either eigenvectors or harmonics.

■ RESTART LAST

Use Condensed Restart File

Description

This option sets up the flags for a condensed restart file, where only the last converged increment is saved.

Note: Upon writing, the last converged increment is written to the restart file. Upon reading, this increment is subsequently read in and the analysis continues. The restart file is closed after it has been read. The REAUTO option can be used to terminate any multi-increment history definition block.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-12	1st	A	Enter the word RESTART LAST.
2nd data block			
1-5	1st	I	Set to 1 to write out last increment of restart data on a file. Set to 2 to read restart data from a file (that is, to restart a problem). Set to 3 to restart a problem and write out the last increment of restart data for subsequent restart.
6-10	2nd	I	Logical unit number for output of restart data; default unit number is 8 if nothing is given here.
11-15	3rd	I	Logical unit number for input of restart data from previous run; default is 9 if nothing is given here.

■ UDUMP

Specify Nodes and Element for Postprocessing

Description

This option allows you to specify which nodes and elements can be referenced for postprocessing through user subroutines. Nodal quantities are accessed through subroutine IMPD, element quantities are accessed through subroutine ELEVAR (see *Volume D: User Subroutines and Special Routines*). During harmonic subincrements, subroutine ELEVEC is used.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-5	1st	A	Enter the word UDUMP.
2nd data block			
1-5	1st	I	First element, defaults to 1.
6-10	2nd	I	Last element, defaults to last element in mesh.
11-15	3rd	I	First node, defaults to 1.
16-20	4th	I	Last node, defaults to last node in mesh.

■ SUMMARY

Create Summary Report

Description

This option produces a summary of the results of the increment and outputs them in a report format. This option is in effect until a NO SUMMARY option is encountered. The summary consists of the maximum and minimum of temperatures, stresses, strains, plastic strains, creep strains, displacements, velocities, accelerations and reaction forces.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word SUMMARY.
11-15	2nd	I	Enter the unit number to be used for output, default is standard output, unit 6.
16-20	3rd	I	Enter the increment frequency of summary, default is every increment.

■ NO SUMMARY

Do Not Create Summary

Description

This option turns off the summary feature. The default is off unless the SUMMARY option has been previously invoked.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words NO SUMMARY.

■ ELEM SORT

Sort Element Results

Description

This option allows various element quantities to be sorted and the output given in report format. This option is in effect until a NO ELEM SORT option is encountered. This option allows you to sort either in ascending or descending order. In addition, you can use either the real numeric value or the absolute value. A range can be given over which to sort.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words ELEM SORT.
2nd data block			
1-5	1st	I	Enter number of sorts to be performed (optional). One data block, as given below, defines each sort.
6-10	2nd	I	Enter unit number to read sort data.
11-15	3rd	I	Enter the unit number to be used for output, default is standard output, unit 6.
16-20	4th	I	Enter the increment frequency, default is every increment.
3rd data block			
The 3rd data block is repeated once for each sort.			
1-5	1st	I	Enter code indicating type of quantity to be sorted (see Table 3-4).
6-10	2nd	I	Enter 0 for sort in descending order. Enter 1 for sort in ascending order. Default is 0; sort in descending order.
11-15	3rd	I	Enter 1 for sort by real numerical value. Enter 0 for sort by absolute value. Default is 0; sort by absolute value.
16-20	4th	I	Enter number of items to be included in sorted list.

Format Fixed	Format Free	Data Type	Entry
21-25	5th	I	Enter lowest element number of range to be sorted. Defaults to 1.
26-30	6th	I	Enter highest element number of range to be sorted. Defaults to last element in mesh.

Table 3-4 Element Sort Codes

Code	Description	Code	Description
1	first stress	28	fourth plastic strain
2	second stress	29	fifth plastic strain
3	third stress	30	sixth plastic strain
4	fourth stress	31	equivalent plastic strain
5	fifth stress	32	mean plastic strain
6	sixth stress	33	Tresca plastic strain
7	equivalent stress	34	first principal plastic strain
8	mean stress	35	second principal plastic strain
9	Tresca stress	36	third principal plastic strain
10	first principal stress	37	first creep strain
11	second principal stress	38	second creep strain
12	third principal stress	39	third creep strain
13	first strain	40	fourth creep strain
14	second strain	41	fifth creep strain
15	third strain	42	sixth creep strain
16	fourth strain	43	equivalent creep strain
17	fifth strain	44	mean creep strain
18	sixth strain	45	Tresca creep strain
19	equivalent strain	46	first principal creep strain
20	mean strain	47	second principal creep strain
21	Tresca strain	48	third principal creep strain

Table 3-4 Element Sort Codes (Continued)

Code	Description	Code	Description
22	first principal strain	49	temperature
23	second principal strain	61	voltage
24	third principal strain	73	first gradient
25	first plastic strain	74	second gradient
26	second plastic strain	75	third gradient
27	third plastic strain		

■ NO ELEM SORT

Do Not Create Report Sorted by Element

Description

This option turns off the ELEM SORT feature. The default is off unless the ELEM SORT option has been previously invoked.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words NO ELEM SORT.

■ NODE SORT

Sort Nodal Results

Description

This option allows various nodal quantities to be sorted and the output given in report format. This option is in effect until a NO NODE SORT is encountered. NODE SORT allows you to sort either in ascending or descending order. In addition, you can use either the real numeric value or the absolute value. A range can be given over which to sort.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words NODE SORT.
2nd data block			
1-5	1st	I	Enter number of sorts to be performed (optional). One data block as given below defines each sort.
6-10	2nd	I	Enter unit number to read sort data.
11-15	3rd	I	Enter the unit number to be used for output, default is standard output, unit 6.
16-20	4th	I	Enter the increment frequency, default is every increment.
3rd data block			
The 3rd data block is entered once for each sort.			
1-5	1st	I	Enter code indicating type of quantity to be sorted (see Table 3-5).
6-10	2nd	I	Enter 0 for sort in descending order. Enter 1 for sort in ascending order. Default is 0, sort in descending value.
11-15	3rd	I	Enter 1 for sort by real numerical value. Enter 0 for sort by absolute value. Default is 0, sort by absolute value.
16-20	4th	I	Enter number of items to be included in sorted list.

Format		Data	
Fixed	Free	Type	Entry
21-25	5th	I	Enter lowest node number of range to be sorted. Defaults to 1.
26-30	6th	I	Enter highest node number of range to be sorted. Defaults to last node in mesh.

Table 3-5 Node Sort Codes

Column	Code	Result
1-12	sort code I	Results in the Ith component of the incremental displacement to be sorted.
13-34	sort code I +12	Results in the Ith component of the total displacement to be sorted.
25-36	sort code I + 24	Results in the Ith component of the velocity to be sorted.
37-48	sort code I + 36	Results in the Ith component of the acceleration to be sorted.
48-60	sort code I + 48	Results in the nodal temperature to be sorted.
61-72	sort code I + 60	Results in the Ith component of the reaction force to be sorted.

■ NO NODE SORT

Cancel Report Sorted by Nodes

Description

This option negates the NODE SORT option. The default is off unless the NODE SORT option has been previously invoked.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words NO NODE SORT.

■ DESIGN OBJECTIVE

Define Objective Function to be Optimized

Description

This option defines the objective function for the optimization process. It is not needed for a pure sensitivity analysis run. If it is specified for a pure sensitivity analysis run, the gradient of the objective function is also computed. Currently, the only option is to "minimize" the objective function.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-16	1st	A	Enter the words DESIGN OBJECTIVE.
2nd data block			
N/A	1st	A	Enter one of the following: MATERIAL VOLUME MATERIAL MASS MATERIAL COST **Note:** For MATERIAL MASS, the mass density, and for MATERIAL COST, the material unit cost should be defined with the material data (for example, see ISOTROPIC)

■ DESIGN VARIABLES

Define Variable Design Parameters

Description

This option defines the design variables. If a sensitivity analysis is required, the derivative of the response with respect to each design variable and the element contributions to the response are reported. If an optimization analysis is performed, then the design variables are modified to optimize the objective function. The option can be used more than once.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-15	1st	A	Enter the words DESIGN VARIABLES.

Data blocks 2 through 5 are repeated for each design variable.

2nd data block			
N/A	1st	I	Enter design variable set id (optional). In general, this is not the same as design variable numbers assigned by the program and defined in the output.
N/A	2nd	A	Enter, as appropriate, one of the words GEOMETRY, MATERIAL, or COMPOSITE.
N/A	3rd	A	For GEOMETRY, enter one of the words: CTHIC – constant thickness over element AREA – cross-sectional area IXX – moment of inertia I_{xx} IYY _ moment of inertia I_{yy} BMHEI – beam height BMWID – beam width RADIU – radius WLLTH – wall thickness

Format Fixed	Format Free	Data Type	Entry
N/A	2nd	A	Enter either: < (for less than or equal to) or > (for greater than or equal to)
N/A	3rd	E	Enter the bound with the correct sign

The 3rd data block is only used if the 1st field of the 2nd data block is DIRTRAN, DIRROTA, DIRLTRA, DIRLROT; otherwise, it is skipped.

3rd data block

N/A	1st	E	Enter first component of vector.
N/A	2nd	E	Enter second component of vector.
N/A	3rd	E	Enter third component of vector. **Note:** The program will extract the direction cosines.

4th data block

N/A	1st	A	Enter the words LOAD CASES.

5th data block

N/A		I	Enter a list of load cases for which this constraint is prescribed.

6th data block

N/A	1st	A	Enter the word NODES.

7th data block

N/A			If first field of 2nd data block does not begin with RL, enter the list of constrained nodes. If first field of 2nd data block begins with RL, enter the first and second node numbers which are constrained relative to one another.

■ DESIGN STRESS CONSTRAINTS

Define Limits on Stress Response

Description

This option is used to specify stress constraints for a design sensitivity/design optimization process. The option can be used more than once.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-13	1st	A	Enter the words DESIGN STRESS.

Data blocks 2 through 7 are repeated for each stress constraint group.

2nd data block			
N/A	1st	I	Enter one of the words (preceded by the word ABSOLUTE where needed; for example, ABSOLUTE STRESS1):

STRESS1, STRESS2, STRESS3, STRESS4, STRESS5, STRESS6 — The stress components as defined for each element separately in *Volume B, Element Library* (see below for "generalized stresses").

VOMSTRS – von Mises equivalent stress
OSHSTRS – octahedral shear stress
MAPSTRS – maximum absolute principal stress
PRSTRS1 – algebraically highest (first) principal stress
PRSTRS2 – second principal stress
PRSTRS3 – third principal stress
TRESTRS – Tresca equivalent stress
STRESSV – normal stress along a vector
SHSTRSP – maximum shear stress on a plane

Format Fixed	Format Free	Data Type	Entry
			GENSTS1 ⋮ GENSTS9 } generalized stresses 1 through 9 obtained by integration through thickness of layered elements as defined in *Volume B, Element Library.*
N/A	2nd	A	Enter either: < (for less than or equal to) or > (for greater than or equal to)
N/A	3rd	E	Enter the bound with the correct sign.

The 3rd data block is only used if the 1st field of the 2nd data block is STRESSV or SHSTRSP; otherwise, it is skipped.

3rd data block

N/A	1st	E	Enter first component of vector.
N/A	2nd	E	Enter second component of vector.
N/A	3rd	E	Enter third component of vector. **Note:** Program extracts the direction cosines.

4th data block

N/A	1str	A	Enter the words LOAD CASES.

5th data block

N/A			Enter a list of load cases for which this constraint is prescribed.

6th data block

N/A	1st	A	Enter the word ELEMENTS.

7th data block

N/A			Enter the list of constrained elements.

■ DESIGN STRAIN CONSTRAINTS

Define Limits on Strain Response

Description

This option is used to specify strain constraints for a design sensitivity/design optimization analysis. The option can be used more than once.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-13	1st	A	Enter the words DESIGN STRAIN.

Data blocks 2 through 6 are repeated for each constraint group.

2nd data block			
N/A	1st	I	Enter one of the words (preceded by the word ABSOLUTE if needed; for example, ABSOLUTE STRAIN1): STRAIN1, STRAIN2, STRAIN3, STRAIN4, STRAIN5, STRAIN6 – The strain components as defined for each element separately in *Volume B, Element Library.* VOMSTRN – von Mises equivalent strain MAPSTRN – maximum absolute principal strain PRSTRN1 – algebraically highest (first) principal strain PRSTRN2 – second principal strain PRSTRN3 – third principal strain TRESTRN – Tresca equivalent strain

Format		Data	
Fixed	**Free**	**Type**	**Entry**
N/A	2nd	A	Enter: < (for less than or equal to) or > (for greater than or equal to)
N/A	3rd	E	Enter the bound with the correct sign.
3rd data block			
N/A	1st	A	Enter the words LOAD CASES.
4th data block			
N/A			Enter a list of load cases for which this constraint is prescribed.
5th data block			
N/A	1st	A	Enter the word ELEMENTS.
6th data block			
N/A			Enter the list of constrained elements.

■ DESIGN FREQUENCY CONSTRAINTS

Define Limits on Eigenfrequency Response

Description

This option is used to specify free vibration frequency constraints for a design sensitivity or design optimization case. The option can be used more than once.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-16	1st	A	Enter the words DESIGN FREQUENCY.
Data blocks 2 though 4 are repeated for each constraint group.			
2nd data block			
N/A	1st	A	Enter one of the following words: FRQCYCL – frequency in cycles per unit time FRQRADS – frequency in radians per unit time FGPCYCL – difference or gap in frequency between any two modes to be prescribed; in cycles per unit time FGPRADS – difference or gap in frequency between any two modes to be prescribed; in radians per unit time
N/A	2nd	A	Enter either: < (for less than or equal to) or > (for greater than or equal to)
N/A	3rd	E	Enter the bound. This is always positive.
3rd data block			
N/A	1st	A	Enter the word FREQUENCIES.

Format		Data	
Fixed	Free	Type	Entry
4th data block			
N/A	1st		If the constraint is on the frequencies of modes, enter a list of constrained mode numbers. If the constraint is on the difference between the frequencies of two modes, enter the numbers of the two modes.

Mechanical Analysis

This section is the first of four sections describing the input format for mechanical analysis. This section describes analysis controls and boundary conditions. The three subsequent sections concentrate on material properties, rate effects, and dynamic analysis.

The CONTROL option is required in all nonlinear analysis. It governs the number of increments and the accuracy associated with the analysis. This section also discusses the procedures for J-integral calculation in fracture mechanics.

The boundary conditions available for performing mechanical analysis are:

- Kinematic constraints of either zero or specified displacements.
- Surface, volumetric or nodal loads.
- Thermal loads.
- Foundation support.
- Surface contact.

These boundary conditions can be specified using a variety of techniques. The boundary conditions when given here in the model definition sections represent the total quantities to be applied in the zeroth increment. Mechanical loads are scaled if the SCALE parameter is included so that the model is at impending yield. Note that thermal loads are not scaled. In addition, as the zeroth increment is treated as linear elastic, the applied boundary conditions should not produce either material or geometry nonlinearities.

■ CONTROL

Control Option for Stress Analysis

Description

This option allows you to input parameters governing the convergence and the accuracy for nonlinear analysis. For heat transfer analysis, see "Heat Transfer Analysis".

For coupled thermal-stress analysis, data block 4 must be used in addition to the 3rd data block.

For nonlinear static analysis, the controls are described in *Volume A: Theory and User Information*. They do not appear on the restart file, and so must be re-entered on a restart run.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-7	1st	A	Enter the word CONTROL.
2nd data block			
1-5	1st	I	Maximum number of load steps/increments in this run. Default is 9999. This is a cumulative number and is usually used to stop the run when restart is being used. If an ELASTIC parameter is included, this field is ignored and all load cases are analyzed.
6-10	2nd	I	Maximum number of recycles/increments during an increment for plasticity, or other tangent modulus nonlinearities. Default is 3. This should usually be increased to 10 for rigid-plastic flow option. If a negative number is entered, then the program will do a maximum of the absolute value entered. If convergence has not been obtained, a warning is given and the program proceeds to the next increment. This is not recommended.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Minimum number of recycles during an increment for plasticity or other tangent modulus nonlinearities. Default is 0. Note that this data field forces this number of recycles to take place at all subsequent increments. **CAUTION:** This value is overwritten by the PROPORTIONAL INCREMENT option.
16-20	4th	I	Flag for convergence testing. If set to 0 or left blank, testing is done on residuals. If set to one, testing is done on displacements. If set to two, testing is done on strain energy. Note that testing on relative displacements or strain energy always requires at least one iteration. If nonlinear analysis is done with the CENTROID parameter, the residuals are not calculated and testing is always done on displacements. Note that nonlinear analysis with the CENTROID parameter is not recommended.
21-25	5th	I	Flag to specify relative or absolute error testing. If equal to 0, testing is done on relative error. If equal to 1, testing is done on absolute value. If set to 2, testing is done on relative error testing unless reactions or incremental displacements are below minimum value in which case absolute tolerances testing is used.
26-30	6th	I	Iterative procedure flag. 1. Full Newton-Raphson (default). 2. Modified Newton-Raphson (no reassembly during iteration). 3. Newton-Raphson with strain correction modification. 8. Secant method.
31-35	7th	I	Nonpositive definite flag. If set to 1, solution of nonpositive definite system is forced. Note that with use of gap and Herrmann elements, the matrix always is nonpositive definite and this entry has no significance.

Format Fixed	Format Free	Data Type	Entry
36-40	8th	I	No longer used; enter 0.
41-45	9th	I	To print convergence control messages to log file, enter 1.
46-50	10th	I	Control on initial stress stiffness. 0 Normal-full contribution. 1 For Mooney material, reduce contribution of hydrostatic pressure on initial stress stiffness according to: $\sigma^{initial} = \sigma - f_r \cdot p \cdot I$ where $\sigma^{initial}$ is the stress tensor used in the initial stress stiffness matrix, σ is the current stress tensor, f_r is entered through the PARAMETERS option, p is the hydrostatic pressure and I is a unit tensor. 2 No initial stress stiffness. 3 Use stress at beginning of increment, not last iteration. Enter a list of elements to be printed.
3rd data block			
1-10	1st	F	If relative residual checking: Maximum allowable value of maximum residual force divided by maximum reaction force. Default is 0.10. If relative displacement checking: Maximum allowable value of the change in displacement increment divided by the displacement increment. Default is 0.10.

Format Fixed	Format Free	Data Type	Entry
11-20	2nd	F	If relative residual checking: Maximum allowable value of maximum residual moment divided by maximum reaction moment. Default is 0.0, in which case, no checking on residual moment occurs.
			If relative displacement checking: Maximum allowable value of the change in rotational increment divided by the rotational increment. Default is 0.0, in which case, no checking on change in rotational increment occurs.
21-30	3rd	F	If relative residual checking: Minimum reaction force, if reaction force is less than this value, checking will be bypassed or absolute testing will be performed.
			If relative displacement checking: Minimum displacement, if displacement increment is less than this value, checking will be bypassed or absolute testing will be performed.
31-40	4th	F	If relative residual checking: Minimum moment: if moment is less than this value, checking will be bypassed or absolute testing will be performed.
			If relative displacement checking: Minimum rotation: if rotation increment is less than this value, checking will be bypassed.
41-50	5th	F	If absolute residual testing: Maximum value of residual force. Default is 0.0 in which case, no checking on residual force will take place.
			If absolute displacement tasking: Maximum value of displacement increment. Default is 0.0; in which case, no checking on displacements take place.

Format Fixed	Format Free	Data Type	Entry
51-60	6th	F	If absolute residual testing: Maximum value of residual moment. Default is 0.0 in which case, no checking on residual moments will take place. If absolute displacement tasking: Maximum value of rotation increment. Default is 0.0; in which case, no checking on rotations takes place.
61-70	7th	F	$\dot{\varepsilon}_o$ initial strain rate. (Rigid-Plastic Analysis only) Default is 1.e-4.
71-80	8th	F	$\dot{\varepsilon}$ cutoff strain rate. (Rigid-Plastic Analysis only) Default is 1.e-12.

4th data block

Only necessary for coupled analysis.

1-10	1st	F	Maximum nodal temperature change allowed. Used to control automatic time step scheme for heat transfer. Default value of 20.
11-20	2nd	F	Maximum nodal temperature change allowed before properties are re-evaluated and matrices reassembled. Default value of 100.
21-30	3rd	F	Maximum error in temperature estimate used for property evaluation. This control provides a recycling capability to improve accuracy in highly nonlinear heat-transfer problems (for example, latent heat, radiation boundary conditions). Default is 0, which bypasses this test. Set to maximum temperature error which is considered acceptable.

■ PARAMETERS

Definition of Parameters used in Numerical Analysis

Description

There are many parameters that are used in the finite element calculations. These parameters can be customized for your particular application. Some of these constants can be entered in other input blocks as well. The last nonzero value is used for the calculation.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word PARAMETERS.
2nd data block			
1-10	1st	E	Enter the scale factor which, when multiplied with the incremental strain, is used to predict the incremental strain in the next increment. Default is 1.0.
11-20	2nd	E	Enter the multiplier used to calculate the penalty used to impose boundary conditions. Default is 1.e9. The penalty used is, hence, 1.e9 times the maximum diagonal of the stiffness matrix. If the APPBC parameter is used, this option is not used.
21-30	3rd	E	Enter the penalty factor used to satisfy incompressibility in rigid plastic analysis for plane strain, axisymmetric, or solid analysis when displacement elements are used. Default is 100.
31-40	4th	E	Enter the penalty factor used to satisfy incompressibility in fluid analysis when displacement elements are used. Default is 1.e6.

Format Fixed	Format Free	Data Type	Entry
41-50	5th	E	Beta parameter used in transient dynamic analysis using Newmark-beta procedure Default is 0.25.
51-60	6th	E	Gamma parameter used in transient dynamic analysis using Newmark-beta procedure. Default is 0.50.
3rd data block			
1-10	1st	E	Enter the angle at which a node separates from a convex corner or becomes stuck in a concave corner for two-dimensional contact. Default is 8.625°.
11-20	2nd	E	Enter the angle at which a node separates from a convex corner or becomes stuck in a concave corner for three-dimensional contact. Default is 20.0°.
21-30	3rd	E	Enter the initial strain rate for rigid plastic analysis. Default is 1.e-4.
31-40	4th	E	Enter the cutoff strain rate for rigid plastic analysis. Default is 1.e-12.
41-50	5th	E	Enter the fraction of the mean strain that is subtracted from the stress tensor in the initial stress calculation. See the tenth field of the CONTROL option. Default is 0.0
51-60	6th	E	Enter the factor used to calculate the drilling mode for shell elements type 22, 75, 138, 139, and 140. Default is 0.0001.

■ FIXED DISP

Define Fixed Displacement

Description

This data defines the fixed displacement that each specified degree of freedom must take during the first and subsequent increments, unless it is further modified using the DISP CHANGE option. The boundary conditions are specified either by giving the kinematic displacement and a list of degrees of freedom and a list of nodal numbers or by the input of boundary conditions generated during mesh generation (MESH2D). The prescribed displacements are with respect to the degrees of freedom associated with the element, unless they have been transformed to a local coordinate system using either the TRANSFORMATION, SHELL TRANSFORMATION, or UTRANFORM options.

Note: In static analysis, the boundary conditions specified must always be sufficient to remove all rigid body modes.

It is advised that boundary conditions not be placed on nodes which might come into contact. Using a symmetry rigid body is preferred.

Further detail is given in *Volume A: Theory and User Information*. See *Volume B: Element Library*, for a definition of the degrees of freedom for each element type.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words FIXED DISP.
2nd data block			
1-5	1st	I	Number of sets of boundary condition data to be read (optional).
6-10	2nd	I	This field is set to nonzero to flag input of boundary conditions set during mesh generation (MESH2D). If this field is activated, no further data is required in this option block.
11-15	3rd	I	Unit number used for MESH2D option. **Note:** The boundary conditions are stored after the connectivity and coordinate data on this file, so that the model definition data must be arranged accordingly.

Format Fixed	Format Free	Data Type	Entry
For each set of boundary conditions use the 3rd, 4th, and 5th data blocks.			
3a data block			
Use only if **not** Fourier Analysis.			
1-10	1st	E	Prescribed displacement for first degree of freedom listed in data block 4.
11-20	2nd	E	Prescribed displacement for second degree of freedom listed in data block 4.
21-30	3rd	F	Prescribed displacement for third degree of freedom listed in data block 4. A maximum of eight kinematic constraints can be specified. The third data block is read as 8E10.3.
3b data block			
Use for Fourier analysis only.			
1-5	1st	I	Enter the series number associated with this boundary condition.
6-15	2nd	F	Prescribed displacement for first degree of freedom listed in data block 4.
16-25	3rd	F	Prescribed displacement for second degree of freedom listed in data block 4.
26-35	4th	F	Prescribed displacement for third degree of freedom listed in data block 4.
36-45	5th	F	Prescribed displacement for fourth degree of freedom listed in data block 4.
46-55	6th	F	Prescribed displacement for fifth degree of freedom listed in data block 4.

Format Fixed	Format Free	Data Type	Entry
4th data block			
			Enter a list of degrees of freedom to which the above prescribed displacements are prescribed. **Note:** List verbs `EXCEPT` and `INTERSECT` are illegal here.
5th data block			
			Enter a list of nodes to which the above boundary conditions are applied.

■ DIST LOADS

Define Distributed Loads

Description

This block of data allows pressure (surface and volumetric) loads to be specified. These values are incremental values unless the ELASTIC parameter is used, in which case they are the total loads. User subroutine FORCEM can be used for nonuniform, time dependent distributed loads.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words DIST LOADS.
2nd data block			
1-5	1st	I	Enter the number of sets of distributed loads to be entered (optional).
6-10	2nd	I	Enter unit number for input of distributed load data, defaults to input.

The 3rd and 4th data blocks are entered as pairs, one for each data set.

3a data block

Use if **not** Fourier Analysis.

1-5	1st	I	Parameter identifying the type of load. See library element description in *Volume B: Element Library.*
6-15	2nd	F	Enter the magnitude of this type of distributed load. For load type 102, enter the magnitude of force per unit mass (gravity acceleration) in first coordinate direction.
16-25	3rd	F	For load type 102, enter the magnitude of force per unit mass (gravity acceleration) in second coordinate direction

Format Fixed	Format Free	Data Type	Entry
26-35	4th	F	For load type 102, enter the magnitude of force per unit mass (gravity acceleration) in third coordinate direction
36-40	5th	I	Distributed load index (optional). (Distributed load index is to be used in subroutine FORCEM.)
3b data block			
Use if Fourier Analysis.			
1-5	1st	I	Parameter identifying the type of load. See library element description in *Volume B: Element Library.*
6-15	2nd	I	Enter the series number associated with this load.
16-25	3rd	F	Enter the magnitude of this type of distributed load. For load type 102, enter the magnitude of force per unit mass (gravity acceleration) in first coordinate direction.
26-35	4th	F	For load type 102, enter the magnitude of force per unit mass (gravity acceleration) in second coordinate direction.
36-40	5th	F	For load type 102, enter the magnitude of force per unit mass (gravity acceleration) in third coordinate direction.
4th data block			
			Enter a list of elements associated with the above distributed loads.

■ POINT LOAD

Define Nodal Point Loads

Description

This block of data allows nodal point loads to be specified. These values are incremental values unless the ELASTIC parameter is used, in which case they are total loads. The prescribed forces are with respect to the degrees of freedom associated with the element, unless they have been transformed to a local coordinate system using either the TRANSFORMATION, SHELL TRANSFORMATION, or UTRANSFORM options.

Note: Enter an upper bound to the number of nodes with point loads on the DIST LOADS parameter.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words POINT LOAD.
2nd data block			
1-5	1st	I	Enter number of sets of point loads to be entered (optional).
6-10	2nd	I	Enter unit number for input of point load data, defaults to input.

The 3rd and 4th data blocks are entered as pairs, one for each data set.

3a data block

Use if **not** Fourier Analysis.

1-10	1st	F	Nodal load associated with first degree of freedom.
11-20	2nd	F	Nodal load associated with second degree of freedom.
21-30	3rd	F	Nodal load associated with third degree of freedom.
31-40	4th	F	Nodal load associated with fourth degree of freedom.
41-50	5th	F	Nodal load associated with fifth degree of freedom.

Format Fixed	Format Free	Data Type	Entry
51-60	6th	F	Nodal load associated with sixth degree of freedom.
61-70	7th	F	Nodal load associated with seventh degree of freedom.
71-80	8th	F	Nodal load associated with eighth degree of freedom.

Continuation data is necessary, must be in 6E10.3 format. Continuation data is needed if more than eight degrees of freedom per node in analysis.

3b data block

Use only if Fourier Analysis.

1-5	1st	I	Enter the series number associated with this load.
6-15	2nd	F	Nodal load associated with first degree of freedom.
16-25	3rd	F	Nodal load associated with second degree of freedom.
26-35	4th	F	Nodal load associated with third degree of freedom.
36-45	5th	F	Nodal load associated with fourth degree of freedom.
46-55	6th	F	Nodal load associated with fifth degree of freedom.

4th data block

Enter a list of nodes having the point load given above.

■ ROTATION A

Define Rotational Axis

Description

This option defines the rotation axis for centrifugal and/or Coriolis loading. The rotational speed is entered though the DIST LOADS option as ω^2. Using an `IBODY=100` results in centrifugal loading, using an `IBODY=103` results in both centrifugal and Coriolis loading.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words ROTATION A.
2nd data block			
1-10	1st	F	Direction cosines of the axis of rotation.
11-20	2nd	F	
21-30	3rd	F	
3rd data block			
1-10	1st	F	Global (x,y,z) coordinates of a point on the axis of rotation.
11-20	2nd	F	
21-30	3rd	F	
31-40	4th	F	Velocity of the point on the axis of rotation, used for Coriolis loading only.
41-50	5th	F	
51-60	6th	F	

■ FLUID DRAG

Define Fluid Drag

Description

This block defines parameters required for the evaluation of drag loads and buoyancy loads on beam type structures immersed in a fluid. The drag forces are defined by Morison's equation. In static analyses, the fluid velocity is constant and defined here. In dynamic analyses, an additional contribution is added due to wave effects.

Note: In two dimensional analyses, the y-axis is considered vertically up, current is required only in x and y directions and the first direction cosine is either a one or minus one. In three dimensional analyses, the z-axis is considered vertically up.

Those elements which require fluid drag must be indicated on the DIST LOADS option using a load type of 11. If an element is above the fluid surface, no drag or buoyancy will be included.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words FLUID DRAG.
2nd data block			
1-10	1st	E	Enter the elevation of the sea bed.
11-20	2nd	E	Enter the surface elevation of the fluid outside the pipe.
21-30	3rd	E	Enter the surface elevation of the fluid inside the pipe.
31-40	4th	E	Enter the gravity constant.
41-50	5th	E	Enter mass density of fluid outside pipe.
51-60	6th	E	Enter mass density of fluid inside pipe.
61-70	7th	E	Enter the drag coefficient.
71-80	8th	E	Enter the inertia coefficient.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
1-10	1st	E	Current in x-direction.
11-20	2nd	E	Current in y-direction.
21-30	3rd	E	Current in z-direction.
31-40	4th	E	Gradient of x-current with elevation.
41-50	5th	E	Gradient of y-current with elevation.
51-60	6th	E	Gradient of x-current with elevation.
4th data block			
1-10	1st	E	Wave height.
11-20	2nd	E	Wave period.
21-30	3rd	E	Wave phase.
31-40	4th	E	1st direction cosine in horizontal plane of wave front.
41-50	5th	E	2nd direction cosine in horizontal plane of wave front.

■ INIT STRESS

Define Initial Stress

Description

This option allows you to enter initial stresses into the model. It is your responsibility to input a self-equilibrating set of stresses. These stresses should not produce any material nonlinearity. As an alternative, user subroutine UINSTR can be used.

Note: It is not recommended to use the CENTROID parameter with initial stresses because the residual load cannot be accurately calculated.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words INIT STRESS.
2nd data block			
1-5	1st	I	Number of sets of initial stress data to be entered (optional)
6-10	2nd	I	Unit number for input of initial stress data. Defaults to input file.

The 3rd, 4th, 5th, and 6th data blocks are repeated once for each data set. The components of stress are given in the order described for each element type in *Volume B: Element Library.*

3rd data block			
1-10	1st	F	Enter the first component of stress.
11-20	2nd	F	Enter the second component of stress.
21-30	3rd	F	Enter the third component of stress.
31-40	4th	F	Enter the fourth component of stress.
41-50	5th	F	Enter the fifth component of stress.
51-60	6th	F	Enter the sixth component of stress.
61-70	7th	F	Enter the seventh component of stress.

Format Fixed	Format Free	Data Type	Entry
4th data block			
			Enter a list of elements for which initial stress prescribed above is applied.
5th data block			
Only necessary if CENTROID parameter is not used.			
			Enter a list of integration points for which the initial stresses are applied.
6th data block			
Only necessary for shell or beam analysis.			
			Enter a list of layers for which the initial stress is prescribed.

■ INITIAL PLASTIC STRAIN

Define Initial Plastic Strain

Description

This option provides various ways of initializing the equivalent plastic strain throughout the model. Occasionally, in metal forming analysis, it is required to define the previous amount of equivalent plastic strain. This history dependent variable represents the amount of plastic deformation that the model was subjected to, and is used in the work (strain) hardening model.

Four ways of specifying the initial equivalent plastic strain values are shown below:

- Read the range of elements, integration points and layers and a corresponding value.
- Read the initial values through user subroutine INITPL.
- Read the initial values from a step of the post output file from a previous analysis with MARC. With this option the program assumes direct correspondence of the post file elements with the elements in the current analysis. Any spatial interpolation must be provided separately by you.
- Read a list of elements, integration points and layers and a corresponding value.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-13	1st	A	Enter the words INITIAL PLASTIC STRAIN.

Format Fixed	Format Free	Data Type	Entry
2nd data block			
1-5	1st	I	Not used, enter 0.
6-10	2nd	I	Enter 1 to initialize the equivalent plastic strain via the 3rd and 4th data blocks below. In this case, the third field must also be defined. Enter 2 to initialize the equivalent plastic strain via user subroutine INITPL. This subroutine will now be called in a loop over all elements in the mesh. Enter 3 to read the initial values of the equivalent plastic strain from the post file written by a previous analysis. In this case, the fourth and fifth field must also be defined. Enter 4 to initialize the equivalent plastic strain via the 5th, 6th, 7th, and 8th data blocks shown below. See also the third filed on this block.
11-15	3rd	I	Only nonzero if the second field is set to 1 or 4. Then this entry gives the number of pairs of data blocks in series 3 and 4 or in series 5, 6, 7 8 used to input the equivalent plastic strain. Defaults to 1.
16-20	4th	I	Only used if the second field is set to 3. Then this entry defines the unit number from which the post file information from the previous run is to be read.
21-25	5th	I	Only used if the second field is set to 3. In that case, this entry defines the step number of the previous analysis.
26-30	6th		Not used, enter 0.
31-35	7th	I	Set to 1 if option 3 is used, and a formatted post file is used. ***For Cray Only:*** Set to 2 for IEEE binary file Set to 3 for VAX binary file Set to 4 for IBM binary file
36-40	8th	I	Only nonzero if the second field is set to 2. Set to 1 to suppress printout of equivalent plastic strains values that are initialized in INITPL.

Format Fixed	Format Free	Data Type	Entry

Data blocks 3 and 4 are only input if the second field above is set to 1. In that case, the number of sets is equal to the number given in the third field above.

3rd data block

1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with this value.
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value.
21-25	5th	I	First layer of cross section point with this value.
26-30	6th	I	Last layer or cross section point with this value can only be bigger than 1 for beam or shell elements.

4th data block

1-10	1st	F	Initial value of the equivalent plastic strain for the above range of points.

Data blocks 5, 6, 7, and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block

1-10	1st	F	Initial value of the equivalent plastic strain at zeroth increment.

6th data block

Enter a list of elements to which the above value is applied.

7th data block

This data block is not necessary if the CENTROID parameter is used.

Enter a list of integration points to which the above value is applied.

8th data block

This data block is only necessary if there are either beams or shells in the mesh.

Enter a list of layer points to which the above value is applied.

■ INITIAL STATE

Initialize State Variables

Description

This option provides various ways of initializing the state variables throughout the model. The number of state variables per integration point is defined in the STATE VARS parameter. The default is one, with temperature always being the first state variable at an integration point. If more than one state variable per integration point has been assigned, this option can be used repeatedly to initialize all the state variables. The default value of state variables not initialized is zero.

Four ways of providing the state variable initial values are shown below:

- Read the range of elements, integration points and layers and a corresponding state variable value.
- Read the initial values through user subroutine INITSV.
- Read the initial values from a step of the binary or formatted post output file from a previous heat transfer analysis with MARC. This technique is most common for thermal stress analysis to initialize temperature (the first state variable at any point). With this option the program assumes direct correspondence of the post file elements with the elements in the current analysis. Any spatial interpolation must be provided separately by you.
- Read a list of elements, integration points and layers and a corresponding state variable value.

Note: Initial temperature values read in by this option are assumed to define the stress-free temperature field. Temperature changes which cause thermal strains are read in through the CHANGE STATE or AUTO THERM options.

In a coupled analysis, the temperatures are not independent state variables and the INITIAL TEMP option must be used.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-13	1st	A	Enter the words INITIAL STATE.
2nd data block			
1-5	1st	I	Enter the state variable identifier for the state variable being set (1,2,etc.). 1=temperature. If more than one state variable is being used, the STATE VARS parameter must be included.
6-10	2nd	I	Enter 1 to initialize the state variable via the 3rd and 4th data block below. See also the third field on this data block. Enter 2 to initialize the state variable via user subroutine INITSV. This subroutine will now be called in a loop over all elements in the mesh. Enter 3 to read the file values of the state variable from the post file written by a previous heat transfer analysis. In this case, the fourth and fifth fields must also be defined. Enter 4 to initialize the state variable via the 5th, 6th, 7th, and 8th data blocks as given below. Also, see the third field on this data block.
11-15	3rd	I	Only nonzero if the second field is set to 1 or 4. Then this entry gives the number of pairs of data blocks in series 3 and 4 or in series 5, 6, 7, and 8 used to input the state variable.
16-20	4th	I	Only used if the second field is set to 3. Then this entry defines the unit number from which the post file information from the previous heat transfer run is to be read. Defaults to unit 24 for a formatted post file and to unit 25 for a binary post file.
21-25	5th	I	Only used if the second field is set to 3. In that case, this entry defines the increment number on the heat transfer run post file to be used as the definition of the initial state variable values.

Format Fixed	Format Free	Data Type	Entry
26-30	6th		Not used, enter 0.
31-35	7th	I	Set to 1 if option 3 is used, and a formatted post file is used. ***For Cray Only:*** Set to 2 for IEEE binary file Set to 3 for VAX binary file Set to 4 for IBM binary file
36-40	8th	I	Only nonzero if the second field is set to 2. Set to 1 to suppress printout of state variable values that are initialized in INITSV.
41-45	9th	I	Enter the post code number to be read into this state variable, default is 9 (temperature).

Data blocks 3 and 4 are only input if the second field above is set to 1. In that case, the number of sets is equal to the number given in the third field above.

3rd data block

1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with the value.
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value.
21-25	5th	I	First layer of cross section point with this value.
26-30	6th	I	Last layer or cross section point with this value can only be bigger than 1 for beam or shell elements.

4th data block

1-10	1st	F	Initial value of this state variable for the above range of points.

Data blocks 5, 6, 7 and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block

1-10	1st	F	New total value of this state variable for the points given below at the start of the zeroth increment.

Format Fixed	Format Free	Data Type	Entry
6th data block			
			Enter a list of elements to which the above state variable is applied.
7th data block			
This data block is not necessary if the CENTROID parameter is used.			
			Enter a list of integration points to which the above state variable is applied.
8th data block			
This data block is only necessary if there are either beams or shells in the mesh.			
			Enter a list of layer points to which the above state variable is applied.

■ CHANGE STATE

Redefine State Variables

Description

This option provides various ways of changing the state variables throughout the model. State variables are initialized in the INITIAL STATE model definition set. The number of state variables per point is defined in the STATE VARS parameter. The default is one, with temperature always being the first state variable at a point. If more than one state variable per point has been assigned, this option can be used repeatedly to change the values of all state variables. The default value is no change if this option is not used. In this option, the values of the state variable at the end of the current increment are read in. When the temperature is being defined, the following points should be noted:

- For "history following analysis", the thermal strains are based on temperature change during this step.
- For elastic re-analysis (ELASTIC parameter) the thermal strains are always based on temperature change between the initial, stress free temperature field and the values read in here.
- The AUTO THERM option is available for automatic control of a nonlinear (elastic-plastic) temperature loaded stress problem, to be used in conjunction with this option.
- The THERMAL LOADS option can be used as an alternate to input the change of temperature.
- The AUTO THERM CREEP option is available for automatic control of a thermally loaded elastic-plastic-creep problem, to be used in conjunction with this option.

Four ways of changing any state variable are shown below:

- Read a range of elements, integration points and layers and a corresponding state variable value for the end of the current step.
- Read the state variable values for the end of the current step through user subroutine NEWSV.
- Read the state variable values for the end of the current step from a named step of the post file output from a previous heat transfer analysis with MARC. With this option the program assumes direct correspondence of the post file elements with the elements in the current analysis. Any spatial interpolation must be provided separately by you.
- Read a list of elements, integration points, and layers and a corresponding state variable value.

Note: Using this option, total state variable values are input.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-12	1st	A	Enter the words CHANGE STATE.
2nd data block			
1-5	1st	I	Enter the state variable identifier for the state variable being changed (1,2,3,etc.) 1 = temperature. If more than one state variable is being used, the STATE VARS parameter must be included.
6-10	2nd	I	Enter 1 to change the state variable via the 3rd and 4th data block below. In this case, the third field must also be defined. Enter 2 to change the state variable via user subroutine NEWSV. This subroutine will now be called in a loop on all the elements in the mesh. Enter 3 to read the new values of the state variable from a post file written by a previous heat transfer analysis. In this case, the fourth and fifth field must be defined. Enter 4 to change the state variable via data blocks 5, 6, 7, and 8 below.
11-15	3rd	I	Only nonzero if the second field is set to 1 or 4. In that case, this entry gives the number of data blocks set in data blocks 3 and 4 used to input the new value of the state variable (optional).
16-20	4th	I	Only nonzero if the second field is set to 3. Then. this entry defines the unit number from which the post file information from the previous heat transfer run will be read. Defaults to unit 24 for a formatted post file, and to unit 25 for a binary post file.
21-25	5th	I	Only nonzero if the second field is set to 3. In that case, this entry defines the step number on the heat transfer run post file to be read as the definition of the new value of the state variable at the end of the current step.

Format Fixed	Format Free	Data Type	Entry
26-30	6th	I	Only used if the AUTO THERM option is in use. Give the number of sets of input to be read to define the temperature history.
31-35	7th	I	Enter 1 if formatted post file is used. ***For Cray Only:*** Set to 2 for IEEE binary file Set to 3 for VAX binary file Set to 4 for IBM binary file
36-40	8th	I	Only nonzero if the second field is set to 2. Set to 1 to suppress printout of state variable values that are defined in NEWSV.
41-45	9th	I	Enter the post code number to be read into this state variable, default is 9 (temperature).

Data blocks 3 and 4 are only input if the second field above set to 1. In that case, the number of sets is equal to the number given in the third field above.

3rd data block

1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with this value.
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value.
21-25	5th	I	First layer or cross section point with this value.
26-30	6th	I	Last layer or cross section point with this value can only be bigger than 1 for beam or shell elements.

4th data block

1-10	1st	F	New value of this state variable for the above range of points at the end of the current step.

Format Fixed	Format Free	Data Type	Entry

Data blocks 5, 6, 7, and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block

1-10	1st	F	New total value of this state variable for the points given below at the start of the zeroth increment.

6th data block

Enter a list of elements to which the above state variable is applied.

7th data block

This data block is not necessary if the CENTROID parameter is used.

Enter a list of integration points to which the above state variable is applied.

8th data block

This data block is only necessary if there are either beams or shells in the mesh.

Enter a list of layer points to which the above state variable is applied.

■ THERMAL LOADS

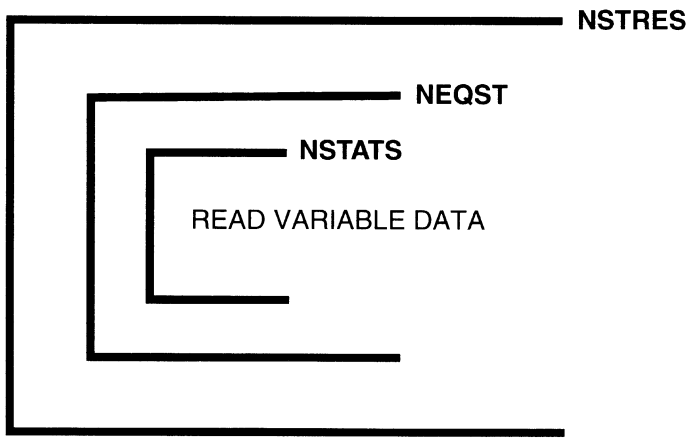
Input Temperature Data

Description

This option allows input of temperature and other state variables (see STATE VARS parameter).

You can specify either a uniform or nonuniform change in temperature (or other state variables). If a nonuniform change is desired, the change of every state variable at every layer of every integration point of every element must be specified. In this case, the program calls user subroutine CREDE for every element in the mesh.

CREDE already exists in the standard version of MARC with coding to read data in loops over integration points, layers and state variables, as illustrated below:



NSTRES = Maximum number of integration points per elements
NEQST = Maximum number of layers per element
NSTATS = Maximum number of state variables

The data to be read should be put in the 4th data block of this option. For particular elements with less than the maximum number of integration points or layers, dummy values should be input when the integration point or layer number exceed the appropriate range. See the description of CREDE in *Volume D: User Subroutines and Special Routines* for other necessary information.

If temperature (state variable) data is on a file, this file can be read from CREDE. However, if temperature data is on a post file from a MARC heat transfer analysis that uses the same mesh, the CHANGE STATE option provides a much simpler method for reading temperature data.

If the Fourier decomposition method is being used to analyze an arbitrarily loaded axisymmetric structure, the THERMAL LOADS option must be invoked separately for each Fourier series term that has temperatures (state variables) associated with it. If there is no variation of these variables in the circumferential direction, only the 0th term of the series should be specified.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-13	1st	A	Enter the words THERMAL LOADS.
2nd data block			
1-5	1st	I	Enter 1 if uniform incremental temperature (state variable) is applied to all elements. Enter 2 if nonuniform incremental temperature (state variable) will be read via user subroutine CREDE. Enter 3 if nonuniform total temperature (state variable) will be read via user subroutine CREDE.
6-10	2nd	I	If a Fourier analysis; the Fourier term with which this loading will be associated.
3rd data block			
1-80	1st	E	If the first field of data block 2 is 1, enter the uniform increments in temperature and any additional state variables; will be applied to all elements.

Format Fixed	Format Free	Data Type	Entry
4th data block			
1-80	1st	E	Include only if the first field of data block 2 is 2 or 3, and using the default subroutine CREDE. Temperature and state variable data to be read in by CREDE. All data blocks should contain 8 values, do not start a new data block for each element.

■ INITIAL TEMP

Define Initial Temperatures

Description

This option provides initial temperatures at nodal points for thermal stress problems. For heat transfer analyses, see INITIAL TEMP for heat transfer.

Note: For shell analyses, a uniform temperature will be used through the thickness direction.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words INITIAL TEMP.
2nd data block			
1-5	1st	I	Enter the number of sets of prescribed temperatures (optional). Enter -1 if user subroutine USINC will be used. In this case, data blocks 3 and 4 are not used.
6-10	2nd	I	Enter file number for input of prescribed temperatures data, defaults to input.
11-15	3rd	I	Flag to indicate that initial conditions are read from previously generated post file. Set to 1.
16-20	4th	I	Only nonzero if the third field is set to 1. Then this entry defines the unit number from which the post file information will be read.
21-25	5th	I	Enter step number to be read.
26-30	6th	I	Enter 1 if a formatted post file is used.

Format Fixed	Format Free	Data Type	Entry
Data blocks 3 and 4 are given in NSET pairs, only if the flag in the third field is not equal to 1.			
3rd data block			
1-10	1st	E	Initial temperature.
4th data block			
			Enter list of nodes for which the above initial temperature is applied.

■ POINT TEMP

Define Point Temperatures

Description

This option defines temperatures at nodal points for an uncoupled thermal stress problem at the end of the increment.

Note: For shell analyses, a uniform temperature will be used through the thickness direction.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words POINT TEMP.
2nd data block			
1-5	1st	I	Enter the number of sets of prescribed temperatures (optional).
6-10	2nd	I	Enter file number for input of prescribed temperatures data; defaults to input.
11-15	3rd	I	Flag to indicate that temperatures are read from previously generated post file. Set to 1.
16-20	4th	I	Only nonzero if the third field is set to 1. Then, this entry defines the unit number from which the post file information will be read.
21-25	5th	I	Enter step number to be read.
26-30	6th	I	Enter 1 if a formatted post file is used. ***For Cray Only:*** Set to 2 for IEEE binary file Set to 3 for VAX binary file Set to 4 for IBM binary file

Format Fixed	Format Free	Data Type	Entry
Data blocks 3 and 4 are given in NSET pairs, only if the flag in the third field is not equal to 1.			
3rd data block			
1-10	1st	E	Temperatures at the end of the increment.
4th data block			
			Enter list of nodes for which the above initial temperature is applied.

■ FORCDT

Input Displacement or Load Histories

Description

This option specifies the nodes for which user subroutine FORCDT is called (see *Volume D: User Subroutines and Special Routines*). Subroutine FORCDT can be used to prescribe the kinematic displacements and point loads. To prescribe displacements/temperatures, they must also be defined in FIXED DISP, FIXED TEMPERATURE, etc.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-6		A	Enter the word FORCDT.
11-15		I	Enter the number of lists to be given below. Default is 1.
2nd data block			
The 2nd data block is entered once for each data set.			
1-80	1st	I	List of nodes for which user subroutine FORCDT will be used.

■ FOUNDATION

Input Elastic Foundation Data

Description

This option allows the specification of elements and associated foundation stiffness to be used with the elastic foundation option (see *Volume A: Theory and User Information*). Nonlinear foundations are available via user subroutine USPRNG (see *Volume D: User Subroutines and Special Routines*).

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word FOUNDATION.
2nd data block			
1-5	1st	I	Number of sets of data blocks to be used to input the lists of element and foundation stiffnesses (optional).
6-10	2nd	I	Enter unit number for input of foundation data, defaults to input.
3rd data block			
The 3rd and 4th data blocks are entered as pairs, once for each list.			
1-5	1st	I	Face identification – same as in DIST LOADS for distributed pressure for element type used. Parameter identifying the type of elastic foundation is the same parameter as used in the DIST LOADS option. See *Volume B: Element Library* for a description of the possible distributed load types for each element type in MARC.
6-15	2nd	E	Spring stiffness per unit surface area (or per unit length for beam elements).
4th data block			
1-80	1st		Enter a list of elements to which the above foundation is applied.

■ FOURIER

Describe Fourier Coefficients

Description

This block is used to describe all the Fourier coefficients for each series used. The FOURIER parameter must be included. If the Fourier coefficients of a series are known, they can be input directly. The function F(θ) to be expanded into a Fourier series can be described by an arbitrary number of pairs of data [θ, F(θ)]. The starting location (θ = 0) as well as the ending location (θ = 360) must be included. They can have different values F(θ). The function F(θ) can also be described via a user subroutine UFOUR for an arbitrary number of stations around the circumference.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-9	1st	A	Enter the word FOURIER.
2nd data block			
1-5	1st	I	Number of Fourier series for which coefficients are input via data blocks.
6-10	2nd	I	Number of Fourier series for which the coefficients are calculated from user input [θ, F(θ)]. The description of F(θ) is given via data blocks. The number of coefficients to be calculated depends on the number of Fourier harmonics specified on the parameter data block.
11-15	3rd	I	Number of Fourier series for which the function F(θ) is determined by a user subroutine. The details on the subroutine UFOUR are found in *Volume D: User Subroutines and Special Routines*.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
Only included if first parameter of 2nd data block is nonzero. This series is repeated for each Fourier expansion for which coefficients are input via data blocks.			
1-5	1st	I	Number of coefficients to be read for this series.
6-15	2nd	E	Fourier coefficients a_0, a_1, b_1, . . . a_n, b_n,. Continuation data is in format 8E10.0.
The following group is included only if second parameter of 2nd data block is nonzero. This group is repeated for each F(θ) function.			
4th data block			
1-5	1st	I	Number of [θ, F(θ)] pairs to be read in for this function.
5th data block			
			Four pairs of [θ, F(θ)] per data block. Continuation data is in format 8E10.0
1-10	1st	F	Value in degrees of first station.
11-20	2nd	F	F(θ) – value for first station.
21-30	3rd	F	Value in degrees of second station.
31-40	4th	F	F(θ) – value for second station (etc.)

■ J-INTEGRAL

Define Path for J-Integral Estimation

Description

This option gives an estimation of the J-Integral for a crack configuration, based on differential stiffness technique. The technique is based upon moving nodes around the crack tip by a small distance and estimating the energy change. This model definition set is used to input the list of nodes to be moved, and the direction and size of those motions. Usually, the motion is 10 times smaller than the crack element size. The program prints out the change in strain energy, which must then be divided by the change in crack surface area to obtain the J-Integral.

Note: For each evaluation the first list must contain the crack tip. The CENTROID parameter should not be used in conjunction with this option. With second order elements, we suggest the use of 1/4 point elements at the crack tip.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word J-INTEGRAL.
2nd data block			
1-5	1st	I	Number of lists in this path.
6-10	2nd	I	Logical unit number for reading this data. Defaults to input.

Format Fixed	Format Free	Data Type	Entry
Data blocks 3 and 4 are entered as paris, one for each list.			
3rd data block			
1-10	1st	F	Motion of these nodes in the 1st coordinate direction.
11-20	2nd	F	Motion in the second coordinate direction, etc. in (8E10) format. (Provide enough data to define the motion in each coordinate direction). **Note:** The motion of the midsize nodes should be proportional to their position along the sides of the elements.
4th data block			
			Enter a list of nodes to be moved.

■ LORENZI

Define Path for Modified J-Integral

Description

As an alternative to the differential stiffness method for evaluating the stress intensity factor, the De Lorenzi method can also be invoked by you. The De Lorenzi method has the advantage that it can also be used for problems with thermal behavior or for dynamic analysis. This procedure is only available for continuum elements. The different lists to be entered by you are shown below. The program prints the change in strain energy, which must then be divided by the change in crack surface to obtain the modified J-Integral.

Note: The CENTROID option should not be used in conjunction with this option. With second order elements, we suggest the use of 1/4 point elements at the crack tip.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word LORENZI.
2nd data block			
1-5	1st	I	Enter the number of paths to be taken.
6-10	2nd	I	Enter the unit number to read the data.

The 3rd, 4th, 5th, 6th, and 7th data blocks are entered once for each path.

3rd data block			
1-5	1st	I	Enter the number of lists of nodes to be given.

Format Fixed	Format Free	Data Type	Entry
Data blocks 4 and 5 are entered as pairs, one for each data set.			
4th data block			
1-10	1st	F	Enter the incremental displacement in the first degree of freedom.
11-20	2nd	F	Enter the incremental displacement in the second degree of freedom
Etc.		Etc.	
71-80		F	Enter the incremental displacement in the eighth degree of freedom. **Note:** If more than eight degrees of freedom, enter an additional data line.
5th data block			
			Enter a list of nodes for which the above displacement are applied.
6th data block			
			Enter a list of elements in the deformed region.
7th data block			
			Enter a list of elements in the rigid region.

Contact

This section describes surface geometry definition, motion definition, and friction description in automatic two- and three-dimensional contact applications. The basic philosophy behind these applications is the existence of one or more bodies that might or might not come into contact with one another, or even contact with themselves during an analysis. As far as the contact is concerned, it is the surface associated with the body that plays a role.

There is a limit of 99 bodies in an analysis. Some can be deformable, others can be rigid. However, deformable surfaces must always be declared in the input file before any rigid surface.

Deformable and Rigid Surfaces

A deformable surface is simply defined by the set of elements that constitute the body to which it is associated. When a node of another body or the same body (in self contact) comes into contact with a deformable surface, information regarding the contacted surface is obtained. This is based upon the coordinates of the nodes on the face of the element or the coordinates and an averaged normal if the SPLINE option is used. This can improve the accuracy of the solution.

A rigid surface does not deform. There are two modes to describe the geometric profile of a rigid surface. In the first, labelled the PieceWise Linear approach (PWL), the profile is defined by sets of geometrical data which can be comprised of straight lines, circles and splines, ruled surfaces, surfaces of revolution and patches, etc. These sets have to be given in a proper sequence around the rigid body they define, even if it is not necessary that the full enclosure be defined.

In the second method, labelled Analytical, the geometric profile is defined by prescribing 2D NURB curves, 3D NURB surfaces, or exact quadratic descriptions. Using this method, the surface is divided into line segments or patches; this is used for visualization (K6 style post file) or by the searching algorithm. The contact condition is based on the true surface geometry. This method is more accurate for curved surfaces, and might reduce the number of iterations, especially if friction is present.

In coupled thermal-stress contact, it is possible to have a surface defined strictly by thermal elements with a rigid body motion applied to it.

Motion of Surfaces

Deformable surfaces can move either because of contact with other surfaces, or because of directly applied displacement boundary conditions or loads.

To each surface, we associate a point (center of rotation) that can be anywhere in space. A translative velocity and a rotational velocity around that point define the instantaneous motion of the surface. These velocities are integrated forward in time to define the motion of the surfaces. It is also possible to directly prescribe the location of the rigid body. As an alternative, you can prescribe a force to the rigid body. The force is applied with the POINT LOAD option.

The CONTACT model definition option can be used for the input of constant rigid body motions which do not change with time during the analysis. However, changes in rigid body motion (time dependent motion) can be simulated either by the load history option MOTION CHANGE or by the user subroutine MOTION activated through the model definition option UMOTION.

Cautions

It is recommended that whenever several deformable bodies are present, the OPTIMIZE option be exercised. In static analysis, it is also necessary to artificially connect (for instance, by very low stiffness springs) deformable bodies that during an analysis might be completely separated from other deformable bodies and have no kinematic boundary conditions applied to them. This is to avoid rigid body motion.

When the debug printout parameter PRINT is used in a contact analysis, `IDEV = 5` or `8`, it produces information on when any node on the boundary comes into contact or separates from any surface. It also produces information on whether a contact node is fixed to a surface or is free to slide along it. In addition to the information printed with `IDEV = 5`, when `IDEV = 8` is entered, the incremental displacement and the reaction forces for those nodes in contact with rigid surfaces are printed in a local coordinate system.

The CONTACT option creates transformations that are internally defined, and, as such, no standard TRANSFORMATION, SHELL TRANSFORMATION, or UTRANSFORM are permitted on boundary nodes which might come in contact with a rigid surface.

The CONTACT option creates ties between nodes which come into contact; hence, you should avoid having the nodes on the exterior boundaries be used in TYING or SERVO LINK.

There are three implied loops in this block of data: the outermost loop is over the number of surfaces; the next loop is over the number of sets of geometrical data for each surface; and the innermost loop is over the number of points comprised in each set. In case of deformable surfaces, the two inner loops reduce to the list of elements.

Upper Bound Quantities

All geometrical data is internally reduced to two types of entities: for 2D contact, it is a linear segment, and for 3D contact, it is a 4-point patch. You must enter an upper bound to the maximum number of entities to be created for any surface. When entering this value, it should be taken into account the fact that a spline splits into linear segments within the contact tolerance error. This is required for Analytical description as well, even though the number of patches does not influence the accuracy of the solution.

One of the first things MARC does when it encounters the CONTACT option is to determine all nodal points that lie on the periphery of each deformable surface. You must enter on the second data block an upper bound to the number of such nodes and this is used for storage definition. If you think you will do rezoning later in the analysis, this should also be an upper bound to the boundary nodes of each of the new meshes.

When an adaptive meshing analysis is performed, additional caution is required. When an element is refined, additional nodes and additional surface patches are generated. It is better to over allocate these values.

Control Variables and Option Flags

The variable RVCNST allows the system to self-adaptively search for sticking zones. RVCNST should be a relative sliding velocity very small compared to the typical sliding velocities in the model, but not so small that it would be overcome by changes between iterations. It is suggested you use values between 10^{-1} and 10^{-2} times a typical relative surface velocity. MARC default is 1.0.

The variable ERROR determines the tolerance for contact. A too small tolerance might provoke too many increment splits. A too coarse tolerance produces unrealistic behavior. If left blank, the code calculates ERROR as the smallest nonzero element dimension divided by 20 or the shell thickness divided by 4. If there are splines in surface definitions, a value should be entered. (See SPLINE definition.)

The variable FNTOL is used for the input of a separation force in a contact analysis. If the contact force of a node, calculated by MARC, is greater than the prescribed separation force (FNTOL), the node is to be separated from the contact surface.

You can control the type of friction in a contact analysis. Either a friction (shear friction or Coulomb friction) or a frictionless condition can be assumed in the analysis. The friction behavior is either continuous or a true-stick slip model can be used.

The computation of Coulomb friction in a contact problem can be based on either nodal stresses or nodal forces.

During each load increment, separations can occur. You can control the maximum number of nodal separations allowed in each increment to reduce computational costs. During each load increment, if the contact of a node (or a group of nodes) is detected, the increment is split (subdivided) in order to accommodate the contact condition. Depending on the occurrence of further contact, the load increment recursively split until the total incremental load is reached. If you suppresses splitting, the load increment is not split (subdivided), and all the penetrations occurring in the current increment are adjusted for equilibrium at the beginning of the next increment.

Optional Heat Transfer Data

In a coupled thermal-stress-contact analysis, a film coefficient is needed for calculating heat transfer from any other surface that contacts the current surface (deformable-to-rigid, or deformable-to-deformable). If the surface is modeled as a rigid surface, the rigid surface temperature is also needed for the analysis. In addition, both a film coefficient and an ambient temperature must be given for the simulation of heat transfer between the surface and surrounding environment.

Time Step Control

The automatic contact procedure is controlled by the time step. This is used to determine the motion of rigid surfaces and to control the splitting of increments if penetration occurs. Even in a quasi-static analysis, a time step must be defined by you. Several procedures can be used to enter this data.

- The AUTO LOAD and TIME STEP history definition options can be used to define several time steps, each of the same magnitude.
- The DYNAMIC CHANGE or TRANSIENT NON AUTO history definition options can be used to define a time period which will be divided into equal time steps.
- With the AUTO STEP, AUTO TIME, or AUTO INCREMENT history definition option, you define a total time period which will be divided into variable size time steps.

Dynamic Contact - Impact

The automatic contact procedure can also be used in dynamic analyses to model impact problems. This can be used with the implicit Newmark-beta operator or the explicit central difference operator. The DYNAMIC parameter is used to control the choice. When the Newmark-beta operator is used, either the DYNAMIC CHANGE, AUTO STEP, or AUTO TIME

option can be used to control the time step. When the central difference procedure is used, the DYNAMIC CHANGE option should be used, and the time step must be less than the stability limit. The stability limit is automatically calculated by the program.

Two-dimensional Rigid Surfaces

In a two-dimensional problem, the rigid surfaces can be represented by any of, or, a combination of the following geometric entities: (1) straight line segments (`ITYPE = 1`), (2) circular arcs (`ITYPE = 2`), and (3) spline (`ITYPE = 3`). The variable `ITYPE` defines the type of the geometric entities to be used for a rigid surface.

Note that the normal vector of the geometric entities (line segments, circular arc, and the spline) always points into the rigid-body. The normal vector direction is determined from the direction of the geometric entity, following a right-handed rule. Care must be taken in entering the coordinates (x, y) data, in a correct direction, for rigid-surfaces.

Line Segments

When option `ITYPE = 1` is chosen, the number `NPOINT` and the coordinates (x, y) of (`NPOINT`) points must be entered for the definition of the rigid surface. The program automatically creates a rigid surface consisting of (`NPOINT -1`) linear segments for the contact problem. A two-dimensional rigid surface consisted of line segments is shown in Figure 3-2.

This entity supports analytic description/procedure.

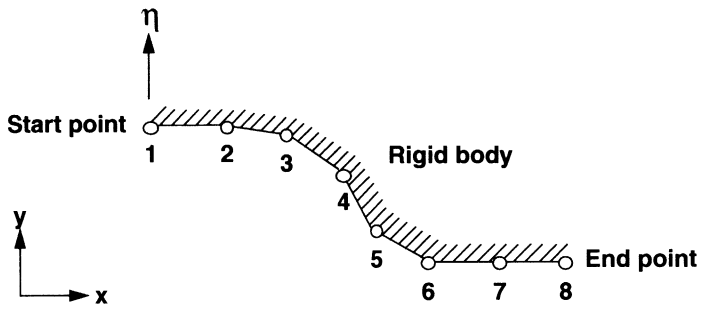


Figure 3-2 Two-Dimensional Rigid Surface (Line Segment, ITYPE = 1)

Circular Arc

When `ITYPE = 2` is chosen, one circular segment is created by the program. There are five different methods available to define a circular arc in two dimensions. Each method requires four data blocks with the following type of data:

Starting Point of Arc	(SP)
Ending Point of Arc	(EP)
Center of Circle	(C)
Radius of Circle	(R)
Tangent Angles	(TA)
Swept Angle	(SA)
Number of Subdivisions	(NS)

Clearly, not all of this information is required for each method. Table 3-6 describes which data is required. The default number of subdivisions is 10. If the analytical approach is used, the number of subdivisions does not influence the accuracy, but is only used for visualization purposes.

Table 3-6 Data Required for Circular Arc Input

Data Block	Method				
	0	1	2	3	4
1	SP	SP	SP	SP	SP
2	EP	EP	EP	EP	blank
3	C	C	C	TA1, TA2	C
4	R, NS	R, NS	R, NS	R, NS	SA, NS

Note: For methods 1 and 3, a positive radius means the center of the circle is on the surface side. A negative radius means the center of the circle is on the outside.

For method 2, the first coordinate of the center will be taken into account, determining whether the center is above (>0) or below (<0) the segment defined by the end points.

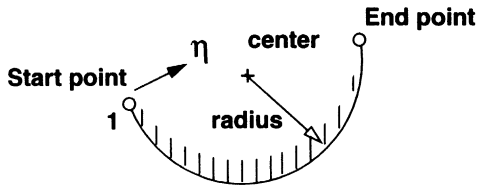
For planar problems, SP, EP and C are X, Y data.

For axisymmetric problems, SP, EP and C are Z, R data.

For methods 0, 1 and 2, if R is zero, it is calculated as distance from the center to the starting point.

This entity supports analytical description/procedure.

A two-dimensional rigid surface represented by a circular arc is shown in Figure 3-3 and Figure 3-4.



Note: For additional circular arc definitions, see Figure 3-4

Figure 3-3 Two-Dimensional Rigid Surface (Circular Arc, ITYPE = 2, METHOD = 0)

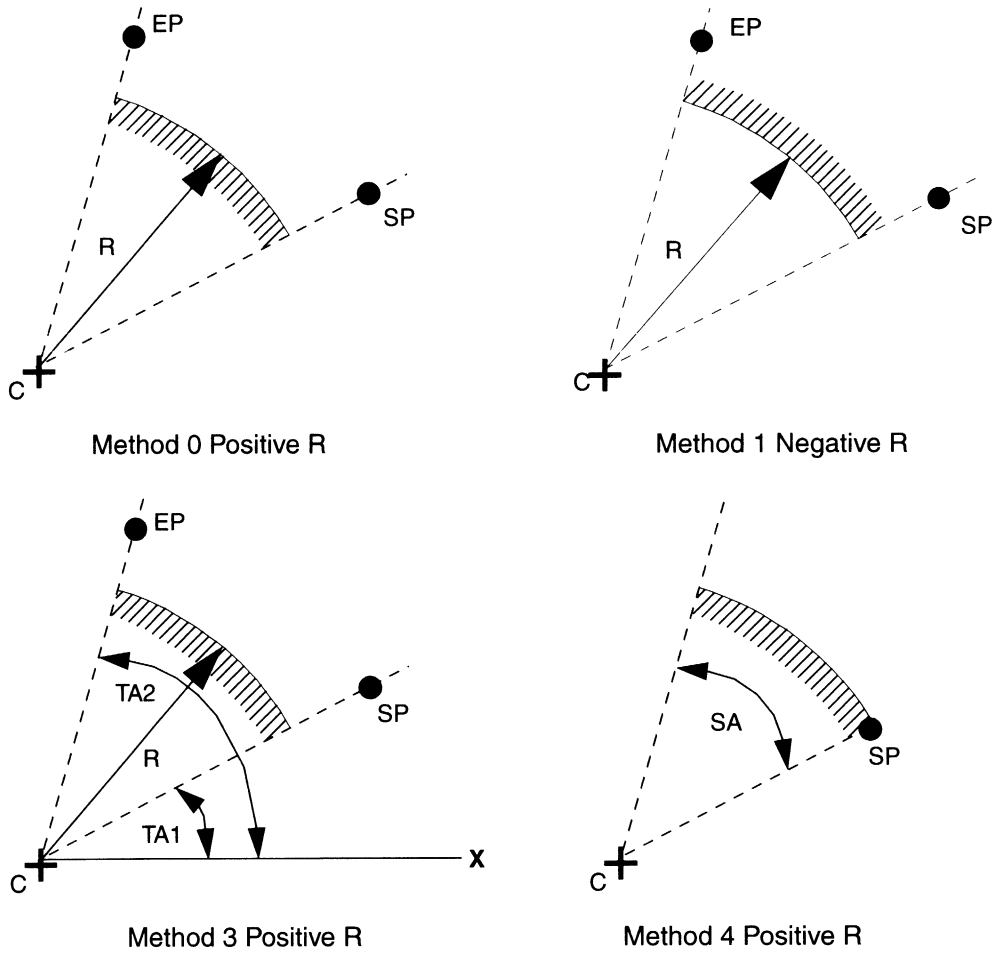


Figure 3-4 Two-Dimensional Rigid Surface (Circular Arc)

Spline

When ITYPE = 3 is chosen, the program creates a spline by passing from the second point through to the second to last point entered. The first and the last points entered are used to define the tangents at the beginning and end of the spline. If a nonanalytical approach is used, then the spline is internally split into linear segments in such a way that the maximum difference between any of them and the spline is less than the contact tolerance ERROR. This operation is done before the automatic tolerance calculation; therefore, a value for ERROR must be entered whenever a spline is used. Figure 3-5 shows a two-dimensional rigid surface defined by a spline.

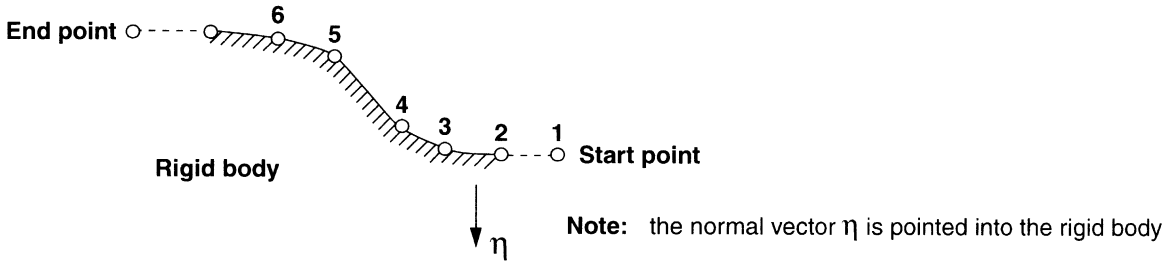


Figure 3-5 Two-Dimensional Rigid Surface (Spline, ITYPE = 3)

This entity supports analytical description/procedure if only one spline is used in a particular rigid body.

Three-Dimensional Rigid Surfaces

In a three-dimensional problem, the rigid surfaces are represented by any of, or, a combination of the following three-dimensional surface entities:

Surface Entity Type	Type Identification (ITYPE)
Ruled surface	4
Surface of revolution	5
Bezier surface	6
4-node patch	7
Poly-surface	8
NURB	9
Cylinder	10
Sphere	11

The variable ITYPE defines the type of surface entity to be used for a rigid surface. Since most of the three-dimensional surfaces can be easily and adequately represented by a finite element mesh of 4-node plate (patch) elements, the option ITYPE = 7 is a very convenient way of representing three-dimensional rigid surfaces. Both the connectivities and the coordinates of the 4-node patches can be generated using Mentat II, or entered through user subroutine DIGEOM.

The three-dimensional surface entities mentioned above, except 4-node patches, can in turn be generated from three-dimensional geometric entities. Available three-dimensional geometric entities are:

Geometric Entity Type	Type Identification (JTYPE)
Straight line segment	1
3-D circular arc	2
Spline	3
Bezier Curve	4
Poly line	5

The variable JTYPE defines the type of geometric entities to be used for the generation of three-dimensional rigid surfaces.

For the (PWL) approach, note that all geometrical data in 3D space is reduced to 4-node patches. The four nodes will probably not be on the same plane. The error in the approximation is determined by the number of subdivisions of the defined surfaces. Note that the normal to a patch is defined by the right-hand rule, based on the sequence in which the four points are entered.

Ruled Surface (ITYPE = 4)

When ITYPE = 4 is chosen, a ruled surface is created by the program based on the input of two surface generators, defined by straight line segment (JTYPE = 1), 3D circular arc (JTYPE = 2), spline (JTYPE = 3) or Bezier curve (JTYPE = 4). If the surface generator is not a 3D circular arc, the number NPOINT1 (NPOINT2) and the coordinates (x, y, z) of these NPOINT1 (NPOINT2) points must be entered for the definition of the surface generators. In case the surface generator is a 3D circular arc, a method (METH) must be selected for the definition of the circular arc. A 3D circular arc is defined by four points. In addition, the number of subdivisions, NDIV1, along the first (surface generator) and the NDIV2 along the second (from the first surface generator to second surface generator) direction must also be entered. For a (PWL) approach, the program creates (NDIV1) x (NDIV2) 4-node patches automatically to represent the prescribed ruled

surface. For analytical approach, (NDIV1 + 1) x (NDIV2 + 1) points are created and a NURB surface is general which passes exactly through these points. The accuracy in general is controlled by the number of points. Figure 3-6 shows a typical ruled surface.

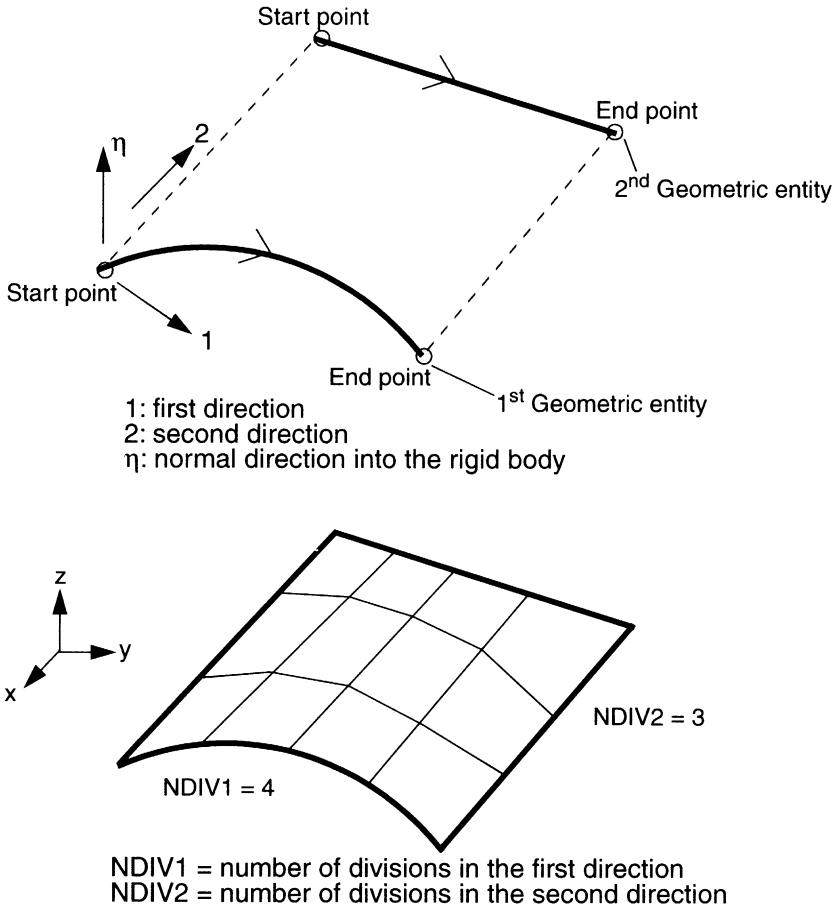


Figure 3-6 Three-Dimensional Rigid Surface (Ruled Surface, ITYPE = 4)

Surface of Revolution (ITYPE = 5)

When ITYPE = 5 is chosen, a surface of revolution is created by the program based on the input of one surface generator, defined by straight line segment (JTYPE = 1), 3D circular arc (JTYPE = 2), spline (JTYPE = 3) or Bezier curve (JTYPE = 4). If the surface generator is not a 3D circular arc, the number NPOINT and the coordinates (x, y, z) of these NPOINT points must be

entered for the definition of the surface generator. In case the surface generator is a 3D circular arc, a method (METH) must be selected for the definition of the circular arc. A 3D circular arc is defined by four points. In addition, the number of subdivisions NDIV1 along the surface generator and NDIV2 along the second (circumferential) direction must also be entered.

The program then creates (NDIV1 x NDIV2) four-node patches automatically, to represent the prescribed surface of revolution. The axis of revolution is defined by the coordinates (x, y, z) of two points in space, and an angle of rotation from the initial position is also needed for the definition of the surface of revolution. A positive rotation is about the axis formed from point 1 to point 2. Figure 3-7 shows a typical surface of revolution.

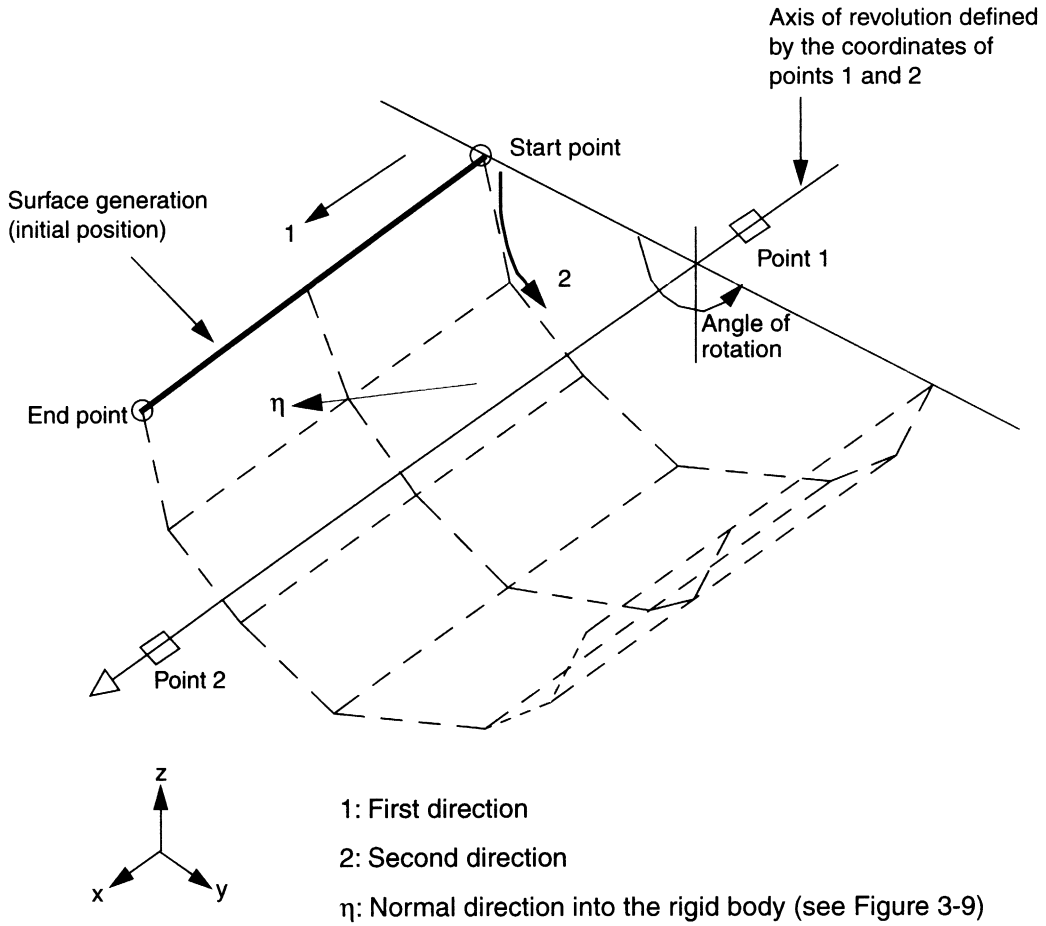
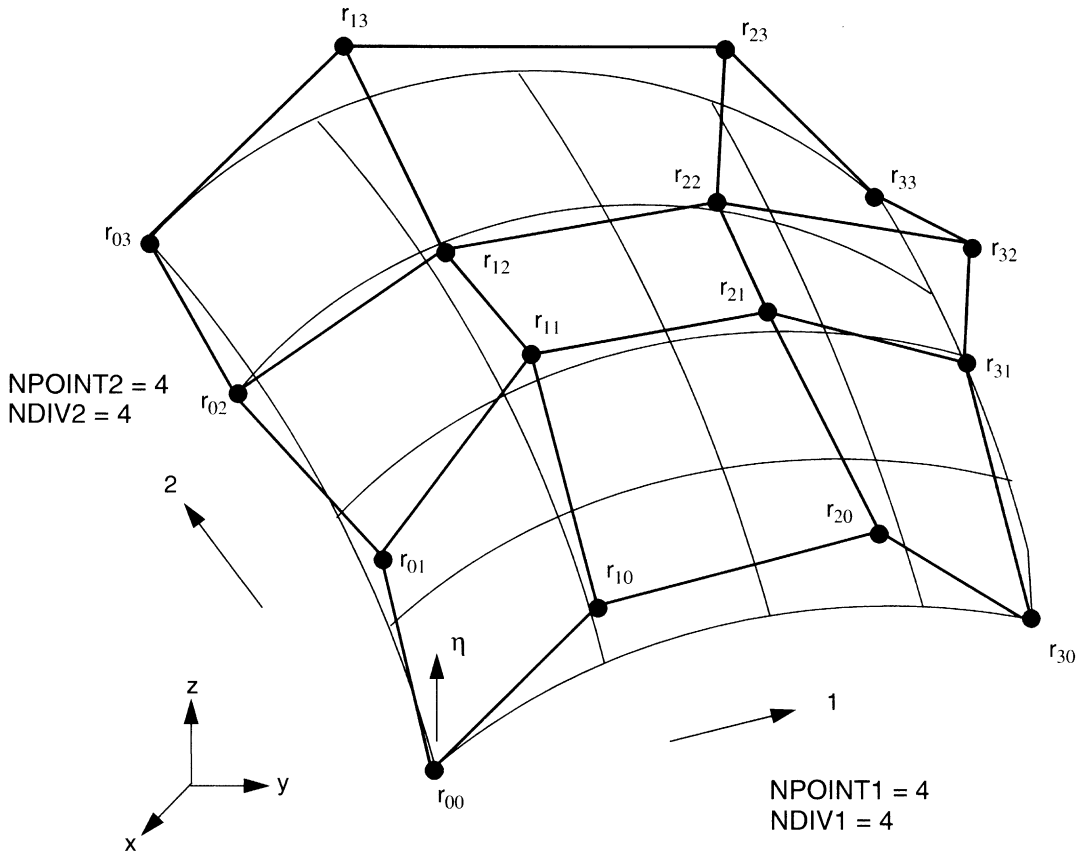


Figure 3-7 Three-Dimensional Rigid Surface (Surface of Revolution, ITYPE = 5)

Bezier Surface (ITYPE = 6)

When `ITYPE = 6` is chosen, a Bezier surface is defined by the coordinates (x, y, z) of `NPOINT1 x NPOINT2` control points. `NPOINT1` points are entered along the first direction and then repeated `NPOINT2` times to fill through the second direction of the surface. `NPOINT1` and `NPOINT2` have to be at least equal to 4. Number of subdivisions (`NDIV1`, `NDIV2`) entered has to be equal or greater than `NPOINT1` and `NPOINT2` for Bezier surface. (`NPOINT1-1`) x (`NPOINT2-1`) 4-node patches are created by the program for the definition of a Bezier surface. Figure 3-8 shows a typical Bezier surface. It can be treated as an analytical surface, an exact conversion to NURBS is performed.



1: first direction
2: second direction
η: normal direction into the rigid body (see Figure 3-9)

Figure 3-8 Three-Dimensional Rigid Surface (Bezier Surface, ITYPE = 6)

Four-node patches (ITYPE = 7)

When `ITYPE = 7` is chosen, you enter directly all the 4-node patches that comprise this surface. They are entered following the same format MARC would use to read the connectivities and coordinates of a mesh of 3D 4-node elements (element type 18 or 75). In this way, a finite element preprocessor can be used to create surfaces. Alternatively, this data can be entered via the user subroutine DIGEOM, further permitting you to read by yourself from any data you have access to. Figure 3-9 shows a typical 4-node patch surface. It cannot be used as an analytical surface.

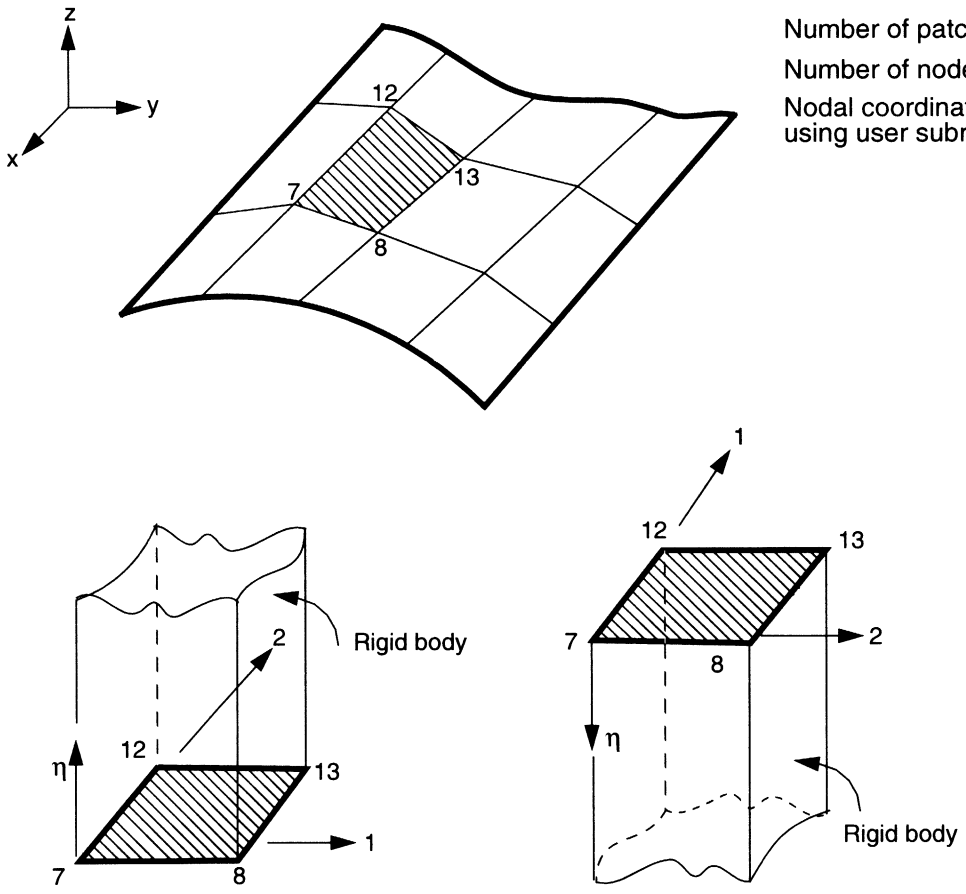


Figure 3-9 Three-Dimensional Rigid Surface (4-Node Patch, ITYPE = 7)

Poly-surface (ITYPE = 8)

When `ITYPE = 8` is chosen, a poly-surface is defined by the coordinates (x, y, z) of `NPOINT1 x NPOINT2` control points. `NPOINT1` points are entered along the first direction and then repeated `NPOINT2` times to fill through the second direction of the surface. `NPOINT1` and `NPOINT2` have to be at least equal to 4 for a poly-surface and there is no need to divide it. A typical poly-surface is shown in Figure 3-10.

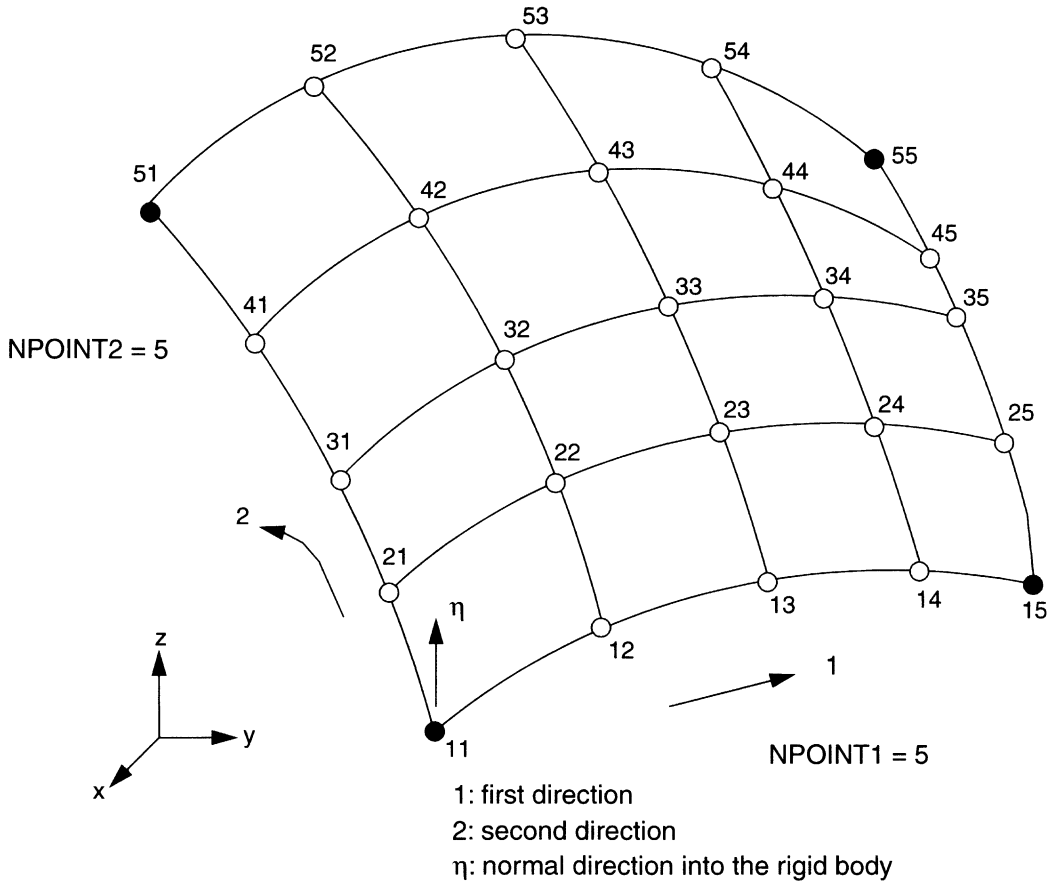


Figure 3-10 Three-Dimensional Rigid Surface (Poly Surface, ITYPE = 8)

In a three-dimensional contact problem, as in a two-dimensional situation, the surface generators can be represented in a variety of ways. It can be treated as an analytical surface. Approximate conversion to NURBS.

Nonuniform Rational Bspline Surface, NURBS (ITYPE = 9)

When `ITYPE = 9` is chosen, a NURBS is defined by the coordinates (x,y,z) of `NPOINT1 x NPOINT2` control points, `NPOINT1 x NPOINT2` homogeneous coordinates and (`NPOINT1+NORDER1`) + (`NPOINT2+NORDER2`) normalized knot vectors. If only the control points are entered, the interpolation scheme is used such that the surface passes through all of control points. The homogeneous coordinates and knot vectors are calculated by MARC program. `NPOINTS` and `NPOINT2` have to be at least equal to 3 for the interpolation scheme. A typical NURBS is shown in Figure 3-11.

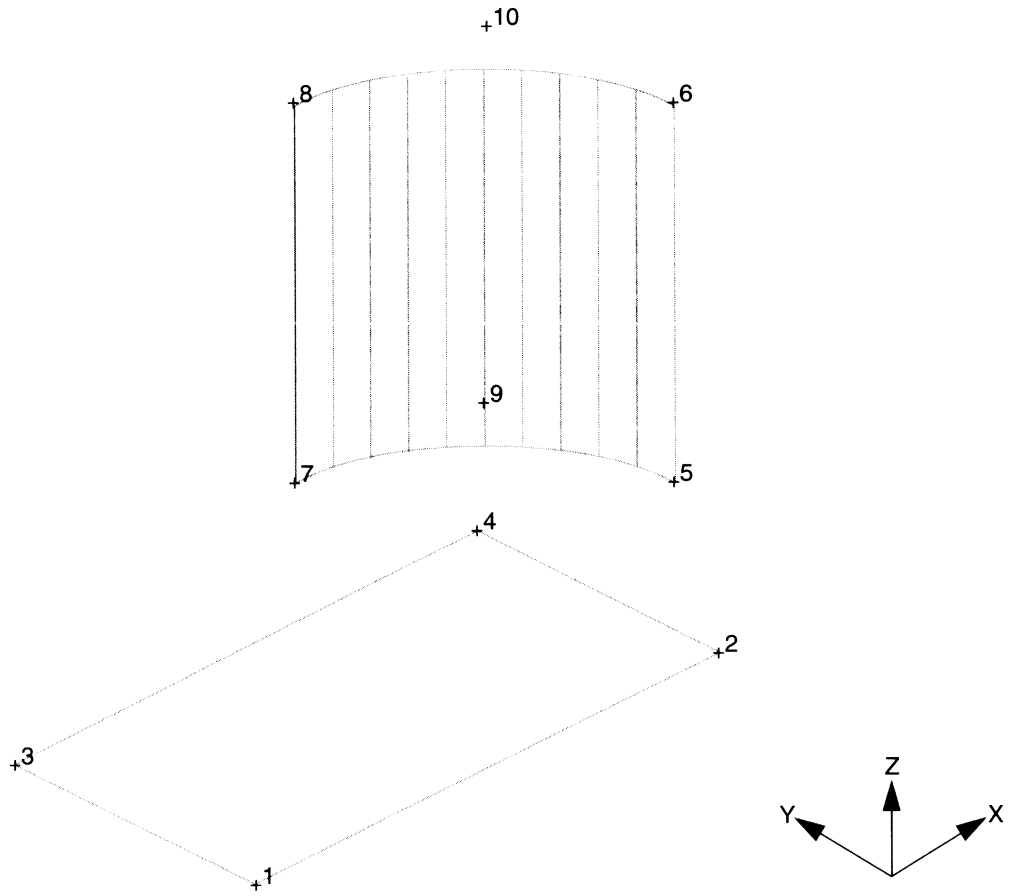


Figure 3-11 Nonuniform Rational Bspline Surface, NURBS (ITYPE = 9)

Cylinder (Cone) Surface (ITYPE = 10)

When `ITYPE = 10` is chosen, a cylinder is defined by the coordinates (x,y,z) of the center, C1, with radius, R1, in top face and the coordinate (x,y,z) of center, C2, with radius, R2, in bottom face. The normal vector of cylinder is inwards. If a negative value of R1 is entered, the normal vector is outwards. A typical cylinder is shown in Figure 3-12.

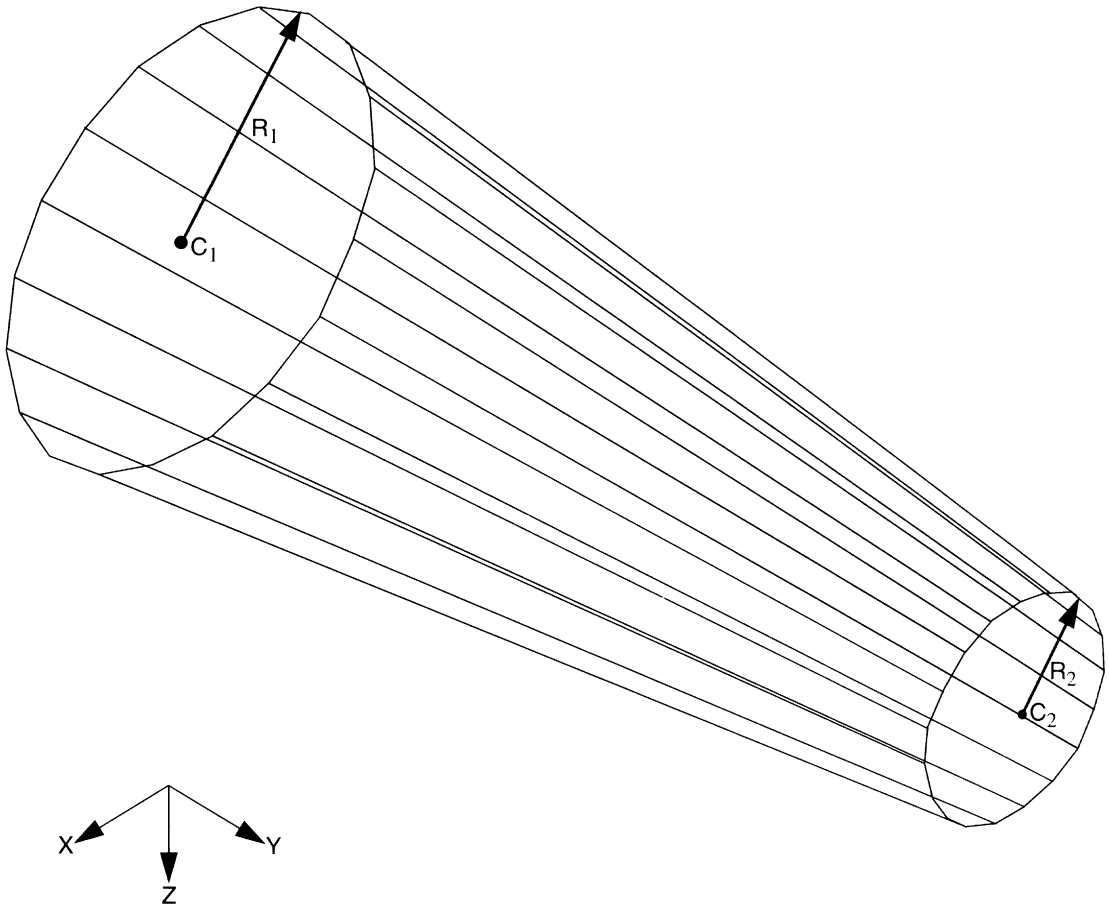


Figure 3-12 Cylinder (Cone) Surface (ITYPE = 10)

Sphere Surface (ITYPE = 11)

When `ITYPE = 11` is chosen, a sphere is defined by the coordinates (x,y,z) of the center, C1, with radius, R1. The normal vector of sphere is inwards. If a negative value of R1 is entered, the normal vector is outwards. A typical sphere is shown in Figure 3-13.

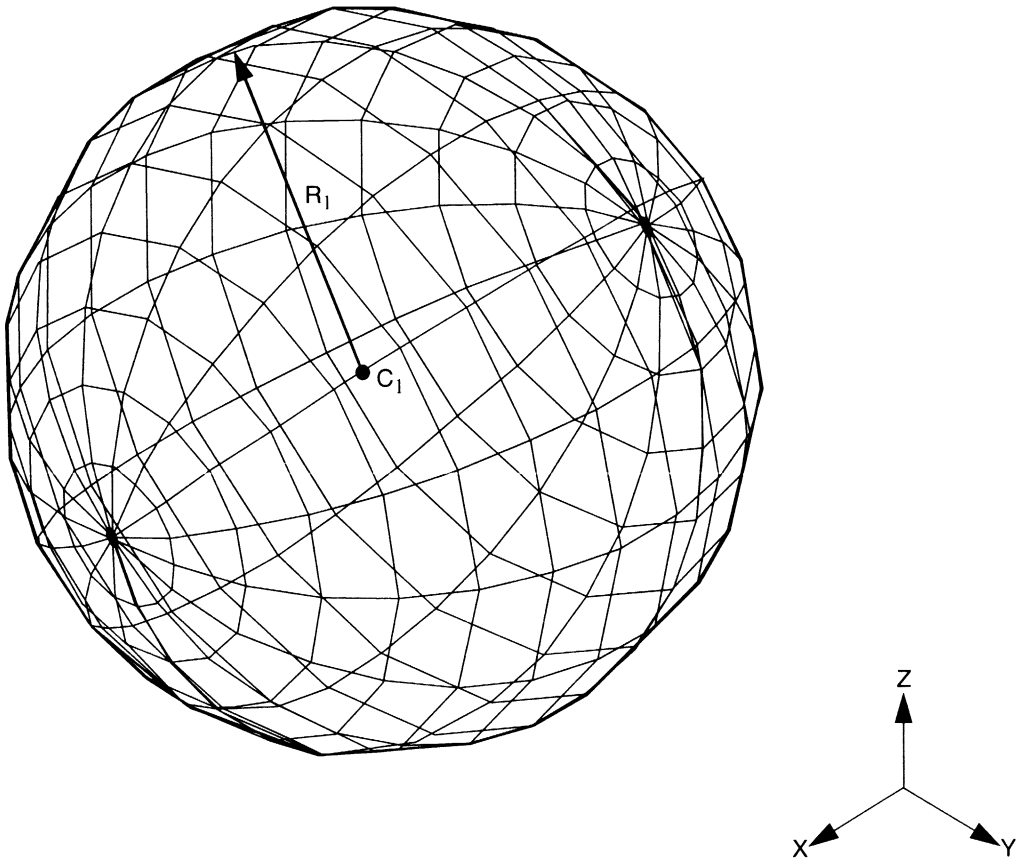


Figure 3-13 Sphere Surface (ITYPE = 11)

3D circular arc

When JTYPE = 2 is chosen, a circular arc is created by the program. There are three different methods (Table 3-7) available to define a circular arc in three dimensions. Each method requires four data blocks, with the following type of data:

Starting point of arc	(SP)
Ending point of arc	(EP)
Enter of circle	(C)
Radius of circle	(R)
Swept angle	(SA)
Swept angle flag	(SAF)
Middle point	(MP)
Arbitrary point (lying in plane of circle)	(AP)

Table 3-7 Defining Circular Arcs

Data Block	Method		
	0	1	2
1	SP	SP	SP
2	EP	MP	AP
3	C	EP	C
4	R	SAF	SA

Note: For Method 1, a positive radius means the center of the circle is on the surface side. A negative radius means the center of the circle is on the outside.

For Method 2, a SAF that is positive means an angle less than 180, a negative value an angle greater than 180.

For Method 3, the starting point, arbitrary point and center define the plane in which the circular arc lies.

SP, EP, C, MP and AP are X, Y, Z data.

For an arc with 180 degrees, either Method 1 or Method 2 is recommended.

A three-dimensional rigid surface represented by a circular arc is shown in Figure 3-14.

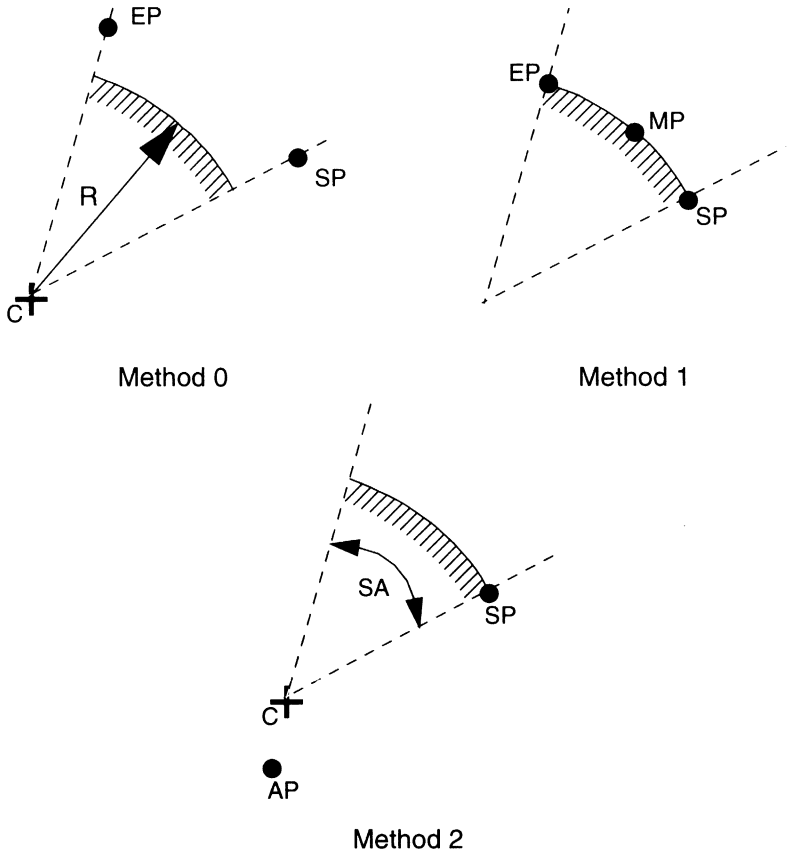


Figure 3-14 Three-Dimensional Rigid Surface (Circular Arc)

Spline

When `JTYPE = 3` is chosen, the spline passes by all `NPOINT` declared, and has zero curvature at the ends (enter at least 4 points).

Bezier curve

When `JTYPE = 4` is chosen, a Bezier curve is defined by `NPOINT` control points (enter at least 4 points).

Poly-line

When `JTYPE = 5` is chosen, a poly-line defined by `NPOINT` control points.

Selective Contact Surfaces

In both the two- and three-dimensional contact problems, contact is always detected between nodes on the surface of a deformable body and the geometrical profile of another surface. There are two modes of the order in which a node checks contact with other bodies. The default version is the double-sided contact procedure. In the single-sided contact procedure, the nodes on a lower numbered body can come into contact with equally or higher numbered surfaces. For instance, the boundary nodes of body number 1 are checked against the surface profiles of bodies 1, 2, 3, ... The boundary nodes of body number 2, however, are only checked against surface profiles of bodies 2, 3, ... It is possible, therefore, that due to surface discretization, a node of body 2 will slightly penetrate the surface of body 1.

The double-sided contact option checks possible contact between any two surfaces (surface i is checked for contact with surface j, and surface j is also checked for contact with surface i, where i, j = 1, 2, 3, ..., total number of surfaces in the problem).

In addition, an option (CONTACT TABLE model definition and load incrementation options) is provided to you for the selection of contact surfaces. Through this option, you can choose, for instance, the surface no. 1 to be in contact with surfaces 3, 5, 6, 7, but not with surfaces 2 and 4. This option can repeatedly be used during an analysis by specifying it in the history definition option.

You can further restrict the potential contact by using the CONTACT NODE or the EXCLUDE option.

User Subroutines

A number of user subroutines are available to you for two- and three-dimensional contact problems. These are listed in Table 3-8.

Table 3-8 User Subroutines for Contact Problems

User Subroutine	Option Required	Contents
MOTION (2D) MOTION (3D)	CONTACT (2D), CONTACT (3D), UMOTION	To define surface motion.
UFRIC	CONTACT (2D), CONTACT (3D), UFRICTION	To define friction data.

Table 3-8 User Subroutines for Contact Problems (Continued)

User Subroutine	Option Required	Contents
UHTCOE	CONTACT (2D), CONTACT (3D), UHTCOEF	To define film coefficient and sink temperature.
UHTCON	CONTACT (2D), CONTACT (3D), UHTCON	To define film coefficient between contact surfaces.
DIGEOM	CONTACT (2D), CONTACT (3D)	To define coordinates of 4-node patches.
SEPFOR	CONTACT (2D), CONTACT (3D)	To define the separation force.
SEPSTR	CONTACT (2D), CONTACT (3D)	To define the separation stress.
UNORST	CONTACT (2D), CONTACT (3D)	To define normal stress at each node in contact.
UCONTACT	CONTACT (2D), CONTACT (3D)	To define new contact schemes.
Note: *Volume D: User Subroutines and Special Routines* manual provides more detailed information on these subroutines.		

Mesh Rezoning in Contact Program

In a contact (or any other) problem, the original mesh can be badly deteriorated during the analysis due to excessive deformation. Rezoning of the deteriorated mesh generally allows the continuation of the analysis to completion. The REZONING parameter, and the rezoning options CONNECTIVITY CHANGE and COORDINATE CHANGE can be used for the rezoning of a deteriorated mesh during an analysis as many times as one wishes. Note that in a contact problem, only the deformable surface definition can be changed. After rezoning occurs, rigid surfaces must be kept the same and cannot be changed.

Spring-Back Analysis

In metal forming analysis, the spring-back condition is of great interest to engineers for the study of the state of residual stresses in the workpiece. However, it is a difficult numerical problem and generally requires a large number of load increments for a final solution. The RELEASE load incrementation option allows the release (separation) of all the nodes in contact with a particular surface at the beginning of the increment. A spring-back condition can be obtained in one load increment. The program iterates the solution within the increment automatically for an equilibrium solution.

Contact Tolerance

A node will come into contact with another body when it enters the contact tolerance zone. This area is dependent upon the value of ERROR and BIAS entered on the third data block. When BIAS is zero, the tolerance is equidistant from the actual surface as shown in Figure 3-15 (a); otherwise, the situation shown in Figure 3-15 (b) is used. If a node would have moved past line B, then increment splitting would occur if the fixed time step procedure is used and the increment splitting procedure is used.

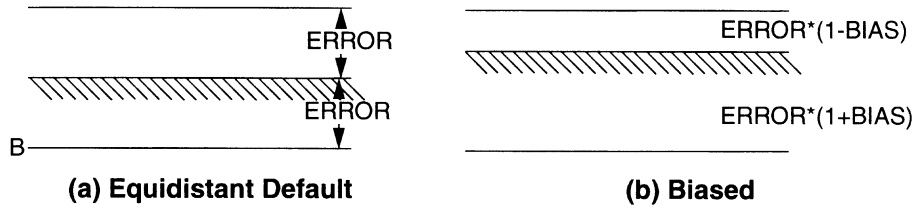


Figure 3-15 Contact Tolerances

Separation

A node on a body will separate from another surface when a tensile load is required to keep it on the surface. The procedure used is either based upon the nodal force or an effective nodal stress. The default separation force is the maximum residual force (separation based upon nodal force method) or the maximum stress at reaction nodes times the convergence tolerance (nodal stress method).

Corner Conditions

When a node slides along a surface composed of multiple segments, three conditions can occur based on the angle that the segments make. This is true for both two-dimensional and three-dimensional problems. The Figure 3-16 shows the two-dimensional case for simplicity. If the angle between the two segments is between $180 - \beta < \alpha < 180° + \beta$, the node smoothly slides between the segments. If the angle is such that $0 < \alpha < 180 - \beta$, the node sticks in the sharp concave corner. If the angle is such that $\alpha > 180 + \beta$, the node separates. The value of β is 8.625° for two-dimensional problems and 20° for three-dimensional problems. These values can be reset using the PARAMETERS option.

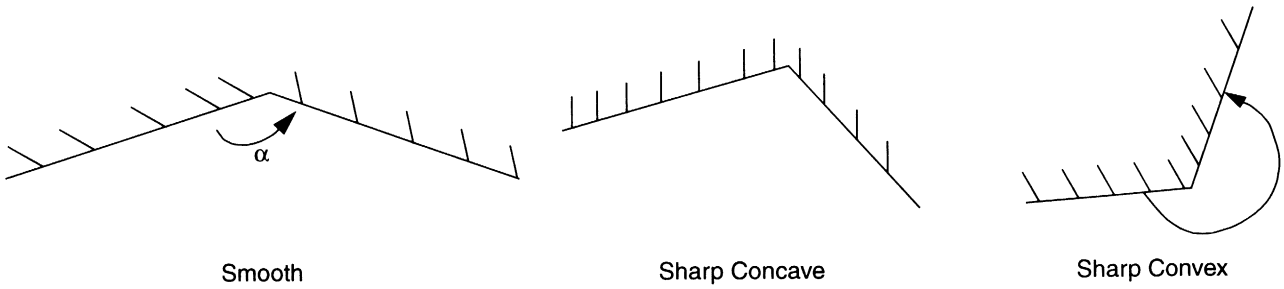


Figure 3-16 Corner Conditions

Friction

There are several friction models available in MARC. The friction types, entered on the second line of the CONTACT block, are as follows:

0	No friction
1	Shear friction
2	Coulomb friction
3	Shear friction for rolling
4	Coulomb friction for rolling
5	Stick-slip model

More details on these models are presented in *Volume A: Theory and User Information*. For model types (1-4), you need to define both the friction constant and the value of `RVCNST(C)`. This value is used to smooth out the discontinuous behavior of friction. The value of `RVCNST` can be interpreted as the value of the relative velocity when sliding occurs. A very small value results in a difficulty to converge; while a very large value results in hardly any effect of friction. This is depicted in Figure 3-17.

Using the stick-slip friction model, three parameters (α, β, and e) are available to control the numerical behavior. a represents a tolerance on the frictional force before sliding occurs. The node changes from stick to slip when

$$|F_t| > \alpha\mu F_n$$

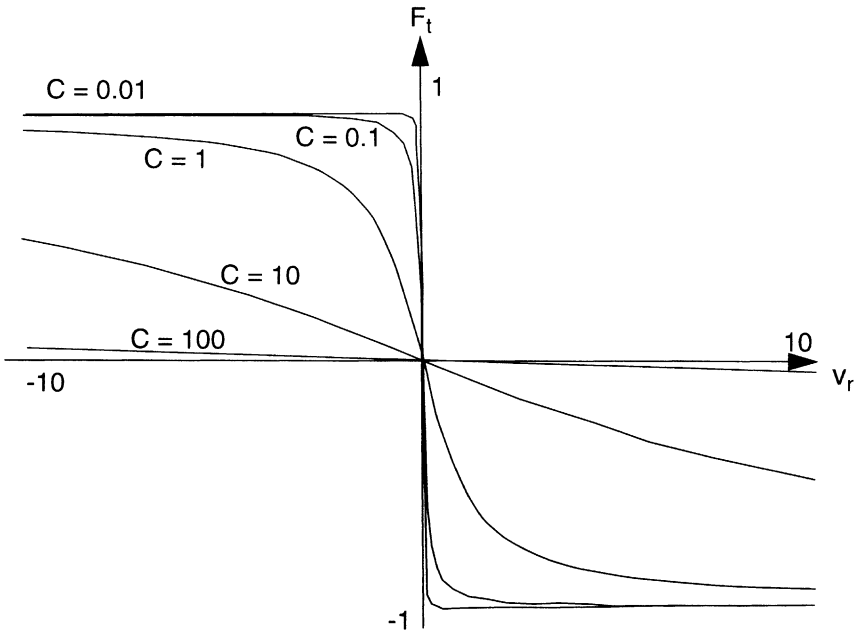


Figure 3-17 Stick-slip Approximation (F_n = 1)

β represents the amount of relative displacement to require slipping conditions. e is a tolerance on the convergence of the solution. It is required that

$$1 - e \leq \frac{F_t}{F_t^{\,p}} \leq 1 + e$$

where $F_t^{\,p}$ is the tangential force at the previous iteration. This is shown in Figure 3-18. The stick-slip model automatically used the nodal based friction procedure.

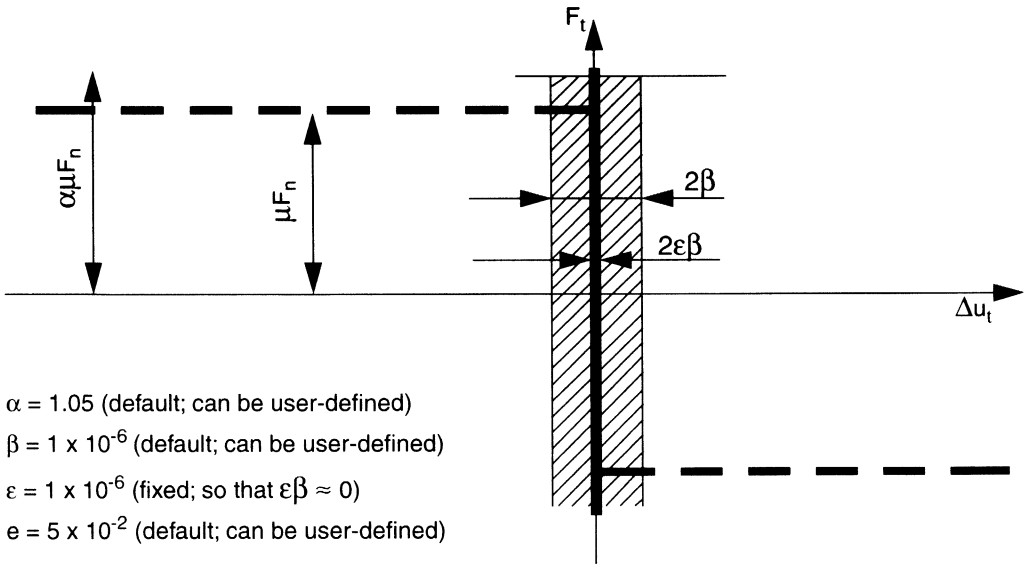


Figure 3-18 Stick-slip Friction Parameters

■ CONTACT

Define Two-Dimensional Contact Surface

Description

This option allows for the input of 2D contact surface definition (rigid or deformable) in contact problems. It also allows you to input friction type, relative sliding velocity for sticking conditions, contact tolerance, average and cut-off strain rates, location of center of rotation, initial angular position of surface, velocity of center of rotation, angular velocity, as well as friction coefficient.

Note: Always define deformable surfaces before rigid surfaces.

If the UMOTION option and user subroutine MOTION are used, velocity data can be skipped.

If the UFRICTION option and user subroutine UFRIC are used, friction data can be skipped, but the friction type must be identified.

If, in a coupled thermal-stress-contact problem, the UHTCOEF option and user subroutine UHTCOE are used, the film coefficient and sink temperature data of a free surface can be skipped.

If the UHTCON option and user subroutine UHTCON are used in a coupled thermal-stress-contact problem, film coefficient data between surfaces in contact can be skipped.

The following data can be changed upon restart:

Friction type	4th field	2nd data line
Maximum number of separations	6th field	2nd data line
Suppression of splitting	7th field	2nd data line
Relative sliding velocity	1st field	3rd data line
Contact distance	2nd field	3rd data line
Average strain rate	3rd field	3rd data line
Cutoff strain rate	4th field	3rd data line
Separation force	5th field	3rd data line
Bias factor	6th field	3rd data line

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-7	1st	A	Enter the word CONTACT.
2nd data block			
1-5	1st	I	Number of surfaces to be defined.
6-10	2nd	I	Maximum number of entities to be created for any surface.
11-15	3rd	I	Upper bound to the number of nodes that lie on the periphery of any deformable surface.
16-20	4th	I	Friction type 0: No Friction 1: Shear Friction 2: Coulomb Friction 3: Shear Friction for Rolling 4: Coulomb Friction for Rolling 5: Stick-slip Coulomb Friction
21-25	5th	I	Enter 0 for the calculation of Coulomb friction based on nodal stress. Enter 1 for the calculation of Coulomb friction based on nodal force. Default is 0.
26-30	6th	I	Maximum number of separations allowed in each increment. Default is 9999.
31-35	7th	I	Enter 0 (default) to use the increment splitting procedure for the fixed time step procedures (AUTO LOAD, DYNAMIC CHANGE, TRANSIENT NON AUTO). Enter 1 for the suppression of the splitting of an increment in fixed time step procedure. Enter 2 for adaptive time step procedure. Default is 0. Enter 3 to use contact procedure which does not require increment splitting.

Format Fixed	Format Free	Data Type	Entry
36-40	8th	I	Enter 1 for the interference kinematic check. Enter 2 to suppress bounding box checking (this might eliminate penetration, but will slow down the solution). Enter 3 to not reset NCYCLE=0 when separation occurs; this speeds up the solution but might result in instabilities. Enter 4 for analytical surfaces only; check for separation only when convergent solution, similar to PWL approach.
41-45	9th	I	Control separations within an increment. When 0 is entered, if the force on a node is greater than the separation force, the node separates and an iteration occurs. When 1 is entered, if a node, which was in contact at the end of the previous increment, has a force greater than the separation force, the node does not separate in this increment, but separates at the beginning of the next increment. When 2 is entered, if a new node comes into contact during this increment, it is not allowed to separate during this increment (prevents chattering). When 3 is entered, both (1) and (2) above are in effect.
46-50	10th	I	Parameter governing normal direction/thickness contribution of shell (ISH). Enter 0 – Check Node Contact with top and bottom surface Enter 1 – Nodes only come into contact with bottom layer Enter 2 – Nodes only come into contact with bottom layer and ignore shell thickness Enter -1 – Nodes only come into contact with top layer Enter -2 – Nodes only come into contact with top layer and ignore shell thickness

Format Fixed	Format Free	Data Type	Entry
51-55	11th	I	Enter 1 to reduce printout of surface definition.
56-60	12th	I	Enter 1 to have separation based upon stresses not forces.
3rd data block			
1-10	1st	F	For friction types 1, 2, 3, or 4 enter the relative sliding velocity between bodies below which sticking is simulated (RVCNST). Default = 1.0. For friction type 5, enter the slip-to-stick transition region (β); Default is 1.e-6.
11-20	2nd	F	Distance below which a node is considered touching a body (ERROR). Leave blank if you want MARC to calculate it. This number is also used to divide splines. If splines are used, this must be defined.
21-30	3rd	F	Average strain rate. Used in rigid-plastic analysis only to start a problem. Default is 1.e-4
31-40	4th	F	Cutoff strain rate below which flow stresses drop linearly to zero. Used in rigid-plastic analysis only. Default is 1.e-12.
41-50	5th	F	Separation force above which a node separates from a body (FNTOL). Default is the maximum residual force. If the 12th field of the second line is 1, enter the separation stress.
51-60	6th	F	Contact tolerance BIAS factor. (0-1)
61-70	7th	F	For stick-slip model, enter the friction coefficient multiplier (α). Defaults to 1.05
71-80	8th	F	For stick-slip model, enter the friction force tolerance (e). Defaults to 0.05.

Format Fixed	Format Free	Data Type	Entry
Data block 4, 5, 6, 7, and 8 are repeated once for each body to be defined.			
4th data block			
1-5	1st	I	Body number.
6-10	2nd	I	Number of sets of geometrical data to be input for this rigid body (NETTY). Enter 0 if deformable body.
11-15	3rd	I	For rigid surfaces, enter 1 if surface is a symmetry plane. For deformable bodies, enter 1 if single-sided deformable-deformable contact will be used. Note that in this case results are dependent upon the order in which contact bodies are defined.
16-20	4th	I	When a node contacts multiple rigid bodies, it contributes a load to a body, the load body being indeterminate. Setting this field to 1 insures that all nodes contacting this body contributes their load to this body. This only influences the load summary table.
21-25	5th	I	Enter 1 if analytic form is to be used.
26-30	6th	I	Enter -1 if body is position controlled. Enter 0 (default) if body is velocity controlled. Enter a positive number if body is load controlled. The number entered is the node number where point loads are applied.
5th data block			
For a deformable body, only the friction coefficients are read and the other entries are ignored.			
1-10	1st	F	First coordinate of initial position of center of rotation.
11-20	2nd	F	Second coordinate of initial position of center of rotation.
21-30	3rd	F	Not used; enter 0.
31-40	4th	F	First component of velocity or target position of center of rotation.
41-50	5th	F	Second component of velocity or target position of center of rotation.
51-60	6th	F	Angular velocity or angular position about center of rotation (radians/time).
61-70	7th	F	Friction coefficient.

Format Fixed	Format Free	Data Type	Entry

6th data block

The 6th data block is only necessary for coupled analysis.

1-10	1st	F	Heat Transfer coefficient (film) to environment.
11-20	2nd	F	Environment sink temperature.
21-30	3rd	F	Contact heat transfer coefficient (film).
31-40	4th	F	Body temperature. (Required for rigid body only.)

A. For 2D Deformable Bodies

7a data block

1-80	1st	I	Enter a list of elements of which the body is comprised.

The 7th and 8th data blocks are repeated for as many geometrical data as required (NETTY).

B. For 2D Rigid Body (Line-Segment)

7b data block

1-5	1st	I	Enter 1 for straight line segments (ITYPE).
6-10	2nd	I	Number of points required to define polyline (NPOINT).

The 8b data block is repeated once for each point entered.

8b data block

1-10	1st	F	First coordinate of point.
11-20	2nd	F	Second coordinate of point.

C. For 2D Rigid Body (Circular Arc)

7c data block

1-5	1st	I	Enter 2 for circular arc (ITYPE).
6-10	2nd	I	Method of describing circular arc (METHOD). See Figure 3-5.

Format Fixed	Format Free	Data Type	Entry

The 8c data block is repeated four times.

8c data block

1-10	1st	F	First coordinate of point.
11-20	2nd	F	Second coordinate of point.

D. For 2D Rigid Body (Spline)

7d data block

1-5	1st	I	Enter 3 for spline (ITYPE).
6-10	2nd	I	Number of points required to define spline (NPOINT).

The 8d data block is repeated for each point to be entered.

8d data block

1-10	1st	F	First coordinate of point.
11-20	2nd	F	Second coordinate of point.

E. For 2D Rigid Body (NURBS)

7e data block

1-5	1st	I	Enter 9 for NURBS.
6-10	2nd	I	Number of control points (NPTU).
11-15	3rd	I	Order (NORU).
16-20	4th	I	Number of subdivisions, default 50.

The 8e data block is repeated NPTU times for control points.

8e data block

1-10	1st	F	First coordinate of point
11-20	2nd	F	Second coordinate of point.

If interpolation scheme is used the following two data blocks are ignored.

The 9e data block is repeated NPTU times for homogeneous coordinate.

9e data block

1-10	1st	F	Homogeneous coordinate between 0 and 1.

The 10e data block is repeated NPTU+ NORU times for knot vectors.

10e data block

1-10	1st	F	Component of knot vector between 0 and 1.

■ CONTACT

Define Three-dimensional Contact Surface

Description

This option allows for the input of 3D contact surface definition (rigid or deformable) in contact problems. It also allows you to input friction type, relative sliding velocity for sticking condition, contact tolerance, average and cut-off strain rates, location of center of rotation, initial angular position of surface, velocity of center of rotation, angular velocity, as well as friction coefficient.

Note: Always define deformable surfaces before rigid surfaces.

If the UMOTION option and user subroutine MOTION are used, velocity data can be skipped.

If the UFRICTION option and user subroutine UFRIC are used, friction data can be skipped, but the friction type must be identified.

If the UHTCOEF option and user subroutine UHTCOE are used, film coefficient and sink temperature data of a free surface in a coupled thermal-stress- contact problem can be skipped.

If the UHTCON option and user subroutine UHTCON are used in a coupled thermal-stress-contact problem, film coefficient data between surfaces in contact can be skipped.

If the 4-node patch option for the definition of rigid surface and the user subroutine DIGEOM are used, the coordinates of the patches can be skipped.

The following data can be changed upon restart:

Friction type	4th field	2nd data line
Maximum number of separations	6th field	2nd data line
Suppression of splitting	7th field	2nd data line
Relative sliding velocity	1st field	3rd data line
Contact distance	2nd field	3rd data line
Average strain rate	3rd field	3rd data line
Cutoff strain rate	4th field	3rd data line
Separation force	5th field	3rd data line
Bias factor	6th field	3rd data line

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-7	1st	A	Enter the word CONTACT.
2nd data block			
1-5	1st	I	Number of surfaces to be defined.
6-10	2nd	I	Maximum number of entities to be created for any surface.
11-15	3rd	I	Upper bound to the number of nodes that lie on the periphery of any deformable surface.
16-20	4th	I	Friction type: 0: No Friction 1: Shear Friction 2: Coulomb Friction 3: Shear Friction for Rolling 4: Coulomb Friction for Rolling 5: Stick-slip Coulomb Friction
21-25	5th	I	Enter 0 for the calculation of Coulomb friction based on nodal stress. Enter 1 for the calculation of Coulomb friction based on nodal force instead of nodal stress. Default is 0.
26-30	6th	I	Maximum of separations allowed in each increment. Default is 9999.
31-35	7th	I	Enter 0 (default) to use the increment splitting procedure for the fixed time step procedures (AUTO LOAD, DYNAMIC CHANGE, TRANSIENT NON AUTO). Enter 1 for the suppression of the splitting of an increment in fixed time procedure. Enter 2 for adaptive time stepping procedure. Default is 0. Enter 3 to use contact procedure which does not require increment splitting.
36-40	8th	I	Enter 1 for the interference kinematic check. Enter 2 to suppress bounding box checking (this might eliminate penetration, but slows down the solution).

Format Fixed	Format Free	Data Type	Entry
			Enter 3 to not reset NCYCLE = 0; this speeds up the solution but might result in instabilities. Enter 4 for analytical surfaces only; check for separation only when convergent solution, similar to PWL approach.
41-45	9th	I	Control separations within an increment. When 0 is entered, if the force on a node is greater than the separation force, the node separates and an iteration occurs. When 1 is entered, if a node which was in contact at the end of the previous increment has a force greater than the separation force, the node does not separate in this increment, but separates at the beginning of the next increment. When 2 is entered, if a new node comes into contact during this increment, it is not allowed to separate during this increment (prevents chattering). When 3 is entered, both (1) and (2) above is in effect.
46-50	10th	I	Parameter governing normal direction/thickness contribution of shell (ISH). Enter 0 – Check Node Contact with top and bottom surface Enter 1 – Nodes only come into contact with bottom layer Enter 2 – Nodes only come into contact with bottom layer and ignore shell thickness Enter -1 – Nodes only come into contact with top layer Enter -2 – Nodes only come into contact with top layer and ignore shell thickness
51-55	11th	I	Enter 1 to reduce printout of surface definition.
56-60	12th	I	Enter 1 to have separation based upon stresses not forces.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
1-10	1st	F	For friction types 1, 2, 3, or 4 enter the relative sliding velocity between bodies below which sticking is simulated (RVCNST). Default is 1.0. For friction type 5, enter the slip-to-stick transition region (β); Default is 1.e-6.
11-20	2nd	F	Distance below which a node is considered touching a body (ERROR). Leave it blank if you want MARC to calculate it.
21-30	3rd	F	Average strain rate. Used in rigid-plastic analysis only to start a problem. Default is 1.e-4.
31-40	4th	F	Cutoff strain rate below which flow stresses drop linearly to zero. Used in rigid-plastic analysis only. Default is 1.e-12.
41-50	5th	F	Separation force, above which a node separates from a body (FNTOL). Default is the maximum residual force. If the 12th field of the second line is 1, enter the separation stress.
51-60	6th	F	Contact tolerance BIAS factor. (0-1)
61-70	7th	F	For stick-slip model, enter the friction coefficient multiplier (α). Defaults to 1.05
71-80	8th	F	For stick-slip model, enter the friction force tolerance (e). Defaults to 0.05.

Data blocks 4, 5, 6, 7, 8, 9, 10 are repeated once for each body to be defined.

4th data block			
1-5	1st	I	Body number.
6-10	2nd	I	Number of sets of body entities, NSURGN, to be input for this rigid body. Enter 0 if deformable body.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	For rigid bodies, enter 1 if body is a symmetry plane. For deformable bodies, enter 1 if single-sided deformable-deformable contact will be used. Note that in this case, results are dependent upon the order in which contact bodies are defined.
16-20	4th	I	When a node contacts multiple rigid bodies, it contributes a load to a body, the load body being indeterminate. Setting this field to 1 insures that all nodes contacting this body contributes their load to this body. This only influences the load summary table.
21-25	5th	I	Enter 1 if analytic form is to be used.
26-30	6th	I	Enter -1 if body is position controlled. Enter 0 (default) if body is velocity controlled. Enter a positive number if body is load controlled. The number entered is the node number where point loads are applied.
5th data block			
1-10	1st	F	First coordinate of initial position of center of rotation.
11-20	2nd	F	Second coordinate of initial position of center of rotation.
21-30	3rd	F	Third coordinate of initial position of center of rotation.
31-40	4th	F	First component of velocity or target position of center of rotation.
41-50	5th	F	Second component of velocity or target position of center of rotation.
51-60	6th	F	Third component of velocity or target position of center of rotation.
6th data block			
1-10	1st	F	Angular velocity or angular position about local axis through center of rotation.
11-20	2nd	F	Not used; enter 0.
21-30	3rd	F	Not used; enter 0.
31-40	4th	F	First component direction cosine of local axis.

Format Fixed	Format Free	Data Type	Entry
41-50	5th	F	Second component direction cosine of local axis.
51-60	6th	F	Third component direction cosine of local axis.
61-70	7th	F	Friction coefficient.

The 7th data block is only necessary for coupled analysis.

7th data block

1-10	1st	F	Heat transfer coefficient (film) to environment.
11-20	2nd	F	Environment sink temperature
21-30	3rd	F	Contact heat transfer coefficient
31-40	4th	F	Body temperature. (Required for rigid body only.)

A. For 3D Deformable Body

8a data block

1-80	1st	I	Enter a list of elements of which the body is comprised.

The 8th, 9th, and 10th data blocks are repeated for each set of body entities (NSURGN).

B. For 3D Rigid Body (Ruled Surface)

8b data block

1-5	1st	I	Enter 4 for ruled surface (ITYPE).
6-10	2nd	I	Entity type of the first surface generator (child) of the surface, JTYPE1.
11-15	3rd	I	If JTYPE1 = 1, 3, 4, 5 enter number of point required to describe first surface generator (NPOINT1). If JTYPE1 = 2 method to describe the circular arc (METH).
16-20	4th	I	Entity type of the second surface generator (child) of the surface (JPOINT2).
21-25	5th	I	If JTYPE1 = 1, 3, 4, 5 enter number of point required to describe second surface generator (NPOINT2). If JTYPE2 = 2 method to describe the circular arc (METH).

Format Fixed	Format Free	Data Type	Entry
26-30	6th	I	Number of subdivisions along first direction (NDIV1), (direction along first and second surface generator).
31-35	7th	I	Number of subdivisions along second direction (NDIV2), (direction from first surface generator to second surface generator).

The 9b data block is repeated (NPOINT1 * NPOINT2) times for ruled surface.

9b data block

1-10	1st	F	First coordinate of point.
11-20	2nd	F	Second coordinate of point.
21-30	3rd	F	Third coordinate of point.

C. For 3D Rigid Body (Surface of Revolution)

8th data block

1-5	1st	I	Enter 5 for surface of revolution (ITYPE).
6-10	2nd	I	Entity type of the surface generator.
11-15	3rd	I	If JTYPE1 = 1, 3, 4, 5 enter number of points required to describe the generator (NPOINT). If JTYPE = 2 Method to describe the circular arc (METH).
16-20	4th	I	Number of subdivisions along the first (surface generator) direction (NDIV1).
21-25	5th	I	Number of subdivisions along the second (circumference) direction (NDIV2).

The 9c data block is repeated NPOINT times for surface of revolution.

9c data block

1-10	1st	F	First coordinate of point.
11-20	2nd	F	Second coordinate of point.
21-30	3rd	F	Third coordinate of point.

Format Fixed	Format Free	Data Type	Entry
10c data block			
1-10	1st	F	First coordinate of 1st point on the axis of revolution.
11-20	2nd	F	Second coordinate of 1st point on the axis of revolution.
21-30	3rd	F	Third coordinate of 1st point on the axis of revolution.
31-40	4th	F	First coordinate of 2nd point on the axis of revolution.
41-50	5th	F	Second coordinate of 2nd point on the axis of revolution.
51-60	6th	F	Third coordinate of 2nd point on the axis of revolution.
61-70	7th	F	Total angle (degree) of rotation (Initial position of the surface generator is given on the 9th data block.)

D. For 3D Rigid Surface (Bezier Surface)

8d data block			
1-5	1st	I	Enter 6 for Bezier surface (ITYPE).
6-10	2nd	I	Number of points along the first direction of surface (NPONT1).
11-15	3rd		Number of points along the second direction of surface (NPOINT2).
16-20	4th	I	Number of subdivisions along first direction (NDIV1).
21-25	5th	I	Number of subdivisions along second direction (NDIV2).

The 9d data block is repeated (NPOINT1 * NPOINT2) times for Bezier surface.

9d data block			
1-10	1st	F	First coordinate of point.
11-20	2nd	F	Second coordinate of point.
21-30	3rd	F	Third coordinate of point.

Format Fixed	Format Free	Data Type	Entry

E. For 3D Rigid Surface (4-Node Patch)

8e data block

1-5	1st	I	Enter 7 for a surface consisting of 4-node patches (ITYPE).
6-10	2nd	I	Number of 4-node patches to be read (which makes the entire surface) NSEG.
11-15	3rd	I	Number of points to be read (NPOINT).
16-20	4th	I	Unit number. Defaults to input. Set KUNIT=-1 if data entered via user subroutine DIGEOM.
21-25	5th	I	Set to 1 if patch data is to be printed. Default: no printing.

The 9e data block is repeated NSEG times for patches not entered by means of user subroutines (ITYPE = 7 and KUNIT not -1)

9e data block

1-5	1st	I	Patch number (not necessary, can be left blank).
6-10	2nd	I	Not used.
11-15	3rd	I	First point number of this patch.
16-20	4th	I	Second point number of this patch.
21-25	5th	I	Third point number of this patch.
26-30	6th	I	Fourth point number of this patch.

The 10e data block is repeated NPOINT times for patches not entered by means number of user subroutine (ITYPE = 7 and KUNIT not -1).

10e data block

1-5	1st	I	Point number.
6-15	2nd	F	First coordinate of this node.
16-25	3rd	F	Second coordinate of this node.
26-35	4th	F	Third coordinate of this node.

Format Fixed	Format Free	Data Type	Entry

F. For 3D Rigid Surface (Poly-Surface)

8f data block

1-5	1st	I	Enter 8 for polysurface (ITYPE).
6-10	2nd	I	Number of points along the first direction of surface (NPOINT1).
11-15	3rd	I	Number of points along the second direction of surface (NPOINT2).

The 9f data block is repeated (NPOINT1 * NPOINT2) times for poly-surfaces.

9f data block

1-10	1st	F	First coordinate of point.
11-20	2nd	F	Second coordinate of point.
21-30	3rd	F	Third coordinate of point.

G. For 3D Rigid Surface (NURBS)

8g data block

1-5	1st	I	Enter 9 for NURBS.
6-10	2nd	I	Number of control points along u-direction (NPTU).
11-15	3rd	I	Number of control points along v-direction (NPTV).
16-20	4th	I	Order along u-direction (NORU).
21-25	5th	I	Order along v-direction (NORV).
26-30	6th	I	Number of subdivisions along u-direction; default 50.
31-35	7th	I	Number of subdivisions along v-direction; default 50.
36-40	8th	I	Number of trimming curves.

Format Fixed	Format Free	Data Type	Entry
The 9g data block is repeated (NPTU * NPTV) for control points.			
9g data block			
1-10	1st	F	First coordinate of point.
11-20	2nd	F	Second coordinate of point.
21-30	3rd	F	Third coordinate of point.
The 10g data block is repeated (NPTU * NPTV) for homogeneous coordinate.			
10g data block			
1-10	1st	F	Homogeneous coordinate ($0 \leq h \leq 1$).
The 11g data block is repeated (NPTU + NORU) + (NPTV + NORV) for knot vectors.			
11g data block			
1-10	1st	F	Knot vector($0 \leq k \leq 1$).

H. For 3D Rigid Surface (Cylinder)

8h data block			
1-5	1st	I	Enter 10 for Cylinder.
6-10	2nd	I	Number of subdivisions.
9h data block			
1-10	1st	F	First coordinate of center point on top surface.
11-20	2nd	F	Second coordinate of center point on top surface.
21-30	3rd	F	Third coordinate of center point on top surface.
31-40	4th	F	Radius of top surface
41-50	5th	F	First coordinate of center point on bottom surface.
51-60	6th	F	Second coordinate of center point on bottom surface.
61-70	7th	F	Third coordinate of center point on bottom surface radius of bottom surface.
71-80	8th	F	Radius of bottom surface.
			Note: If the radius is negative value in 4th field the normal of cylinder is outward. Default is inward.

Format Fixed	Format Free	Data Type	Entry

I. For 3D Rigid Surface (Sphere)

8i data block

1-5	1st	I	Enter 11 for Sphere.
6-10	2nd	I	Number of subdivisions.

9i data block

1-10	1st	F	First coordinate of center point.
11-20	2nd	F	Second coordinate of center point.
21-30	3rd	F	Third coordinate of center point.
31-40	4th	F	Radius of sphere.

Note: If the radius is negative value in 4th field, the normal of sphere is outward. Default is inward.

■ CONTACT TABLE

Define Contact Table

Description

This option is useful for deactivating or activating bodies when the CONTACT option is used. To avoid unnecessary detection of contact between bodies, you can control the detection of contact. The default for contact analysis is that every body detects the possibility of contact relative to all other bodies and itself if it is a flexible body. When the CONTACT TABLE option is entered, the default of detection for every body is overridden. Instead, you specify the relationship of detection between bodies for contact. The touching body does not contact itself unless you request it. Whenever the touched body is a flexible one, the capability of double-sided contact is applied between the contacting bodies even if you do not explicitly request it. A positive value of the interference closure implies that there is an overlap between the bodies; a negative value implies that a gap exists.

The following control variables of contact between bodies can be modified throughout the table: contact tolerance, separation force, friction coefficient, interference closure and contact heat transfer coefficient (for coupled thermal-stress-contact analysis). In addition, you can invoke the glue option. The previous values of those control variables is not overridden unless nonzero values are entered here.

In the glue option when a node contacts a rigid body, the relative tangential displacement is zero. When a node contacts a deformable body, all of the degrees of freedom are tied.

Notes: This option should be placed after the CONTACT option.

In a restart analysis, if these values are to be changed, use the REAUTO option and specify the CONTACT TABLE after the END OPTION.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word CONTACT TABLE.
2nd data block			
1-5	1st	I	Enter the number of sets of bodies to be input.

Format Fixed	Format Free	Data Type	Entry
The 3rd and 4th data blocks are entered once for each set of bodies to be input.			
3rd data block			
1-5	1st	I	Enter the touching body number.
6-15	2nd	F	Enter the contact tolerance (ERROR).
16-25	3rd	F	Enter the contact separation force (FNTOL).
26-35	4th	F	Enter the friction coefficient.
36-45	5th	F	Enter the interference closure amount, normal to the contact surface.
46-55	6th	F	Enter the contact heat transfer coefficient (coupled analysis only).
56-60	7th	I	Enter 1 to invoke the glue option.
4th data block			
			Enter a list of bodies for which the touching body detects contact with the parameters above.

■ SPLINE

Analytical Surface used to Represent a Deformable Body

Description

In order to improve the accuracy for a deformable-deformable contact analysis, the outer surface of a contacted body can be described based on a spline (2D) or Coons surface (3D) description. The analytical surface is then used to calculate the normal to the deformable body.

In 2D, for a contacted segment, a spline is created based on:

tangent at first and second point of segment

position of first and second point of segment

In 3D, for a contacted segment, a Coons surface is created based on:

tangent vectors at corner points of segment

position of corner points of segment

zero twist vectors

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-6	1st	A	Enter the word SPLINE.
2nd data block			
1-5	1st	I	Enter the number of deformable bodies for which the spline description must be applied.

Format Fixed	Format Free	Data Type	Entry
The 3rd and 4th data blocks are repeated for each deformable body with a spline description.			
3rd data block			
1-5	1st	I	Body number.
4th data block			
			Enter a list of nodes defining nodes/edges to be excluded from the spline description. **Note:** Typically, this list contains nodes at which the direction of the tangent vector at the outer contour of the structure shows a discontinuity. In 3D, when an element edge should be excluded, the nodes defining the edge must be entered one after another.

■ UMOTION

Invoke User Subroutine to Prescribe Surface Motion

Description

This option calls the user subroutine MOTION to define surface motions. See *Volume D: User Subroutines and Special Routines*.

Note: This option should be placed after the CONTACT option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-7	1st	A	Enter the word UMOTION.

■ UFRICTION

Invoke User Subroutine to Define Surface Friction Behavior

Description

This option calls the user subroutine UFRIC to define friction coefficients (see *Volume D: User Subroutines and Special Routines*).

Note: Use this option only in conjunction with the CONTACT option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-9	1st	A	Enter the word UFRICTION.

■ UHTCOEF

Invoke User Subroutine to Define Surface/Environment Thermal Behavior

Description

This option calls the user subroutine UHTCOE to define heat transfer coefficients (film coefficients) and sink temperatures of a free surface subjected to convective or radiative heat transfer (see *Volume D: User Subroutines and Special Routines*).

Note: Use only in conjunction with the CONTACT option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-7		A	Enter the word UHTCOEF.

■ UHTCON

Invoke User Subroutine to Define Surface/Surface Thermal Behavior

Description

This option calls the user subroutine UHTCON to define heat transfer coefficients (film coefficients) between surfaces in contact (see *Volume D: User Subroutines and Special Routines*).

Note: Use only in conjunction with the CONTACT option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-6	1st	A	Enter the word UHTCON.

■ CONTACT NODE

Define Nodes for Surface Contact

Description

This option is used to define which nodes in a body might potentially contact other surfaces. This option can be used to reduce the computational cost if a body has many exterior nodes, yet it is known for which nodes contact might occur. If this option is not used, all exterior surface nodes are checked for contact.

Notes: If this option is used and a node number is not explicitly listed, that node might penetrate other bodies.

In a restart analysis, if these values are to be changed, use the REAUTO option and specify the CONTACT NODE option after the END OPTION.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-12	1st	A	Enter the words CONTACT NODE.
2nd data block			
1-5	1st	I	Enter the number of bodies for which exterior nodes are defined
3rd data block			
1-5	1st	I	Body number.
4th data block			
1-80	1st	I	Enter a list of nodes that are potential contact nodes.

■ EXCLUDE

Ignore Contact with Certain Regions

Description

For certain contact problems, you might wish to influence the decision regarding the deformable segment a node contacts. By means of the EXCLUDE option, you can specify a list of nodes defining segments to be excluded from the contacted bodies.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-6	1st	A	Enter the word EXCLUDE.
2 data block			
1-5	1st	I	Enter the number of deformable bodies for which the EXCLUDE option must be applied.

The 3rd and 4th data blocks are repeated for each deformable body with the EXCLUDE option.

3rd data block			
1-5	1st	I	Body number.
4th data block			
			Enter a list of nodes defining segments to be excluded from contacted bodies. **Note:** In 2D, each segment must be defined by two nodes. In 3D, each segment must be defined by four nodes.

■ UCONTACT

Invoke User Subroutine for User-defined Contact Conditions

Description

This option calls the user subroutine UCONTACT to define contact conditions (see *Volume D: User Subroutines and Special Routines*). In general, the four stages in the MARC contact procedure are: checking contact, enforce contact constraint, check separation, and check penetration. You can substitute your own procedure to replace the default MARC one.

Note: Use only in conjunction with the CONTACT option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-6	1st	A	Enter the word UCONTACT.

Material Properties

This section describes the material properties that can be associated with the model. These consist of both the specification of the constitutive model used to describe the material behavior and the actual material data necessary to represent the material. The default material for stress analysis is an isotropic elastic-plastic compressible material with no work hardening.

The model definition block ISOTROPIC can be used for the input of simple engineering material properties, including the fluid density in fluid-solid interaction problems. Additional model definition options such as ORTHOTROPIC, ANISOTROPIC, MOONEY, OGDEN, FOAM, and HYPOELASTIC are available for more complex material representations. The STRAIN RATE option allows the definition of a strain-rate dependent yield stress; the WORK HARD option allows you to specify work hardening slopes for elastic-plastic behavior. Variations of material properties with temperatures such as Young's modulus, Poisson's ratio, etc. can be entered through TEMPERATURE EFFECTS, ORTHO TEMP, and TIME-TEMP options; cracking data and failure criteria data can be entered using CRACK DATA and FAIL DATA, respectively. Finally, the GAP DATA option can be used for the input of gap width, frictional coefficient, etc. for the gap-friction element (element type 12 and 97).

In addition, the COMPOSITE model definition option allows you to input layer information for a laminated composite material and the ORIENTATION option allows for the definition of the preferred material directions.

A brief description of the material models which MARC has the capability of representing is given below. For additional details, refer to *Volume A: User Information*.

A. Elastic Behavior

1. Isotropic Elastic Compressible Material – This is a material represented by Hooke's Law. This material has a linear relation between stress and strain and its behavior is not path dependent. The stress strain relations can be expressed as $\sigma_{ij} = \lambda\delta_{ij}\varepsilon_{kk} + 2G\varepsilon_{ij}$

   where λ, the Lame constant, and G, the shear modulus, can be expressed as:

   $\lambda = \nu E/((1+\nu)(1-2\nu))$ and $G = E/2(1+\nu)$

   E and ν are the familiar Young's modulus and Poisson's ratio, respectively, and can be specified using the ISOTROPIC model definition option.

2. Orthotropic Elastic Compressible Material – For an isotropic material, every plane is a plane of symmetry and every direction is an axis of symmetry. An orthotropic material, however, has only three mutually orthogonal planes of symmetry. With respect to a coordinate system parallel to these planes, the constitutive law for this material is given by the following more general form of Hooke's Law:

$$\begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{23} \\ \gamma_{31} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_{11}} & -\frac{\nu_{21}}{E_{22}} & -\frac{\nu_{31}}{E_{33}} & 0 & 0 & 0 \\ -\frac{\nu_{12}}{E_{11}} & \frac{1}{E_{22}} & -\frac{\nu_{32}}{E_{33}} & 0 & 0 & 0 \\ -\frac{\nu_{13}}{E_{11}} & -\frac{\nu_{23}}{E_{22}} & \frac{1}{E_{33}} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{12}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{23}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{31}} \end{bmatrix} \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \tau_{12} \\ \tau_{23} \\ \tau_{31} \end{Bmatrix}$$

Due to symmetry of this compliance matrix, $E_{11}\,\nu_{21} = E_{22}\,\nu_{12}$, $E_{22}\,\nu_{32} = E^{33}\,\nu_{23}$, and $E_{33}\,\nu_{13} = E_{11}\,\nu_{31}$. Using these relations, a general orthotropic material has nine independent constants: E_{11}, E_{22}, E_{33}, ν_{12}, ν_{23}, ν_{31}, G_{12}, G_{23}, G_{31}. These nine constants can be specified using the ORTHOTROPIC option. Note that the inequalities $E_{22} > \nu_{23}^2\,E_{33}$, $E_{11} > \nu_{12}^2\,E_{22}$, and $E_{33} > \nu_{31}^2\,E_{11}$ must be satisfied in order for the orthotropic material to be stable. This is checked by the program.

3. Anisotropic Elastic Compressible Material – This is a material represented by the generalized Hookes' Law. This material has a linear relation between stress and strain and its behavior is not path dependent. The stress-strain relation can be expressed as:

   $\sigma_{ij} = C_{ijkl}\,e_{kl}$.

   All of the values of C can be specified using either user subroutine ANELAS or HOOKLW. As an alternative, you can supply the compliance matrix in subroutine HOOKLW. The ANISOTROPIC model definition option is used to direct MARC to call these user subroutines when necessary.

4. Isotropic Incompressible Material Mooney-Rivlin form – The nonlinear elastic material can be represented by a certain class of strain energy functions. The form of this function is:

$$W = C_{10}(I_1 - 3) + C_{01}(I_2 - 3) + C_{11}(I_1 - 3)(I_2 - 3) + C_{20}(I_1 - 3)^2 + C_{30}(I_1 - 3)^3$$

where I_1, I_2 are the first and second invariants of the elastic strain. This strain energy function can represent the Neo-Hookean materials (C_{01}, C_{11}, C_{20}, and C_{30} are zero) or Mooney-Rivlin materials (C_{11}, C_{20}, and C_{30} are zero). The material has a nonlinear relation between stress and strain; hence, an incremental procedure must be performed. Alternative energy functions can be specified via user subroutine UENERG.

The stress-strain relations can be expressed as $\sigma_{ij} = \dfrac{\partial W}{\partial \varepsilon_{ij}}$.

The constants are supplied by you through the MOONEY option. Note that this material model can be used to represent large-strain elastic materials.

5. Isotropic Elastic Incompressible Material-Ogden Formulation - Another representation of nonlinear elastic material is by the Ogden strain energy function. This model can be used to represent large-strain behavior in elastic materials. For plane stress, displacement elements are always used. The strain energy function is

$$W = \sum_{n=1} \frac{\mu_n}{\alpha_n} J^{-\alpha_n/3}(\lambda_1^{\alpha_n} + \lambda_2^{\alpha_n} + \lambda_3^{\alpha_n} - 3) + 4.5K(J^{1/3} - 1)^2$$

Note: There are two modes for performing Mooney or Ogden analysis for plane strain, generalized plane strain, axisymmetric, or solid analysis. This mode is set on the ELASTICITY parameter. If the total Lagrange formulation is invoked, the elements, in this case, must be of the Herrmann formulation. If the updated Lagrange formulation is invoked, the elements must be conventional displacement formulation.

6. Isotropic Elastic Foam Material - This nonlinear elastic material has the characteristic that it can have both large strain deviatoric and volumetric behavior. The material model is used in conjunction with the displacement elements. The strain energy function is

$$W = \sum_{n=1} \frac{\mu_n}{\alpha_n}(\lambda_1^{\alpha_n} + \lambda_2^{\alpha_n} + \lambda_3^{\alpha_n} - 3) + \sum_{n=1} \frac{\mu_n}{\beta_n}(1 - J^{\beta_n})$$

7. General Anisotropic Nonlinear Elastic Material – This can be represented by the hypoelastic material model. The material has a nonlinear relation between stress and strain; hence, an incremental procedure must be performed. The stress-strain relation can be expressed as $\dot{\sigma}_{ij} = C_{ijkl}\dot{\varepsilon} + g_{ij}$ where C_{ijkl} and g_{ij} are functions of elastic strain

and temperature. The material data is described by the user subroutine HYPELA. Model definition block HYPOELASTIC should be used to direct the MARC program to call user subroutine HYPELA when necessary.

B. Elastic-Plastic Behavior

Elastic-plastic material can be described using a variety of models. The differentiation between these models is due to:

A. the inclusion or exclusion of elastic effects;

B. the yield function;

C. the flow rule, and

D. the hardening rule.

1. Rigid Plastic Material – This is the only material model which excludes the elastic strains. The capability is based on the iteration for the velocity field in an incompressible, non-Newtonian fluid. The nonlinear stress-strain relation can be expressed as $S_{ij} = G(\dot{\bar{\varepsilon}})\dot{\varepsilon}_{ij}$. Note that as the material is incompressible, a traction boundary condition must be specified; otherwise the stress field is only known to an arbitrary hydrostatic pressure. Only the yield stress need be entered.
2. Elastic-Perfectly-Plastic Material – This is a material which behaves elastically until it reaches the yield stress. The material has no ability to support additional load (in a uniaxial sense) upon yield. The material has a nonlinear relation between the stress rate and strain rate and is path dependent. You need only specify the elastic constants and the yield stress. The program uses the von Mises yield function and the associated flow law.
3. Elastic-Plastic Isotropic-Hardening Material – This is a model which behaves elastically until it reaches the yield stress. After yielding the material strains or work hardens according to the isotropic model. The yield surface uniformly expands in all directions with increasing equivalent plastic strain. The additional information necessary to define the rate of growth of the yield surface is prescribed through the WORK HARD option.
4. Elastic-Plastic Kinematic-Hardening Material – This is a material whose yield surface behavior is governed by the kinematic hardening model. In this model, the yield surface translates in stress space depending upon the change in plastic strain. The rate of translation is prescribed through the WORK HARD option. This option is set through the ISOTROPIC, ORTHOTROPIC or ANISOTROPIC model definition option.

5. Elastic-Plastic Combined-Hardening Material – This is a material whose yield surface behavior is governed by a combination of both the isotropic and kinematic hardening models. That is, the yield surface both expands in size and shifts in space. This behavior is given through the WORK HARD option. This option is set through the ISOTROPIC, ORTHOTROPIC, or ANISOTROPIC model definition option.

6. Elastic-Plastic-ORNL Hardening – This is a model whose yield surface is governed by the ORNL constitutive theory. This model also allows plastic creep interaction. This option is set through the ISOTROPIC, ORTHOTROPIC, or ANISOTROPIC model definition option.

   As discussed above, models 6, 7, 8, 9, 10, and 11 use the von Mises yield function. It is also possible to use the anisotropic yield function of Hill. You must flag the anisotropy option and use the user subroutine ANPLAS. This data can also be given through the ORTHOTROPIC model definition option. In addition, the yield function can be dependent upon the strain rate; this can be specified by using the STRAIN RATE option.

7. Elastic-Plastic Material with Mohr-Coulomb Yield Surface – is a model which allows the representation of materials in which the yield surface is dependent on the hydrostatic stress. It is appropriate for modeling soils. The yield function dependence on the shear stress is given using the WORK HARD option; the dependence on the hydrostatic stress is given in the ISOTROPIC option. This option is set through the ISOTROPIC model definition option.

8. The yield surface can also be modified based upon the Gurson model for void damage. In this model, a modified von Mises criteria, with hydrostatic stress and void dependency, is used. The DAMAGE option is used to activate this model.

9. A special viscoplastic model for powder materials can be entered through the POWDER option. In this model, the material properties are also dependent on the relative density of the material. These materials are always assumed to be isotropic.

10. A general viscoplastic option is available for materials that are modeled using unified creep-plasticity rules. In this procedure, the elastic properties are defined in either the ISOTROPIC or ORTHOTROPIC option and the inelastic properties are provided through user subroutine UVSCPL.

C. Temperature Dependent Material Properties

All of the parameters (except density) necessary to represent the material properties can be given as a function of temperature. These effects are specified using the TEMPERATURE EFFECTS or ORTHO TEMP option.

D. Relative Density Dependent Material Properties

In powder materials, the materials can be given as a function of the relative density where a relative density of 1.0 implies a fully compacted material. This data is entered through the DENSITY EFFECTS option.

E. Low Tension Material

In addition to using the ISOTROPIC model definition block for the input of Young's modulus, Poisson's ratio, yield stress, etc., the CRACK DATA model definition option must be used for the specification of critical cracking stress, tension-softening modulus, crushing strain and shear retention factor. These data can be alternatively specified by user subroutine UCRACK and TENSOF. Detailed discussion on low tension material can be found in Volume A: User Information.

F. Soil Materials

When performing a coupled fluid-soil analysis, the soil material can be modeled either as a linear elastic material, a nonlinear elastic material or using the modified CAM-CLAY model.

G. Material Dependent Failure Criteria

Five failure criteria are available in the program. They are maximum stress (`MX STRESS`), maximum strain (`MX STRAIN`), Tsai-Wu, Hoffman and Hill. Detailed discussion on failure criteria can be found in *Volume A: User Information*. During each analysis, up to three fail criteria can be selected; failure indices are calculated and printed for every integration point. The model definition block FAIL DATA (or user subroutine UFAIL) is used for the input of failure criteria data.

H. Characterization of Gap Elements

The GAP DATA model definition block allows for the input of gap closure distance, gap elastic stiffness, contact coefficient of friction and momentum ratio. Detailed discussion on gap elements (elements 12 and 97) can be found in *Volume B: Element Library*.

I. Laminated Composite

A laminated composite is a "material" made of several thin layers of separate materials with different material behavior, layer thicknesses, and orientations from one layer to the next. To model laminated composite plates, shells, or beams with MARC, use the COMPOSITE option. In this option, three quantities are specified on a layer-by-layer basis: material identification

number, layer thickness, and ply angle. The entire set of data (a "composite group") is then associated with a list of elements. For each individual layer, all of the above mentioned constitutive laws can be used with the exception of the low tension material. The layer thickness can be constant or variable (in the case of variable total thickness elements), and the ply angle can change from one layer to the next. The orientation of the 0 ply angle within each element is defined in the ORIENTATION option. For more information on the specific assumptions employed by the COMPOSITE option, see *Volume A: User Information.*

J. Material Preferred Direction

Every element type in MARC has a default orientation (that is, a default coordinate system) within which element stress-strain calculations take place. This system is also assumed to be the coordinate system of material symmetry. This is especially important for nonisotropic materials (orthotropic, anisotropic, or composite materials). With the ORIENTATION option, you specify the orientation of the material axes of symmetry (or the 0 ply angle line, if composite) in one of four different ways: 1) as a specific angle offset from an element edge, 2) as a specific angle offset from the line created by two intersecting planes, 3) as a particular coordinate system specified by user-supplied unit vectors, or 4) as specified by user subroutine ORIENT. For more information on these options, see *Volume A: Theory and User Information.*

K. Material Property (Element) Coordinate Systems in MARC

When defining material properties in MARC, you should be aware of the three coordinate systems used by MARC. They are:

Global Coordinate System

The material data supplied in the ISOTROPIC and ORTHOTROPIC options are always considered to be defined with respect to the material principal axes of symmetry.

For continuum problems (that is, those using 2D or 3D solid elements), this coordinate system is aligned (by default) with the global xyz coordinate system. (For truss, beam and shell problems, see MARC (or Local) Coordinate System.) This is not normally a problem for isotropic materials since every direction is then a principal direction. For orthotropic materials, however, the material principal coordinates are seldom aligned with the global coordinates. For this reason, a second coordinate system is needed.

User-defined (or "preferred") Coordinate System

If the material principal axes of symmetry are not aligned with the global coordinate system, a second coordinate system is used. This is the user-defined "preferred" coordinate system. This coordinate system is usually taken to be coincident with the material principal coordinates. The orientation of the preferred coordinate system and, hence, of the material principal axes, is

defined by the ORIENTATION option on an element-by-element basis. In this way, you can completely define through the input deck, the material data and orientation of a general orthotropic material.

MARC (or Local) Coordinate System

For truss, beam, and shell problems, the material principal axes of symmetry are aligned (by default) with special local element dependent coordinate systems. For example, for shell element 72, these local coordinates are the $\tilde{v}^1\tilde{v}^2\tilde{v}^3$ surface coordinates so that material property data are assumed to begin with respect to these $\tilde{v}^1\tilde{v}^2\tilde{v}^3$ coordinates. For isotropic materials, this is not normally a problem. For orthotropic materials, the material principal axes cannot be aligned with the $\tilde{v}^1\tilde{v}^2\tilde{v}^3$ axes. As in continuum problems, the ORIENTATION option is used to define a second set of preferred coordinates. This allows you to arbitrarily orient orthotropic materials in shells with local coordinates.

Composite Shells

In composite shells, the orientation of the materials in each shell layer can vary from layer to layer. In this case, the ORIENTATION option is used to locate the 0^0 ply angle direction in the shell surface. (If the ORIENTATION option is omitted, the 0^0 ply angle direction coincides with the $\tilde{V}^1$ axis. See previous section.) For each layer, additional ply angle offsets from this 0^0 ply angle direction are given in the COMPOSITE option. This allows you to arbitrarily orient an arbitrary composite layup in shells with local coordinates.

■ ISOTROPIC

Define Mechanical Data for Isotropic Materials

Description

This option allows you to define material properties, a yield criterion, and a strain hardening law for an isotropic material. You can also associate these material properties with a list of element numbers.

To define the dependence of these properties on temperature, use the TEMPERATURE EFFECTS model definition.

Defaults for this option are von Mises yield criterion, isotropic strain hardening law (with a slope of 0.0 if the WORK HARD model definition is omitted), and an equivalent yield stress of 10^{20}. Therefore, the default is a perfectly elastic nonyielding isotropic material.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word ISOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of sets of isotropic material data to follow (optional).
6-10	2nd	I	Enter the unit number for input. Defaults to input deck.
Data blocks 3, 4, 5, and 6 are repeated as a set, once for each set of isotropic material defined.			
3d data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.), for cross-referencing TEMPERATURE EFFECTS and WORK HARD data.
6-15	2nd	A	Enter one of the following yield criteria: VON MISES – Von Mises (Default). LIN MOHRC – Linear Mohr-Coulomb. PBL MOHRC – Parabolic Mohr-Coulomb.

Format Fixed	Format Free	Data Type	Entry
			BUY MOHRC– Buyukozturk Concrete Model.
			NORM ORNL – Normal ORNL.
			CRMO ORNL– 2-1/4 Cr-Mo ORNL.
			REVP ORNL – Reversed Plasticity ORNL.
			ARST ORNL – Full alpha reset ORNL.
			GEN-PLAST – Generalized Plasticity Model.
			VISCO PLAS – Viscoplastic model through subroutine UVSCPL.
16-25	3rd	A	Enter one of the following hardening rules:
			ISOTROPIC – Isotropic hardening (Default).
			KINEMATIC – Kinematic hardening.
			COMBINED – Combined (isotropic kinematic) hardening.
26-30	4th	I	Not used; enter 0.
31-35	5th	I	Enter 1 to turn on concrete cracking.
36-40	6th	I	Not used; enter 0.
41-45	7th	I	Number of viscoplastic parameters to be read through data block 5b.
46-55	8th	A	Enter the material name to cross-reference with material data base for temperature dependent properties, strain rate, and work hardening effects.

4th data block

The data entered in the following blocks should be the values at the lowest temperature expected during an analysis, not necessarily at the stress-free temperature.

1-10	1st	F	Young's modulus.
11-20	2nd	F	Poisson's ratio.
21-30	3rd	F	Mass density (stress analysis).
31-40	4th	F	Coefficient of thermal expansion.

Format Fixed	Format Free	Data Type	Entry
41-50	5th	F	Equivalent (Von Mises) tensile yield stress. (For Mohr-Coulomb behavior, this is at zero hydrostatic stress. For implicit viscoplasticity, back stress.)
51-60	6th	F	For ORNL yield criteria, equivalent 10th cycle tensile yield stress. (For Mohr-Coulomb yield criteria, α-β parameter.)
61-70	7th	F	Enter the cost per unit volume.
71-80	8th	F	Enter the cost per unit mass.

5a data block

Necessary only in a coupled thermal-stress analysis.

1-10	1st	F	Thermal conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis).
31-40	4th	F	Not used; enter 0.
41-50	5th	F	Emissivity.

5b data block

The following blocks are only used if viscoplastic material and the seventh field of data block 3 is entered. Enter 8 fields per data line.

1-10	1st	Enter the first viscoplastic parameter
11-20	2nd	Enter the second viscoplastic parameter.

6th data block

Enter a list of elements associated with this material. (Do not enter composite elements which use this material in its layers.)

■ ISOTROPIC

Define Mechanical Data for Rigid-Plastic Materials

Description

This option allows you to define material properties for a rigid-plastic material. You must also associate these material properties with a list of element numbers.

To define the dependence of these properties on temperature, use the TEMPERATURE EFFECTS model definition.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word ISOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of sets of rigid-plastic material data to follow (optional).
6-10	2nd	I	Enter the unit number for input. Defaults to input deck.

Data blocks 3, 4, 5, and 6 are repeated as a set, once for each set of isotropic material defined.

3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing TEMPERATURE EFFECTS and WORK HARD data.
46-55	8th	A	Enter the material name to cross-reference with material data base for temperature dependent properties, strain rate, and work hardening effects.

4th data block

The data entered in the following blocks should be the values at the lowest temperature expected during an analysis, not necessarily at the stress-free temperature.

1-10	1st	F	Not used; enter 0.
11-20	2nd	F	Not used; enter 0.

Format Fixed	Format Free	Data Type	Entry
21-30	3rd	F	Mass density (stress analysis).
31-40	4th	F	Coefficient of thermal expansion.
41-50	5th	F	Equivalent (von Mises) tensile yield stress.
51-60	6th	F	Not used; enter 0.
61-70	7th	F	Enter the cost per unit volume.
71-80	8th	F	Enter the cost per unit mass.

5th data block

Necessary only in a coupled thermal-stress analysis.

1-10	1st	F	Thermal conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis).
31-40	4th	F	Not used; enter 0.
41-50	5th	F	Emissivity.

6th data block

Enter a list of elements associated with this material.

■ ORTHOTROPIC

Define Mechanical Data for Orthotropic Materials

Description

This option allows you to define material properties, a yield criterion, and a strain hardening law for an orthotropic material. You can also associate these material properties with a list of element numbers.

To define the dependence of these properties on temperature, use the ORTHO TEMP model definition.

Defaults for this option are von Mises yield criterion, isotropic strain hardening law (with a slope of 0. if the WORK HARD model definition is omitted), and an equivalent yield stress of 10^{20}. Therefore, the default is a perfectly elastic nonyielding orthotropic material.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word ORTHOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of sets of ortho-tropic material data to follow.
6-10	2nd	I	Enter the unit number for input. Defaults to input deck.
Data blocks 3-8 are entered as a set, once for each data set.			
3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.); for cross-referencing ORTHO TEMP, WORK HARD data and user subroutines.
6-15	2nd	A	Enter one of the following yield criteria: VON MISES – Von Mises (Default). NORM ORNL– Normal ORNL. CRMO ORNL– 2-1/4 Cr-Mo ORNL.

Format Fixed	Format Free	Data Type	Entry
			REVP ORNL – Reversed Plasticity ORNL.
			ARST ORNL – Full alpha reset ORNL.
			GEN-PLAST – Generalized Plasicity Model.
			VISCO PLAS – Viscoplastic model through subroutine UVSCPL.
16-25	3rd	A	Enter one of the following hardening rules:
			ISOTROPIC – Isotropic hardening (Default).
			KINEMATIC – Kinematic hardening.
			COMBINED – Combined (isotropic/kinematic) hardening.
26-30	4th	I	IANELS flag, enter 1 to call user subroutines ANELAS, HOOKLW, ANPLAS, ANEXP, ANKOND, and ORIENT.
31-35	5th	I	Not used, enter 0.
36-40	6th	I	Not used, enter 0.
41-45	7th	I	Number of viscoplastic parameters to be read through data block 7b.

4th data block

Notes: Since all material properties in an orthotropic material are independent, it is your responsibility to enter all the data required to match the dimension of the stress-strain law of the elements listed for this material (See *Volume B: Element Library*, if necessary). No defaults for this data are provided by MARC.

The data entered in the following data lines should be the values at the lowest temperature expected during an analysis, not necessarily at the stress-free temperature.

These values are with respect to the user coordinate (1, 2, 3) system.

1-10	1st	F	E_{11} – Young's modulus.
11-20	2nd	F	E_{22} – Young's modulus.
21-30	3rd	F	E_{33} – Young's modulus.
31-40	4th	F	ν_{12} – Poisson's ratio.
41-50	5th	F	ν_{23} – Poisson's ratio.
51-60	6th	F	ν_{31} – Poisson's ratio.
61-70	7th	F	ρ – Mass density (stress analysis).

Format Fixed	Format Free	Data Type	Entry
5th data block			
1-10	1st	F	G_{12} – Shear modulus.
11-20	2nd	F	G_{23} – Shear modulus.
21-30	3rd	F	G_{31} – Shear modulus.
31-40	4th	F	α_{11} – Coefficients of thermal expansion.
41-50	5th	F	α_{22} – Coefficients of thermal expansion.
51-60	6th	F	α_{33} – Coefficients of thermal expansion.
61-70	7th	F	Enter the cost per unit volume.
71-80	8th	F	Enter the cost per unit mass.
6th data block			
1-10	1st	F	Equivalent (Von Mises) tensile yield stress. Default: 10^{20}. (For implicit viscoplasticity, back stress.)
11-20	2nd	F	For ORNL, 10th cycle equivalent yield stress.
21-30	3rd	F	YRDIR1 Direct stress anisotropic yield ratios of Hill.
31-40	4th	F	YRDIR2 (Default: YRDIR1=YRDIR2=YRDIR3=1.0).
41-50	5th	F	YRDIR3.
51-60	6th	F	YRSHR1 Shear stress anisotropic yield ratios of Hill.
61-70	7th	F	YRSHR2 (Default: YRSHR1=YRSHR2=YRSHR3=1.0).
71-80	8th	F	YRSHR3.
7a data block			
Necessary only in a coupled thermal-stress analysis			
1-10	1st	F	K_{11} – Thermal conductivities.
11-20	2nd	F	K_{22} – Thermal conductivities.
21-30	3rd	F	K_{33} – Thermal conductivities.
31-40	4th	F	ρ – Mass density (heat transfer analysis).
41-50	5th	F	Specific heat
51-60	6th	F	R_{11} – If Joule heating analysis, restivities.
61-70	7th	F	R_{22} – If Joule heating analysis, restivities.
71-80	8th	F	R_{33} – If Joule heating analysis, restivities.

Format Fixed	Format Free	Data Type	Entry
7b data block			
The following data blocks are only required if viscoplastic material and the seventh field, data block 3, is entered. Enter 8 fields per block.			
1-10	1st	F	Enter first viscoplastic parameter.
11-20	2nd	F	Enter second viscoplastic parameter.
8th data block			
			Enter a list of elements associated with this material. (Do not enter composite elements which use this material in its layers.).

■ ANISOTROPIC

Model Definition Block for Stress or Coupled-Thermal Stress Analysis

Description

A general temperature dependent orthotropic material model is available through the MARC input deck by the use of the ORTHOTROPIC and ORTHO TEMP options. If a more general model is needed, you can supply such a model through the user subroutines ANELAS, HOOKLW, ANEXP, ANKOND, ANPLAS, or ORIENT.

Two ways to request a call to these subroutines are shown below:

- Use the flag (3rd data block, fourth field) on the ORTHOTROPIC option to modify the material data entered there.
- Use the ANISOTROPIC model definition block to call these subroutines.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word ANISOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of anisotropic material data sets to follow.
6-10	2nd	I	Enter the unit number for input. Defaults to input deck.

Format Fixed	Format Free	Data Type	Entry
Data block 3,4, and 5 are repeated as a set NSET times.			
3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing ORTHO TEMP, WORK HARD data and user subroutines.
6-15	2nd	A	Enter one of the following yield criteria: VON MISES – Von Mises (default) NORM ORNL– Normal ORNL CRMO ORNL– 2-1.3 Cr-Mo ORNL REVP ORNL – Reversed Plasticity ORNL ARST ORNL – Full alpha reset ORNL
16-25	3rd	A	Enter one of the following hardening rules: ISOTROPIC – Isotropic hardening (default) KINEMATIC – Kinematic hardening COMBINED – Combined hardening (isotropic/kinematic)
26-30	4th	I	Enter 1 if user subroutines ANELAS, ANEXP, ANPLAS, and HOOKLW are to be called. Enter 2 if the anisotropic stress strain law, etc., is to be entered in data blocks (4a, 4b, 4c, 4d, 4e, 4f).
4th data block			
1-10	1st	F	Mass density (stress analysis)
11-20	2nd	F	Equivalent (von Mises) yield stress
21-30	3rd	F	If ORNL yielding, 10th cycle yield stress
31-40	4th	F	Mass density (heat transfer analysis)
41-50	5th	F	Specific Heat
51-60	6th	I	Leave blank.
61-70	7th	F	Cost of material per unit volume (optional).
71-80	8th	F	Cost of material per unit mass (optional).

Format Fixed	Format Free	Data Type	Entry

Data blocks 4a, 4b, and 4c used to define anisotropic elastic stress strain relation. Data block 4a only required if the fourth field is a "2".

4a data block

1-10	1st	F	C_{11}
11-20	2nd	F	C_{12}
21-30	3rd	F	C_{13}
31-40	4th	F	C_{14}
41-50	5th	F	C_{15}
51-60	6th	F	C_{16}
61-70	7th	F	C_{22}
71-80	8th	F	C_{23}

Data block 4b only required if the fourth field is a "2".

4b data block

1-10	1st	F	C_{24}
11-20	2nd	F	C_{25}
21-30	3rd	F	C_{26}
31-40	4th	F	C_{33}
41-50	5th	F	C_{34}
51-60	6th	F	C_{35}
61-70	7th	F	C_{36}
71-80	8th	F	C_{44}

Format Fixed	Format Free	Data Type	Entry

Data block 4c only required if the fourth field is a "2"

4c data block

1-10	1st	F	C_{45}
11-20	2nd	F	C_{46}
21-30	3rd	F	C_{55}
31-40	4th	F	C_{56}
41-50	5th	F	C_{66}

Use only as many terms as are required for the element type chosen. All three blocks must be used. For plane stress only C_{11}, C_{12}, C_{13}, C_{22}, C_{24}, C_{33} must be entered.

Data block 4d is only required if the fourth field is a "2". It defines the anisotropic thermal expansion coefficients.

4d data block

1-10	1st	F	α_{11}
11-20	2nd	F	α_{12}
21-30	3rd	F	α_{13}
31-40	4th	F	α_{22}
41-50	5th	F	α_{23}
51=6-	6th	F	α_{33}

Data block 4e is only required if the fourth field is a "2". It defines the anisotropic plasticity.

4e data block

1-10	1st	F	YRDIR1	Direct stress anisotropic yield ratios.
11-20	2nd	F	YRDIR2	
21-30	3rd	F	YRDIR3	
31-40	4th	F	YRSHR1	Shear stress anisotropic yield ratios.
41-50	5th	F	YRSHR2	
51-60	6th	F	YRSHR3	

Format Fixed	Format Free	Data Type	Entry
Data block 4f only required if the fourth field is a "2" on a coupled analysis.			
4f data block			
1-10	1st	F	K_{11}
11-20	2nd	F	K_{12}
21-30	3rd	F	K_{13}
31-40	4th	F	K_{22}
41-50	5th	F	K_{23}
51-60	6th	F	K_{33}
5th data block			
			Enter a list of elements associated with this material. (Do not enter composite elements which use this material in its layers.)

■ HYPOELASTIC

Define Data for Hypoelastic Materials

Description

This option allows you to input data associated with MARC's hypoelastic material model. You must define the material stress/strain law through user subroutine HYPELA or HYPELA2 (or, for element type 52 or 98, user subroutine UBEAM).

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word HYPOELASTIC.
2nd data block			
1-5	1st	I	Enter the number of hypoelastic material data sets to follow.
6-10	2nd	I	Enter the unit number for input. Defaults to input deck.
Data block 3, 4, 5 are repeated as a set, once for each data set.			
3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing TEMPERATURE EFFECTS data and user subroutines.
6-10	2nd	I	Enter 1 to call user subroutines ANEXP and ORIENT.
11-15	3rd	I	Flag to use user subroutine HYPELA2. Enter 1 to pass in deformation gradient (F) and rotation (R). Enter 2 to pass in deformation gradient (F) and stretch ratios (λ). Enter 3 to pass in F, R, and λ.

Format Fixed	Format Free	Data Type	Entry
4th data block			
1-10	1st	F	ρ mass density (stress analysis).
11-20	2nd	F	α coefficient of thermal expansion.
21-30	3rd	F	K thermal conductivity.
31-40	4th	F	Specific heat.
41-50	5th	F	Resistivity.
51-60	6th	F	ρ mass density (heat transfer analysis).
			Note: In a coupled thermal-stress analysis, the thermal material model defaults to isotropic heat transfer behavior.
5th data block			
			Enter a list of elements using this material model. (Do not enter composite elements using this material in a layer.)

■ MOONEY

Define Data for Mooney-Rivlin Materials

Description

This option allows you to enter all the material data for a Mooney-Rivlin rubber material. User subroutine UMOONY can be used to enter temperature dependent coefficients. User subroutine UENERG can be used to enter a general strain energy function. If rate effects are also present, VISCELMOON can also be required. A list of elements can also be associated with this material.

Notes: For a Neo-Hookean material model, only C_{10} is needed.

For a Mooney/Rivlin material model, only C_{10} and C_{01} are needed.

For the full 3rd-order invariant model of Jamus, Green and Simpson, use all C_{10}, C_{01}, C_{11}, C_{20}, C_{30}.

The procedure used for the Mooney formulation is defined on the ELASTICITY parameter. If the total Lagrange formulation is invoked, the elements in this case must be of the Herrmann formulation except for plane stress. If the updated Lagrange formulation is invoked, the elements must be conventional displacement formulation. (Near-incompressibility is imposed using mixed approach and condensing out pressure degrees of freedom.) For plane stress, displacement elements are always used.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word MOONEY.
2nd data block			
1-5	1st	I	Enter the number of data sets to follow.
6-10	2nd	I	Unit number for data input. Defaults to input deck.
Data block 3, 4, and 5 are repeated as a set, once for each data set.			
3rd data block			
1-5	1st	I	Material identification number.
4th data block			
1-10	1st	F	C_{10} – Mooney-Rivlin constant.
11-20	2nd	F	C_{01} – Mooney-Rivlin constant.
21-30	3rd	F	ρ – mass density (stress analysis).
31-40	4th	F	α – coefficient of thermal expansion.

Format		Data	
Fixed	**Free**	**Type**	**Entry**
41-50	5th	F	C_{11} – higher order constants
51-60	6th	F	C_{20} – higher order constants.
61-70	7th	F	C_{30} – higher order constants.

5th data block

(Only necessary in a coupled thermal-stress analysis)

1-10	1st	F	Conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis).

Note: In a coupled thermal-stress analysis, the thermal material model defaults to isotropic heat transfer behavior.

6th data block

Enter a list of element numbers associated with this particular elastomeric material.

Notes: These material identifications cannot be referenced by any composite group.

The values C_{10}, C_{01}, C_{11}, C_{20}, and C_{30} can be redefined using the subroutine UMOONY.

Although a general strain energy function can be defined by using the subroutine UENERG, it is still required to define the elements associated with the material identifier here.

■ OGDEN

Define Data for Ogden or Principal Stretch Based Material Model

Description

This option allows you to define the data associated with the Ogden model for incompressible and nearly incompressible rubber material. The strain energy function for this model has the form:

$$W = \sum_{n=1}^{N} \frac{\mu_n}{\alpha_n}[J^{-\alpha_n/3}(\lambda_1^{\alpha_n} + \lambda_2^{\alpha_n} + \lambda_3^{\alpha_n}) - 3] + 4.5K(J^{1/3} - 1)^2$$

This option can also be used to activate the general principal stretch based models through user subroutine UPSTRECH and UOGDEN.

Note: The procedure used for the Ogden model is defined on the ELASTICITY parameter. If the total Lagrange formulation is invoked, the elements in this case must be of the Herrmann formulation except for plane stress. If the updated Lagrange formulation is invoked, the elements must be conventional displacement formulation. (Near-incompressibility is imposed using mixed approach and condensing out pressure degrees of freedom.) For plane stress, displacement elements are always used.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word OGDEN.
2nd data block			
1-5	1st	I	Enter the number of sets of Ogden material data to follow (optional).
6-10	2nd	I	Enter the logical unit number for input. Defaults to input deck.

Format Fixed	Format Free	Data Type	Entry
Data block 3, 4, 5, 6, and 7 are repeated for each data set.			
3rd data block			
1-5	1st	I	Enter the material identification.
6-10	2nd	I	Enter the number of terms (N) that defines the strain energy function.
11-15	3rd	I	Enter 1 for Ogden model (default). Enter 2 for generalized principal stretch based model.
4th data block			
1-10	1st	F	Enter the bulk modulus (K), default is such that material is incompressible.
11-20	2nd	F	Enter the mass density.
21-30	3rd	F	Enter the coefficient of thermal expansion.
5th data block			
Only necessary in a coupled thermal-stress analysis.			
1-10	1st	F	Conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis). **Note:** In a coupled thermal-stress analysis, the thermal material model defaults to isotropic heat transfer behavior.
Data block 6 is repeated once for each term specified in the 3rd data blocks. Not used if generalized stretch based model.			
6th data block			
1-10	1st	F	Enter the modulus.
11-20	2nd	F	Enter the power.
7th data block			
			Enter a list of element numbers associated with this particular elastomeric material.

■ FOAM

Define Data for Foam Material Model

Description

This option allows you to define the data associated with the foam model for highly compressible rubber material. The foam model can be used for plane strain, axisymmetric, and solid elements using the conventional displacement elements.

The strain energy function for this model has the form.

$$W = \sum_{n=1}^{N} \frac{\mu_n}{\alpha_n}(\lambda_1^{\alpha_n} + \lambda_2^{\alpha_n} + \lambda_3^{\alpha_n} - 3) + \sum_{n=1} \frac{\mu_n}{\beta_n}(1 - J^{\beta_n})$$

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word FOAM.
2nd data block			
1-5	1st	I	Enter the number of sets of Foam material data to follow (optional).
6-10	2nd	I	Enter the logical unit number for input. Defaults to input deck.

Data block 3, 4, 5, 6, and 7 are repeated for each data set.

3rd data block			
1-5	1st	I	Enter the material identification.
6-10	2nd	I	Enter the number of terms (N) that defines the strain energy function.
4th data block			
1-10	1st	F	Not used; enter 0
11-20	2nd	F	Enter the mass density.
21-30	3rd	F	Enter the coefficient of thermal expansion.

Format Fixed	Format Free	Data Type	Entry
5th data block			
(Only necessary in a coupled thermal-stress analysis)			
1-10	1st	F	Conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis).
			Note: In a coupled thermal-stress analysis, the thermal material model defaults to isotropic heat transfer behavior.
Data block 6 is repeated once for each term specified in the 3rd data blocks.			
6th data block			
1-10	1st	F	Enter the modulus (μ_n).
11-15	2nd	F	Enter the power for deviatoric behavior (α_n).
21-30	3rd	F	Enter the power for volumetric behavior (β_n).
7th data block			
			Enter a list of element numbers associated with this particular elastomeric material.

Note: If the bulk modulus is entered, then $\beta i = 0$ for all values of i.
If the bulk modulus is zero and all βi are 0, then the material is treated as an Ogden material.

■ STRAIN RATE

Define Strain Rate Dependent Yield Stress

Description

This option allows the definition of a strain rate dependent yield stress, for use in dynamic and flow (for example, extrusion) problems. This can also be used in static analysis by entering a fictitious time using the TIME STEP option. The zero strain rate yield stress is given on the ISOTROPIC, ORTHOTROPIC, and ANISOTROPIC options. This block must be repeated for each different material for which strain rate data is necessary. The yield stress variation with strain rate is given using one of two options:

A. The breakpoints and slopes for a piecewise linear approximation to the yield stress strain rate curve are given. The strain rate breakpoints should be in ascending order, or

B. The yield stress and stain rate data points lying on the yield stress, strain rate curve are input directly. The data is entered in ascending order of strain rate. This method is flagged by entering the word DATA on the first data block.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-11	1st	A	Enter the words STRAIN RATE.
13-80	2nd	A	Enter the word DATA to indicate that option B is being used.
2nd data block			
1-5	1st	I	For option A, enter the number of slopes of yield versus strain rate curve. For option B, enter the number of data points.
6-10	2nd	I	Material type identification (1,2,3...) for cross-reference to ISOTROPIC, ORTHOTROPIC, and ANISOTROPIC blocks.
11-15	3rd	I	Unit number for input of the set of this block. Defaults to blocks.

Format		Data	
Fixed	Free	Type	Entry
3a data block			
Data block 3a is used in conjunction with option A. The number of blocks in this series is equal to that given in the first field of data block 2.			
1-10	1st	F	Enter the slope of the yield versus strain rate curve.
11-20	2nd	F	Enter the strain rate value above which the above slope becomes operational. Note, the first strain rate breakpoint must be zero.
3b data block			
Data block 3b is used in conjunction with option B.			
1-10	1st	F	Enter the value of the yield strength.
11-20	2nd	F	Enter the associated strain rate. Note that the first strain rate must be zero.

■ WORK HARD

Define Workhardening Data

Description

This block allows you to specify the material stress-strain relation for elastic-plastic behavior. Further details on this block are given in *Volume A: Theory and User Information*. The workhardening data can be entered in one of three forms.

A. The breakpoints and slopes for a piecewise linear approximation to the stress-strain curves are given. The piecewise linear curve is entered in ascending order of equivalent plastic strain.

B. The stress and plastic strain data points lying on the stress-strain curve are input directly. The data is entered in ascending order of plastic strains. This method is flagged by entering the word DATA on the WORK HARD option. These data points are used to calculate slope breakpoint data.

C. By user subroutine WKSLP. This routine is called for every integration point where elastic-plastic behavior occurs. See *Volume D: User Subroutines and Special Routines* for details. Note that if this option is used, it must be used for ALL material types.

This block must be repeated for each different material for which workhardening data is necessary.

Note: When performing a small deformation analysis without the LARGE DISP, UPDATE, FINITE, or PLASTICITY (3, 4, 5, or 6) parameter, the work hard data should be given in terms of engineering stress and engineering strain.

If a large displacement analysis is performed where the LARGE DISP parameter is included but without either the UPDATE, FINITE, or PLASTICITY (3, 4, 5, or 6) parameter, you should use the second Piota-Kirchhoff stress and the Green Lagrange strain.

If either the UPDATE or PLASTICITY (3, 4, 5, or 6) parameter is included, you should enter the work hard information in terms of true stress and true strain.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-9	1st	A	Enter the words WORK HARD.
11-80	2nd	A	Enter the word DATA to indicate that option B is being used.
2nd data block			
1-5	1st	I	For option A enter number of slopes of the workhardening curve. For option B, enter the number of data points. For option C, enter -1.
6-10	2nd	I	This is the same in the first field except it is for the data associated with the 10th cycle yield used in the ORNL constitutive theory.
11-15	3rd	I	Material type identification (1,2,3 etc.) for cross-referencing with the ISOTROPIC, ORTHOTROPIC, and ANISOTROPIC blocks.
16-20	4th	I	Enter unit number for input of workhardening data, defaults to input.

Data blocks 3a and 4a are used in conjunction with Option A.

3a data block

The number of blocks entered is equal to the number of slopes entered above.

Included only if the first field of 2nd data blocks > 0.

1-15	1st	F	Enter the slope of the workhardening curve.
16-30	2nd	F	Enter the breakpoint when the slope becomes operative. The breakpoint and slope data should be described in ascending order of the equivalent plastic strain, the first slope starting at zero plastic strain.

Note: The workhardening slope should be for a uniaxial tension specimen, and is the change in stress per unit of plastic strain, not per unit of total strain. See *Volume A: Theory and User Information.*

Format Fixed	Format Free	Data Type	Entry
4a data block			
Included only if the first field of 2nd data line > 0. Then, number of blocks is equal to that number.			
1-15	1st	F	Slope of 10th cycle workhardening curve (stress change per plastic strain change).
16-30	2nd	F	Breakpoint when above slope becomes operative. First breakpoint should be at zero plastic strain.
Data block 3b and 4b are used in conjunction with Option B.			
3b data block			
The number of blocks entered is equal to the number of data points entered above.			
1-15	1st	F	Enter the equivalent stress.
16-30	2nd	F	Enter the equivalent plastic strain. The data should be described in ascending order of equivalent plastic strain; the first data set starting at zero plastic strain.
4b data block			
Included only if the first field of 2nd data line > 0. Then, number of data lines is equal to that number.			
1-15	1st		Enter the equivalent stress associated with the 10th cycle work-hardening curve.
16-30	2nd		Enter the equivalent plastic strain.

■ TEMPERATURE EFFECTS

Define Effects of Temperature

Description

This data block defines the variation of element properties (Young's modulus, yield stress, Poisson's ratio, and coefficient of thermal expansion) with temperature. The values read in through either the ISOTROPIC or POWDER option are those at the lowest temperature specified. Properties are not defined below the lowest temperature. The temperature dependency can be entered using one of the following two options:

- The variation of a particular property with temperature is specified as a piecewise linear curve. Breakpoints must be given in ascending order of temperature.
- The particular property value and temperature lying on the relevant curve are input directly. Data points must be given in increasing order of temperature. This option is flagged by entering the word DATA on the first data line.

Note: For Mooney materials, the temperature dependence for C10 and C01 can be defined by replacing C10 for "Young's modulus" and C01 for "Poisson's ratio". The other constants can be specified by utilizing user subroutine UMOONY.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-19	1st	A	Enter the words TEMPERATURE EFFECTS.
21-80	2nd	A	Enter the word DATA to indicate that option B is used.

For option A, use data blocks 2a, 3a, 4a, 5a, 6a, 7a and 8a. For option B, use data blocks 2b, 3b, 4b, 5b, 6b, 7b and 8b, below.

Format Fixed	Format Free	Data Type	Entry

Option A

2a data block

1-5	1st	I	Number of slopes of yield stress versus temperature curve.
6-10	2nd	I	Number of slopes of Young's modulus versus temperature curve.
11-15	3rd	I	Number of slopes of Poisson's ratio versus temperature curve
16-20	4th	I	Number of slopes for instantaneous coefficient of thermal expansion versus temperature.
21-25	5th	I	Number of slopes of 10th cycle yield stress versus temperature curve of ORNL constitutive theory option, or for powder materials, number of slopes of viscosity versus temperature.
26-30	6th	I	Number of slopes of the workhardening versus temperature curve.
31-35	7th	I	Material type identification (1,2,3,...) for cross-referencing the ISOTROPIC blocks.
36-40	8th	I	Logical unit number for input of this set of data. Defaults to data lines.

3a data block

The number entered in the first field of data line 2 defines the number of data lines in data block 3.

1-15	1st	F	Enter the slope of yield stress versus the temperature curve.
16-30	2nd	F	Enter the temperature at which the above slope becomes operative.

Format		Data	
Fixed	Free	Type	Entry
4a data block			
The number entered in the second field of data line 2 defines the number of data lines required in data block 4.			
1-15	1st	F	Enter the slope of Young's modulus versus the temperature curve.
16-30	2nd	F	Enter the temperature at which the above slope becomes operative.
5a data block			
The number in the third field of data line 2 defines the number of data lines required in data block 5.			
1-15	1st	F	Enter the slope of Poisson's ratio versus the temperature curve.
16-30	2nd	F	Enter the temperature at which the above slope becomes operative.
6a data block			
The number in the fourth field of data line 2 defines the number of data lines required in data block 6.			
1-15	1st	F	Enter the slope of instantaneous coefficient of thermal expansion.
16-30	2nd	F	Enter the temperature at which the above slope becomes operative. These are instantaneous values.
7a data block			
Slopes/breakpoints for 10th cycle yield or viscosity versus temperature curve. The number in the fifth field of data block 2 defines the number of data lines required in data block 7.			
1-15	1st	F	Slope of 10th cycle yield or viscosity versus temperature curve.
16-30	2nd	F	Temperature at which this slope becomes operative.

Format Fixed	Format Free	Data Type	Entry
8a data block			
Slopes and breakpoints of the curve describing the ratio of the workhardening curve at temperature to the workhardening curve at the first breakpoint of this set; that is, $H(\bar{\varepsilon}^p, \underline{T}_o) \bullet R(T)$ is the workhardening slope at T given in terms of $H(\bar{\varepsilon}^p, \underline{T}_o)$ the workhardening slope at plastic strain $\bar{\varepsilon}^p$ and the first breakpoint of this set, T_o, and R(T), the ratio parameter. In these data lines $\frac{dR}{dT}$ the slope of the ratio curve, is input. The ratio R(T) is not dependent on $\bar{\varepsilon}^p$, only on T. The number in the sixth field of data block 2 defines the number of lines required in data block 8.			
1-15	1st	F	Slope of ratio of workhardening slope versus temperature curve.
16-30	2nd	F	Temperature at which this slope becomes operative.

Option

2b data block			
1-5	1st	I	Number of data points on the yield stress versus temperature curve.
6-10	2nd	I	Number of data points on the Young's modulus versus temperature curve.
11-15	3rd	I	Number of data points on the Poisson's ratio versus temperature curve.
16-20	4th	I	Number of data points on the instantaneous coefficient of thermal expansion versus temperature.
21-25	5th	I	Number of data points on the 10th cycle yield stress versus temperature curve of ORNL constitutive theory option, or number of data points on the viscosity versus temperature curve for powder materials.
26-30	6th	I	Number of data points on the workhardening versus temperature curve.
31-35	7th	I	Material type identification (1,2,3,...) for cross-referencing the ISOTROPIC blocks.
36-40	8th	I	Logical unit number for input of this set of data. Defaults to data lines.

Format Fixed	Format Free	Data Type	Entry
3b data block			
The number entered in the first field of data line 2 defines the number of data lines in data block 3.			
1-15	1st	F	Enter the value of the yield stress.
16-30	2nd	F	Enter the associated temperature.
4b data block			
The number entered in the second field of data line 2 defines the number of data lines required in data block 4.			
1-15	1st	F	Enter the value of the Young's modulus.
16-30	2nd	F	Enter the associated temperature.
5b data block			
The number in the third field of data line 2 defines the number of data lines required in data block 5.			
1-15	1st	F	Enter the value of the Poissons's ratio.
16-30	2nd	F	Enter the associated temperature.
6b data block			
The number in the fourth field of data line 2 defines the number of data lines required in data block 6.			
1-15	1st	F	Enter the value of the instantaneous coefficient of thermal expansion.
16-30	2nd	F	Enter the associated temperature.
7b data block			
The number in the fifth field of data line 2 defines the number of data lines required in data block 7.			
1-15	1st	F	Enter the value of the 10th cycle yield or viscosity.
16-30	2nd	F	Enter the associated temperature.

Format		Data	
Fixed	Free	Type	Entry

8b data block

Data points on the curve describing the ratio of the workhardening curve at a given temperature to the workhardening curve at the first temperature of this set; that is, $H(\bar{\varepsilon}^p, \underline{T}_0) \bullet R(T)$ is the workhardening slope at plastic strain, $\bar{\varepsilon}^p$, and the first temperature of this set, T, and R(T), the ratio parameter.

The same temperature effects are applied for all values of $\bar{\varepsilon}^p$; that is, the ratio R(T) is not dependent on $\bar{\varepsilon}^p$, only on T. The number in the sixth field of data block 2 defines the number of data lines required in data block 8.

1-15	1st	F	Enter the value of the ratio of the workhardening slope vs. the temperature curve, *R(T)*.
16-30	2nd	F	Enter the associated temperature.

Note: In calculating a particular temperature dependent property, the program averages the value of this property at the start and at the end of the increment. Also, a first order correction is made for the effect of temperature change on the total elastic stresses because of changes in material properties. Even in the elastic case, you should not expect a simple calculation with property values corresponding to the temperature at the end of the increment. More details are given on the discussion of temperature-dependent plasticity in *Volume A: Theory and User Information*.

■ TEMPERATURE EFFECTS

Temperature Effects in Coupled Thermal-Stress Analysis

Description

This data block defines the variation of element properties (Young's modulus, yield stress, Poisson's ratio, and coefficient of thermal expansion) with temperature. The values read in through either the ISOTROPIC or POWDER options are those at the lowest temperature specified. Properties are not defined below the lowest temperature. The temperature dependency can be entered using one of the following two options:

- The variation of a particular property with temperature is specified as a piecewise linear curve. Breakpoints must be given in ascending order of temperature.
- The particular property value and temperature lying on the relevant curve are input directly. Data points must be given in increasing order of temperature. This option is flagged by entering the word DATA on the first data line.

Note: For Mooney materials, the temperature dependence for C10 and C01 can be defined by replacing C10 for "yield stress" and C01 for "Young's modulus". The other constants can be specified by utilizing user subroutine UMOONY.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-19	1st	A	Enter the words TEMPERATURE EFFECTS.
21-80	2nd	A	Enter the word DATA to indicate that option B is used.

For option A, use data blocks 2a, 3a, 4a, 5a, 6a, 7a and 8a. For option B, use data blocks 2b, 3b, 4b, 5b, 6b, 7b and 8b, below.

Format Fixed	Format Free	Data Type	Entry

Option A

2a data block

1-5	1st	I	Number of slopes of yield stress versus temperature curve.
6-10	2nd	I	Number of slopes of Young's modulus versus temperature curve.
11-15	3rd	I	Number of slopes of Poisson's ratio versus temperature curve.
16-20	4th	I	Number of slopes for instantaneous coefficient of thermal expansion versus temperature.
21-25	5th	I	Number of slopes of 10th cycle yield stress versus temperature curve of ORNL constitutive theory option, or for powder materials, number of slopes of viscosity versus temperature.
26-30	6th	I	Number of slopes of the workhardening versus temperature curve.
31-35	7th	I	Number of slopes of conductivity versus temperature curve
36-40	8th	I	Number of slopes of specific heat versus temperature curve.
41-45	9th	I	Number of latent heats to be entered.
46-50	10th	I	Number of slopes of emissivity versus temperature curve.
31-35	11th	I	Material type identification (1,2,3,...) for cross-referencing the ISOTROPIC block.
36-40	12th	I	Logical unit number for input of this set of data. Defaults to blocks.

Format Fixed	Format Free	Data Type	Entry
3a data block			
The number entered in the first field of data line 2 defines the number of data lines in data block 3.			
1-15	1st	F	Enter the slope of yield stress versus the temperature curve.
16-30	2nd	F	Enter the temperature at which the above slope becomes operative.
4a data block			
The number entered in the second field of data line 2 defines the number of data lines required in data block 4.			
1-15	1st	F	Enter the slope of Young's modulus versus the temperature curve.
16-30	2nd	F	Enter the temperature at which the above slope becomes operative.
5a data block			
The number in the third field of data line 2 defines the number of data lines required in data block 5.			
1-15	1st	F	Enter the slope of Poisson's ratio versus the temperature curve.
16-30	2nd	F	Enter the temperature at which the above slope becomes operative.
6a data block			
The number in the fourth field of data line 2 defines the number of data lines required in data block 6.			
1-15	1st	F	Enter the slope of instantaneous coefficient of thermal expansion.
16-30	2nd	F	Enter the temperature at which the above slope becomes operative. These are instantaneous values.

Format Fixed	Format Free	Data Type	Entry

7a data block

Slopes/breakpoints for 10th cycle yield or viscosity versus temperature curve. The number in the fifth field of data line 2 defines the number of data lines required in data block 7.

1-15	1st	F	Enter the slope of 10th cycle yield stress or viscosity versus temperature curve.
16-30	2nd	F	Enter the temperature at which this slope becomes operative.

8a data block

Slopes and breakpoints of the curve describing the ratio of the workhardening curve at temperature to the workhardening curve at the first breakpoint of this set; that is, $H(\bar{\varepsilon}^p, \underline{T}_0) \bullet R(T)$ is the workhardening slope at T given in terms of $H(\bar{\varepsilon}^p, \underline{T}_0)$ the workhardening slope at plastic strain $\bar{\varepsilon}^p$ and the first breakpoint of this set, T_0, and R(T), the ratio parameter. In these data lines $\frac{dR}{dT}$ the slope of the ratio curve, is input.

The ratio R(T) is not dependent on $\bar{\varepsilon}^p$, only on T. The number in the sixth field of data line 2 defines the number of data lines required in data block 8.

1-15	1st	F	Enter the slope of ratio of workhardening slope versus temperature curve.
16-30	2nd	F	Enter the temperature at which this slope becomes operative.

9a data block

Conductivity variation. Number of data lines as given on data line 2, seventh field.

1-15	1st	F	Enter the slope of conductivity versus temperature curve.
16-30	2nd	F	Enter the temperature at which the above slope becomes operative.

10a data block

Specific heat variation. Number of data lines as given on data line 2, eighth field.

1-15	1st	F	Enter the slope of specific heat versus temperature curve.
16-30	2nd	F	Temperature above which this slope becomes operative.

Format Fixed	Format Free	Data Type	Entry
11a data block			
Latent heat. Number of data lines given on data line 2, ninth field.			
1-15	1st	F	Enter the latent heat.
16-30	2nd	F	Enter the solidus temperature (lower phase change limit).
31-45	3rd	F	Enter the liquidus temperature (upper phase change limit).

Option B

2b data block

1-5	1st	I	Number of data points on the yield stress versus temperature curve.
6-10	2nd	I	Number of data points on the Young's modulus versus temperature curve.
11-15	3rd	I	Number of data points on the Poisson's ratio versus temperature curve.
16-20	4th	I	Number of data points on the instantaneous coefficient of thermal expansion versus temperature.
21-25	5th	I	Number of data points on the 10th cycle yield stress versus temperature curve of ORNL constitutive theory option, or number of data points on the viscosity versus temperature curve for powder materials.
26-30	6th	I	Number of data points on the workhardening versus temperature curve.
31-35	7th	I	Number of data points on the conductivity versus temperature curve.
36-40	8th	I	Number of data points on the specific heat versus temperature curve.
41-45	9th	I	Number of latent heats to be entered.
46-50	10th	I	Number of data points on the emissivity versus temperature curve.

Format Fixed	Format Free	Data Type	Entry
51-55	11th	I	Material type identification (1,2,3,...) for cross-referencing the ISOTROPIC blocks.
56-60	12th	I	Logical unit number for input of this set of data. Defaults to data lines.

3b data block

The number entered in the first field of data line 2 defines the number of data lines in data block 3.

1-15	1st	F	Enter the value of the yield stress.
16-30	2nd	F	Enter the associated temperature.

4b data block

The number entered in the second field of data line 2 defines the number of data lines required in data block 4.

1-15	1st	F	Enter the value of the Young's modulus.
16-30	2nd	F	Enter the associated temperature.

5b data block

The number in the third field of data line 2 defines the number of data lines required in data block 5.

1-15	1st	F	Enter the value of the Poissons's ratio.
16-30	2nd	F	Enter the associated temperature.

6b data block

The number in the fourth field of data line 2 defines the number of data lines required in data block 6.

1-15	1st	F	Enter the value of the instantaneous coefficient of thermal expansion.
16-30	2nd	F	Enter the associated temperature.

7b data block

The number in the fifth field of data line 2 defines the number of data lines required in data block 7.

1-15	1st	F	Enter the value of the 10th cycle yield or viscosity.
16-30	2nd	F	Enter the associated temperature.

Format Fixed	Format Free	Data Type	Entry

8b data block

Data points on the curve describing the ratio of the workhardening curve at a given temperature to the workhardening curve at the first temperature of this set; that is, $H(\bar{\varepsilon}^p, \underline{T}_0) \bullet R(T)$ is the workhardening slope at plastic strain, $\bar{\varepsilon}^p$, and the first temperature of this set, T, and R(T), the ratio parameter.

1-15	1st	F	Enter the value of the ratio of the workhardening slope vs. the temperature curve, *R(T)*.
16-30	2nd	F	Enter the associated temperature.

Note: The same temperature effects are applied for all values of $\bar{\varepsilon}^p$, that is, the ratio *R(T)* is not dependent on $\bar{\varepsilon}^p$, only on *T*. The number in the sixth field of data line 2 defines the number of data lines required in data block 8.

9b data block

Conductivity variation. Number of data lines as given on data line 2, seventh field.

1-15	1st	F	Enter the value of the conductivity.
16-30	2nd	F	Enter the associated temperature.

10b data block

Specific heat variation. Number of data lines as given on data line 2, eighth field.

1-15	1st	F	Enter the value of the specific heat.
16-30	2nd	F	Enter the associated temperature.

11b data block

Latent heat. Number of data lines as given on data line 2, ninth field.

1-15	1st	F	Enter the value of the latent heat.
16-30	2nd	F	Enter the solidus temperature (lower phase change limit).
31-45	3rd	F	Enter the liquidus temperature (upper phase change limit).

Format		Data	
Fixed	Free	Type	Entry

12b data block

Emissivity variation. Number of data lines as given on data line 2, tenth field.

1-15	1st	F	Enter the value of the emissivity.
16-30	2nd	F	Enter the associated temperature.

Note: In calculating a particular temperature dependent property, the program averages the value of this property at the start and at the end of the increment. Also, a first order correction is made for the effect of temperature change on the total elastic stresses because of changes in material properties. Even in the elastic case, you should not expect a simple calculation with property values corresponding to the temperature at the end of the increment. More details are given on the discussion of temperature-dependent plasticity in *Volume A: User Information.*

■ ORTHO TEMP

Define Temperature Effects for Orthotropic Materials

Description

This data block defines the variation of all orthotropic material properties with temperature. Note that the values read in through the ORTHOTROPIC model definition block are those at the lowest temperature defined. Properties at temperatures below this temperature are defined to be equal to properties at this temperature.

The variation of a particular property is defined as a piecewise linear curve. Two options are available to define this curve.

A. Slope/breakpoint data in ascending order of temperature can be given.

B. Property value/temperature data in ascending order of temperature can be given. This option is flagged by entering the word DATA after the string ORTHO TEMP on data block 1.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words ORTHO TEMP.
12-15	2nd	A	Enter the word DATA to indicate that Option B defined above is to be used. (There must be a space or comma between ORTHO TEMP and DATA.)

For Option A, use the 2a-22a data blocks. For Option B, use the 2b-22b data blocks.

Option A

2a data block			
1-5	1st	I	Number of slopes of yield vs. temperature curve.
6-10	2nd	I	Number of slopes of E_{11} vs temperature curve.
11-15	3rd	I	Number of slopes of E_{22} vs temperature curve. Enter -1 to have (E_{22} vs. temp.) ≡ (E_{11} vs. temp.).

Format Fixed	Format Free	Data Type	Entry
16-20	4th	I	Number of slopes of E_{33} vs. temperature curve. Enter -1 to have (E_{33} vs. temp.) ≡ (E_{11} vs. temp.).
21-25	5th	I	Number of slopes of ν_{12} vs. temperature curve.
26-30	6th	I	Number of slopes of ν_{23} vs. temperature curve. Enter -1 to have (ν_{23} vs. temp.) ≡ (ν_{12} vs. temp.).
31-35	7th	I	Number of slopes of ν_{31} vs. temperature curve. Enter -1 to have (ν_{31} vs. temp.) ≡ (ν_{12} vs. temp.)
36-40	8th	I	Number of slopes of G_{12} vs. temperature curve.
41-45	9th	I	Number of slopes of G_{23} vs. temperature curve. Enter -1 to have (G_{23} vs. temp.) ≡ (G_{12} vs. temp.).
46-50	10th	I	Number of slopes of G_{31} vs. temperature curve. Enter -1 to have (G_{31} vs. temp.) ≡ (G_{12} vs. temp.).
51-55	11th	I	Number of slopes of α_{11} vs. temperature curve.
56-60	12th	I	Number of slopes of α_{22} vs. temperature curve. Enter -1 to have (α_{22} vs. temp.) ≡ (α_{11} vs. temp.).
61-65	13th	I	Number of slopes of α_{33} vs. temperature curve. Enter -1 to have (α_{33} vs. temp.) ≡ (α_{11} vs. temp.).
66-70	14th	I	Number of slopes of the workhardening vs. temperature curve.
71-75	15th	I	Enter the material identification for this data set.
76-80	16th	I	Enter the unit number for input of this data. Defaults to input deck.

3a data block

Include this data block only in a coupled thermal-stress analysis.

1-5	1st	I	Number of slopes of K_{11} vs. temperature curve.
6-10	2nd	I	Number of slopes of K_{22} vs. temperature curve. Enter -1 to have (K_{22} vs. temp.) ≡ (K_{11} vs. temp.).
11-15	3rd	I	Number of slopes of K_{33} vs. temperature curve. Enter -1 to have (K_{33} vs. temp.) ≡ (K_{11} vs. temp.).

Format Fixed	Format Free	Data Type	Entry
16-20	4th	I	Number of slopes of specific heat vs. temperature curve.
21-25	5th	I	Number of latent heats.

4a data block

The number of data lines in this block is the number in data block 2a, first field.

1-15	1st	F	Enter the slope of yield vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

5a data block

The number of data lines in this block is the number in data block 2a, second field.

1-15	1st	F	Enter the slope of E_{11} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

6a data block

The number of data lines in this block is n, the number in data block 2a, third field, or 0 if n = -1.

1-15	1st	F	Enter the slope of E_{22} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

7a data block

The number of data lines in this block is n, the number in data block 2a, fourth field, or 0 if n = -1.

1-15	1st	F	Enter the slope of E_{33} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

8a data block

The number of data lines in this block is the number in data block 2a, fifth field.

1-15	1st	F	Enter the slope of ν_{12} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

9a data block

The number of data lines in this block is n, the number in data block 2a, sixth field, or 0 if n = -1.

1-15	1st	F	Enter the slope of ν_{23} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

Format Fixed	Format Free	Data Type	Entry
10a data block			
The number of data lines in this block is n, the number in data block 2a, seventh field, or 0 if n = -1.			
1-15	1st	F	Enter the slope of v_{31} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.
11a data block			
The number of data lines in this block is the number in data block 2a, eighth field.			
1-15	1st	F	Enter the slope of G_{12} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.
12a data block			
The number of data lines in this block is n, the number in data block 2a, ninth field, or 0 if n = -1.			
1-15	1st	F	Enter the slope of G_{23} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.
13a data block			
The number of data lines in this block is n, the number in data block 2a, tenth field, or 0 if n = -1.			
1-15	1st	F	Enter the slope of G_{31} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.
14a data block			
The number of data lines in this block is the number in data block 2a, eleventh field.			
1-15	1st	F	Enter the slope of α_{11} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

Format Fixed	Format Free	Data Type	Entry
15a data block			
The number of data lines in this block is n, the number in data block 2a, twelfth field, or 0 if n = -1.			
1-15	1st	F	Enter the slope of α_{22} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.
16a data block			
The number of data lines in this block is n, the number in data block 2a, thirteenth field, or 0 if n=-1.			
1-15	1st	F	Enter the slope of α_{33} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.
17a data block			
The number of data lines in this block is the number in data block 2a, fourteenth field.			
1-15	1st	F	Enter the slope of the ratio of work hardening slope vs. temperature.
16-30	2nd	F	Temperature at which above slope becomes operative.
			Note: To define the dependence of 10th cycle yield on temperature, use user subroutine YIEL.
The following data blocks are used only in a coupled thermal-stress analysis.			
18a data block			
The number of data lines in this block is the number in the first field of the 3a data block.			
1-15	1st	F	Enter the slope of K_{11} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.
19a data block			
The number of data lines in this block is n, the number in the second field of the 3a data block, or 0 if n = -1.			
1-15	1st	F	Enter the slope of K_{22} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

Format Fixed	Format Free	Data Type	Entry
20a data block			
The number of data lines in this block is n, the number in the third field of the 3a data block, or 0 if n = -1.			
1-15	1st	F	Enter the slope of K_{33} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.
21a data block			
The number of data lines in this block is the number in the fourth field of the 3a data block.			
1-15	1st	F	Enter the slope of specific heat vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.
22a data block			
The number of data lines in this block is the number in the fifth field of the 3a data block.			
1-15	1st	F	Enter latent heat value.
16-30	2nd	F	Solidus temperature (lower phase change limit).
31-45	3rd	F	Liquidus temperature (upper phase change limit).

Option B

2b data block			
1-5	1st	I	Number of data points of yield.
6-10	2nd	I	Number of data points of E_{11} vs temperature curve.
11-15	3rd	I	Number of data points of E_{22} vs temperature curve. Enter -1 to have (E_{22} vs. temp.) ≡ (E_{11} vs. temp.).
16-20	4th	I	Number of data points of E_{33}. Enter -1 to have (E_{33} vs. temp.) ≡ (E_{11} vs. temp.).
21-25	5th	I	Number of data points of ν_{12}.
26-30	6th	I	Number of data points of ν_{23}. Enter -1 to have (ν_{23} vs. temp.) ≡ (ν_{12} vs. temp.).
31-35	7th	I	Number of data points of ν_{31}. Enter -1 to have (ν_{31} vs. temp.) ≡ (ν_{12} vs. temp.).
36-40	8th	I	Number of data points of G_{12}.

Format Fixed	Format Free	Data Type	Entry
41-45	9th	I	Number of data points of G_{23}. Enter -1 to have (G_{23} vs. temp.) ≡ (G_{12} vs. temp.).
46-50	10th	I	Number of data points of G_{31}. Enter -1 to have (G_{23} vs. temp.) ≡ (G_{12} vs. temp.).
51-55	1th1	I	Number of data points of α_{11}.
56-60	12th	I	Number of data points of α_{22}. Enter -1 to have (α_{22} vs. temp.) ≡ (α_{11} vs. temp.).
61-65	13th	I	Number of data points of α_{33}. Enter -1 to have (α_{33} vs. temp.) ≡ (α_{11} vs. temp.).
66-70	14th	I	Number of data points of the workhardening vs. temperature curve.
71-75	15th	I	Enter the material identification for this data set.
76-80	16th	I	Enter the unit number for input of this data. Defaults to input deck.

3b data block

Include this data block only in a coupled thermal-stress analysis.

1-5	1st	I	Number of data points of K_{11}
6-10	2nd	I	Number of data points of K_{22}. Enter -1 to have (K_{22} vs. temp.) ≡ (K_{11} vs. temp.).
11-15	3rd	I	Number of data points of K_{33}. Enter -1 to have (K_{33} vs. temp.) ≡ (K_{11} vs. temp.).
16-20	4th	I	Number of data points of specific heat
21-25	5th	I	Number of latent heats.

4b data block

The number of data lines in this block is the number in data block 2b, first field.

1-15	1st	F	Enter the value of yield stress.
16-30	2nd	F	Enter the associated temperature.

Format Fixed	Format Free	Data Type	Entry
5b data block			
The number of data lines in this block is the number in data block 2b, second field.			
1-15	1st	F	Enter the value of E_{11}.
16-30	2nd	F	Enter the associated temperature.
6b data block			
The number of data lines in this block is n, the number in data block 2b, third field, or 0 if n = -1.			
1-15	1st	F	Enter the value of E_{22}.
16-30	2nd	F	Enter the associated temperature.
7b data block			
The number of data lines in this block is n, the number in data block 2b, fourth field, or 0 if n = -1.			
1-15	1st	F	Enter the value of E_{33}.
16-30	2nd	F	Enter the associated temperature.
8b data block			
The number of data lines in this block is the number in data block 2b, fifth field.			
1-15	1st	F	Enter the value of ν_{12}.
16-30	2nd	F	Enter the associated temperature.
9b data block			
The number of data lines in this block is n, the number in data block 2b, sixth field, or 0 if n = -1.			
1-15	1st	F	Enter the value of ν_{23}.
16-30	2nd	F	Enter the associated temperature.
10b data block			
The number of data lines in this block is n, the number in data block 2b, seventh field, or 0 if n = -1.			
1-15	1st	F	Enter the value of ν_{31}.
16-30	2nd	F	Enter the associated temperature.

Format Fixed	Format Free	Data Type	Entry
11b data block			
The number of data lines in this block is the number in data block 2b, eighth field.			
1-15	1st	F	Enter the value of G_{12}.
16-30	2nd	F	Enter the associated temperature.
12b data block			
The number of data lines in this block is n, the number in data block 2b, ninth field, or 0 if n = -1.			
1-15	1st	F	Enter the value of G_{23}.
16-30	2nd	F	Enter the associated temperature.
13b data block			
The number of data lines in this block is n, the number in data block 2b, tenth field, or 0 if n = -1.			
1-15	1st	F	Enter the value of G_{31}.
16-30	2nd	F	Enter the associated temperature.
14b data block			
The number of data lines in this block is the number in data block 2b, eleventh field.			
1-15	1st	F	Enter the value of α_{11}.
16-30	2nd	F	Enter the associated temperature.
15b data block			
The number of data lines in this block is n, the number in data block 2b, twelfth field, or 0 if n = -1.			
1-15	1st	F	Enter the value of α_{22}.
16-30	2nd	F	Enter the associated temperature.
16b data block			
The number of data lines in this block is n, the number in data block 2b, thirteenth field, or 0 if n = -1.			
1-15	1st	F	Enter the value of α_{33}.
16-30	2nd	F	Enter the associated temperature.

Format Fixed	Format Free	Data Type	Entry

17b data block

The number of data lines in this block is the number in data block 2b, fourteenth field.

1-15	1st	F	Enter the value of work hardening slope.
16-30	2nd	F	Enter the associated temperature.

Note: To define the dependence of 10th cycle yield on temperature, use user subroutine YIEL.

The following five data blocks are used only in a coupled thermal-stress analysis.

18b data block

The number of data lines in this block is the number in data block 3b, first field.

1-15	1st	F	Enter the value of K_{11}.
16-30	2nd	F	Enter the associated temperature.

19b data block

The number of data lines in this block is n, the number in data block 3b, second field, or 0 if n = -1.

1-15	1st	F	Enter the value of K_{22}.
16-30	2nd	F	Enter the associated temperature.

20b data block

The number of data lines in this block is n, the number in data block 3b, third field, or 0 if n = -1.

1-15	1st	F	Enter the value of K_{33}.
16-30	2nd	F	Enter the associated temperature.

21b data block

The number of data lines in this block is the number in data block 3b, fourth field.

1-15	1st	F	Enter the value of specific heat.
16-30	2nd	F	Enter the associated temperature.

22b data block

The number of data lines in this block is the number in data block 3b, fifth field.

1-15	1st	F	Enter latent heat value.
16-30	2nd	F	Solidus temperature (lower phase change limit).
31-45	3rd	F	Liquidus temperature (upper phase change limit).

■ TIME-TEMP

Define Effects of Time/Temperature Transformation

Description

This data block provides the user with the option of defining either mechanical or thermal (heat transfer) properties as a function of both temperature and the rate at which the temperature changes. It can also be used to describe the effects of phase transformations on these properties. You are reminded to include the T-T-T parameter when invoking this option. If the material properties are strictly dependent on temperature and not time as well, the TEMPERATURE EFFECTS option should be used instead.

In performing a thermal-stress analysis, the mechanical properties, which can be defined as a function of time-temperature-transformation (T-T-T), are the Young's modulus, Poisson's ratio, yield stress, work or strain hardening rate, and thermal coefficient of expansion. The effects of volumetric change due to phase transformation can be included through the definition of the thermal coefficient of expansion.

In performing a transient heat transfer analysis, the thermal properties, which can be defined as a function of T-T-T, are the thermal conductivity and the specific heat per unit reference mass. Here, the effects of heat or phase transformation can be included through the definition of the specific heat.

You are expected to have the test data for each property of each material group in a tabular form. For a given cooling rate, the value of a property must be known at discrete points over a range of temperature. There can be several sets of this data corresponding to measurements at several different cooling rates. The cooling tests must be of a specific type known as "Newton Cooling"; for example, the temperature change in the material is controlled such that

$$T(t) = A \exp(-at) + B$$

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words TIME-TEMP.

Format Fixed	Format Free	Data Type	Entry
2nd data block			
1-5	1st	I	Enter the total number of different material groups with time-temperature-transformation dependent properties.
3rd data block			
This data block is used to specify the minimum and maximum temperatures bracketing the range over which the property values given below are meant to apply.			
1-10	1st	F	Enter the value of the minimum temperature.
11-20	2nd	F	Enter the value of the maximum temperature.
Data blocks 4 through 8 are repeated for each material group.			
4th data block			
1-5	1st	I	Enter the material group number whose properties will be defined in the following data block.
6-10	2nd	I	Enter the element number corresponding to the first element in a consecutive list of elements for this material group.
11-15	3rd	I	Enter the element number corresponding to the last elements in a consecutive list of elements for this material group.
5th data block			
a) Stress Analysis			
1-5	1st	I	Enter the number of cooling rates used to define the Young's modulus for this material group. If this number is preceded by a minus sign, all properties for this material group will be assigned the same number of cooling rates. In this case, the remaining fields on this block need not be specified.
6-10	2nd	I	Enter the number of cooling rates used to define the Poisson's ratio for this material group.
11-15	3rd	I	Enter the number of cooling rates used to define the yield point for this material group.

Format Fixed	Format Free	Data Type	Entry
16-20	4th	I	Enter the number of cooling rates used to define the work hardening rate for this material group.
21-25	5th	I	Enter the number of cooling rates used to define the coefficient of thermal expansion for this material group.
b) Heat Transfer (Thermal) Analysis			
1-5	1st	I	Enter the number of cooling rates used to define the thermal conductivity for this material group.
6-10	2nd	I	Enter the number of cooling rates used to define the specific heat per unit reference mass for this material group.

6th data block

The coefficients associated with the definition of each cooling rate are specified in this data block. The cooling rates are described by the formula

$T(t) = A_{ij} \exp(-a_{ij} t) + B_{ij}$

where i refers to a particular cooling rate and j refers to a particular property

This data block is comprised of sets of blocks. There may be as many as five sets for a thermal-stress analysis and a maximum of two sets for a heat transfer analysis corresponding to the maximum number of properties which may be a function of time-temperature-transformation in each case. The number of blocks in each set corresponds to the five (stress analysis) or two (heat transfer analysis) entries given in the fifth data block. The order of the block sets is the same as for the entries on data block 5. On each block, specify the coefficients for a particular rate. The order within each set must be from the fastest to the slowest rate.

1-10	1st	F	Enter the value of the coefficient "A" in the above equation.
11-20	2nd	F	Enter the value of the coefficient "a" in the above equation.

Format		Data	
Fixed	**Free**	**Type**	**Entry**
21-30	3rd	F	Enter the value of the constant "B" in the above equation.
31-50	4th	I	If the number of cooling rates associated with each property is the same and magnitude (that is, coefficients) of these rates is also the same, enter a positive integer in this field on the first block (only) in this series. It is then necessary to specify the coefficients of each cooling rate for only a single property of the material group being used.
7th data block			
1-80	1st	I	Specify the number of temperature points at which a property value is specified for each cooling rate associated with a given property. If more than sixteen cooling rates are being used, use additional blocks. The order is from the fastest to the slowest cooling rate. Beginning on a new block repeat this block set for each property. The order in which properties are considered corresponds to the order specified in data block 5. (16I5) If the number of temperature points is the same for all cooling rates of all properties, enter this value in the first field preceded by a minus sign.

Format		Data	
Fixed	Free	Type	Entry

8th data block

The actual property values are defined in this data block. This series should be considered as a data block. There are as many sub-blocks in it as there are properties with time-temperature-transformation dependent effects for this material group. The order in which the property sub-blocks should be specified is given in data blocks 5(a) or 5(b).

Each property sub-block can be considered as consisting of a series of block sets. The number of block sets per sub-block corresponds to the number of cooling rates for the particular property in question. These block sets are specified in the order of fastest to slowest cooling rate.

Each block set of a sub-block, corresponding to a given cooling rate for a given property, contains as many blocks as there are temperature levels where a discrete value of the property is defined. The property values must be defined in the order of increasing temperature levels.

1-10	1st	F	Enter the value of the material property for a particular cooling rate and temperature level.
11-20	2nd	F	Enter the temperature level corresponding to the property value defined above.

■ CRACK DATA

Define Material Properties for Concrete Cracking

Description

This data block inputs the uniaxial cracking data for a low tension (concrete) material. The standard material properties, like Young's modulus, Poisson's ratio etc., are given in the ISOTROPIC option. Cross reference is given by the material identification number. Cracking data can alternatively be specified by user subroutine UCRACK, TENSOF, and USHRET.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word CRACK DATA.
2nd data block			
1-5	1st	I	Enter the number of distinct sets of element properties to be input (optional).
6-10	2nd	I	Enter the unit number for input of cracking data. Default to input.

The 3rd data block is repeated once for each distinct data set.

3rd data block			
1-5	1st	I	Material type identification (1, 2, 3, ...) for cross-referencing the ISOTROPIC block.
6-15	2nd	F	Critical cracking stress.
16-25	3rd	F	Modulus for tension softening material. (If no value is entered, stress goes to zero upon cracking.)
26-35	4th	F	Strain at which crushing of the material occurs. If this value is not given, it is automatically set to some large value.
36-45	5th	F	Shear retention factor.

■ FAIL DATA

Define Failure Criteria Data

Description

This option is used to define material dependent failure criteria data. Up to three failure criteria per material can be specified. Failure indices are calculated and printed for every integration point at which material dependent failure data exists. This option can also be used to invoke the progressive failure of a material.

The supported failure criteria types are:

MX STRESS
MX STRAIN
TSAI-WU
HOFFMAN
HILL
UFAIL

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words FAIL DATA.
2nd data block			
1-5	1st	I	Enter the number of materials with program calculated failure criteria. If blank, program will read until no more failure criteria data are left.
6-10	2nd	I	Enter the unit number for reading. Defaults to input.

Data blocks 3, 4 and 5 are repeated as a set, once for each material with program calculated failure criteria, as described above.

3rd data block			
1-5	1st	I	Material identification number.
6-10	2nd	I	Number of sets of failure criteria data for this material. No more than three sets are allowed.
11-15	3rd	I	Enter 1 to invoke the progressive failure option.

Format Fixed	Format Free	Data Type	Entry

Data blocks 4 and 5 are entered as pairs, once for each set of failure criteria data.

For MX Stress:

4th data block

1-10	1st	A	Enter the words MX STRESS.
11-20	2nd	F	X – Maximum tensile stress in x-direction.
21-30	3rd	F	X_c – Maximum absolute value of compressive stress in x-direction.
31-40	4th	F	Y – Maximum tensile stress in y-direction.
41-50	5th	F	Y_c – Maximum absolute value of compressive stress in y-direction.
51-60	6th	F	Z – Maximum tensile stress in z-direction.
61-70	7th	F	Z_c – Maximum absolute value of compressive stress in z-direction.

5th data block

1-10	1st	F	S_{xy} – Maximum absolute value of shear stress in xy-plane.
11-20	2nd	F	S_{yz} – Maximum absolute value of shear stress in yz-plane.
21-30	3rd	F	S_{zx} – Maximum absolute value of shear stress in zx-plane.

Note: X_c, Y_c and Z_c default to the values of X, Y, and Z, respectively, if left undefined.

For MX Strain:

4th data block

1-10	1st	A	Enter the words MX STRAIN.
11-20	2nd	F	ε_x – Maximum tensile strain in x-direction.
21-30	3rd	F	ε_{xc} – Maximum absolute value of compressive strain in x-direction.

Format Fixed	Format Free	Data Type	Entry
31-40	4th	F	ε_y – Maximum tensile strain in y-direction.
41-50	5th	F	ε_{yc} – Maximum absolute value of compressive strain in y-direction.
51-60	6th	F	ε_z – Maximum tensile strain in z-direction.
61-70	7th	F	ε_{zc} – Maximum absolute value of compressive strain in z direction
5th data block			
1-10	1st	F	γ_{xy} – Maximum absolute value of shear strain in xy-plane.
11-20	2nd	F	γ_{yz} – Maximum absolute value of shear strain in yz-plane.
21-30	3rd	F	γ_{zx} – Maximum absolute value of shear strain in zx-plane.

For Hoffman or Hill:

4th data block			
1-10	1st	A	Enter either HOFFMAN or HILL.
11-20	2nd	F	X – Maximum tensile stress in x-direction.
21-30	3rd	F	X_c – Maximum absolute value of compressive stress in x-direction.
31-40	4th	F	Y – Maximum tensile stress in y-direction.
41-50	5th	F	Y_c – Maximum absolute value of compressive stress in y-direction.
51-60	6th	F	Z – Maximum tensile stress in z-direction.
61-70	7th	F	Z_c – Maximum absolute value of compressive stress in z-direction.

Note: X_c, Y_c and Z_c default to the values of X, Y, and Z, respectively, if left undefined for Hoffman. For Hill, X_c, Y_c and Z_c are assumed to be equal to X, Y, and Z respectively and are not used.

Format Fixed	Format Free	Data Type	Entry
5th data block			
1-10	1st	F	S_{xy} – Maximum absolute value of shear stress in xy-plane.
11-20	2nd	F	S_{yz} – Maximum absolute value of shear stress in yz-plane.
21-30	3rd	F	S_{zx} – Maximum absolute value of shear stress in zx-plane.
31-40	4th	F	Failure index. Default is 1.0.

For Tsai-Wu:

4th data block

1-10	1st	A	Enter the word TSAI-WU.
11-20	2nd	F	X – Maximum tensile stress in x-direction.
21-30	3rd	F	X_c – Maximum absolute value of compressive stress in x-direction.
31-40	4th	F	Y – Maximum tensile stress in y-direction.
41-50	5th	F	Y_c – Maximum absolute value of compressive stress in y-direction.
51-60	4th	F	Z – Maximum tensile stress in z-direction.
61-70	5th	F	Z_c – Maximum absolute value of compressive stress in Z-direction.
			Note: X_c, Y_c and Z_c default to the values of X, Y, and Z, respectively, if left undefined.

5th data block

1-10	1st	F	S_{xy} – Maximum absolute value of shear stress in xy-plane.
11-20	2nd	F	S_{yz} – Maximum absolute value of shear stress in yz-plane.
21-30	3rd	F	S_{zx} – Maximum absolute value of shear stress in zx-plane.

Format Fixed	Format Free	Data Type	Entry
31-40	4th	F	Failure index. Default is 1.0.
41-50	5th	F	F_{xy} – Interactive strength tensor constant for the xy-plane.
61-60	6th	F	F_{yz} – Interactive strength tensor constant for the yz-plane.
61-70	7th	F	F_{zx} – Interactive strength tensor constant for the zx-plane.
			Note: F_{xy} should be such that $F_{xy}^2 < \frac{1}{XX_c} \cdot \frac{1}{YY_c}$, etc.

For UFAIL:

For UFAIL type failure criteria, enter only the word UFAIL in the 4th data block and leave all other fields blank. The 5th data block is not used for UFAIL.

4th data block

1-10	1st	A	Enter the word UFAIL.

■ DAMAGE

Define Properties for Damaging Materials

Description

This option allows you to define a set of data for a specific material which includes a damage model. The specific model and associated data can also be specified with this option. For elasto-plastic materials, the damage model is based on a Gurson model for the yield surface definition for materials with voids:

$$F = \left(\frac{\bar{\sigma}}{\sigma_y}\right)^2 + 2q_1 f^* \cosh\left(\frac{q_2 + \sigma_{kk}}{2\sigma_y}\right) - [1 - (q_1 f^*)^2] = 0$$

Void nucleation and void growth are based on a model by Tuergaard and Needleman. Here, f^* is introduced to model the rapid decrease in load carrying capacity if void coalenscence occurs:

$$f^* = f \qquad \text{if } f \le f_c$$

$$f^* = f_c + \left[\frac{f_u - f_c}{f_F - f_c}\right](f - f_c) \text{ if } f > f_c$$

the nucleation can be either stress or strain controlled. The strain controlled nucleation is given by:

$$\dot{f}_{\text{nucleation}} = \frac{f_N}{S\sqrt{2\pi}} \exp\left[-\frac{1}{2}\left(\frac{\varepsilon_m^p - \varepsilon_n}{S}\right)^2\right]\dot{\varepsilon}_m^p$$

For elastomeric materials, the model is based on the undamaged strain energy function W° multiplied by a Kachanov damage factor, K. The damage capability is available for Ogden strain energy model using the total Lagrange formulation as well as the Mooney-Rivlin, Ogden, and general principal stretch based model using the updated Lagrange formulation.

$$W = K \cdot W^\circ$$

Both continuous damage (Miehe's formulation) as well as the discontinuous damage (Mullin's effect) can be modeled within an additive:

$$K = d^\infty + \sum_n d_n^\alpha \exp\left(-\frac{\alpha}{\eta_n}\right) + \sum_m d_m^\beta \exp\left(-\frac{\beta}{\lambda_m}\right)$$

or a multiplicative format:

$$K = d^{\infty} + \sum d_n \exp\left(-\frac{\alpha + \delta_n \beta}{\eta_n}\right)$$

d^{∞} is automatically calculated by the program. The normal data for a specific material are defined with the ISOTROPIC, WORK HARD, and OGDEN options. Cross-reference to this material is made with the material number. Additional data for the initial void volume fraction can be defined with user subroutine UVOID. Other nucleation models are allowed via the user subroutine UVOIDN.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words DAMAGE.
2nd data block			
1-5	1st	I	Enter the number of distinct sets of material properties to be input (optional).
6-10	2nd	I	Enter the logical unit number for reading damage data. Defaults to input.

The 3rd and 4th data blocks are entered as pairs, once for each distinct data set.

3rd data block			
1-5	1st	I	Material type identification (1, 2, 3, etc.) for cross-referencing to ISOTROPIC or ODGEN option.
6-10	2nd	I	Method of void nucleation. Enter 0 for no nucleation. Enter 1 for plastic-strain controlled nucleation. Enter 2 for stress controlled nucleation. Enter 3 for nucleation controlled by user subroutine UVOIDN. Enter 4 for elastomeric damage model; additive decomposition of the Kachanov factor. Enter 5 for elastomeric damage model; multiplicative decomposition of the Kachanov factor. Enter 6 for elastomeric damage model controlled by user subroutine UELDAM.

Format Fixed	Format Free	Data Type	Entry
4a data block			
Use only if the method of void nucleation (shown above) is 0, 1, or 2.			
1-10	1st	F	First yield surface multiplier q_1 (recommended is $q_1 = 1.5$).
11-20	2nd	F	Second yield surface multiplier q_2 (recommended is $q_2 = 1$).
21-30	3rd	F	Initial void volume fraction.
31-40	4th	F	Critical void volume fraction. This value represents the value at which coalescence of voids start (f_c).
41-50	5th	F	Failure void volume fraction (f_F). This is the value of the void volume fraction at which the stiffness of the material has reduced to zero.
51-60	6th	F	If strain controlled, enter the mean strain for nucleation. If stress controlled, enter the mean stress for nucleation.
61-70	7th	F	Standard deviation in nucleation relation (S).
71-80	8th	F	Volume fraction of void nucleating particles f_N.
			Note: The presence of these blocks in the model definition block automatically overwrite the yield criterion specified for a specific material on the ISOTROPIC block. Currently, the model can only be used for isotropic hardening materials.
4b data block			
Use only for elastomeric damage model, additive decomposition, two term Prony series.			
1-10	1st	F	First scalar factor, continuous damage (d_1^{β}).
11-20	2nd	F	First relaxation parameter, continuous damage (λ_1).
21-30	3rd	F	Second scalar factor, continuous damage (d_2^{β}).
31-40	4th	F	Second relaxation parameter, continuous damage (λ_2).
41-50	5th	F	First scalar factor, discontinuous damage (d_1^{α}).

Format Fixed	Format Free	Data Type	Entry
51-60	6th	F	First relaxation parameter, discontinuous damage (η_1).
61-70	7th	F	Second scalar factor, discontinuous damage (d_2^{α}).
71-80	8th	F	Second relaxation parameter, discontinuous damage (η_2).

4c data block

Use only for elastomeric damage model, multiplicative decomposition, two term Prony Series.

1-10	1st	F	First scalar factor (d_1).
11-20	2nd	F	First proportioning term (δ_1).
21-30	3rd	F	First relaxation rate constant (η_1).
31-40	4th	F	Second scalar factor (d_2).
41-50	5th	F	Second proportioning term (δ_2).
51-60	6th	F	Second relaxation rate constant (η_2).

■ GAP DATA

Define Data for Gap Elements

Description

This option allows you to specify all of the data associated with gap elements (types 12 and 97). These data include gap closure distance, gap elastic stiffness, contact coefficient of friction, and momentum ratio.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words GAP DATA.
2nd data block			
1-5	1st	I	Number of sets of gap data to be input.
6-10	2nd	I	Unit number for reading data. Defaults to input.
The 3rd and 4th data blocks are entered as pairs, once for each set of gap data.			
3rd data block			
1-10	1st	F	For a fixed direction gap, enter the gap closure distance U_{cl}. For a true distance gap, enter the minimum distance d between end points. **Note:** If d > 0, the two end points will never be closer than a distance |d| apart. If d < 0, the two end points are never farther apart than |d|.
11-20	2nd	F	μ, the contact coefficient of friction.
21-30	3rd	F	K_{GAP} the elastic stiffness of the closed gap in the contact direction. Default: Gap is rigid when closed.
31-40	4th	F	$K_{FRICTION}$, the elastic stiffness of the closed gap in the friction direction. Default: Gap is rigid when closed.
41-50	5th	F	User supplied momentum ratio for first gap node. Default: MARC calculates this ratio internally.

Format		Data	
Fixed	**Free**	**Type**	**Entry**
51-60	6th	F	User supplied momentum ratio for fourth gap node. Default: MARC calculates this ratio internally.
61-65	7th	I	Enter 0 for fixed direction gap.
			Enter 1 for true distance gap. Default is 0.
66-70	8th	I	Enter 0 if gap is open during increment 0.
			Enter 1 if gap is closed during increment 0. Default is 0.
4th data block			
			Enter a list of gap elements to be associated with this set of gap data.

■ COMPOSITE

Define Properties for Laminated Composite Materials

Description

This option allows you to define the layer-by-layer material identifications, layer thicknesses, and orientation angles for a laminated composite material and to associate this information with an element number. Property data for each material identification is entered using the ISOTROPIC, ORTHOTROPIC, ANISOTROPIC, or HYPOELASTIC options.

To specify a user-defined orientation, use the ORIENTATION option. Note that an input error results if the COMPOSITE option is specified for nonlayered elements.

This option is available for shell elements or beams in a plane (type 16). It is not available for open and closed section beam elements.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word COMPOSITE.
2nd data block			
1-5	1st	I	Enter the number of composite group data sets to follow.
6-10	2nd	I	Unit number for input. Defaults to standard input (unit 5).
3rd data block			
1-5	1st	I	Composite group number.
6-10	2nd	I	Number of layers in this group.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Enter 0 to input actual layer thicknesses in the second field of the 4th data block. (Default is 0; that is, the sum of the layer thicknesses will override thickness data entered in the GEOMETRY block. Enter 1 to input percentage of total thickness in the second field of the 4th data block. In this case, element thickness is entered using the GEOMETRY block or the NODAL THICKNESS block. If you are using the variable thickness capability for those elements which have such an option, you must enter 1 here and then enter percentages of total thickness in the second field of the 4th data block below.
16-25	4th	F	Enter position of user-defined reference plane. This is the value of the local z-coordinate of the user-defined plane with respect to the geometric midplane. Default is 0.

The 4th data block is repeated once for each layer defined in the 3rd data block.

4th data block

1-5	1st	I	Material identification number for this layer.
6-15	2nd	F	Actual layer thickness if default (0) is used in data block 3, Field 3 (above). If 1, percentage of total thickness.
16-25	3rd	F	Ply orientation angle in degrees. Location of principal material axes with respect to element coordinate system measured positive about local z-coordinate, (that is, angle defining orientation of preferred frame w.r.t. element frame). The element orientation is either defined in the ORIENTATION option or defaults to the v_1, v_2, v_3 system defined in *Volume B: Element Library*.

5th data block

Enter a list of elements to be associated with this particular composite group.

■ REBAR

Define Rebar Positions, Areas, and Orientations

Description

This option allows the rebar positions, areas, and orientations to be read in by means of input file instead of user subroutine REBAR. This option is used only when rebar layers within an element are similar/parallel to one of the element surfaces (edges). For more general cases, use user subroutine REBAR.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-5	1st	A	Enter the word REBAR.
2nd data block			
1-5	1st	I	Number of data sets to be read in.
6-10	2nd	I	Unit number for input; defaults to standard input (unit 5).
Data blocks 3, 4, and 5 are given for each data set.			
3rd data block			
1-5	1st	I	Rebar data set id.
6-10	2nd	I	Number of rebar layers to be read in. Maximum is 5.
11-15	3rd	I	Enter 2 if the considered structure is an axisymmetric expansion of cylinders made of bias plies and the cords are nearly inextensible relative to matrix materials. In this case, all parameters below describe the cylinders from which the considered structure was made. The rebar positions, areas, and orientations for real structure will be calculated by MARC.
16-25	4th	F	First direction cosine of reference axis.
26-35	5th	F	Second direction cosine of reference axis.
36-45	6th	F	Third direction cosine of reference axis. The default reference axis is (1, 0, 0). **Note:** Reference axis should not be perpendicular to rebar layer.

Format Fixed	Format Free	Data Type	Entry
Data block 4 is given for each rebar layer.			
4th data block			
1-5	1st	I	Material id.
6-15	2nd	F	$\frac{p \cdot r}{T}$, relative position of the rebar layer (ratio of the distance between the reference surface (edge) and the rebar layer to the distance across the element); not used for membrane elements.
16-25	3rd	F	A, area of cross section of each rebar.
26-35	4th	F	S, number of rebars per unit length in each layer. Equivalent thickness of the rebar layer is $A \cdot S$.
36-45	5th	F	Angle (α) between the rebar and the projection of the reference axis on rebar layer plane [-90, 90]. See Figure 3-19.
46-55	6th	F	Radius of the cylinder; only used when the third field of the 3rd data block is 2.
5th data block			
			Enter a list of elements.

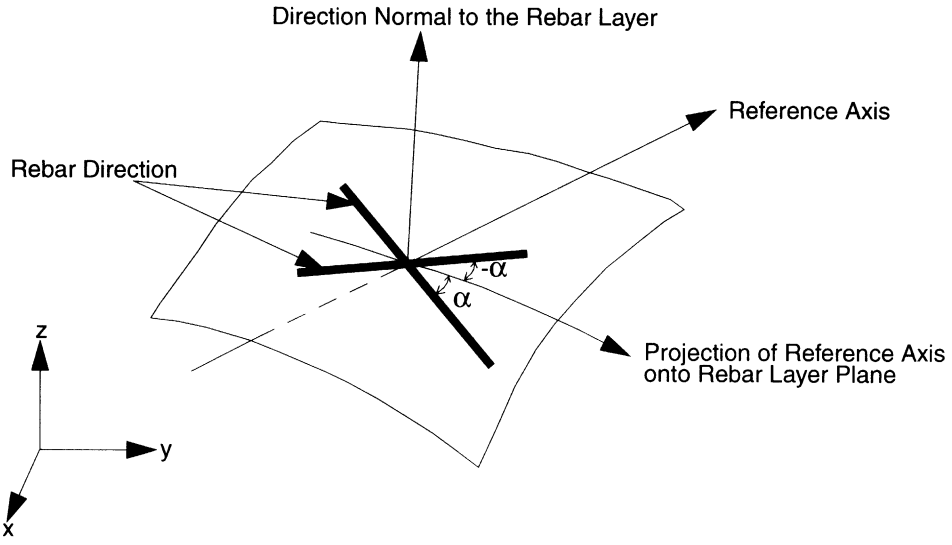


Figure 3-19 Description of Rebar Orientation on a Single Rebar Layer

■ ORIENTATION

Define Orientation of Elements

Description

The ORIENTATION option is used to specify orientation angle data as follows:

1. Edge orientation types (`EDGE i-j`). For two-dimensional elements (both continuum and shells), you choose a particular element edge with respect to which the preferred coordinates are specified. With these types, the direction vector along the edge from the first to the second edge node is projected onto the surface tangent plane (xy plane if continuum, or $\tilde{V}^1\tilde{V}^2$ plane if shell) at each integration point.

   The first preferred direction is given by a rotation about the surface normal (z axis if continuum, $\tilde{V}^3$ axis if shell) equal to the orientation angle. The third preferred direction is given by the surface normal, and the second preferred direction is given by a cross product of the third and first directions. See Figure 3-20.

2. Global intersecting plane types (`ij PLANE`). These types are also for two-dimensional elements. Here, a particular global coordinate plane (selected by the orientation type) is intersected with the surface tangent plane.

   The first preferred direction is given by a rotation about the surface normal from this intersection by an amount equal to the orientation angle. The third preferred direction is given by the surface normal and the second direction by a cross product of the third and first. See Figure 3-21.

3. User-defined intersecting plane. These types are also for two-dimensional elements. Here, a plane, defined by you, with one coordinate direction and a user-defined vector or by two user-defined vectors is intersected with the surface tangent plane.

   The first preferred direction is given by a rotation about the surface normal from this intersection by an amount equal to the orientation angle. The third preferred direction is given by the surface normal and the second direction by a cross product of the third and first. See Figure 3-22.

4. Three-dimensional orientation types (3D ANISO). For three-dimensional elements, you directly enter vectors in the first and second preferred directions. The third preferred direction is given by a cross product of the first and second direction. See Figure 3-23.

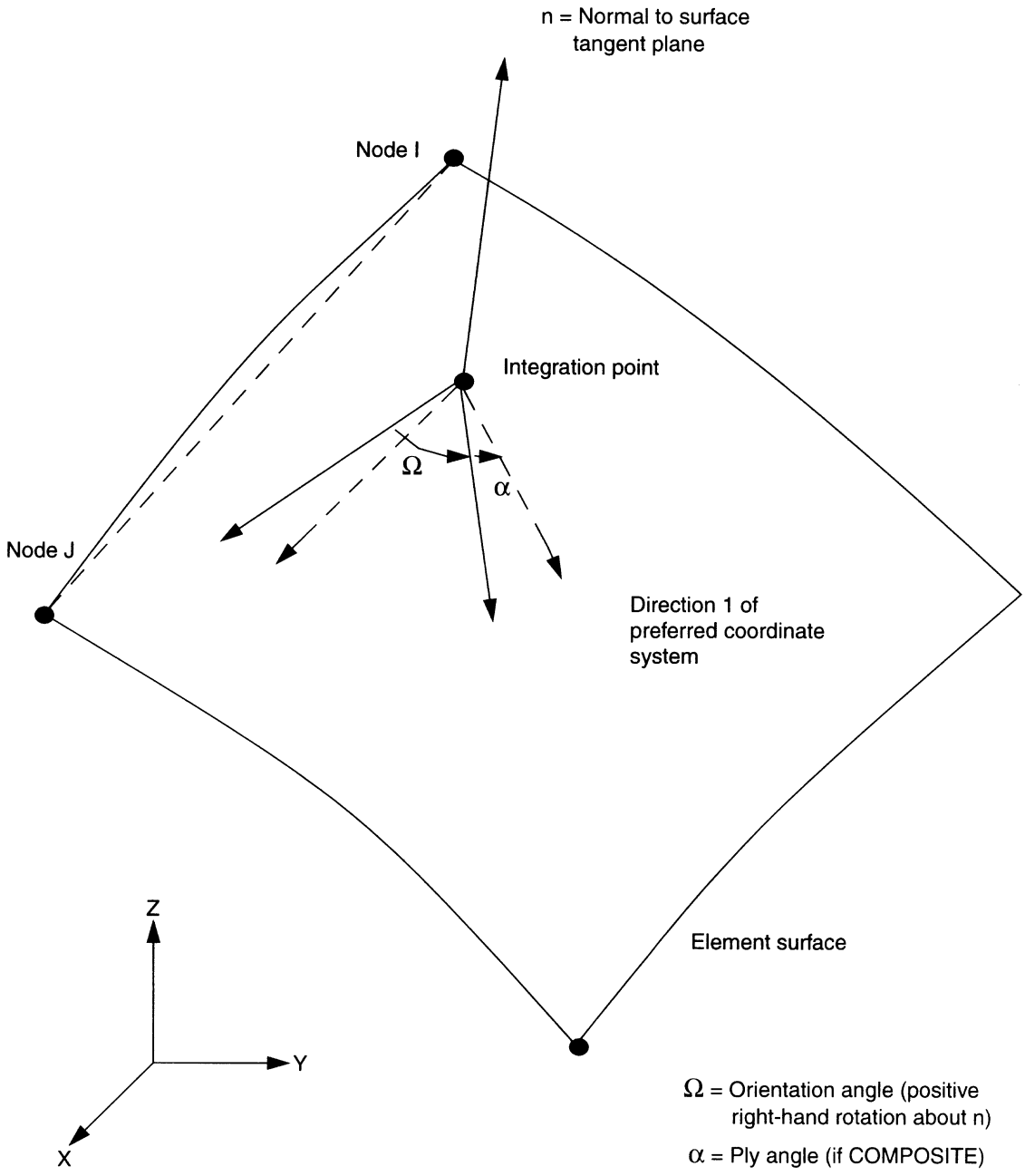


Figure 3-20 Edge I-J Orientation Type

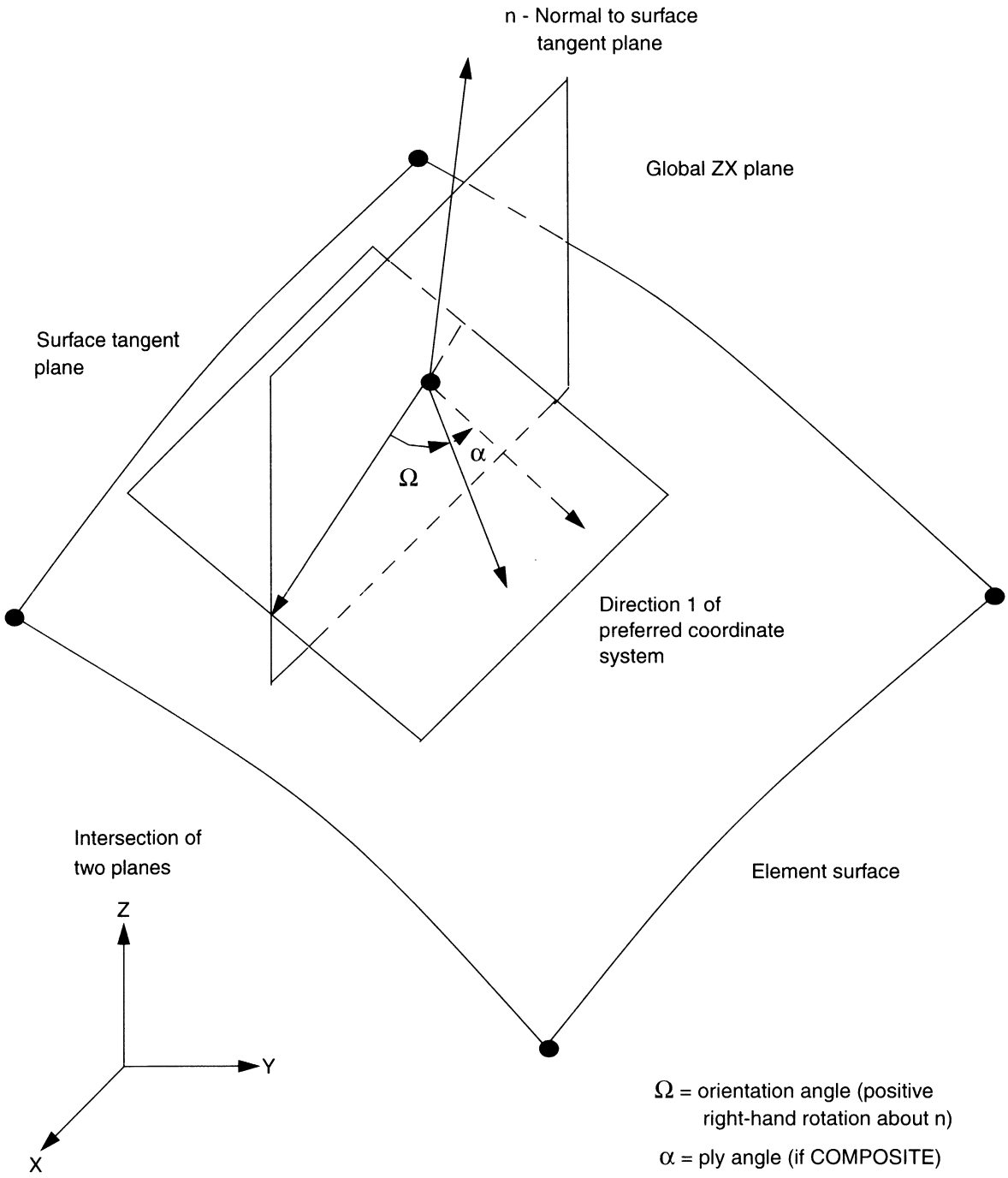


Figure 3-21 ZX Plane Orientation Type

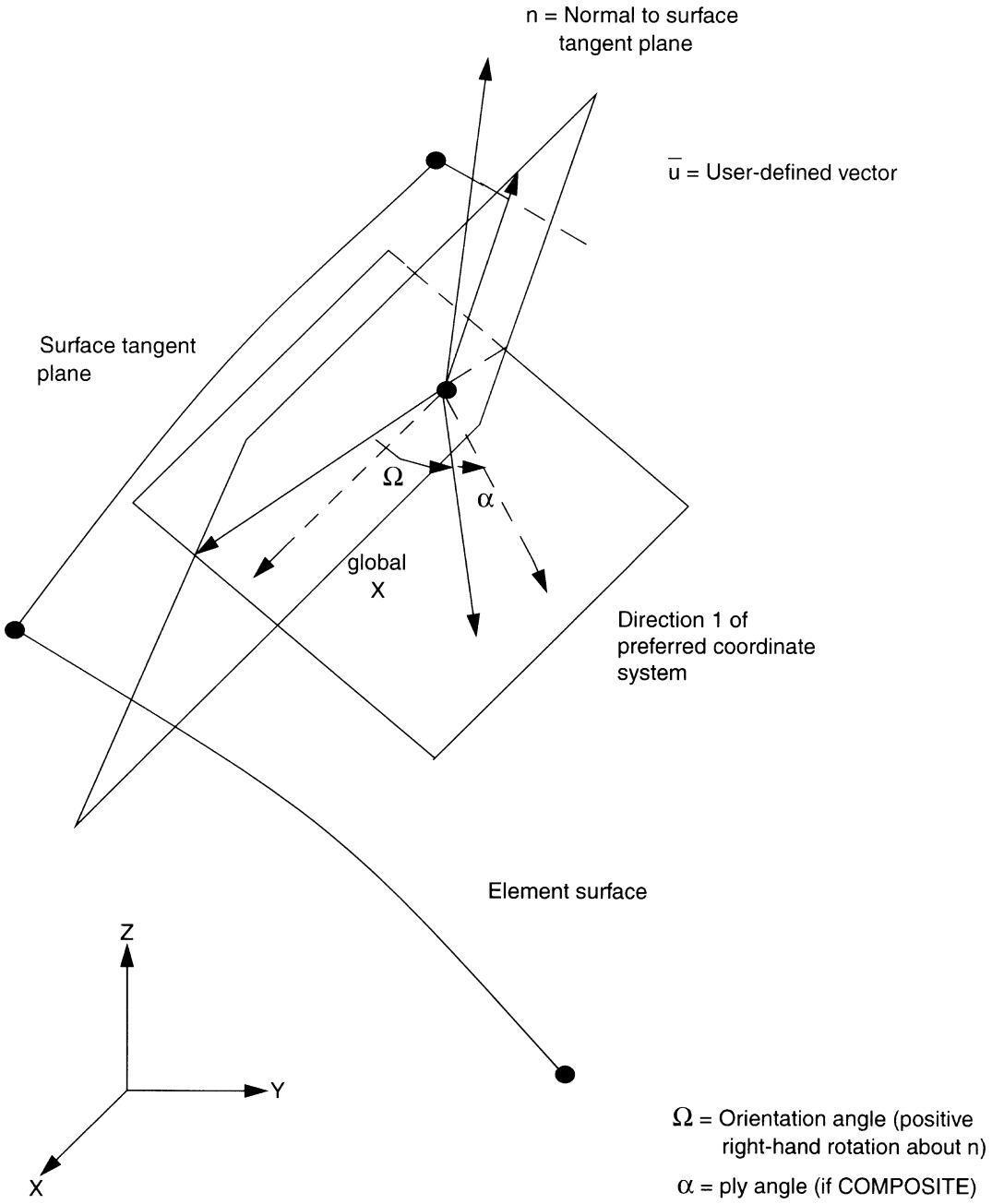


Figure 3-22 XU Plane Orientation Type

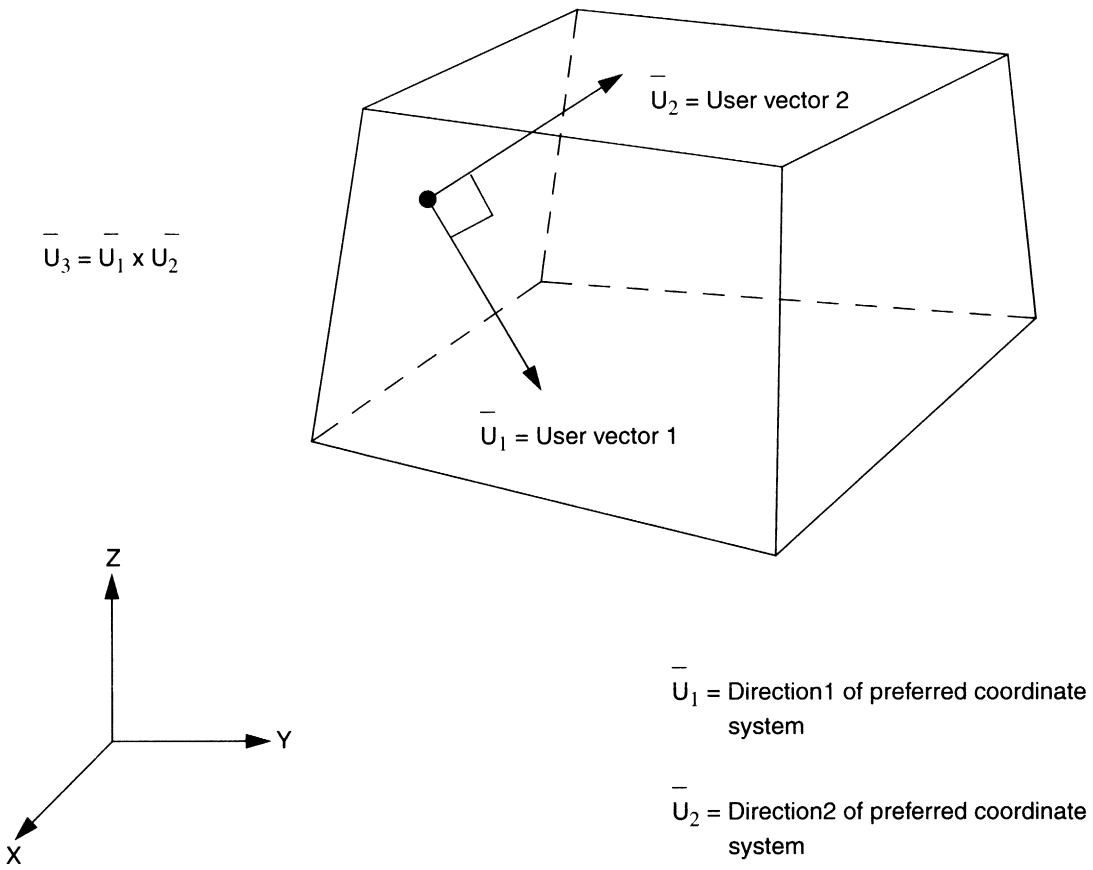


Figure 3-23 Three-Dimensional ANISO Orientation Type

5. UORIENT orientation type. Here, you define the transformation matrix between global coordinates (if continuum elements) or local coordinates (if beams, plates or shells) directly in user subroutine ORIENT.

Notes: The ORIENTATION option is ignored for 1-D elements, gaps, pipe bend, shear panel and cable elements.

The ORIENTATION option is turned on for composite elements. If no ORIENTATION data is given for these elements, the default is no preferred orientation; that is, the default material orientation of the element.

The ORIENTATION option, UORIENT, is turned on for particular material numbers if the IANELS flag is set during data input (see ISOTROPIC, ORTHOTROPIC, ANISOTROPIC, MOONEY and HYPOELASTIC options). You can override this default by entering your own ORIENTATION block.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word ORIENTATION.
2nd data block			
1-5	1st	I	Enter the number of orientation angle data sets to follow.
6-10	2nd	I	Unit number for input. Defaults to input file.
The 3rd and 4th data blocks are entered as pairs, once for each angle data set.			
3rd data block			
1-10	1st	A	Enter one of the following to specify orientation angle type. EDGE 1-2 EDGE 2-3 EDGE 3-4 EDGE 3-1 EDGE 4-1 XY PLANE YZ PLANE ZX PLANE XU PLANE YU PLANE ZU PLANE UU PLANE UORIENT 3D ANISO
11-20	2nd		Orientation angle

Format Fixed	Format Free	Data Type	Entry	
For XU PLANE, YU PLANE, ZU PLANE, UU PLANE, and 3D ANISO, complete the following:				
21-30	1st	F	1	
31-40	2nd	F	2	component of user vector 1 w.r.t. global coords.
41-50	3rd	F	3	
For UU PLANE and 3D ANISO, complete the following:				
51-60	4th	F	1	
61-70	5th	F	2	component of user vector 2 w.r.t. global coords.
71-80	6th	F	3	

4th data block

Enter a list of elements to be associated with this orientation angle.

■ POWDER

Define Powder Material Model

Description

This option allows you to input data associated with MARC's powder material model. The material parameters can be entered here or through user subroutine UPOWDR. The influences of temperature and relative density are entered through the TEMPERATURE EFFECTS and DENSITY EFFECTS options. The data entered here is at the beginning of the analysis; for example, at the temperatures given through the INITIAL STATE option and the relative density given through the RELATIVE DENSITY option.

The yield function for powder material is:

$$F = \frac{1}{\gamma}\left(\frac{3}{2}S_{ij}S_{ij} + \frac{p^2}{\beta^2}\right)^{1/2} - \sigma_y$$

where s is the deviatoric stresses and p is the hydrostatic stress. γ and β are material properties which can be expressed as:

$$\beta = (b_1 + b_2\rho^{b_3})^{b_4}$$

$$\gamma = (q_1 + q_2\rho^{q_3})^{q_4}$$

where ρ is the relative density.

This data is entered through this option. Additional details can be found in *Volume A: Theory and User Information.*

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word POWDER.
2nd data block			
1-5	1st	I	Enter the number of sets of powder material data to be defined (optional).
6-10	2nd	I	Enter the logical unit number for data input. Defaults to input data file.

Format Fixed	Format Free	Data Type	Entry
Data blocks 3, 4, 5, 6, 7, and 8 are entered as a set, once for each data associated with this material definition.			
3rd data block			
1-5	1st	I	Material identification number.
4th data block			
1-10	1st	F	Young's modulus.
11-20	2nd	F	Poisson's ratio.
21-30	3rd	F	Mass density
31-40	4th	F	Coefficient of thermal expansion.
41-50	5th	F	Compressive yield stress.
51-60	6th	F	Gamma (γ).
61-70	7th	F	Beta (β).
71-80	8th	F	Viscosity.
5th data block			
The 5th data block is only required in a coupled thermal-stress analysis.			
1-10	1st	F	Thermal conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density.
6th data block			
1-10	1st	F	q_1
11-20	2nd	F	q_2
21-30	3rd	F	q_3
31-40	4th	F	q_4
7th data block			
1-10	1st	F	b_1
11-20	2nd	F	b_2
21-30	3rd	F	b_3
31-40	4th	F	b_4
8th data block			
			Enter a list of elements to be associated with this material definition.

■ DENSITY EFFECTS

Define Effects of Density on Powder Materials

Description

This option defines the variation of powder material data with respect to the relative density. The base values are those read in through the POWDER option at the initial state. The data in this option is used in conjunction with the TEMPERATURE EFFECTS option to give bilinear variations in the material properties. The relative density can be entered using one of the following options:

A. The variation of a particular property relative to its base value with respect to the relative density as a piecewise linear curve. Breakpoints must be given in ascending order of relative density.

B. The particular value relative to its base value and with respect to the relative density lying on the relevant curve are input directly. Data points must be given in increasing order of relative density. This option is flagged by entering the word DATA on the first block.

Note: In this option, relative density is the density relative to the fully compacted density having the range (0-1). The relative Young's modulus, Poisson's ratio, relative conductivity, and relative specific heat are their respective values relative to those given on the POWDER option at the base temperature.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-15	1st	A	Enter the words DENSITY EFFECTS.
16-80	2nd	A	Enter the word DATA to indicate that option B is used.

OPTION A

2a data block			
1-5	1st	I	Number of slopes of relative Young's modulus versus relative density curve.
6-10	2nd	I	Number of slopes of relative Poisson's ratio versus relative density curve.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Number of slopes of relative Conductivity versus relative density curve (coupled analysis only).
16-20	4th	I	Number of slopes of relative Specific Heat versus relative density curve (coupled analysis only).
21-25	5th	I	Material type identification number for cross-reference with POWDER model definition option.

3a data block

The number entered in the first field of data block 2a defines the number of blocks required in data block 3a.

1-15	1st	F	Enter the slope of relative Young's modulus versus the relative density curve.
16-30	2nd	F	Enter the relative density at which the above slope becomes operative.

4a data block

The number in the second field of block 2a defines the number of blocks required in data block 4a.

1-15	1st	F	Enter the slope of relative Poisson's ratio versus the relative density curve.
16-30	2nd	F	Enter the relative density at which the above slope becomes operative.

5a data block

The number in the third field of block 2a defines the number of blocks required in data block 5a.

1-15	1st	F	Enter the slope of relative conductivity versus relative density.
16-30	2nd	F	Enter the relative density at which the above slope becomes operative.

Format Fixed	Format Free	Data Type	Entry
6a data block			
The number in the fourth field of block 2a defines the number of blocks required in data block 6a.			
1-15	1st	F	Enter the slope of the relative specific heat versus relative density.
16-30	2nd	F	Relative density at which this slope becomes operative.

Option B

2b data block			
1-5	1st	I	Number of data points on relative Young's modulus versus relative density curve.
6-10	2nd	I	Number of data points on relative Poisson ratio versus relative density curve.
11-15	3rd	I	Number of data points on relative Conductivity versus relative density curve (coupled analysis only).
16-20	4th	I	Number of data points on relative specific heat versus relative density curve (coupled analysis only).
21-25	5th	I	Material type identification number for cross-reference with POWDER model definition option.
3b data block			
The number in the first field of block 2b defines the number of blocks required in data block 3b.			
1-15	1st	F	Enter the value of the relative Young's modulus.
16-30	2nd	F	Enter the associated relative density.
4b data block			
The number in the second field of block 2b defines the number of blocks required in data block 4b.			
1-15	1st	F	Enter the value of the relative Poisson's ratio.
16-30	2nd	F	Enter the associated relative density.

Format Fixed	Format Free	Data Type	Entry
5b data block			
The number in the third field of block 2b defines the number of blocks required in data block 5b.			
1-15	1st	F	Enter the value of the relative conductivity.
16-30	2nd	F	Enter the associated relative density.
6b data block			
The number in the fourth field of block 2b defines the number of blocks required in data block 6b.			
1-15	1st	F	Enter the value of the relative specific heat.
16-30	2nd	F	Enter the associated relative density.

■ RELATIVE DENSITY

Define Initial Relative Density for Stress or Coupled Thermal Stress Analysis

Description

This option allows the user to define initial relative density in a powder material analysis.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words RELATIVE DENSITY.
2nd data block			
1-5	1st	I	Enter the number of sets to be input with this option.
6-10	2nd	I	Enter the logical unit number of the data input. Defaults to input data file.
The 3rd, 4th and 5th data blocks are entered as pairs, once for each data set.			
3rd data block			
1-10	1st	F	Initial relative density.
4th data block			
			Enter a list of elements to be associated with the above defined initial relative density.
5th data block			
			Enter a list of integration points for which the defined initial relative density is used.

■ SOIL

Define Material Properties for Soil Analysis

Description

This option allows you to define material data for soil analysis. It is assumed that the soil skeleton is composed of a collection of randomly oriented grains resulting in effectively isotropic behavior. You must define here both the material properties of the soil and the fluid. For additional details, see *Volume A: Theory and User Information.*

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-4	1st	A	Enter the word SOIL.
2nd data block			
1-5	1st	I	Enter the number of sets of data used to define the soil data.
6-10	2nd	I	Enter the unit number of input of soil data. Defaults to input.

Data blocks 3, 4, and 5 are entered as a set, once for each data set.

3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing TEMPERATURE EFFECTS and WORK HARD data.
6-15	2nd	A	Enter one of the following soil models: LINEAR – linear elastic. NON LINEAR – nonlinear elastic via HYPELA. CAMCLAY – Cam clay model.

Format Fixed	Format Free	Data Type	Entry
4th data block			
1-10	1st	F	Young's modulus of soil.
11-20	2nd	F	Poisson ratio of soil.
21-30	3rd	F	Mass density.
31-40	4th	F	Coefficient of thermal expansion.
41-50	5th	F	Yield strength.
51-60	6th	F	Not used; enter 0.
61-70	7th	F	Bulk modulus of fluid.
71-80	8th	F	Dynamic viscosity of fluid.
5th data block			
1-10	1st	F	Permeability of the soil.
11-20	2nd	F	Virgin compression ratio.
21-30	3rd	F	Recompression ratio.
31-40	4th	F	Slope of critical state line.
6th data block			
			Enter a list of elements associated with this particular soil data.

■ INITIAL POROSITY

Define Initial Porosity for Soil Analysis

Description

This option provides the ability to initialize the porosity throughout the soil analysis model. You can either specify the porosity or use the INITIAL VOID RATIO option to specify the void ratio.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-13	1st	A	Enter the words INITIAL PR.
2nd data block			
1-5	1st	I	Enter the number of sets of data required to define the initial porosity.
6-10	2nd	I	Enter the unit number; defaults to input.
The number of sets is equal to the number given in the first field above.			
3rd data block			
1-10	1st	F	Enter the initial porosity for the points given below at the start of increment zero.
4th data block			
			Enter a list of elements to which the above porosity is to be applied.
5th data block			
This block is not necessary if the CENTROID parameter is used.			
			Enter a list of integration points to which the above porosity is to be applied.
6th data block			
This data block is necessary only when there are either beams or shells in the mesh.			
			Enter a list of layer points to which the above porosity is to be applied.

■ INITIAL VOID RATIO

Define Initial Void Ratio for Soil Analysis

Description

This option provides the ability to initialize the porosity throughout the soil analysis model. You can either specify the void ratio or use the INITIAL POROSITY option to specify the porosity.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-13	1st	A	Enter the words INITIAL VOID.
2nd data block			
1-5	1st	I	Enter the number of sets of data required to define the initial void ratio.
6-10	2nd	I	Enter the unit number; defaults to input.

The number of sets is equal to the number given in the first field above.

3rd data block			
1-10	1st	F	Enter the initial void ratio for the points given below at the start of increment zero.
4th data block			
			Enter a list of elements to which the above void ratio is to be applied.
5th data block			

This block is not necessary if the CENTROID parameter is used.

			Enter a list of integration points to which the above void ratio is to be applied.
6th data block			

This data block is necessary only when there are either beams or shells in the mesh.

			Enter a list of layer points to which the above void ratio is to be applied.

■ INITIAL PC

Define Initial Preconsolidation Pressure

Description

This option provides the ability to define the preconsolidation pressure throughout the model for soil analysis when using the Cam-Clay model.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-13	1st	A	Enter the words INITIAL PC.
2nd data block			
1-5	1st	I	Enter the number of sets of data required to define the preconsolidation pressure.
6-10	2nd	I	Enter the unit number; defaults to input.

The number of sets is equal to the number given in the first field above.

3rd data block			
1-10	1st	F	Initial value of the preconsolidation pressure for the points given below at the start of the zeroth increment.
4th data block			
			Enter a list of elements for which the above data is to be applied.
5th data blockS			

This block is not necessary if the CENTROID parameter is used.

			Enter a list of integration points for which the data is to be applied.
6th data block			

This data block is only necessary if there are either beams or shells in the mesh.

			Enter a list of layer points to which the above data is to be applied.

■ SPECIFIC WEIGHT

Define Specific Weight Constant for Soil Analysis

Description

This option allows you to enter the specific weight constant with respect to the global coordinate system for soil analysis.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words SPECIFIC WEIGHT.
2nd data block			
1-10	1st	F	Enter gravity constant in first coordinate direction.
11-20	2nd	F	Enter gravity constant in second coordinate direction.
21-30	3rd	F	Enter gravity constant in third coordinate direction.

■ INITIAL PORE

Define Initial Pore Pressure for Soil Analysis

Description

This option provides the ability to initialize the pore pressure throughout the model for soil analysis.

Four ways of providing the initial pore pressures are given below.

1. Read the range of elements, integration points and layers, and a corresponding pore pressure.
2. Read the initial values through user subroutine INITPO.
3. Read the initial values from a step of the binary or formatted post output file from a previous pore pressure analysis with MARC. This technique is most common for uncoupled soil analysis to initialize the pore pressure. With this option, the program assumes direct correspondence of the post file elements with the elements in the current analysis. Any spatial interpolation must be provided separately by you.
4. Read a list of elements, integration points and layers and a corresponding pore pressure.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-13	1st	A	Enter the words INITIAL PORE.
2nd data block			
1-5	1st	I	Enter 1.
6-10	2nd	I	Enter 1 to initialize the pore pressure via block 3 series below. See also the third field on this block. Enter 2 to initialize the pore pressure via user subroutine INITPO. This subroutine will now be called in a loop on all elements in the mesh.

Format Fixed	Format Free	Data Type	Entry
			Enter 3 to read the initial values of the pore pressure from the post file written by a previous pore pressure analysis. In this case, the fourth and fifth field must also be defined.
			Enter 4 to initialize the pore pressure via data blocks 5, 6, 7 and 8 given below. See also the third field on this block.
11-15	3rd	I	Only nonzero if the second field is set to 3 or 4. Then, this entry gives the number of pairs of blocks in series 3 and 4 or in series 5, 6, 7, 8 used to input the pore pressure.
16-20	4th	I	Only used if the second field is set to 3. Then this entry defines the unit number from which the post file information from the previous pore pressure run is to be read.
21-25	5th	I	Only used if the second field is set to 3. In that case this entry defines the step number on the pore pressure run post file to be used as the definition of the initial pore pressure values.
26-30	6th		Not used; enter 0.
31-35	7th	I	If option 3 and a formatted post file, are used, enter 1.
36-40	8th	I	Only nonzero if the second field is set to 2.
			Enter 1 to suppress printout of pore pressure values that are initialized in INITPO.

3rd data block

Data blocks 3 and 4 are only input if the second field above is set to 1. In that case, the number of blocks is equal to the number given in the third field above.

1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with the value.
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value.
21-25	5th	I	First layer of cross section point with this value.
26-30	6th	I	Last layer or cross section point with this value can only be bigger than 1 for beam or shell elements.

Format Fixed	Format Free	Data Type	Entry
4th data block			
1-10	1st	F	Initial value of the pore pressure for the above range of points.

Data blocks 5, 6, 7 and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block

1-10	1st	F	Initial pore pressure for the points given below at the start of the zeroth increment.

6th data block

Enter a list of elements to which the above pore pressure is to be applied.

7th data block

This block is not necessary if the CENTROID parameter is used.

Enter a list of integration points to which the above pore pressure is to be applied.

8th data block

This data block is only necessary if there are either beams or shells in the mesh.

Enter a list of layer points to which the above pore pressure is to be applied.

■ CHANGE PORE

Define Pore Pressures for Uncoupled Soil Analysis

Description

This option provides various ways of changing the pore pressure throughout the model. This option is only used in uncoupled soil analysis.

Four ways of providing the pore pressures are given below.

1. Read a range of elements, integration points and layers, and corresponding pore pressures for the end of the current step.
2. Read the pore pressure values for the end of the current step through user subroutine NEWPO.
3. Read the pore pressure values for the end of the current step from a named step of the post file output from a previous pore pressure analysis with MARC. With this option, the program assumes direct correspondence of the post file elements with the elements in the current analysis. Any spatial interpolation must be provided separately by you.
4. Read a list of elements, integration points and layers, and corresponding pore pressure.

   Note: On this option, total pore pressures are input.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-13	1st	A	Enter the words CHANGE PORE.
2nd data block			
1-5	1st	I	Enter 1.
6-10	2nd	I	Enter 1 to change the pore pressure via data block 3 below. In this case, the third field must also be defined. Enter 2 to change the pore pressure via user subroutine NEWPO. This subroutine will then be called in a loop on all the elements in the mesh.

Format Fixed	Format Free	Data Type	Entry
			Enter 3 to read the initial values of the pore pressure from the post file written by a previous pore pressure analysis. In this case, the fourth and fifth field must also be defined.
			Enter 4 to change the pore pressure via data blocks 5, 6, 7 and 8 below.
11-15	3rd	I	Only nonzero if the second field is set to 1 or 4. In that case, this entry gives the number of block set in series 3 and 4 used to input the new value of the pore pressure (optional).
16-20	4th	I	Only nonzero if the second field is set to 3. Then, this entry defines the unit number from which the post file information from the previous pore pressure run will be read.
21-25	5th	I	Only nonzero if the second field is set to 3. In that case, this entry defines the step number on the pore pressure run post file to be read as the definition of the new value of the pore pressure at the end of the current step.
26-30	6th	I	Not used; enter 1.
31-35	7th	I	Enter 1 if a formatted post file is used.
36-40	8th	I	Only nonzero if the second field is set to 2. Set to 1 to suppress printout of pore pressure values that are defined in user subroutine NEWPO.

Data blocks 3 and 4 are only input if the second field above set to 1. In that case, the number of block sets is equal to the number given in the third field above.

3rd data block

1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with this value.
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value can only be bigger than 1 if the ALL POINTS parameter is used.

Format Fixed	Format Free	Data Type	Entry
21-25	5th	I	First layer or cross-section point with this value.
26-30	6th	I	Last layer or cross-section point with this value.
4th data block			
1-10	1st	F	New value of the pore pressure for the above range of points at the end of the current step.

Data blocks 5, 6, 7 and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block			
1-10	1st	F	Pore pressure for the points given below at the end of the current increment.
6th data block			
			Enter a list of elements to which the above pore pressure is to be applied.
7th data block			

This block is not necessary if the CENTROID parameter is used.

Enter a list of integration points to which the above pore pressure is to be applied.

8th data block

This data block is only necessary if there are either beams or shells in the mesh.

Enter a list of layer points to which the above pore pressure is to be applied.

Rate Effects

This section describes the input of material behavior associated with rate effects. There are three models available in MARC.

- The Maxwell Model – invoked by use of the CREEP parameter.
- The Generalized Kelvin Model – invoked by the use of the CREEP parameter.
- In addition, options 1 and 2 can be combined.

For the creep model, the function defining the creep strain rate is given here through the CREEP option. The creep function as given through either the CREEP option or user subroutine CRPLAW defines a deviatoric creep strain increment. In addition, dilatational creep strain can be given through user subroutine VSWELL. For the generalized Kelvin model, user subroutine CRPVIS is used.

There are two methods for defining a viscoplastic material. In these models, the creep and plastic strains are treated together. Using CREEP parameter and model definition option, an explicit procedure is used. User subroutines CRPLAW, NASSOC, ZERO, and YIEL are provided to allow you to define the increment of inelastic strain. Using the "VISCOPLASTIC" yield type on the ISOTROPIC or ORTHOTROPIC option results in an implicit formulation. User subroutine UVSCPL is available to define the increment of inelastic strain.

■ CREEP

Define Creep Constitutive Data

Description

This block of data defines the parameters and material properties used in a creep analysis. The creep data can be specified in either an exponent form or in a piecewise linear curve. Attention is drawn to the existence of subroutine CRPLAW, which allows alternative forms of creep behavior to be programmed indirectly. Further detail on creep is given in *Volume D: User Subroutines and Special Routines* and *Volume A: Theory and User Information*. In addition, the CREEP parameter must be included.

The three possible modes of input of creep constitutive data are:

1. Express the dependence of equivalent creep strain rate on any independent parameter through a piecewise-linear relationship. The equivalent creep strain rate is then assumed to be a piecewise linear approximation to

$$\dot{\bar{\varepsilon}}^c = A \cdot f(\bar{\sigma}) \cdot g(\bar{\varepsilon}^c) \cdot h(T) \cdot \frac{dk(t)}{dt}$$

Note that the function k relates total equivalent creep strain to time. Any of the functions f, g, h, or k can be set to unity by setting the number of slopes to zero for that relation on the input data. This is done using one of two methods. Note that these methods cannot be mixed for different functions (f, g, h, k).

a. The slopes and breakpoints of the piecewise linear functions are given using data blocks 3a, 4a, 5a, and 6a. Note that the independent variable either σ, ε^c, T, or t should be given in ascending order. The format is as follows:

Column	Field	Entry
1-15	1st	Slope of curve.
16-30	2nd	Breakpoint at which slope begins with the number of blocks describing each curve (up to a maximum of five) given in the appropriate field on block 2 of this set.

b. The data points describing the curve of $\bar{\varepsilon}^c$ are given directly using data blocks 3b, 4b, 5b, and 6b. This method is flagged by entering the word "DATA" on the CREEP block. These data points are used to calculate slope breakpoint data. Note that the value of $\bar{\varepsilon}^c$ at the lowest data point should equal the value A. The format is as follows:

Column	Field	Entry
1-15	1st	Value of $\dot{\bar{\varepsilon}}^c$
16-30	2nd	Value of either σ, ε^c, or T for data blocks 3b, 4b or 5b. or for data block 6b

Column	Field	Entry
1-15	1st	Value of $\bar{\varepsilon}^c$
16-30	2nd	Value of t

2. The dependence of equivalent creep strain rate on any independent parameter can be given directly in power law form by giving the appropriate exponent (as a floating-point number) in the first field of blocks 3, 4, 5, or 6. The equivalent creep strain rate is

$$\dot{\bar{\varepsilon}}^c = A\bar{\sigma}^n \cdot (\bar{\varepsilon}^c)^n \cdot T^n \cdot (nt^{n-1})$$

Note that the time dependence is specified as a function of total equivalent creep strain. $\bar{\varepsilon}^c = At^n$ The power law form is indicated by setting the corresponding field on block 2 to -1.

3. For a user-supplied creep law (using subroutine CRPLAW, see *Volume D: User Subroutines and Special Routines*), set the first five fields of block 2 to 0.

Note: The default numerical procedure for creep analysis is explicit. In case of Norton creep, an alternative implicit procedure can be used. This should be set using the CREEP parameter.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-5	1st	A	Enter the word CREEP.
11-80	2nd	A	Enter the word DATA to indicate that option B is being used.

Format Fixed	Format Free	Data Type	Entry
2nd data block			
1-5	1st	I	Number of blocks defining creep-strain rate versus temperature relation.
6-10	2nd	I	Number of blocks defining creep-strain rate versus equivalent stress relation.
11-15	3rd	I	Number of blocks defining creep-strain rate versus equivalent creep-strain curve.
16-20	4th	I	Number of blocks defining total creep-strain increment versus time curve.
21-35	5th	F	Enter the numerical constant in total creep strain relation. Set to zero if a creep law is being supplied through subroutine CRPLAW.
36-50	6th	F	If the entry in the tenth field is 0, enter tolerance on the creep strain increment relative to the elastic strain. Default = 0.50. Note: a higher value is likely to cause stability problems. If the entry in the tenth field is 1, enter the maximum allowable creep strain increment. Default is .01. **Note:** Use of the AUTO CREEP option to input this value is preferred.
51-65	7th	F	If the entry in the tenth field is 0, enter the tolerance on the stress change per stress during creep. Default is 0.10. If the entry in the tenth field is 1, enter the maximum stress increment. Default is 100. This control is included primarily for accuracy purposes. Default value is adequate for creep laws of the type $\varepsilon = a\sigma^n$ where $3 < n < 6$. For lower values of n, tolerance can be increased; for higher values, it should be decreased. **Note:** Use of the AUTO CREEP option to input this value is preferred.
66-70	8th	I	Enter the unit number for input of creep data, defaults to input.

Format Fixed	Format Free	Data Type	Entry
71-75	9th	I	Material id number.
76-80	10th	I	Enter 1 if absolute rather than relative testing is to be performed.

3a data block

Slope and breakpoint data for equivalent creep strain rate versus temperature curve. The number entered in the first field of the second data block defines the number of blocks required in data block 3.

1-15	1st	F	Enter the slope of the curve or the exponent of temperature in the exponential creep law.
16-30	2nd	F	Enter the temperature above which the slope (above) becomes operative. This entry is left blank for exponential creep law.

4a data block

Enter the slope and breakpoint data for equivalent creep strain rate versus equivalent total stress curve. The number entered in the second field of the second data block defines the number of blocks required in data block 4.

1-15	1st	F	Enter the slope of the curve or the exponent of stress in the exponential creep law.
16-30	2nd	F	Enter the equivalent total stress above which the slope becomes operative. This entry is left blank for exponential creep law.

5a data block

Slope and breakpoint data for equivalent creep strain rate versus total equivalent creep strain curve. The number entered in the third field of the second data block defines the number of blocks required in data block 5.

1-15	1st	F	Enter the slope of the curve or the exponent of total equivalent creep strain in the exponential creep law.
16-30	2nd	F	Enter the equivalent total creep strain above which the slope becomes operative. This entry is left blank for exponential creep law.

Format Fixed	Format Free	Data Type	Entry
6a data block			
Slope and breakpoint data for total equivalent creep strain versus time curve. The number entered in the fourth field of the second data block defines the number of blocks required in data block 6.			
1-15	1st	F	Enter the slope of the curve or the exponent of time in the exponential creep law.
16-30	2nd	F	Enter the total time above which the slope becomes operative. This entry is left blank for exponential creep law.
3b data block			
Data points for the equivalent creep strain rate versus temperature curve. The number entered in the first field of the second data block defines the number of blocks required in data block 3.			
1-15	1st	F	Enter the equivalent creep strain rate or the exponent of temperature in the exponential creep law.
16-30	2nd	F	Enter the associated temperature. This entry is left blank for exponential creep law.
4b data block			
Data points for the equivalent creep strain rate versus equivalent total stress curve. The number entered in the second field of the second data block defines the number of blocks required in data block 4.			
1-15	1st	F	Enter the equivalent creep strain rate or exponents of stress in the exponential creep law.
16-30	2nd	F	Enter the associated equivalent total stress. This entry is left blank for exponential creep law.

Format Fixed	Format Free	Data Type	Entry
5b data block			
Data points for the equivalent creep strain rate versus total equivalent creep strain curve. The number entered in the third field of the second data block defines the number of blocks required in data block 5.			
1-15	1st	F	Enter the equivalent creep strain rate or the exponent of total equivalent creep strain in the exponential creep law.
16-30	2nd	F	Enter the associated total creep strain. This entry is left blank for exponential creep law.
6b data block			
Data point for the equivalent creep strain versus time curve. The number entered in the fourth field of the second data block defines the number of blocks required in data block 6.			
1-15	1st	F	Enter the equivalent creep strain or the exponent of time in the exponential creep law.
16-30	2nd	F	Enter the associated total time. This entry is left blank for exponential creep law.

■ PHI-COEFFICIENTS

Define Phi-Coefficients for Rubber Viscoelastic Model

Description

This option allows the input of phi function value vs. frequency for one out of the seven possible PHI functions ϕ_0, ϕ_1, ϕ_2, ϕ_{11}, ϕ_{12}, ϕ_{21}, ϕ_{22}. This option can be repeated up to seven times to completely define the seven PHI functions. These PHI functions are used in a harmonic analysis with rubber materials using the Mooney material model. This can be used only in the total Lagrange formulation. See *Volume A: Theory and User Information* for more detail.

Note: For symmetry of the relaxation data, ϕ_{12} should equal ϕ_{21}.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words PHI-COEFFI.
2nd data block			
1-5	1st	I	Number of different frequencies per PHI function to be read in. If a negative value is entered, user subroutine UPHI will be called to supply PHI coefficients as a function of frequency for all PHI functions.
6-10	2nd	I	PHI function identifier. = 1 for ϕ_0 = 2 for ϕ_1 = 3 for ϕ_2 = 4 for ϕ_{11} = 5 for ϕ_{12} = 6 for ϕ_{21} = 7 for ϕ_{22}
11-15	3rd	I	Material type identifier.
3rd data block *(Not used if user subroutine UPHI requested)*			
1-10	1st	F	Frequency in radians/time unit.
11-20	2nd	F	Real PHI coefficient.
21-30	3rd	F	Imaginary PHI coefficient.

■ VISCELPROP

Define Properties for Isotropic Viscoelastic Materials

Description

This series of blocks is used to specify the time dependent part of the material behavior of a small strain viscoelastic material. Here, only isotropic quantities can be specified. Note that the instantaneous moduli for small-strain viscoelasticity are specified on the ISOTROPIC blocks. Orthotropic time-dependent behavior can be specified using the VISCELORTH option.

Note: Thermo-rheologically simple behavior is specified with the SHIFT FUNCTION option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word VISCELPROP.
2nd data block			
1-5	1st	I	Enter the number of distinct sets of visco-elastic properties to be input.
6-10	2nd	I	Enter the unit number for input of visco-elastic properties, default to input.
3rd data block			
1-5	1st	I	Material type identification number. This number is used for cross reference with the ISOTROPIC, TEMPERATURE EFFECTS, and SHIFT FUNCTION options.
6-10	2nd	I	Maximum number of terms in the Prony series expansion (maximum value of either the deviatoric terms or the volumetric terms).
11-15	3rd	I	Number of terms in the Prony series expansion for deviatoric behavior.
16-20	4th	I	Number of terms in the Prony series expansion for volumetric behavior.

Format Fixed	Format Free	Data Type	Entry
4th data block			
Repeated for the maximum number of terms.			
1-10	1st	F	Shear constant.
11-20	2nd	F	Relaxation time for deviatoric behavior.
21-30	3rd	F	Bulk constant.
31-40	4th	F	Relaxation time for volumetric behavior

■ VISCELORTH

Define Properties for Viscoelastic Orthotropic Materials

Description

This option inputs the time dependent material data used in conjunction with the ORTHOTROPIC option for viscoelastic materials. It can also be used to specify the anisotropic time dependent constants for an anisotropic material exhibiting small strain viscoelastic behavior by use of user subroutine HOOKVI. The instantaneous elastic behavior is specified on the ORTHOTROPIC option.

Note: Since the material properties for orthotropic materials are independent, it is your responsibility to enter all required data. No defaults are provided by MARC.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word VISCELORTH.
2nd data block			
1-5	1st	I	Number of sets of VISCELORTH data to read.
6-10	2nd	I	Unit number for reading data, defaults to input.

The 3rd, 4th, and 5th data blocks are entered as a set, once for each set of VISCELORTH data.

3rd data block			
1-5	1st	I	Material identification number for cross-referencing with ORTHOTROPIC data.
6-10	2nd	I	Number of terms in the Prony series expansion (note that deviatoric and volumetric behavior are treated together).

Format		Data	
Fixed	**Free**	**Type**	**Entry**
4th data block			
1-10	1st	F	Time constant for this term in the Prony series expansion.
11-20	2nd	F	E^n_{xx}
21-30	3rd	F	E^n_{yy}
31-40	4th	F	E^n_{zz}
41-50	5th	F	ν^n_{xy}
51-60	6th	F	ν^n_{yz}
61-70	7th	F	ν^n_{zx}
5th data block			
1-10	1st	F	G^n_{xy}
11-20	2nd	F	G^n_{yz}
21-30	3rd	F	G^n_{zx}

■ VISCELMOON

Define Properties for Large Strain Viscoelastic Materials

Description

This option is used to input the time dependent data for a Mooney-Rivlin viscoelastic material. The instantaneous elastic behavior is specified using the MOONEY model definition option.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word VISCELMOON.
2nd data block			
1-5	1st	I	Number of sets of VISCELMOON data to read.
6-10	2nd	I	Unit number for reading data. Defaults to input.

The 3rd data block is entered once for each set of VISCELMOON data.

3rd data block			
1-5	1st	I	Material id number for cross-referencing with the MOONEY option.
6-10	2nd	I	Number of terms in the Prony series expansion for deviatoric behavior.

The 4th data block is entered once for each term in the Prony series.

4th data block			
1-10	1st	F	Multiplier for energy function.
11-20	2nd	F	Relaxation time.

■ VISCELOGDEN

Define Properties for Large Strain Viscoelastic Ogden Materials

Description

This option is used to input the time dependent data for a Ogden viscoelastic material. The instantaneous elastic behavior is specified using the OGDEN model definition option.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word VISCELOGDEN.
2nd data block			
1-5	1st	I	Number of sets of data to read.
6-10	2nd	I	Unit number for reading data. Defaults to input.
The 3rd data block is entered once for each set of data.			
3rd data block			
1-5	1st	I	Material id number for cross-referencing with the OGDEN option.
6-10	2nd	I	Maximum number of terms of either Prony series expansion.
11-15	3rd	I	Number of terms for deviatoric behavior.
16-20	4rd	I	Number of terms for dilatational behavior.
The 4th data block is entered once for each term in the Prony series.			
4th data block			
1-10	1st	F	Multiplier for deviatoric energy function.
11-20	2nd	F	Deviatoric relaxation time.
21-30	3rd	F	Multiplier for dilatational energy function.
31-40	4rd	F	Dilitational relaxation time.

■ SHIFT FUNCTION

Define Properties for Thermo-Rheologically Simple Viscoelastic Materials

Description

This block allows you to define the shift function parameters for viscoelastic material groups that exhibit thermo-rheologically simple behavior.

Note that for the Narayanaswamy model, the initial value of the fictive temperature for each term must be specified via the CHANGE STATE option.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word SHIFT FUNCTION.
2nd data block			
1-5	1st	I	Number of sets used to define different shift functions.
6-10	2nd	I	Unit number from which the data block will be read. Defaults to block input.

The 3rd and 4th data blocks are entered as pairs, once for each data set.

3rd data block			
1-5	1st	I	Material number for cross-referencing with the instantaneous and time dependent visco-elastic properties.
6-10	2nd	I	Enter the code number denoting the shift function type: 1 = Williams-Landel-Ferry equation 2 = Power series expansion 3 = Narayanaswamy model -N = any negative integer value denotes that the shift function will be specified in the user subroutine.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	If the second field is 2, enter the number of coefficients in the power series representation. If the second field is 3, enter the number of terms in Prony series expansion.
16-25	4th	E	Enter the reference or glass transition temperature T_g, for this shift function. For the Narayanaswamy model, enter the temperature for stress relaxation data.
26-35	5th	E	For the Narayanaswamy model, enter the activation energy divided by the gas constant H/R.
36-45	6th	E	For the Narayanaswamy model, enter the fraction parameter.
46-55	7th	E	For the Narayanaswamy model, enter the temperature shift between your temperature and absolute temperature for calculating fictitive temperatures.
56-65	8th	E	For the Narayanaswamy model, enter the reference temperature for the structural relaxation data.

If Williams-Landel-Ferry form, use the following block.

4th data block

1-10	1st	F	Enter the constant C^1.
11-20	2nd	F	Enter the constant C^2.

If power series expansion, use the following block.

If shift function is defined in user subroutine TRSFAC, the 4th data block is not required.

4th data block

1-80	1st	F	Enter the constants C_o to C_m in increasing order of subscript, using additional blocks if necessary to define all constants.

If Narayanaswamy model, use the 4th and 5th data block.

4th data block

1-80	1st	F	Enter the weighting factors W_g in increasing order of subscript. Use additional blocks if necessary to define all constants.

5th data block

1-80	1st	F	Enter the relaxation time $\tau_{i,ref}$ in increasing order of subscript. Use additional blocks if necessary to define all constants.

■ VISCEL EXP

Viscoelastic Thermal Expansion

Description

This option is used to define the thermal expansion behavior often observed in viscoelastic materials. It is used in conjunction with the viscoelastic material models and the Narayanaswamy thermal rheologically simple shift function. The fictive temperature is stored in the second state variable and can be postprocessed by selecting post code 29.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words VISCEL EXP.
2nd data block			
1-5	1st	I	Enter the number of materials to be defined.
6-10	2nd	I	Enter the unit number for input. Defaults to input.
3rd data block			
1-5	1st	I	Material number for cross-referencing with the instantaneous and time dependent viscoelastic properties.
6-15	2nd	E	Enter the solid coefficient of thermal expansion, α_g.
16-25	3rd	E	Enter the liquid coefficient of thermal expansion, α_l.

Dynamic Analysis

This section describes the data input required for dynamic analysis. There are several options available for specifying the initial conditions of the problem. The mass density of the object is specified in the ISOTROPIC, ORTHOTROPIC, and ANISOTROPIC options. The program uses this to calculate a consistent mass matrix, which can be converted into a diagonal mass matrix by using the LUMP parameter. This is not recommended for either higher order elements or shells and beams. In addition, you can apply concentrated masses to particular degrees of freedom using options given in this section. Damping and dashpot can be applied using the DAMPING and SPRINGS options. Note that stiffness damping should not be applied to either the Herrmann elements or the gap elements.

In addition, the FLUID SOLID option is included so that you can specify the interface between the fluid and solid boundary.

If a response spectrum analysis is to be performed, the spectral density is provided through the RESPONSE SPECTRUM option.

■ DAMPING

Define Damping Factors

Description

This option allows the input of damping factors for use with the dynamic analysis options. Two damping inputs are available depending on your choice of dynamic option. For modal superposition analysis, you give the fraction of critical damping associated with each mode of the solution. For direct integration options, you input the factors weighting the mass and stiffness matrices to form the damping matrix. In both cases, the damping matrix is assumed to be formed as a linear combination of the mass and stiffness matrices of the system, see *Volume A: Theory and User Information.*

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-7	1st	A	Enter the word DAMPING.
			For modal analysis, the second data block is used.
			For direct integration analysis, damping is given on a element basis, data blocks 3, 4, and 5 are used.
2nd data block			
For modal analysis only.			
1-10	1st	F	Fraction of critical damping for 1st mode.
11-20	2nd	F	Fraction of critical damping for 2nd mode.
Etc.	Etc.	Etc.	Etc.

Format Fixed	Format Free	Data Type	Entry
3th data block			
For direct integration (Newmark-beta, Houbolt or Central Difference).			
1-5	1st	I	Number of damping sets (NDMPST) to be read in. The 4th and 5th data blocks are given in pairs NDMPST times.
6-10	2nd	I	Enter the unit number for input of the damping data, defaults to input.
4th data block			
1-10	1st	F	Multiplier (α) for mass matrix contribution to damping matrix.
11-20	2nd	F	Multiplier (β) for stiffness matrix contribution to damping matrix.
21-30	3rd	F	Multiplier (γ) for numerical damping.
5th data block			
1-5	1st	I	First element to have these damping values.
6-10	2nd	I	Last element to have these damping values.

In a dynamic analysis, the damping matrix is evaluated as

$$C = \alpha M + \left(\beta + \frac{\gamma \Delta t}{\pi}\right) K$$

In a harmonic analysis, the damping matrix is evaluated as

$$C = \alpha M + \left(\beta + \frac{2\gamma}{\omega}\right) K$$

■ FLUID SOLID

Define Fluid-Solid Interface

Description

This option is used with the added mass approach to fluid-solid problems. In such analysis, the fluid is considered incompressible and inviscid. This model definition set is necessary to identify the element faces in the solid which abut the fluid. Note that with this feature, the fluid density is entered on the ISOTROPIC option in the Young's modulus field for the fluid elements.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-11	1st	A	Enter the words FLUID SOLID.
2nd data block			
One data line per solid/fluid interface pair – for each abutting pair specify:			
1-5	1st	I	Solid element number.
6-10	2nd	I	Solid element face number (as identified for distributed load option).
11-15	3rd	I	Fluid element number.
16-20	4th	I	Fluid element face number (as identified in distributed flux option).

■ INITIAL DISP

Define Initial Displacements

Description

This data block provides initial displacements for dynamic problems or a spatially varying interference fit in contact analyses.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words INITIAL DISP.
2nd data block			
1-5	1st	I	Enter the number of sets of prescribed displacements (optional). If a -1 is entered, user subroutine USINC is used for all nodes. Data blocks 3 and 4 are not required.
6-10	2nd	I	Enter file number for input of prescribed displacement data, defaults to input.
3rd data block			
1-10	1st	F	Initial displacement in first degree of freedom or the interference fit normal to the contact surface.
11-20	2nd	F	Initial displacement in second degree of freedom.
21-30	3rd	F	Initial displacement in third degree of freedom.
31-40	4th	F	Initial displacement in fourth degree of freedom.
41-50	5th	F	Initial displacement in fifth degree of freedom.
51-60	6th	F	Initial displacement in sixth degree of freedom.
61-70	7th	F	Initial displacement in seventh degree of freedom.
71-80	8th	F	Initial displacement in eighth degree of freedom.

Continuation data lines, if necessary, must be in 6E10.3 format. Continuation data lines are needed if there are more than eight degrees of freedom per node in the analysis.

4th data block			
			Enter list of nodes for which the above initial displacements are applied.

■ INITIAL VEL

Define Initial Velocity

Description

This option allows the input of initial velocity for dynamic problems.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words INITIAL VEL.
2nd data block			
1-5	1st	I	Enter the number of sets of prescribed velocity (optional). Enter -1 if user subroutine USINC is to be used for all nodes. In this case, data blocks 3 and 4 are not used.
6-10	2nd	I	Enter unit number for input of prescribed velocity data, defaults to input.

The 3rd and 4th data blocks are entered as pairs, one for each data set.

3rd data block			
1-10	1st	E	Initial velocity in first degree of freedom.
11-20	2nd	E	Initial velocity in second degree of freedom.
21-30	3rd	E	Initial velocity in third degree of freedom.
31-40	4th	E	Initial velocity in fourth degree of freedom.
41-50	5th	E	Initial velocity in fifth degree of freedom.

Format		Data	
Fixed	Free	Type	Entry
51-60	6th	E	Initial velocity in sixth degree of freedom.
61-70	7th	E	Initial velocity in seventh degree of freedom.
71-80	8th	E	Initial velocity in eighth degree of freedom.

Continuation data lines, if necessary, must be in 6E10.3 format. Continuation data lines are needed if there are more than eight degrees of freedom per node in the analysis.

4th data block

Enter list of nodes for which the above initial velocities are applied.

■ FIXED ACCE

Define Fixed Acceleration

Description

This option defines the fixed accelerations that each specified degree of freedom must take during the first and subsequent increments, unless it is further modified using the ACC CHANGE option. This option can only be used in dynamic analyses. It is usually used to prescribe base motion accelerations.

Further detail is given in *Volume A: Theory and User Information*. See *Volume B: Element Library*, for a definition of the degrees of freedom for each element type.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words FIXED ACCE.
2nd data block			
1-5	1st	I	Number of sets of boundary condition data lines to be read (optional).

For each set of boundary conditions, use the 3rd, 4th and 5th data blocks.

3d data block			
1-10	1st	E	Prescribed acceleration for first degree of freedom listed in data block 4.
11-20	2nd	E	Prescribed acceleration for second degree of freedom listed in data block 4.
21-30	3rd	E	Prescribed acceleration for third degree of freedom listed in data block 4.

Format Fixed	Format Free	Data Type	Entry
4th data block			
			Enter a list of degrees of freedom to which the above prescribed displacements are prescribed. **Note:** List verbs `EXCEPT` and `INTERSECT` are illegal here.
5th data block			
			Enter a list of nodes to which the above accelerations are applied.

■ MASSES

Define Concentrated Masses

Description

This option is used to input any concentrated masses for use with the dynamic analysis options. Each concentrated mass is associated with a single degree of freedom.

Note: Rotational degrees of freedom might have mass depending on the element types used in the data.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-6	1st	A	Enter the word MASSES.
2nd data block			
1-5	1st	I	Number of sets of data used to enter mass points (optional).
6-10	2nd	I	Enter unit number for reading mass point data. Defaults to input.
3rd data block			
1-5	1st	I	Degree of freedom to which mass is applied.
6-10	2nd	F	Value of concentrated mass.
4th data block			
			Enter a list of nodes having the above concentrated masses.

■ RESPONSE SPECTRUM

Define Density for Spectral Response

Description

This option allows you to define the response spectral density. Note that the RESPONSE parameter must also be included. A spectrum response calculation is performed based on the last set of extracted modes when a SPECTRUM load incrementation block is encountered.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-20		A	Enter the words RESPONSE SPECTRUM.
2nd data block			
1-5		I	Enter the number of spectral density pairs (optional).
6-10		I	Enter the unit number to read data. Default is to input.
3rd data block			
1-10		F	Enter the frequency in cycles per time unit.
11-20		F	Enter the displacement spectral response density.

■ MODAL INCREMENT

Define Increments for Eigenvalue Extraction

Description

This option allows you to specify at which increments an eigenvalue extraction is performed. It can be used as either a replacement to the MODAL SHAPE history definition option or in conjunction with it. This option allows you to extract modes within an AUTO LOAD, AUTO CREEP, or AUTO INCREMENT period. Note that the increment numbers specified here cannot be changed upon restart.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-20	1st	A	Enter the words MODAL INCREMENT.
2a data block			
Data block 2a is used if the inverse power sweep method is selected on the DYNAMIC parameter.			
1-5	2nd	I	Maximum number of iterations per mode in the power sweep. Maximum number of iterations for all modes if subspace iteration is used. Default is 40.
6-15	3rd	F	Convergence tolerance. The power sweep terminates when the difference between the eigenvalues in two consecutive sweeps divided by the eigenvalue is less than the tolerance. Default is 0.0001.
16-25	4th	F	Initial shift in cycles per time. The power shift is likely to start converging to the eigenvalue closest to this value. Default is 0.

Format Fixed	Format Free	Data Type	Entry
26-35	5th	F	Maximum frequency to be extracted in cycles per time. If this is left blank or zero, the number of modes requested on the DYNAMIC parameter are extracted. If this is nonzero, the extraction ends when this frequency is exceeded or when the number of modes requested on the DYNAMIC parameter is reached, whichever occurs first.
36-40	6th	I	Number of modes extracted per shift. This data field determines if auto shifting occurs. If auto shift is not required, set equal to or greater than number of modes requested on DYNAMIC parameter. Default is 5.
41-50	7th	F	Auto shift parameter. The program determines the new shift point (in frequency squared) as the highest frequency square plus this entry times the difference between the highest and next highest distinct frequency squared. Default is 1.0.
51-55	8th	I	Enter 1 if eigenvectors are to be written to the post file.

2b data block

Data block 2b is used if the Lanczos method is selected on the DYNAMIC parameter.

1-10	1st	F	SHFMIN, lowest frequency of mode to be extracted (in cycles/time). This is also the initial shift point. The shift point is SHFMIN*SHFMIN. If a negative frequency is given, shift point is -SHFMIN*SHFMIN. This cannot be changed upon restart.
11-20	2nd	F	SHFMAX, highest frequency of mode to extract. If set to 0, NSNRM modes are extracted. This can be changed upon restart.
21-25	3rd	I	Number of requested modes (NSNRM). If zero, all modes between SHFMIN and SHFMAX are extracted. This can be increased upon restart.
26-30	4th	I	Sturm sequence flag. Enter 1 to perform Sturm sequence check on the extracted eigenvalues. Enter 0 to specify no checking.

Format Fixed	Format Free	Data Type	Entry
31-35	5th	I	Restart option flag. 0 do not write modes onto restart file. 1 write previously extracted modes onto restart file. 2 read modes from restart file. 3 read previously extracted modes and write new modes. Note the RESTART model definition option must also be included.
36-40	6th	I	Enter 1 if eigenvectors are to be written to the post file.
3rd data block			
			Enter a list of increment numbers at which modes are to be extracted.

■ BUCKLE INCREMENT

Define Increments for Buckling Analysis

Description

This option allows you to specify at which increments a buckling analysis is performed. It can be used as either a replacement to the BUCKLE history definition option or in conjunction with it. This option allows you to extract modes within an AUTO LOAD, AUTO CREEP, or AUTO INCREMENT period. Note that the increment numbers specified here cannot be changed upon restart.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-20	1st	A	Enter the words BUCKLE INCREMENT.
2nd data block			
1-5	1st	I	Maximum number of iterations per mode in the power sweep. Default is 40.
6-15	2nd	F	Convergence tolerance. The power sweep terminates when the difference between the eigenvalues in two consecutive sweeps divided by the eigenvalue is less than the tolerance. Default is 0.0001.
16-20	3rd	I	Enter 1 if Fourier Buckling is to be performed.
21-25	4th	I	Enter 1 if the eigenvectors are to be written to the POST file.
26-30	5th	I	Enter 1 for automatic buckling perturbation. Enter 2 for manual buckling perturbation.
31-35	6th	I	Enter buckling mode to be used in perturbation.
36-45	7th	F	Enter the scale factor to be multiplied with the normalized buckle mode and added to coordinates.

Format		Data	
Fixed	Free	Type	Entry
3rd data block			
Used only with Fourier buckling.			
			Enter a list of Fourier harmonics at which to perform extractions. Positive numbers are symmetric modes; negative numbers are antisymmetric modes.
4th data block			
			Enter a list of increment numbers at which modes are to be extracted.

Heat Transfer Analysis

This section describes the input of material data and boundary conditions applicable for heat transfer problems. The boundary conditions discussed in this section are also used for coupled thermal-mechanical problems or coupled fluid-thermal problems. The ISOTROPIC, ORTHOTROPIC, and ANISOTROPIC options are used to define the conductivity, specific heat and density. If these material properties are influenced with temperature, this variation can be prescribed by the TEMPERATURE EFFECTS or ORTHO TEMP options. In problems where temperature effects are important, a steady-state analysis performed in one increment requires recycling. A transient analysis recycles and reassembles based upon the tolerances given in the CONTROL option. The initial temperatures can be prescribed using INITIAL TEMP option. Surface, volumetric or nodal fluxes can be prescribed and convective boundary conditions can be imposed through the FILMS option.

■ FIXED TEMPERATURE

Define Fixed Temperature

Description

This data defines the fixed temperature that each node must take during the first and subsequent increments, unless it is further modified using the TEMP CHANGE option. The boundary conditions are specified either by giving the temperature and a list of nodal numbers, or by the input of boundary conditions generated during mesh generation (MESH2D).

Note: The boundary conditions are stored after the connectivity and coordinate data on this file, so that the model definitions must be arranged accordingly.

For each set of boundary conditions, use the 3a and 4a data blocks or the 3b, 4b, and 5b blocks.

(3a); (4a) For analyses which do not include heat transfer shell elements.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-19	1st	A	Enter the words FIXED TEMPERATURE.
2nd data block			
1-5	1st	I	Number of sets of boundary conditions to be read (optional).
6-10	2nd	I	This field is set to nonzero to flag input of boundary conditions set during mesh generation (MESH2D). If this field is activated, no further data lines are required in this option.
11-15	3rd	I	Unit number used for MESH2D option.

Use 3a,4a for analyses which do not include heat transfer shell elements.

3a data block			
1-10	1st	F	Prescribed temperature.
4a data block			
			Enter a list of nodes for which the above temperature is applied.

Format Fixed	Format Free	Data Type	Entry
3b data block			
Use 3b, 4b, and 5b for analyses which include heat transfer shell elements.			
1-10	1st	F	Prescribed temperature for first degree of freedom listed in data block 4b.
11-20	2nd	F	Prescribed temperature for second degree of freedom listed in data block 4b.
21-30	3rd	F	Prescribed temperature for third degree of freedom listed in data block 4b. (See *Volume B: Element Library* for the definition of nodal degrees of freedom.)
4b data block			
			Enter a list of degrees of freedom to which the above prescribed temperatures are given. **Note:** List verbs `EXCEPT` and `INTERSECT` are illegal here.
5b data block			
			Enter a list of nodes for which the above boundary conditions are applied.

■ FILMS

Define Convection Film Coefficient Input

Description

This option allows film coefficients and associated sink temperatures to be input. Nonuniform films or sink temperatures can be obtained via user subroutine FILM (see *Volume D: User Subroutines and Special Routines*).

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-5	1st	A	Enter the word FILMS.
2nd data block			
1-5	1st	I	Number of sets of data used to input film (optional).
6-10	2nd	I	Unit number for input of film data, defaults to input.
3rd data block			
1-5	1st	I	Face identification. Same as for user subroutine FLUX – see *Volume B, Element Library.*
6-15	2nd	F	Reference value of film coefficient.
16-25	3rd	F	Reference value of sink temperature (reference values can be modified by user subroutine FILM).
26-30	4th	I	Film coefficient index (optional).
31-35	5th	I	Sink temperature index (optional). (Film coefficient and sink temperature indices are to be used in user subroutine FILM).
4th data block			
			Enter a list of elements to which the above film data is applied.

■ DIST FLUXES

Define Distributed Fluxes

Description

This option allows distributed (surface and volumetric) fluxes to be specified. Distributed fluxes are converted to consistent nodal fluxes by the program. Note that for a given element type, there is an established convention for the application of surface flux on a particular face. User subroutine FLUX can be used to input time and spatial dependent fluxes.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words DIST FLUXES.
2nd data block			
1-5	1st	I	Enter the number of sets of distributed fluxes to be entered (optional).
6-10	2nd	I	Enter unit number for input of distributed flux data, defaults to input.
3rd data block			
1-5	1st	I	Parameter identifying the type of load. See library element description in *Volume B: Element Library.*
6-15	2nd	F	Enter the magnitude of this type of distributed fluxes.
16-20	3rd	I	Flux index (optional). (Flux index is to be used in user subroutine FLUX.)
4th data block			
			Enter a list of elements associated with the above distributed fluxes.

■ POINT FLUX

Define Point Fluxes

Description

This option allows total nodal point fluxes to be specified. User subroutine FORCDT can be used for the time dependent fluxes. Enter an upper bound to the number of nodes with point fluxes on the FLUXES parameter.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words POINT FLUX.
2nd data block			
1-5	1st	I	Enter the number of sets of point fluxes to be entered (optional).
6-10	2nd	I	Enter unit number for input of point flux data, defaults to input.
3rd data block			
1-10	1st	F	Magnitude of point flux.
11-20	2nd	F	Magnitude of point flux for second degree of freedom, (heat transfer shell elements only).
21-30	3rd	F	Magnitude of point flux for third degree of freedom, (heat transfer shell elements only).
4th data block			
			Enter a list of nodes to which the above nodal sources are applied.

■ INITIAL TEMP

Define Initial Temperatures

Description

This option provides initial temperatures for heat transfer problems.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words INITIAL TEMP.
2nd data block			
1-5	1st	I	Enter the number of sets of prescribed temperatures (optional). Enter -1 if user subroutine USINC is used. In this case, data blocks 3 and 4 are not used.
6-10	2nd	I	Enter unit number for input of prescribed temperatures data, defaults to input.
11-15	3rd	I	Flag to indicate that initial conditions read from previously generated post file. Set to 1.
16-20	4th	I	Only nonzero if the third field is set to 1. Then this entry defines the unit number which the post file information is read.
21-25	5th	I	Enter step number to be read.
26-30	6th	I	Enter 1 if a formatted post file is used. ***For Cray Only:*** Set to 2 for IEEE binary file Set to 3 for VAX binary file Set to 4 for IBM binary file
3a data block			
For analyses which do not include heat transfer shell elements:			
1-10	1st	F	Initial temperature.

Format		Data	
Fixed	Free	Type	Entry
3b data block			
For analyses which include heat transfer shell elements:			
1-10	1st	F	Initial temperature in first degree of freedom.
11-20	2nd	F	Initial temperature in second degree of freedom.
21-30	3rd	F	Initial temperature in third degree of freedom.
			Note: See *Volume B: Element Library* for the definition of nodal degrees of freedom.
4th data block			
			Enter list of nodes for which the above initial temperature is applied.

■ ISOTROPIC

Define Thermal Properties for Isotropic Materials

Description

This option allows you to define thermal properties for an isotropic material. You can also associate these material properties with a list of element numbers.

To define the dependence of these properties on temperature, use the TEMPERATURE EFFECTS model definition option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word ISOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of sets of isotropic material data to follow (optional).
6-10	2nd	I	Enter the unit number for input. Defaults to input file.
3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-reference to TEMPERATURE EFFECTS option.
6-10	2nd	I	Enter 1 to call user subroutine ANKOND and ORIENT.
46-55	8th	A	Enter the material name to cross-reference with material data base for temperature dependent properties.

The data entered in data blocks 4 and 5 should be the values at the lowest temperature expected during an analysis, not necessarily at the initial temperature.

4th data block			
1-10	1st	F	Thermal conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density.
31-40	4th	F	Resistivity (for Joule heating analysis).
41-50	5th	F	Emissivity (for radiating cavities).
5th data block			
			Enter a list of elements associated with this material.

Note: Do not enter composite elements which use this material in its layers.

■ ORTHOTROPIC

Define Thermal Properties for Orthotropic Materials

Description

This option allows you to define thermal properties for an orthotropic material. You can also associate these material properties with a list of element numbers.

To define the dependence of these properties on temperature, use the ORTHO TEMP model definition.

Note: Since the material properties in an orthotropic material are independent, it is your responsibility to enter all data required to match the dimension of the conductivity matrix for the elements listed below. (See *Volume B: Element Library,* if necessary). No defaults for this data are provided by MARC.

The data entered in the following blocks should be the values at the lowest temperature expected during an analysis, not necessarily at the initial temperature.

These values are with respect to the user coordinate (1, 2, 3) system.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word ORTHOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of sets of orthotropic material data to follow (optional).
6-10	2nd	I	Enter the unit number for input. Defaults to input file.
3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-reference to ORTHO TEMP option.
6-10	2nd	I	Enter 1 to call user subroutines ANKOND and ORIENT.

Format Fixed	Format Free	Data Type	Entry
4th data block			
1-10	1st	F	K_{11} Thermal conductivities.
11-20	2nd	F	K_{22} Thermal conductivities.
21-30	3rd	F	K_{33} Thermal conductivities.
31-40	4th	F	ρ Mass density.
41-50	5th	F	Specific heat per unit mass.
51-60	6th	F	R_{11} If Joule heating analysis, resistivities.
61-70	7th	F	R_{22} If Joule heating analysis, resistivities.
71-80	8th	F	R_{33} If Joule heating analysis, resistivities.
5th data block			
Only required if radiating cavities are included.			
1-10	1st	F	Emissivity.
6th data block			
			Enter a list of elements associated with this material. (Do not enter composite elements that use this material in their layers.)

■ ANISOTROPIC

Model Definition Block for Heat Transfer Analysis

Description

This option specifies thermal properties defined by a call to user subroutines ANKOND and ORIENT. The subroutine ANKOND must be used for the input of constant or temperature dependent anisotropic thermal conductivities (K_{11}, K_{22}, K_{33}) and/or resistives (R_{11}, R_{22}, R_{33}) defined in the user coordinate (1,2,3) system. The TEMPERATURE EFFECTS model definition option can be used for the input of variations of specific heat with temperatures. Note that the data entered in this option should be the values at the lowest temperature expected during an analysis, not necessarily at the initial temperature.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word ANISOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of anisotropic data sets to follow (optional)
6-10	2nd	I	Enter the unit number for input. Defaults to input file.
Data blocks 3,4, and 5 are repeated as a set NSET times.			
3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing ANKOND and ORIENT, as well as to the TEMPERATURE EFFECTS option.
6-10	2nd	I	Enter 1 if the user subroutine ANKOND is to be called. Enter 2 if the anisotropic conductivity is to be entered in the 4a data block.

Format Fixed	Format Free	Data Type	Entry
4th data block			
1-10	1st	F	Mass density.
11-20	2nd	F	Specific heat per unit mass.
21-30	3rd	F	Emissivity.
Data block 4a only required if the second fields 6-10 is a "2"			
4a data block			
1-10	1st	F	K_{11}
11-20	2nd	F	K_{12}
21-30	3rd	F	K_{13}
31-40	4th	F	K_{22}
41-50	5th	F	K_{23}
51-60	6th	F	K_{33}
5th data block			
			Enter a list of elements associated with this material. (Do not enter composite elements which use this material in its layers.)

■ TEMPERATURE EFFECTS

Define Variation of Element Properties in Heat Transfer Analysis

Description

This option defines the variation of element properties (conductivity, specific heat and electrical resistance) with temperature. The values read in through the ISOTROPIC option are those at the lowest temperature specified. Properties are not defined below the lowest temperature. The temperature dependency can be entered using one of the following two options:

A. The variation of a particular property with temperature is specified as a piecewise linear curve. Breakpoints must be given in ascending order of temperature.

B. The particular property value and temperature lying on the relevant curve are input directly. Data points must be given in ascending order of temperature. This option is flagged by entering the word "DATA" on the first data line.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-19	1st	A	Enter the word TEMPERATURE EFFECTS.
21-80	2nd	A	Enter the word DATA to indicate that Option B is used.

For option A, use data blocks 2a, 3a, 4a, 5a, and 6a. For option B, use data blocks 2b, 3b, 4b, 5b, and 6b.

Format Fixed	Format Free	Data Type	Entry

Option A

2a data block

1-5	1st	I	Number of slopes of conductivity versus temperature curve.
6-10	2nd	I	Number of slopes of specific heat versus temperature curve.
11-15	3rd	I	Number of latent heats to be entered.
16-20	4th	I	Number of slopes of resistivity versus temperature curve for Joule heating problem.
21-25	5th	I	Number of slopes for emissivity versus temperature curve for radiating cavities.
26-30	6th	I	Not used; enter 0.
31-35	7th	I	Material type identification (1,2,3...) for cross-reference to the ISOTROPIC option.
36-40	8th	I	Logical unit number for input of this set of data. Defaults to blocks.

3a data block

Conductivity variation. Number of blocks as given on data block 2, first field.

1-15	1st	F	Slope of conductivity versus temperature curve.
16-30	2nd	F	Temperature above which above slope becomes operative.

4a data block

Specific heat variation. Number of blocks as given on data block 2, second field.

1-15	1st	F	Slope of specific heat versus temperature curve.
16-30	2nd	F	Temperature above which above slope becomes operative.

Format Fixed	Format Free	Data Type	Entry
5a data block			
Latent heat. Number of blocks given on data block 2, third field.			
1-15	1st	F	Latent heat.
16-30	2nd	F	Solidus temperature (lower phase change limit).
31-45	3rd	F	Liquidus temperature (upper phase change limit).
6a data block			
Resistivity variation for Joule heating problem. Number of blocks given on data block 2, fourth field.			
1-15	1st	F	Slope of resistivity versus temperature curve.
16-30	2nd	F	Temperature above which above slope becomes operative.
7a data block			
Emissivity variation for radiating cavity problems. Number of blocks given on data block 2, fifth field.			
1-15	1st	F	Slope of emissivity versus temperature curve.
16-30	2nd	F	Temperature above which the above slope becomes operative.

Option B

2b data block			
1-5	1st	I	Number of data points on the conductivity versus temperature curve.
6-10	2nd	I	Number of data points on the specific heat versus temperature curve.
11-15	3rd	I	Number of latent heats to be entered.
16-20	4th	I	Number of data points on the resistivity versus temperature curve for Joule heating problem.
21-25	5th	I	Number of data points on the emissivity versus the temperature curve.

Format Fixed	Format Free	Data Type	Entry
26-30	6th	I	Not used; enter 0.
31-35	6th	I	Material type identification (1,2,3...) for cross-reference to ISOTROPIC option.
36-40	7th	I	Logical unit number for input of this set of data. Defaults to blocks.

3b data block

Conductivity variation. Number of blocks as given on data block 2, first field.

1-15	1st	F	Enter the value of the conductivity.
16-30	2nd	F	Enter the associated temperature.

4b data block

Specific heat variation. Number of blocks as given on data block 2, second field.

1-15	1st	F	Enter the value of the specific heat.
16-30	2nd	F	Enter the associated temperature.

5b data block

Latent heat. Number of blocks given on data block 2, third field.

1-15	1st	F	Latent heat.
16-30	2nd	F	Solidus temperature (lower phase change limit).
31-45	3rd	F	Liquidus temperature (upper phase change limit).

6b data block

Resistivity variation for Joule heating problem. Number of blocks given on data block 2, fourth field.

1-15	1st	F	Enter the value of the resistivity.
16-30	2nd	F	Enter the associated temperature.

7b data block

Emissivity variation for radiating cavity problem. Number of blocks given on data block 2, fifth field.

1-15	1st	F	Enter the value of the emissivity.
16-30	2nd	F	Enter the associated temperature.

Note: In calculating a particular temperature dependent property, the program averages the value of this property at the start and at the end of the increment. The temperature at the end of the increment is an estimated value.

■ ORTHO TEMP

Define Variation of Orthotropic Thermal Properties

Description

This option defines the variation of all orthotropic thermal properties with temperature. Note that the values read in through the ORTHOTROPIC model definition are those at the lowest temperature defined. Properties at temperatures below this temperature are defined to be equal to properties at this temperature.

The variation of a particular property is defined as a piecewise linear curve. Two options are available to define this curve.

A. Slope/breakpoint data in ascending order of temperature can be given.

B. Property value/temperature data in ascending order of temperature can be given. This option is flagged by entering the word "DATA" after the string ORTHO TEMP on the first data block.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the string ORTHO TEMP.
11-80	2nd	A	Enter the word DATA to indicate that option B defined above is to be used. **Note:** For option A, use data block 2a-10a. For option B, use data block 2b-10b.

Option A

2a data block			
1-5	1st	I	Number of slopes of K_{11} vs. temperature curve.
6-10	2nd	I	Number of slopes of K_{22} vs. temperature curve. Enter -1 to have (K_{22} vs. temp.) ≡ (K_{11} vs. temp.).

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Number of slopes of K_{33} vs. temperature curve. Enter -1 to have (K_{33} vs. temp.) ≡ (K_{11} vs. temp.).
16-20	4th	I	Number of slopes of specific heat vs. temperature curve.
21-25	5th	I	Number of latent heats. Column Format Entry.
26-30	6th	I	In a Joule heating analysis, number of slopes of R_{11} vs. temperature curve.
31-35	7th	I	In a Joule heating analysis, number of slopes of R_{22} vs. temperature curve. Enter -1 to have (R_{22} vs. temp.) ≡ (R_{11} vs. temp.).
36-40	8th	I	In a Joule heating analysis, number of slopes of R_{33} vs. temperature curve. Enter -1 to have (R_{33} vs. temp.) ≡ (R_{11} vs. temp.).
41-45	9th	I	Number of slopes of emissivity vs. temperature curve for radiating cavities.
46-50	10th	I	Enter the material identification (1,2,3, etc.) for this data set.
51-55	11th	I	Enter the unit number for input. Defaults to input file.

3a data block

The number of blocks in this series is the number in the 2a data block, first field.

1-15	1st	F	Enter the slope of K_{11} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

4a data block

The number of blocks in this series is n, the number in the 2a data block, second field, or 0 if n = -1.

1-15	1st	F	Enter the slope of K_{22} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

Format Fixed	Format Free	Data Type	Entry
5a data block			
The number of blocks in this series is n, the number in the 2a data block, third field, or 0 if n = -1.			
1-15	1st	F	Enter the slope of K_{33} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.
6a data block			
The number of blocks in this series is the number in the 2a data block, fourth field.			
1-15	1st	F	Enter the slope of specific heat vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.
7a data block			
The number of blocks in this series is the number in the 3a data block, fifth field.			
1-15	1st	F	Enter latent heat value.
16-30	2nd	F	Solidus temperature (lower phase change limit).
31-45	3rd	F	Liquidus temperature (upper phase change limit).
8a data block			
The number of blocks in this series is the number in the 2a data block, sixth field.			
1-15	1st	F	Enter the slope of R_{11} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.
9a data block			
The number of blocks in this series is n, the number in the 2a data block, seventh field, or 0 if n = -1.			
1-15	1st	F	Enter the slope of R_{22} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.

Format Fixed	Format Free	Data Type	Entry
10a data block			
The number of blocks in this series is n, the number in the 2a data block, eighth field, or 0 if n = -1.			
1-15	1st	F	Enter the slope of R_{33} vs. temperature curve.
16-30	2nd	F	Temperature at which above slope becomes operative.
11a data block			
The number of block in this series is the number in the 2a data block, ninth field.			
1-15	1st	F	Enter the slope of emissivity vs. temperature curve.
16-30	2nd	F	Temperature at which the above slope becomes operative.

Option B

2b data block

1-5	1st	I	Number of data points of K_{11} vs. temperature curve.
6-10	2nd	I	Number of data points of K_{22} vs. temperature curve. Enter -1 to have (K_{22} vs. temp.) ≡ (K_{11} vs. temp.)
11-15	3rd	I	Number of data points of K_{33} vs. temperature curve. Enter -1 to have (K_{33} vs. temp.) ≡ (K_{11} vs. temp.)
16-20	4th	I	Number of data points of specific heat vs. temperature curve.
21-25	5th	I	Number of latent heats.
26-30	6th	I	In a Joule heating analysis, number of data points of R_{11} vs. temperature curve.
31-35	7th	I	In a Joule heating analysis, number of data points of R_{22} vs. temperature curve. Enter -1 to have (R_{22} vs. temp.) ≡ (R_{11} vs. temp.).
36-40	8th	I	In a Joule heating analysis, number of data points of R_{33} vs. temperature curve. Enter -1 to have (R_{33} vs. temp.) ≡ (R_{11} vs. temp.).

Format Fixed	Format Free	Data Type	Entry
41-45	9th	I	Number of data points of emissivity versus temperature curve.
46-50	10th	I	Enter the material identification (1, 2, 3, etc.) for this data set.
51-55	11th	I	Enter the unit number for input. Defaults to input file.

3b data block

The number of blocks in this series is the number in the 2b data block, first field.)

1-15	1st	F	Enter the value of K_{11}.
16-30	2nd	F	Enter the associated temperature.

4b data block

The number of blocks in this series is n, the number in the 2b data block, second field, or 0 if n = -1.

1-15	1st	F	Enter the value of K_{22}.
16-30	2nd	F	Enter the associated temperature.

5b data block

The number of blocks in this series is n, the number in the 2b data block, third field, or 0 if n = -1.

1-15	1st	F	Enter the value of K_{33}.
16-30	2nd	F	Enter the associated temperature.

6b data block

The number of blocks in this series is the number in the 2b data block, fourth field.

1-15	1st	F	Enter the value of specific heat
16-30	2nd	F	Enter the associated temperature.

7b data block

The number of blocks in this series is the number in the 2b data block, fifth field.)

1-15	1st	F	Enter latent heat value.
16-30	2nd	F	Solidus temperature (lower phase change limit).
31-45	3rd	F	Liquidus temperature (upper phase change limit).

Format Fixed	Format Free	Data Type	Entry
8b data block			
The number of blocks in this series is the number in the 2b data block, sixth field.)			
1-15	1st	F	Enter the value of R_{11}.
16-30	2nd	F	Enter the associated temperature.
9b data block			
The number of blocks in this series is n, the number in the 2b data block, seventh field, or 0 if n = -1.			
1-15	1st	F	Enter the value of R_{22}.
16-30	2nd	F	Enter the associated temperature.
10b data block			
The number of blocks in this series is n, the number in the 2b data block, eighth field, or 0 if n = -1.			
1-15	1st	F	Enter the value of R_{33}.
16-30	2nd	F	Enter the associated temperature.
11b data block			
The number of blocks in this series is the number in the 2b data block, ninth field.			
1-15	1st	F	Enter the value of the emissivity.
16-30	2nd	F	Enter the associated temperature.

■ CONTROL

Define Control Parameters for Heat Transfer Analysis

Description

This option allows you to input parameters governing the convergence solution and accuracy for heat transfer analysis.

For heat transfer analysis, the only data field required to be set is the maximum number of steps, the first field in the second data block. All other fields can, in these cases, be left blank but notice that the 3rd data block must be included.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-7	1st	A	Enter the word CONTROL.
2nd data block			
1-5	1st	I	Maximum number of load steps in this run. Default is 99999. This is a cumulative number and is usually used to stop the run when RESTART is being used.
6-10	2nd	I	Maximum number of recycles during an increment due to temperature dependent material properties. Default value is 3.
11-15	3rd	I	Minimum number of recycles during an increment. Note that this data field forces this number of recycles to take place in all subsequent increments.
3rd data block			
1-10	1st	F	Maximum nodal temperature change allowed. Used to control automatic time step scheme for heat transfer. Default value of 20.
11-20	2nd	F	Maximum nodal temperature change allowed before properties are reevaluated and matrices reassembled. Default value of 100.

Format		Data	
Fixed	Free	Type	Entry
21-30	3rd	F	Maximum error in temperature estimate used for property evaluation. This control provides a recycling capability to improve accuracy in highly nonlinear heat-transfer problems (for example, latent heat, radiation boundary conditions). Default is 0, which bypasses this test. Set to maximum temperature error which is considered acceptable.
31-40	4th	F	Maximum nodal voltage change allowed. Used to control automatic time-step scheme for Joule heating. Default value is 100.

■ CONVERT

Define Conversion Factors

Description

In the coupled thermal-stress analysis, the unit of heat generated from the mechanical problem is, in general, not consistent with the units used in a heat transfer problem. Depending on the units used for the problem, different conversion factors must be used. In addition, following the work of Farren and Taylor, not all inelastic work is dissipated into heat; for most metals, about 90% is converted[1]. This should also be considered when defining a conversion factor.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word CONVERT.
2nd data block			
1-10	1st	F	Heat generation conversion factor between inelastic mechanical energy and heat transfer flux in a coupled analysis. Default is1.0.

1. S. W. Farren, G. I. Taylor. "The Heat Developed During Plastic Extension of Metals", Proceedings of the Royal Society, London, A107, p. 422, 1925.

■ CONRAD GAP

Define Convection/Radiation Gap

Description

This option allows you to input emissivity, Stefan-Boltzmann constant, absolute temperature conversion factor, film coefficient, and gap closure temperature for convection/radiation gap option.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word CONRAD GAP.
2nd data block			
1-5	1st	I	Number of sets of data used to input CONRAD GAP.
6-10	2nd	I	Unit number for input of CONRAD GAP data, defaults to input.
3rd data block			
1-5	1st	I	Face identification – see *Volume B: Element Library.* Note that these identifiers are different from those used for DIST FLUXES.
6-15	2nd	F	Emissivity
16-25	3rd	F	Stefan-Boltzmann constant (default is 0.1714×10^{-8} BTU/HR-FT^2 -R^4).
26-35	4th	F	Absolute temperature conversion factor (for example, degrees Rankine = 459.7 + degrees Fahrenheit; or, degrees Kelvin = 273.15 + degrees centigrade; default is 459.7).
36-45	5th	F	Film coefficient.
46-55	6th	F	GAP closure temperature.
4th data block			
			Enter a list of elements to which the above CONRAD GAP data is applied.

■ CHANNEL

Define Fluid Channel Input

Description

This option allows you to input inlet temperature, fluxes, and film coefficient for a fluid channel in a heat transfer analysis.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word CHANNEL.
2nd data block			
1-5	1st	I	Number of sets of data used to input fluid channels.
6-10	2nd	I	Unit number for input of fluid channels data; defaults to input.
3rd data block			
1-5	1st	I	Face identification – see *Volume B: Element Library.* Note that these identifiers are different from those used for DIST FLUXES.
6-10	2nd	I	First (inlet) element number in the channel.
11-20	3rd	F	Inlet temperature.
21-30	4th	F	Mass flow rate.
31-40	5th	F	Film coefficient.
4th data block			
			Enter a list of elements to which the above fluid channel data is applied.

■ VIEW FACTOR

Read in Radiation View Factors

Description

This option initiates the reading of radiation view factors created by Mentat. This file is read as `vfid.vfs` where `vfid` is entered using the `-vf` option when invoking MARC. The RADIATION parameter is required.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-11	1st	A	Enter the words VIEW FACTOR.

■ RADIATING CAVITY

Define Outline of Radiating Cavity

Description

This option allows for the input of the outlines of radiating cavities. Each cavity outline is defined by a group of nodal points in a counter clockwise direction. The RADIATION parameter is required

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-16	1st	A	Enter the word RADIATING CAVITY.
2nd data block			
1-5	1st	I	Total number of cavities.
6-10	2nd	I	Unit number for input. Default is 5 (input).
3rd data block			
			Enter a list of nodes defining the outline of a cavity. Repeat for each cavity to be defined.
			Note: List verbs `EXCEPT` and `INTERSECT` are illegal in these nodal lists.

■ VELOCITY

Define Nodal Velocity Components

Description

This option allows the specification of the nodal velocity components in a heat transfer analysis, where the convective terms are to be included. The convective option is specified by placing a 2 in the fifth field of the HEAT parameter. The nodal velocity components are defined by specifying the velocity magnitude of a series of components for sets of nodes. This data can be input from data blocks or from an auxiliary input device. Moreover, the velocity values can be respecified, or initialized if no previous data was entered via user subroutine UVELOC (see *Volume D: User Subroutines and Special Routines*).

A summary of nodal velocities appears in the printout. This can be suppressed by specifying a nonzero value for the print-suppress parameter.

If the motion of the media is to be calculated, a coupled fluid-thermal analysis should be performed.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word VELOCITY.
2nd data block			
1-5	1st	I	Number of data blocks used to input nodal velocity components. If a negative value is entered, user subroutine UVELOC is called for every node.
6-10	2nd	I	Enter the unit number for input of the velocity field. Default to unit 5, unless the INPUT TAPE parameter has been used.
11-15	3rd	I	Set to 1 to suppress printout of the summary of nodal velocity components.

Format Fixed	Format Free	Data Type	Entry
Data blocks 3, 4 and 5 should be repeated for each data set.			
3rd data block			
1-10	1st	F	Enter the magnitude of the velocity in the first coordinate-direction for which the velocity is given. Additional velocity components in other coordinate directions must be specified on the same data line in F10 format. The number of components must equal the number of directions.
4th data block			
			Enter a list of coordinate directions in which the velocity is specified. **Note:** List verbs `EXCEPT` and `INTERSECT` are illegal here.
5th data block			
			Enter a list of nodes for which the velocity vector as defined in data block 3 and 4.

Joule Heating Analysis

This section describes the input of additional data required for coupled Joule heating analysis. All of the options in the previous subsection referring to heat transfer are also applicable. You have the ability to apply surface, volumetric and nodal currents, and prescribe nodal voltages. In addition, you can input a conversion factor so that you can work in convenient units for both the heat transfer and electrical conduction problems.

■ JOULE

Define Conversion Factor for Joule Heating Analysis

Description

In the analysis of Joule heating, the unit of heat generation computed from the electrical problem is, in general, not consistent with the unit required for the heat transfer analysis. Depending on the units used for the problems different conversion factors must be used. For example, in an electric problem the heat generation can be expressed in terms of current and resistance as $q = I^2R$. If the units of current and resistance are (amp) and (ohm/ft), respectively, then the unit of heat generation in the electric problem must be (watt/ft). Since 1 (watt) is equal to 3.4129 (btu/hr), a factor of 3.4129 must be used in a Joule heating problem, for the purpose of converting the unit of heat generation from (watt/ft) to (btu/hr-ft) for heat transfer analysis.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-5	1st	A	Enter the word JOULE.
2nd data block			
1-10	1st	F	Heat generation unit conversion factor between electrical and heat transfer analyses in a Joule heating problem. Default is 1.0.

■ DIST CURRENT

Define Distributed Current

Description

This option allows distributed (surface and volumetric) current to be specified. Distributed current is converted to a consistent nodal current by the program. Note that for a given element type, there is an established convention for the application of surface current of a particular face. User subroutine FLUX can be used to input time and spatial dependent current.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words DIST CURRENT.
2nd data block			
1-5	1st	I	Enter the number of sets of distributed currents to be entered (optional).
6-10	2nd	I	Enter unit number for input of distributed current data, defaults to input.

The 3rd and 4th data blocks are entered in pairs, once for each data block.

3rd data block			
1-5	1st	I	Parameter identifying the type of current. See library element description in *Volume B: Element Library.*
6-15	2nd	F	Enter the magnitude of this type of distributed current.
4th data block			
			Enter a list of elements associated with the above distributed current.

■ POINT CURRENT

Define Nodal Point Current

Description

This option allows total nodal point current to be specified. User subroutine FORCDT can be used for the time dependent current.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10		A	Enter the words POINT CURRENT.
2nd data block			
1-5		I	Enter the number of sets of point currents to be entered (optional).
6-10		I	Enter unit number of input of point current data, defaults to input.
3rd data block			
1-10		F	Magnitude of point current.
4th data block			
			Enter a list of nodes to which the above nodal current are applied.

■ VOLTAGE

Define Nodal Fixed Voltage

Description

This option defines the fixed voltage that each associated nodal point must take during the first and subsequent increments.

Note that a number equal to or exceeding the total number of degrees of freedom constrained must appear on the SIZING parameter.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-7		A	Enter the word VOLTAGE.
2nd data block			
1-5		I	Number of sets of voltage boundary condition data lines to be read (optional).
6-10		I	Enter unit number for input of voltages; defaults to input.
3rd data block			
1-10		F	Enter the prescribed voltage.
4th data block			
			Enter a list of nodes for which the voltage is prescribed.

Hydrodynamic Bearing Analysis

This section describes the input of data necessary for a hydrodynamic bearing analysis. You have the ability to define the lubricant film thickness and initial velocity in a variety of ways. In addition, restrictors and pump pressures can be imposed on the film pressure.

■ VELOCITY

Define Nodal Velocity Components

Description

This option allows the specification of the nodal velocity components in a bearing analysis or a heat transfer analysis, where the convective terms are to be included. The nodal velocity components are defined by specifying the velocity magnitude of a series of components for sets of nodes. This data can be input from data blocks or from an auxiliary input device. Moreover, the velocity values can be respecified, or initialized if no previous data was entered via user subroutine UVELOC. See *Volume D: User Subroutines and Special Routines.*

A summary of nodal velocities will appear in the printout. This can be suppressed by specifying a nonzero value for the print-suppress parameter. Note that the velocity in a stationary bearing analysis is specified with respect to the (moving) lubricant. This means that, in case both the adjacent surfaces as well as the lubricant move with respect to some global coordinate system, the velocity vector to be defined equals the sum of both surface velocities relative to the stationary film profile.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word VELOCITY.
2nd data block			
1-5	1st	I	Number of data blocks used to input nodal velocity components. If a negative value is entered, user subroutine UVELOC is called for every node.
6-10	2nd	I	Enter the unit number for input of the velocity field. Default to unit 5, unless the INPUT TAPE parameter has been used.
11-15	3rd	I	Set to 1 to suppress printout of the summary of nodal velocity components.

Format Fixed	Format Free	Data Type	Entry
Data blocks 3, 4 and 5 should be repeated for each data set.			
3rd data block			
1-10	1st	F	Enter the magnitude of the velocity in the first coordinate-direction for which the velocity will be given. Additional velocity components in other coordinate directions must be specified on the same data line in F10 format. The number of components must equal the number of directions.
4th data block			
			Enter a list of coordinate directions in which the velocity will be specified. **Note:** List verbs `EXCEPT` and `INTERSECT` are illegal here.
5th data block			
			Enter a list of nodes for which the velocity vector as defined in data blocks 3 and 4 applied

■ THICKNESS

Define Lubrication Thickness

Description

This option defines the thickness of the lubricant film in a bearing analysis. The nodal thicknesses are specified by giving the thickness values for a list of nodes. This data can be input from data blocks or from a tape. Moreover, the nodal thickness values can be respecified, or initialized in case no previous data was input, via user subroutine UTHICK. See *Volume D: User Subroutines and Special Routines*. A summary of nodal thickness values will appear in the printout. This can be suppressed by specifying a nonzero value for the print-suppress parameter.

The input of element thicknesses can be done via the GEOMETRY block or by means of user subroutine UGROOV. Element thickness values usually only have to be defined in case of film discontinuities (grooves). See *Volume A: Theory and User Information* for a description of the various ways to specify the contributions to the total lubricant film.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word THICKNESS.
2nd data block			
1-5	1st	I	Number of data blocks used to input nodal thickness values (optional). If a negative value is entered, user subroutine UTHICK is called for every node.
6-10	2nd	I	Enter the unit number for input of film thicknesses. Default to unit 5.
11-15	3rd	I	Set to 1 to suppress printout of the nodal thickness summary.

Format Fixed	Format Free	Data Type	Entry
The third and fourth blocks should be entered in pairs, one pair for each distinct nodal data set.			
3rd data block			
1-10	1st	F	Enter nodal thickness value.
4th data block			
			Enter a list of nodes for which the thickness as specified in data block 3 is applied.

■ RESTRICTOR

Coefficient Input for Bearing Analysis

Description

This option allows restrictor coefficients and associated pump pressures to be input. Nonuniform restrictors or pump pressures can be obtained via user subroutine URESTR. See *Volume D: User Subroutines and Special Routines.*

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-5	1st	A	Enter the word RESTRICTOR.
2nd data block			
1-5	1st	I	Number data blocks used to input the restrictor coefficient data (optional).
6-10	2nd	I	Enter the unit number for input of restrictor data. Default to unit 5, unless the INPUT TAPE parameter has been used.

Data Blocks 3 and 4 are entered in pairs, one pair for each distinct data set.

3rd data block			
1-5	1st	I	Face identification. Same as for user subroutine FLUX. See *Volume B: Element Library.*
6-15	2nd	F	Reference value of restrictor coefficient.
16-25	3rd	F	Reference value of pump pressure (reference values can be modified by user subroutine URESTR).
4th data block			
			Enter a list of elements for which restrictor data as defined in data block 3 is applied.

■ TEMPERATURE EFFECTS

Define Effect of Temperature in Bearing Analysis

Description

This option defines the variation of element property (viscosity) temperature. The values read in through the ISOTROPIC option are those at the lowest temperature specified. Properties are not defined below the lowest temperature. The temperature dependency can be entered using one of the following two options:

A. The variation of a particular property with temperature is specified as a piecewise linear curve. Breakpoints must be given in ascending order of temperature.

B. The particular property value and temperature lying on the relevant curve are input directly. Data points must be given in ascending order of temperature. This option is flagged by entering the word DATA on the 1st data line.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-19	1st	A	Enter the words TEMPERATURE EFFECTS.
21-80	2nd	A	Enter the word DATA to indicate that option B is used.

For option A, use data blocks 2a and 3a. For option B, use data blocks 2b and 3b, below.

Option A

2a data block

1-5	1st	I	Number of slopes of viscosity versus temperature curve.
6-30	2nd	I	Not used; enter 0.
31-35	3rd	I	Material type identification (1,2,3,...) for cross-reference to ISOTROPIC block.
36-40	4th	I	Logical unit number for input of this set of data. Defaults to data lines.

Format Fixed	Format Free	Data Type	Entry
3a data block			
Viscosity variation. Number of data lines as given on data block 2a, first field.			
1-15	1st	F	Slope of viscosity versus temperature curve.
16-30		F	Temperature above which slope becomes operative.

Option B

2b data block

1-5	1st	I	Number of data points on the viscosity versus temperature curve.
6-30	2nd	I	Not used; enter 0.
31-35	3rd	I	Material type identification (1,2,3...) for cross-reference to ISOTROPIC block.
36-40	4th	I	Logical unit number for input of this set of data. Defaults to data lines.

3b data block

Viscosity variation. Number of data lines as given on data block 2b, first field.

1-15	1st	F	Enter the value of the viscosity.
16-30	2nd	F	Enter the associated temperature.

Note: Since in bearing analysis no incrementation is performed, the value of the viscosity is always taken at the current temperature. No averaging is performed.

■ CONTROL

Define Maximum Number of Increments for Bearing Analysis

Description

This option allows you to input the maximum number of increments in a hydrodynamic bearing analysis.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-7	1st	A	Enter the word CONTROL.
2nd data block			
1-5	1st	I	Maximum number of bearing increments in this run. Default is 4.
3rd data block			
			Not used. Enter a blank data line.

■ ISOTROPIC

Define Element Lubricant Properties

Description

This option is used to define the lubricant properties for all of the elements.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-8	1st	A	Enter the word ISOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of distinct sets of element properties to be input (optional). For temperature dependent properties, these are values corresponding to the first (lowest temperature) breakpoint (see TEMPERATURE EFFECTS option). A temperature dependent property is undefined below its lowest breakpoint.
6-10	2nd	I	Enter unit number for input of data. Defaults to input.
The 3rd, 4th, and 5th data blocks should be entered as pairs, one for each distinct data block.			
3rd data block			
1-5	1st	I	Material identification number (1,2,3, etc.) for cross-reference to TEMPERATURE EFFECTS option.
4th data block			
1-10	1st	F	Enter the reference temperature value of the viscosity.
11-20	2nd	F	Enter the value of the cavitation pressure.
21-30	3rd	F	Enter the reference value of the mass density of the lubricant.
5th data block			
			Enter element data set for which the properties as specified in data block 3 applies.

Acoustic Analysis

This section describes the data required for an acoustic analysis where the pressure distribution in a cavity with reflecting surfaces is calculated and where the fluid in the cavity is treated as an inviscid compressible fluid.

■ FIXED PRESSURE

Define Nodal Fixed Pressure

Description

This option defines the fixed pressure that each node must take during the first and subsequent increments, unless it is further modified using the PRESS CHANGE option. The boundary conditions are specified either by giving the pressure and a list of nodal numbers, or by the input of boundary conditions generated during mesh generation (MESH2D).

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-19	1st	A	Enter the words FIXED PRESSURE.
2nd data block			
1-5	1st	I	Number of sets of boundary condition blocks to be read (optional).
6-10	2nd	I	This field is set to nonzero to flag input of boundary conditions set during mesh generation (MESH2D). If this field is activated, no further data lines are required in this option block.
11-15	4th	I	Unit number used for MESH2D option. Note that the boundary conditions are stored after the connectivity and coordinate data on this file, so that the model definition options must be arranged accordingly.

For each set of boundary conditions, use the 3a and 4a data blocks, or the 3b, 4b, and 5b data blocks.

For analyses which do not include heat transfer shell elements, use the 3a and 4a data blocks.

3a data block			
1-10	1st	F	Prescribed nodal pressure.

Format Fixed	Format Free	Data Type	Entry
4a data block			
			Enter a list of nodes for which the above pressure is applied. 3b, 4b and 5b data blocks for analyses which include heat transfer shell elements.
3b data block			
1-10	1st	E	Prescribed pressure for first degree of freedom listed in data block 4b.
11-20	2nd	E	Prescribed pressure for second degree of freedom listed in data block 4b.
21-30	3rd	E	Prescribed pressure for third degree of freedom listed in data block 4b.

See *Volume B: Element Library* for the definition of nodal degrees-of-freedom.

4b data block

Enter a list of degrees of freedom to which the above prescribed pressures are given.

Note: List verbs `EXCEPT` and `INTERSECT` are illegal here.

5b data block

Enter a list of nodes for which the above boundary conditions are applied.

■ DIST SOURCES

Define Distributed Sources

Description

This option allows incrementally distributed (surface and volumetric) sources to be specified in an acoustic analysis. Distributed sources are converted to consistent nodal sources by the program. Note that for a given element type, there is an established convention for the application of surface source on a particular face. User subroutine FLUX can be used to input time and spatial dependent fluxes.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words DIST SOURCES.
2nd data block			
1-5	1st	I	Enter the number of sets of distributed sources to be entered (optional).
6-10	2nd	I	Enter unit number for input of distributed source data, defaults to input.

The following 3rd and 4th data blocks are repeated for each set of distributed sources.

3rd data block			
1-5	1st	I	Parameter identifying the type of load. See library element description in *Volume B: Element Library.*
6-15	2nd	F	Enter the magnitude of this type of distributed sources.
16-20	3rd	I	Source index (optional). (Source index is to be used in user subroutine FLUX).
4th data block			
			Enter a list of elements associated with the above distributed sources.

■ POINT SOURCE

Define Point Sources

Description

This option allows incremental nodal point sources to be specified in an acoustic analysis. User subroutine FORCDT can be used for the time dependent sources. Enter an upper bound to the number of nodes with point sources on the DIST LOADS parameter.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10		A	Enter the words POINT SOURCE.
2nd data block			
1-5		I	Enter the number of sets of point sources to be entered (optional).
6-10		I	Enter unit number for input of point source data, defaults to input.

The following 3rd and 4th data blocks are repeated for each set of distributed sources.

3rd data block			
1-10		F	Magnitude of incremental point source.
11-20		F	Magnitude of point source for second degree of freedom, (heat transfer shell elements only).
21-30		F	Magnitude of point source for third degree of freedom, (heat transfer shell elements only).
4th data block			
			Enter a list of nodes to which the above nodal sources are applied.

■ ISOTROPIC

Define Properties for Acoustic Cavity

Description

This option allows you to define properties for the fluid/gas in the acoustic cavity. You can also associate these material properties with a list of element numbers.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word ISOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of sets of isotropic material data to follow (optional).
6-10	2nd	I	Enter the unit number for input. Defaults to input deck.
The 3rd, 4th, and 5th data blocks should be entered as pairs and repeated for each data block.			
3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.).
4th data block			
1-10	1st	F	Bulk modulus.
11-20	2nd	F	Enter the mass density.
5th data block			
			Enter a list of elements associated with this material.

Electrostatic Analysis

This section describes the input of material data and boundary conditions applicable for electrostatic problems. The ISOTROPIC and ORTHOTROPIC options are used to define dielectric constants in electrostatic analysis. A steady-state solution can be obtained in one increment using the STEADY STATE option. In addition, user subroutine FLUX can be used for variable distributions of charges; user subroutine UEPS can be used for the anisotropic dielectric constants.

■ FIXED POTENTIAL

Define Fixed Nodal Potential

Description

This option defines the fixed potential that each node must take during the first and subsequent increments. The boundary conditions are specified either by giving the potential and a list of nodal numbers, or by the input of boundary conditions generated during mesh generation (MESH2D).

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-15	1st	A	Enter the words FIXED POTENTIAL.
2nd data block			
1-5	1st	I	Number of sets of boundary condition blocks to be read (optional).
6-10	2nd	I	This field is set to nonzero to flag input of boundary conditions set during mesh generation (MESH2D). If this field is activated, no further data lines are required in this option block.
11-15	3rd	I	Unit number used for MESH2D option. Note that the boundary conditions are stored after the connectivity and coordinate data on this file, so that the model definition options must be arranged accordingly.

Data blocks 3a and 4a are for analyses which do not include shell elements.

3rd data block			
1-10	1st	F	Prescribed nodal potential.
4a data block			
			Enter a list of nodes for which the above potential is applied.

Format Fixed	Format Free	Data Type	Entry
Data blocks 3b, 4b, and 5b are for analyses which include shell elements.			
3b data block			
1-10	1st	F	Prescribed potential for first degree of freedom listed in data block 4b.
11-20	1st	F	Prescribed potential for second degree of freedom listed in data block 4b.
21-30	2nd	F	Prescribed potential for third degree of freedom listed in data block 4b. (See *Volume B: Element Library* for the definition of nodal degrees of freedom.)
4b data block			
			Enter a list of degrees of freedom to which the above prescribed potential is given. **Note:** List verbs `EXCEPT` and `INTERSECT` are illegal here.
5b data block			
			Enter a list of nodes for which the above fixed potential conditions are applied.

■ DIST CHARGES

Define Distributed Charges

Description

This option allows distributed (surface and volumetric) charges to be specified. Distributed charges are converted to consistent nodal charges by the program. Note that for a given element type, there is an established convention for the application of surface charge on a particular face. User subroutine FLUX can be used to input spatially dependent charges.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-12	1st	A	Enter the words DIST CHARGES.
2nd data block			
1-5	1st	I	Enter the number of sets of distributed charges to be entered (optional).
6-10	2nd	I	Enter unit number for input of distributed charge data, defaults to input.

The 3rd and 4th data blocks are repeated for each set of distributed charges.

3rd data block			
1-5	1st	I	Parameter identifying the type of distributed charge. See description of library element in *Volume B: Element Library.*
6-15	2nd	F	Enter the magnitude of this type of distributed charges.
16-20	3rd	I	Charge index (optional). Charge index is to be used in user subroutine FLUX.
4th data block			
			Enter a list of elements associated with the above distributed charges.

■ POINT CHARGE

Define Nodal Point Charges

Description

This option allows total nodal point charges to be specified. User subroutine FORCDT can be used for nonuniform point charges.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-12	1st	A	Enter the words POINT CHARGE.
2nd data block			
1-5	1st	I	Enter the number of sets of point charges to be entered (optional).
6-10	2nd	I	Enter unit number for input of point charge data, defaults to input.

The 3rd and 4th data blocks are entered as pairs, one for each data set.

3rd data block			
1-10	1st	F	Magnitude of point charge.
11-20	2nd	F	Magnitude of point charge for second degree of freedom (shell elements only).
21-30	3rd	F	Magnitude of point charge for third degree of freedom (shell elements only).
4th data block			
			Enter a list of nodes to which the above nodal charges are applied.

■ ISOTROPIC

Define Electrical Properties for Isotropic Materials

Description

This option allows you to define electrical properties for an isotropic material. You can also associate these material properties with a list of element numbers.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word ISOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of sets of isotropic material data to follow.
6-10	2nd	I	Enter the unit number for input. Defaults to input data file.

The 3rd and 4th data blocks are entered as pairs, one for each data set.

3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.)
4th data block			
1-10	1st	F	Permittivity constant.
5th data block			
			Enter a list of elements associated with this material.

■ ORTHOTROPIC

Define Electrical Properties for Orthotropic Materials

Description

This option allows you to define electrical properties for an orthotropic material. You can also associate these material properties with a list of element numbers.

Note: Since the material properties in an orthotropic material are independent, it is your responsibility to enter all data required to match the dimension of the conductivity matrix for the elements listed below. (See *Volume B: Library Elements*, if necessary). No defaults for this data are provided by MARC.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word ORTHOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of sets of orthotropic material data to follow (optional).
6-10	2nd	I	Enter the unit number for input. Defaults to input deck.
The 3rd and 4th data blocks are entered as pairs, one for each data set.			
3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-reference to ORTHO TEMP option.
6-10	2nd	I	Enter 1 to call user subroutines UEPS and ORIENT.
4th data block			
These values are with respect to the user coordinate (1, 2, 3) system.			
1-10	1st	F	ε_{11} Electric permittivity
11-20	2nd	F	ε_{22} Electric permittivity
21-30	3rd	F	ε_{33} Electric permittivity
5th data block			
			Enter a list of elements associated with this material. (Do not enter composite elements which use this material in its layers.)

Magnetostatic Analysis

This section describes the input of material data and boundary conditions applicable for magnetostatic problems. The ISOTROPIC and ORTHOTROPIC options are used to define magnetic permeability in magnetostatic analysis. The variation of magnetic permeability with either magnetic field density or magnetic field vector can be prescribed by the B-H RELATION option. A steady state solution can be obtained in one increment using the STEADY STATE option. In addition, user subroutine FLUX can be used for variable distributions of currents; user subroutine UMU can be used for anisotropic magnetic permeabilities.

■ FIXED POTENTIAL

Define Nodal Fixed Potential

Description

This option defines the fixed potential that each node must take during the first and subsequent increments. The boundary conditions are specified either by giving the potential and a list of nodal numbers, or by the input of boundary conditions generated during mesh generation (MESH2D).

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-19	1st	A	Enter the words FIXED POTENTIAL.
2nd data block			
1-5	1st	I	Number of sets of boundary condition data blocks to be read (optional).
6-10	2nd	I	This field is set to nonzero to flag input of boundary conditions set during mesh generation (MESH2D). If this field is activated, no further data lines are required in this option block.
11-15	3rd	I	Unit number used for MESH2D option. Note that the boundary conditions are stored after the connectivity and coordinate data on this file, so that the model definition options must be arranged accordingly.

Format Fixed	Format Free	Data Type	Entry

For each set of boundary conditions, use the 3a and 4a data blocks, or the 3b, 4b, and 5b data blocks.

Data blocks 3a and 4a are used for analyses which are planar or axisymmetric.

3a data block

1-10	1st	F	Prescribed potential ϕ.

4a data block

Enter a list of nodes for which the above potential is applied.

Data blocks 3b, 4b, and 5b are used for analyses which include solid brick or shell elements.

3b data block

1-10	1st	F	Prescribed potential for first degree of freedom listed in data block 4b.
11-20	2nd	F	Prescribed potential for second degree of freedom listed in data block 4b.
21-30	3rd	F	Prescribed potential for third degree of freedom listed in data block 4b. (See *Volume B: Element Library* for the definition of nodal degrees of freedom.)

4b data block

Enter a list of degrees of freedom to which the above prescribed potentials are given.

Note: List verbs `EXCEPT` and `INTERSECT` are illegal here.

5b data block

Enter a list of nodes for which the above boundary conditions are applied.

■ DIST CURRENT

Define Distributed Current

Description

This option allows distributed (surface and volumetric) currents to be specified. Distributed currents are converted to consistent nodal currents by the program. Note that for a given element type, there is an established convention for the application of surface current of a particular face. User subroutine FLUX can be used to input spatially dependent current.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-12	1st	A	Enter the words DIST CURRENT.
2nd data block			
1-5	1st	I	Enter the number of sets of distributed currents to be entered (optional).
6-10	2nd	I	Enter unit number for input of distributed current data, defaults to input.

The 3rd and 4th data blocks are entered as pairs, one for each data set.

3rd data block			
1-5	1st	I	Parameter identifying the type of current. See library element description in *Volume B: Element LIbrary.*
6-15	2nd	E	Enter the magnitude of this type of distributed current.
16-20	3rd	I	Current index (current index is to be used in user subroutine FLUX).
4th data block			
			Enter a list of elements associated with the above distributed current.

■ POINT CURRENT

Define Nodal Point Current

Description

This option allows total nodal point current to be specified.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words POINT CURRENT.
2nd data block			
1-5	1st	I	Enter the number of sets of point currents to be entered (optional).
6-10	2nd	I	Enter unit number of input of point current data, defaults to input.

The 3rd and 4th data blocks are entered as pairs, one for each data set.

3rd data block			
1-10	1st	F	Magnitude of point current.
11-20	2nd	F	Magnitude of point current for second degree of freedom (brick elements only).
21-30	3rd	F	Magnitude of point current for third degree of freedom (brick elements only).
4th data block			
			Enter a list of nodes to which the above nodal currents are applied.

■ ISOTROPIC

Define Magnetic Properties for Isotropic Materials

Description

This option allows you to define magnetic properties for isotropic materials. You can also associate these material properties with a list of element numbers. Note that either the permeability or its inverse can be entered.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word ISOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of sets of isotropic material data to follow (optional).
6-10	2nd	I	Enter the unit number for input. Defaults to input file.
Data blocks 3, 4, and 5 are repeated for each data set.			
3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-reference to B-H RELATION option.
6-10	2nd	I	Enter 1 to call user subroutines UMU and ORIENT.
4th data block			
1-10	1st	F	Permeability.
11-20	2nd	F	Inverse permeability.
5th data block			
			Enter a list of elements associated with this material.

■ ORTHOTROPIC

Define Magnetic Properties for Orthotropic Materials

Description

This option allows you to define magnetic properties for an orthotropic material. You can also associate these material properties with a list of element numbers. Note that either the permeability or the inverse permittivity can be entered.

Note: Since the material properties in an orthotropic material are independent, it is your responsibility to enter all data required to match the dimension of the conductivity matrix for the elements listed below. (See *Volume B: Element Library*, if necessary). No defaults for this data are provided by MARC.

These values are with respect to the user coordinate (1, 2, 3) system.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word ORTHOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of sets of orthotropic material data to follow.
6-10	2nd	I	Enter the unit number for input. Defaults to input file.
Data blocks 3 through 8 are repeated for each data set.			
3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-reference to B-H RELATION option.
6-10	2nd	I	Enter 1 to call user subroutines UMU and ORIENT.
4th data block			
1-10	1st	F	μ_{11} Magnetic permeability
11-20	2nd	F	μ_{22} Magnetic permeability
21-30	3rd	F	μ_{33} Magnetic permeability

Format Fixed	Format Free	Data Type	Entry
31-40	4th	F	$1/\mu_{11}$ Inverse magnetic permeability
41-50	5th	F	$1/\mu_{22}$ Inverse magnetic permeability
51-60	6th	F	$1/\mu_{22}$ Inverse magnetic permeability

5th data block

Enter a list of elements associated with this material.

■ B-H RELATION

Define Magnetization Curve for Nonlinear Magnetic Material

Description

This option can be used to specify the magnetization curve(s) for nonlinear isotropic or orthotropic material. Depending on the material type, a different method of entering data must be used:

isotropic material: (OPTION A) Enter one set of data points (|H|, |B|) representing the magnitude of H as a function of the magnitude of B. For |H| = 0 the value of |B| should be zero. If not, the corresponding offset of the curve will be disregarded.

orthotropic material: (OPTION B) For every component of H, a set of data points (H,B) is entered, relating this component of H to the corresponding component of B. A component of the remanence vector can be specified by choosing a nonzero value of B for H = 0.

Note: In either cases the curve(s) represented by the data sets must be monotone and uniquely defined. Furthermore, the data points must be given in ascending order of B.

An alternative way of specifying the magnetization curve(s) is to supply the reluctivity $1/\mu$ in user subroutine UMU.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-12	1st	A	Enter the word B-H RELATION.

For option A, use data blocks 2a, 3a, 4a, and 5a. For option B, use data blocks 2b, 3b, 4b, and 5b.

Format Fixed	Format Free	Data Type	Entry

Option A Isotropic Behavior

2a data block

1-5	1st	I	Number of data points of |H|-|B| curve.
6-10	2nd	I	Not used; 0.
11-15	3rd	I	Not used; 0.
16-20	4th	I	Not used; 0.
21-25	5th	I	Not used; 0.
26-30	6th	I	Not used; 0.
31-35	7th	I	Material type identification (1,2,3...) for cross-reference to ISOTROPIC blocks.
36-40	8th	I	Logical unit number for input of this set of data. Defaults to data blocks.

3a data block

|H| - |B| variation. Number of blocks as given on data block 2, first field.

1-15	1st	F	Enter value of |H|.
16-30	2nd	F	Enter value of |B|.

Option B Orthotropic Behavior

2b data block

1-5	1st	I	Number of data points of H_1 - B_1 curve.
6-10	2nd	I	Number of data points of H_2 - B_2 curve.
11-15	3rdt	I	Number of data points of H_3 - B_3 curve.
16-20	4th	I	Not used; enter zero.
21-25	5th	I	Not used; enter zero.
26-30	6th	I	Not used; enter zero.

Format Fixed	Format Free	Data Type	Entry
31-35	7th	I	Material type identification (1,2,3...) for cross-reference to ORTHOTROPIC blocks.
36-40	8th	I	Logical unit number for input of this set of data. Defaults to data blocks.

3b data block

H_1 - B_1 variation. Number of blocks given on data block 2, first field.

1-15	1st	F	Enter value of H_1.
16-30	2nd	F	Enter value of B_1.

4b data block

H_2 - B_2 variation. Number of blocks given on data block 2, second field.

1-15	1st	F	Enter value of H_2.
16-30	2nd	F	Enter value of B_2.

5b data block

H_3 - B_3 variation. Number of blocks given on data block 2, third field.

1-15	1st	F	Enter value of H_3.
16-30	2nd	F	Enter value of B_3.

■ PERMANENT

Define Permanent Magnet

Description

This option provides various ways of defining a permanent magnet in parts of the model. The default is that no permanent magnets are present.

You need to enter the remanence vector B_r, which is the product of the magnet vector M_0 and the permeability of the vacuum μ_0.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-13	1st	A	Enter the word PERMANENT.
2nd data block			
1-5	1st	I	Not used; enter 0.
6-10	2nd	I	Enter 1 to initialize the permanent magnet via data blocks 3 and 4 below. See also the third field on this block. Enter 4 to initialize the permanent magnet via data blocks 5, 6, 7, and 8 given below. See also the third field on this block.
11-15	3rd	I	This entry gives the number of pairs of bocks in data blocks 3 and 4 or in data blocks 5, 6, 7, and 8 used to input the permanent magnet.

Data blocks 3 and 4 are only input if the second field above is set to 1. In that case, the number of blocks is equal to the number given in the third field above.

3rd data block			
1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with the value.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value.
21-25	5th	I	First layer of cross section point with this value.
26-30	6th	I	Last layer or cross section point with this value can only be bigger than 1 for beam or shell elements.

4th data block

1-10	1st	F	First component of remanence vector.
11-20	2nd	F	Second component of remanence vector.
21-30	3rd	F	Third component of remanence vector.

Data blocks 5, 6, 7 and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block

1-10	1st	F	First component of remanence vector.
11-20	2nd	F	Second component of remanence vector.
21-30	3rd	F	Third component of remanence vector.

6th data block

Enter a list of elements to which the above state variable is applied.

7th data block

This data block is **not** necessary if CENTROID parameter is used.

Enter a list of integration points to which the above remanence is applied.

8th data block

This data blocks is only necessary if there are either beams or shells in the mesh.

Enter a list of layer points to which the above remanence is applied.

■ CONTROL

Control for Magnetostatic Analysis

Description

This option allows you to input parameters governing the convergence and accuracy for magnetostatic analysis. This option is only required if the B-H RELATION option is used to enter a nonlinear permeability.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-7	1st	A	Enter the word CONTROL.
2nd data block			
1-5	1st	I	Maximum number of steps in this run. Default is 9999.
6-10	2nd	I	Maximum number of recycles during an increment. Default is 3.
11-15	3rd	I	Minimum number of recycles during an increment.
3rd data block			
1-10	1st	F	Maximum allowed relative error in residual current.
11-20	2nd	F	Maximum allowed absolute error in residual current.

Electromagnetic Analysis

This section describes the input of material data and boundary conditions applicable for electromagnetic problems. The ISOTROPIC and ORTHOTROPIC options are used to define magnetic permeability, electrical permittivity, conductivity and susceptibility in the electromagnetic analysis. The variation of magnetic permeability, with either magnetic field density or magnetic field vector, can be prescribed by the B-H RELATION option. In addition, user subroutine FLUX can be used for variable distributions of currents and charges.

Electromagnetic analysis can be performed using either a harmonic or transient approach. If the harmonic approach is used, the steady state sinusoidal result is obtained. Using this method, the excitation frequency is given using the HARMONIC option. If the transient approach is used, the time step is defined using the DYNAMIC CHANGE option.

■ FIXED POTENTIAL

Define Nodal Fixed Potential

Description

This option defines the fixed potential that each node must take during the first and subsequent increments. The boundary conditions are specified either by giving the potential and a list of nodal numbers, or by the input of boundary conditions generated during mesh generation (MESH2D). In electromagnetic analysis, the potential consists of vector and scalar potentials, the first three of which are associated with the magnetic vector potential, and the fourth degree of freedom with the scalar potential.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-19	1st	A	Enter the words FIXED POTENTIAL.
2nd data block			
1-5	1st	I	Number of sets of boundary condition blocks to be read (optional).
6-10	2nd	I	This field is set to nonzero to flag input of boundary conditions set during mesh generation (MESH2D). If this field is activated, no further blocks are required in this option block.
11-15	3rd	I	Unit number used for the MESH2D option. Note that the boundary conditions are stored after the connectivity and coordinate data on this tape, so that the model definition options must be arranged accordingly.

Format Fixed	Format Free	Data Type	Entry
Data blocks 3, 4, and 5 are repeated for each set.			
3rd data block			
1-10	1st	F	Prescribed potential for first degree of freedom listed in data block 4.
11-20	2nd	F	Prescribed potential for second degree of freedom listed in data block 4.
21-30	3rd	F	Prescribed potential for third degree of freedom listed in data block 4.
31-40	4th	F	Prescribed potential for fourth degree of freedom list in data block 4.
4th data block			
			Enter a list of degrees of freedom to which the above prescribed potentials are given. **Note:** List verbs `EXCEPT` and `INTERSECT` are illegal here.
5th data block			
			Enter a list of nodes for which the above boundary conditions are applied.

■ DIST CURRENT

Define Distributed Currents

Description

This block of data allows distributed (surface and volumetric) currents to be specified. Distributed currents are converted to consistent nodal currents by the program. Note that for a given element type, there is an established convention for the application of surface current of a particular face. User subroutine FORCEM can be used to input spatially dependent current.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-12	1st	A	Enter the words DIST CURRENT.
2nd data block			
1-5	1st	I	Enter the number of sets of distributed currents to be entered (optional).
6-10	2nd	I	Enter unit number for input of distributed current data, defaults to input.

The 3rd and 4th data blocks are entered as pairs, repeated for each data set.

3rd data block			
1-5	1st	I	Parameter identifying the type of current. See library element description in *Volume B: Element Library.*
6-15	2nd	F	Enter the magnitude of this type of distributed current.
16-20	3rd	I	Current index (current index) is to be used in user subroutine FORCEM).
4th data block			
			Enter a list of elements associated with the above distributed current.

■ DIST CHARGE

Define Distributed Charges

Description

This option allows distributed (surface and volumetric) charges to be specified. Distributed charges are converted to consistent nodal charges by the program. Note that for a given element type, there is an established convention for the application of surface charge on a particular face. User subroutine FORCEM can be used to input spatially dependent charges.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words DIST CHARGE.
2nd data block			
1-5	1st	I	Enter the number of sets of distributed charges to be entered(optional).
6-10	2nd	I	Enter unit number for input of distributed charge data, defaults to input.

The 3rd and 4th data blocks are entered as pairs, repeated for each data set.

3rd data block			
1-5	1st	I	Parameter identifying the type of distributed charge. See description of library element in *Volume B: Element Library.*
6-15	2nd	F	Enter the magnitude of this type of distributed charges.
16-20	3rd	I	Charge index (optional). Charge index is to be used in user subroutine FORCEM.
4th data block			
			Enter a list of elements associated with the above distributed charges.

■ POINT CURRENT-CHARGE

Define Nodal Point Currents and Point Charges

Description

This block of data allows nodal point currents and point charges to be specified. User subroutine FORCDT can be used for nonuniform loading conditions.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words POINT CURRENT-CHARGE.
2nd data block			
1-5	1st	I	Enter number of sets of point current and charge to be entered (optional).
6-10	2nd	I	Enter unit number for input of point current and charge data; defaults to input.

The 3rd and 4th data blocks should be entered as pairs and repeated for each data set.

3rd data block			
1-10	1st	F	Magnitude of point current for first degree of freedom.
11-20	2nd	F	Magnitude of point current for second degree of freedom.
21-30	3rd	F	Magnitude of point current for third degree of freedom.
31-40	4th	F	Nodal charge.
4th data block			
			Enter a list of nodes to which the above point current-charge applies.

■ ISOTROPIC

Define Electromagnetic Properties for Isotropic Materials

Description

This option allows you to define electromagnetic properties for an isotropic material. You can also associate these material properties with a list of element numbers.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10		A	Enter the word ISOTROPIC.
2nd data block			
1-5		I	Enter the number of sets of isotropic material data to follow (optional).
6-10		I	Enter the unit number for input. Defaults to input file.

The 3rd, 4th, and 5th data blocks should be entered as pairs and repeated for each data set.

3rd data block			
1-5		I	Material identification number (1, 2, 3, etc.) for cross-reference to B-H RELATION option.
4th data block			
1-10		F	Permeability (μ).
11-20		F	Permittivity (ε).
21-30		F	Permeability of air.
31-40		F	Electric conductivity (σ).
5th data block			
			Enter a list of elements associated with this material.

■ ORTHOTROPIC

Define Electromagnetic Properties for Orthotropic Materials

Description

This option allows you to define electromagnetic properties for an orthotropic material. You can also associate these material properties with a list of element numbers.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word ORTHOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of sets of orthotropic material data to follow.
6-10	2nd	I	Enter the unit number for input. Defaults to input file.

Data blocks 3-6 are repeated once for each data set.

3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-reference to B-H RELATION option.
6-10	2nd	I	Enter 1 to call user subroutines UMU, UEPS, and USIGMA.
4th data block			

Note: Since the material properties in an orthotropic material are independent, and a full vector potential is used, three values must be entered for the permeability, permittivity and conductivity.

These values are with respect to the global coordinate system.

1-10	1st	F	μ_{11} Magnetic permeability
1-20	2nd	F	μ_{22} Magnetic permeability
21-30	3rd	F	μ_{33} Magnetic permeability
31-40	4th	F	ε_{11} Permittivity

Format Fixed	Format Free	Data Type	Entry
1-50	5th	F	ε_{22} Permittivity
51-60	6th	F	ε_{33} Permittivity
61-70	7th	F	Permeability of air
5th data block			
1-10		F	σ_{11} Electrical Conductivity
11-20		F	σ_{22} Electrical Conductivity
21-30		F	σ_{33} Electrical Conductivity
6th data block			
			Enter a list of elements associated with this material.

■ B-H RELATION

Define Magnetization Curve for Nonlinear Magnetic Material

Description

This option can be used to specify the magnetization curve(s) for nonlinear isotropic or orthotropic material. Depending on the material type, a different method of entering data must be used:

isotropic material: (OPTION A) Enter one set of data points (|H|, |B|) representing the magnitude of H as a function of the magnitude of B. For |H| = 0 the value of |B| should be zero. If not, the corresponding offset of the curve is disregarded.

orthotropic material: (OPTION B) For every component of H a set of data points (H,B) is entered, relating this component of H to the corresponding component of B. A component of the remanence vector can be specified by choosing a nonzero value of B for H = 0.

Note: In either case, the curve(s) represented by the data sets must be monotone and uniquely defined. Furthermore, the data points must be given in ascending order of B.

An alternative way of specifying the magnetization curve(s) is to supply the reluctivity 1/μ in user subroutine UMU.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-12	1st	A	Enter the words B-H RELATION.

For option A, use data blocks 2a, 3a, 4a, and 5a. For option B, use data blocks 2b, 3b, 4b, and 5b.

Format Fixed	Format Free	Data Type	Entry

Option A Isotropic Behavior

2a data block

1-5	1st	I	Number of data points of |H|-|B| curve.
6-10	2nd	I	Not used; 0.
11-15	3rd	I	Not used; 0.
16-20	4th	I	Not used; 0.
21-25	5th	I	Not used; 0.
26-30	6th	I	Not used; 0.
31-35	7th	I	Material type identification (1,2,3...) for cross-reference to ISOTROPIC blocks.
36-40	8th	I	Logical unit number for input of this set of data. Defaults to data blocks.

3a data block

|H| - |B| variation. Number of blocks as given on data block 2, first field.

1-15	1st	F	Enter value of |H|.
16-30	2nd	F	Enter value of |B|.

Option B Orthotropic Behavior

2b data block

1-5	1st	I	Number of data points of H_1 - B_1 curve.
6-10	2nd	I	Number of data points of H_2 - B_2 curve.
11-15	3rdt	I	Number of data points of H_3 - B_3 curve.
16-20	4th	I	Not used; enter zero.
21-25	5th	I	Not used; enter zero.
26-30	6th	I	Not used; enter zero.

Format Fixed	Format Free	Data Type	Entry
31-35	7th	I	Material type identification (1,2,3...) for cross-reference to ORTHOTROPIC blocks.
36-40	8th	I	Logical unit number for input of this set of data. Defaults to data blocks.

3b data block

H_1 - B_1 variation. Number of blocks given on data block 2, first field.

1-15	1st	F	Enter value of H_1.
16-30	2nd	F	Enter value of B_1.

4b data block

H_2 - B_2 variation. Number of blocks given on data block 2, second field.

1-15	1st	F	Enter value of H_2.
16-30	2nd	F	Enter value of B_2.

5b data block

H_3 - B_3 variation. Number of blocks given on data block 2, third field.

1-15	1st	F	Enter value of H_3.
16-30	2nd	F	Enter value of B_3.

■ PERMANENT

Define Permanent Magnet

Description

This option provides various ways of defining a permanent magnet in parts of the model. The default is that no permanent magnets are present.

You need to enter the remanence vector B_v, which is the product of the magnet vector M and the permeability of the vacuum μ_0.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-13	1st	A	Enter the word PERMANENT.
2nd data block			
1-5	1st	I	Not used; enter 0.
6-10	2nd	I	Enter 1 to initialize the permanent magnet via data blocks 3 and 4 below. See also the third field on this block. Enter 4 to initialize the permanent magnet via data blocks 5, 6, 7, and 8 given below. See also the third field on this block.
11-15	3rd	I	This entry gives the number of pairs of blocks in data blocks 3 and 4 or in data blocks 5, 6, 7, and 8 used to input the permanent magnet.

Data blocks 3 and 4 are only input if the second field above is set to 1. In that case, the number of blocks is equal to the number given in the third field above.

3rd data block			
1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with the value.
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value.

Format Fixed	Format Free	Data Type	Entry
21-25	5th	I	First layer of cross section point with this value.
26-30	6th	I	Last layer or cross section point with this value can only be bigger than 1 for beam or shell elements.

4th data block

1-10	1st	F	First component of remanence vector.
11-20	2nd	F	Second component of remanence vector.
21-30	3rd	F	Third component of remanence vector.

Data blocks 5, 6, 7 and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block

1-10	1st	F	First component of remanence vector.
11-20	2nd	F	Second component of remanence vector.
21-30	3rd	F	Third component of remanence vector.

6th data block

Enter a list of elements to which the above state variable is applied.

7th data block

This data block is **not** necessary if CENTROID parameter is used.

Enter a list of integration points to which the above remanence is applied.

8th data block

This data block is only necessary if there are either beams or shells in the mesh.

Enter a list of layer points to which the above remanence is applied.

■ CONTROL

Control for Electromagnetostatic Analysis

Description

This option allows you to input parameters governing the convergence and accuracy for magnetostatic analysis. This option is only required if the B-H RELATION option is used to enter a nonlinear permeability.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-7	1st	A	Enter the word CONTROL.
2nd data block			
1-5	1st	I	Maximum number of steps in this run. Default is 9999.
6-10	2nd	I	Maximum number of recycles during an increment. Default is 3.
11-15	3rd	I	Minimum number of recycles during an increment.
3rd data block			
1-10	1st	F	Maximum allowed relative error in residual current.
11-20	2nd	F	Maximum allowed absolute error in residual current.

Fluid Analysis

This section describes the input of material data and boundary conditions applicable for fluid, fluid-thermal, fluid-solid and fluid-thermal-solid interaction analyses. The ISOTROPIC option is used to define the fluid material properties, viscosity, density, and, if necessary, the conductivity and specific heat. Non-Newtonian fluid behavior can be defined through the STRAIN RATE option, while temperature dependent properties are defined through the TEMPERATURE EFFECTS option. The boundary conditions on a fluid can be either defined through the FIXED VELOCITY, or the POINT LOAD, and DIST LOADS options. The POINT LOAD and DIST LOADS options are described in the Mechanical Analysis section. In a fluid-thermal analysis, the additional boundary conditions are FIXED TEMPERATURE, POINT FLUX, DIST FLUXES, and FILMS which are defined in the Heat Transfer Analysis section. In a fluid-solid analysis, the boundary conditions on the solid region are specified using the FIXED DISP, POINT LOAD, DIST LOAD, and FOUNDATION options. Initial conditions in a transient analysis can be specified using the INITIAL VEL or INITIAL TEMP options.

Note the degrees of freedom in an analysis are dependent upon the type of analysis and the procedure used. This is important when applying boundary conditions in these analyses and is summarized as follows:

	Fluid Parameter	**2D-planar**	**2D-axisymmetric**	**3D**
Fluid only, mixed	10	v_x , v_y , p	v_z , v_r , p	v_x , v_y , v_z , p
Fluid only, penalty	11	v_x , v_y	v_z , v_r	v_x , v_y , v_z
Fluid-thermal, mixed, strong coupling	12	v_x , v_y , p, T	v_z , v_r , p, T	v_x , v_y , v_z , p, T
Fluid-thermal, penalty, strong coupling	13	v_x , v_y , T	v_x , v_r , T	v_x , v_y , v_z , T
Fluid-thermal, mixed, weak coupling	2	v_x , v_y, p	v_z , v_r , p	v_x , v_y , v_z , p
		T	T	T
Fluid-thermal, penalty, weak coupling	3	v_x , v_y	v_z , v_r	v_x , v_y , v_z
		T	T	T

	Fluid Parameter	2D-planar	2D-axisymmetric	3D
Fluid-solid, mixed, weak coupled	40	v_x , v_y , p	v_z , v_r , p	v_x , v_y , v_z
		u_x , u_y	u_z , u_r	u_x , u_y , u_z
Fluid-solid, penalty, weak coupling	41	v_x , v_y	v_z , v_r	v_x , v_y , v_z
		u_x , u_y	u_z , u_r	u_x , u_y , u_z
Fluid-thermal-solid, mixed, strong-weak	42	v_x , v_y , p, T	v_x , v_r , p, T	v_x , v_y , v_z , p, T
		u_x , u_y	u_z , u_r	u_x , u_y , u_z
Fluid-thermal-solid, penalty, strong-weak	43	v_x , v_y , T	v_z , v_r , T	v_x , v_y , v_z , T
		u_x , u_y	u_z , u_r	u_x , u_y , u_z
Fluid-thermal-solid, mixed, weak-weak	44	v_x , v_y , p	v_z , v_r , p	v_x , v_y , v_z
		T	T	T
		u_x , u_y	u_z , u_r	u_x , u_y , u_z
Fluid-thermal-solid, penalty, weak-weak	45	v_x , v_y	v_z , v_r	v_x , v_y , v_z
		T	T	T
		u_x , u_y	u_z , u_r	u_x , u_y , u_z

■ REGION

Define Elements in a Region

Description

This option allows you to define which elements are part of a region. In fluid-solid, or fluid-thermal-solid analysis, it is necessary to divide the model into different regions depending on whether only a fluid analysis is performed in an area or a structural analysis is performed. This is used in conjunction with using the weakly coupled formulations. See *Volume A: Theory and User Information* for more details.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word REGION.
2nd data block			
1-5	1st	I	Enter the number of regions.
6-10	2nd	I	Enter the unit number for reading data. Defaults to input.
Repeat data blocks 3 and 4 for region.			
3rd data block			
1-5	1st	I	Enter the region type: 1 – solid region 3 – fluid region
4th data block			
			Enter a list of elements.

■ FIXED DISP

Define Fixed Displacement

Description

This data defines the fixed displacement that each specified degree of freedom must take during the first and subsequent increments, unless it is further modified using the DISP CHANGE option. The boundary conditions are specified either by giving the kinematic displacement and a list of degrees of freedom and a list of nodal numbers or by the input of boundary conditions generated during mesh generation (MESH2D).

Note: In static analysis, the boundary conditions specified must always be sufficient to remove all rigid body modes.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words FIXED DISP.
2nd data block			
1-5	1st	I	Number of sets of boundary condition data to be read (optional).
6-10	2nd	I	This field is set to nonzero to flag input of boundary conditions set during mesh generation (MESH2D). If this field is activated, no further data is required in this option block.
11-15	3rd	I	Unit number used for MESH2D option. Note that the boundary conditions are stored after the connectivity and coordinate data on this file, so that the model definition data must be arranged accordingly.

Format Fixed	Format Free	Data Type	Entry
For each set of boundary conditions use the 3rd, 4th and 5th data blocks.			
3rd data block			
1-10	1st	E	Prescribed displacement for first degree of freedom listed in data block 4.
11-20	2nd	E	Prescribed displacement for second degree of freedom listed in data block 4.
21-30	3rd	E	Prescribed displacement for third degree of freedom listed in data block 4. A maximum of eight kinematic constraints can be specified. 3rd data block is read as 8E10.3.
4th data block			
			Enter a list of degrees of freedom to which the above prescribed displacements are prescribed. **Note:** List verbs `EXCEPT` and `INTERSECT` are illegal here.
5th data block			
			Enter a list of nodes to which the above boundary conditions are applied.

■ FIXED VELOCITY

Define Fixed Velocity

Description

This data defines the fixed velocity that each specified degree of freedom must take during the first and subsequent increments, unless it is further modified using the VELOCITY CHANGE option in a fluid analysis. The boundary conditions are specified either by giving the kinematic velocity and a list of degrees of freedom and a list of nodal numbers or by the input of boundary conditions generated during mesh generation (MESH2D).

Note: In steady state analysis, the boundary conditions specified must always be sufficient to remove all rigid body modes.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words FIXED VELOCITY.
2nd data block			
1-5	1st	I	Number of sets of boundary condition data to be read (optional).
6-10	2nd	I	This field is set to nonzero to flag input of boundary conditions set during mesh generation (MESH2D). If this field is activated, no further data is required in this option block.
11-15	3rd	I	Unit number used for MESH2D option. Note that the boundary conditions are stored after the connectivity and coordinate data on this file, so that the model definition data must be arranged accordingly.

Format Fixed	Format Free	Data Type	Entry
For each set of boundary conditions use the 3rd, 4th and 5th data blocks.			
3rd data block			
1-10	1st	E	Prescribed velocity for first degree of freedom listed in data block 4.
11-20	2nd	E	Prescribed velocity for second degree of freedom listed in data block 4.
21-30	3rd	F	Prescribed velocity for third degree of freedom listed in data block 4. A maximum of eight kinematic constraints can be specified. 3rd data block is read as 8E10.3.
4th data block			
			Enter a list of degrees of freedom to which the above prescribed velocities are prescribed. **Note:** List verbs `EXCEPT` and `INTERSECT` are illegal here.
5th data block			
			Enter a list of nodes to which the above boundary conditions are applied.

■ ISOTROPIC

Define Material Properties for Fluid Analysis

Description

This option is used to define the fluid properties for all of the elements. To define the dependence of these properties on temperature, use the TEMPERATURE EFFECTS model definition option.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-8	1st	A	Enter the word ISOTROPIC.
2nd data block			
1-5	1st	I	Enter the number of distinct sets of element properties to be input (optional). For temperature dependent properties, these are values corresponding to the first (lowest temperature) breakpoint (see TEMPERATURE EFFECTS option). A temperature dependent property is undefined below its lowest breakpoint.
6-10	2nd	I	Enter unit number for input of data. Defaults to input.

The 3rd, 4th, and 5th data blocks should be entered as pairs, one for each distinct data block.

3rd data block			
1-5	1st	I	Material identification number (1,2,3, etc.) for cross-reference to TEMPERATURE EFFECTS option.
6-10	2nd	A	Enter the word FLUID if fluid-solid interaction and this is a fluid region.

Format Fixed	Format Free	Data Type	Entry
4th data block			
1-10	1st	F	Enter the reference temperature value of the viscosity.
11-20	2nd	F	Not used; enter 0.
21-30	3rd	F	Enter the reference value of the mass density of the fluid.
31-40	4th	F	Enter the coefficient of volumetric expansion.
5th data block			
Necessary only in a coupled fluid-thermal analysis or a fluid-thermal-solid analysis.			
1-10	1st	F	Thermal conductivity
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis)
31-40	4th	F	Reference temperature.
6th data block			
			Enter element data set for which the properties as specified in data block 3 applies.

■ STRAIN RATE

Define Strain Rate Dependent Viscosity

Description

This option allows the definition of a strain rate dependent viscosity for use in fluid flow problems. It also allows specification of different non-Newtonian viscosity models. The zero strain rate viscosity is given on the ISOTROPIC option. This block must be repeated for each different material for which strain rate data is necessary. The yield stress variation with strain rate is given using the following options:

A. The breakpoints and slopes for a piecewise linear approximation to the viscosity strain rate curve are given. The strain rate breakpoints should be in ascending order, or

B. The viscosity and stain rate data points lying on the viscosity strain rate curve are input directly. The data is entered in ascending order of strain rate. This method is flagged by entering the word "DATA" on the first data block.

C. Bingham Fluid – enter 3 in the fourth field on the second data block.

D. Fluid in the form of Power Law Relation – enter 4 in the fourth field on the second data block.

E. Fluid in the form of Generalized Power Law Relation – enter 5 in the fourth field on the second data block.

F. Fluid in the form of Carreau Model - enter 6 in the fourth field on the second data block.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-11	1st	A	Enter the words STRAIN RATE.
13-80	2nd	A	Enter the word DATA to indicate that option B is being used.
2nd data block			
1-5	1st	I	For option A, enter the number of slopes of viscosity versus strain rate curve. For option B, enter the number of data points. For other options, enter 0.

Format Fixed	Format Free	Data Type	Entry
6-10	2nd	I	Material type identification (1,2,3...) for cross-reference to ISOTROPIC blocks.
11-15	3rd	I	Unit number for input of the set of this block. Defaults to blocks.
16-20	4th	I	Non-Newtonian Viscosity Model type. Enter 0 for piecewise linear. Enter 3 for Bingham Fluid. Enter 4 for Power Law Relation. Enter 5 for Generalized Power Law Relation. Enter 6 for Carreau model.

3a data block

Data block 3a is used in conjunction with piecewise representations, Option A. The number of blocks in this series is equal to that given in the first field of data block 2.

1-10	1st	F	Enter the slope of the viscosity versus strain rate curve.
11-20	2nd	F	Enter the strain rate value above which the above slope becomes operational. Note, the first strain rate breakpoint must be zero.

3b data block

Data block 3b is used in conjunction with piecewise representation, Option B.

1-10	1st	F	Enter the value of the viscosity.
11-20	2nd	F	Enter the associated strain rate. Note that the first strain-rate must be zero.

3c data block

Data block 3c is used for Bingham Fluid.

1-10	1st	F	Enter the value of g (stress, see *Volume A: User Information*).
11-20	2nd	F	Enter the associated strain rate. Note that the first strain rate must be zero.

Format Fixed	Format Free	Data Type	Entry
3d data block			
Data block 3d is used for Power Law Relation Fluid.			
1-10	1st	F	Enter the value of K (stress, see *Volume A: Theory and User Information*).
11-20	2nd	F	Enter the value of N (power, see *Volume A: Theory and User Information*).
21-30	3rd	F	Enter the value of D (cutoff shear rate, see *Volume A: Theory and User Information*).
3e data block			
Data block 3e is used for Generalized Power Law Relation Fluid.			
1-10	1st	F	Enter the value of K (stress, see *Volume A: Theory and User Information*).
11-20	2nd	F	Enter the value of N (power, see *Volume A: Theory and User Information*).
21-30	3rd	F	Enter the value of D (cutoff shear rate, see *Volume A: Theory and User Information*).
31-40	4th	F	Enter the value of A1 (see *Volume A: Theory and User Information*).
41-50	5th	F	Enter the value of A2 (see *Volume A: Theory and User Information*).
51-60	6th	F	Enter the value of A3 (see *Volume A: Theory and User Information*).
61-70	7th	F	Enter the value of A4 (see *Volume A: Theory and User Information*).
3f data block			
Data block 3f is used for Carreau Model Fluid.			
1-10	1st	F	Enter the value of K (stress, see *Volume A: Theory and User Information*).
11-20	2nd	F	Enter the value of N (power, see *Volume A: Theory and User Information*).
21-30	3rd	F	Enter the value of μ (infinite shear rate viscosity, see *Volume A: Theory and User Information*).

■ TEMPERATURE EFFECTS

Temperature Effects in Coupled Fluid-Thermal Analysis

Description

This data block defines the variation of element properties (viscosity, thermal conductivity, specific heat) with temperature. The values read in through the ISOTROPIC option are those at the lowest temperature specified. Properties are not defined below the lowest temperature. The temperature dependency can be entered using one of the following two options:

- The variation of a particular property with temperature is specified as a piecewise linear curve. Breakpoints must be given in ascending order of temperature.
- The particular property value and temperature lying on the relevant curve are input directly. Data points must be given in increasing order of temperature. This option is flagged by entering the word DATA on the 1st data line.

Note: For Mooney materials, the temperature dependence for C10 and C01 can be defined by replacing C10 for "yield stress" and C01 for "Young's modulus". The other constants can be specified by utilizing user subroutine UMOONY.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-19	1st	A	Enter the words TEMPERATURE EFFECTS.
21-80	2nd	A	Enter the word DATA to indicate that option B is used.

For option A, use data blocks 2a, 3a, 4a, 5a, 6a, 7a and 8a. For option B, use data blocks 2b, 3b, 4b, 5b, 6b, 7b and 8b, below.

Format Fixed	Format Free	Data Type	Entry

Option A

2a data block

1-5	1st	I	Not used; enter 0.
6-10	2nd	I	Not used; enter 0.
11-15	3rd	I	Not used; enter 0.
16-20	4th	I	Not used; enter 0.
21-25	5th	I	Number of slopes of viscosity versus temperature curve.
26-30	6th	I	Not used; enter 0.
31-35	7th	I	Number of slopes of conductivity versus temperature curve
36-40	8th	I	Number of slopes of specific heat versus temperature curve.
41-45	9th	I	Number of latent heats to be entered.
46-50	10th	I	Number of slopes of emissivity versus temperature curve.
31-35	11th	I	Material type identification (1,2,3,...) for cross-referencing the ISOTROPIC blocks.
36-40	12th	I	Logical unit number for input of this set of data. Defaults to blocks.

3a data block

Slopes/breakpoints for viscosity versus temperature curve. The number in the fifth field of data line 2 defines the number of data lines required in data block 3.

1-15	1st	F	Enter the slope of viscosity versus temperature curve.
16-30	2nd	F	Enter the temperature at which this slope becomes operative.

Format Fixed	Format Free	Data Type	Entry
4a data block			
Conductivity variation. Number of data lines as given on data line 2, seventh field.			
1-15	1st	F	Enter the slope of conductivity versus temperature curve.
16-30	2nd	F	Enter the temperature at which the above slope becomes operative.
5a data block			
Specific heat variation. Number of data lines as given on data line 2, eighth field.			
1-15	1st	F	Enter the slope of specific heat versus temperature curve.
16-30	2nd	F	Temperature above which this slope becomes operative.
6a data block			
Latent heat. Number of data lines given on data line 2, ninth field.			
1-15	1st	F	Enter the latent heat.
16-30	2nd	F	Enter the solidus temperature (lower phase change limit).
31-45	3rd	F	Enter the liquidus temperature (upper phase change limit).

Option B

2b data block			
1-5	1st	I	Not used; enter 0.
6-10	2nd	I	Not used; enter 0.
11-15	3rd	I	Not used; enter 0.
16-20	4th	I	Not used; enter 0.
21-25	5th	I	Number of data points on the viscosity versus temperature curve for powder materials.
26-30	6th	I	Not used; enter 0.
31-35	7th	I	Number of data points on the conductivity versus temperature curve.

Format Fixed	Format Free	Data Type	Entry
36-40	8th	I	Number of data points on the specific heat versus temperature curve.
41-45	9th	I	Number of latent heats to be entered.
46-50	10th	I	Number of data points on the emissivity versus temperature curve.
51-55	11th	I	Material type identification (1,2,3,...) for cross-referencing the ISOTROPIC blocks.
56-60	12th	I	Logical unit number for input of this set of data. Defaults to data lines.

3b data block

The number in the fifth field of data line 2 defines the number of data lines required in data block 7.

1-15	1st	F	Enter the value of the viscosity.
16-30	2nd	F	Enter the associated temperature.

4b data block

Conductivity variation. Number of data lines as given on data line 2, seventh field.

1-15	1st	F	Enter the value of the conductivity.
16-30	2nd	F	Enter the associated temperature.

5b data block

Specific heat variation. Number of data lines as given on data line 2, eighth field.

1-15	1st	F	Enter the value of the specific heat.
16-30	2nd	F	Enter the associated temperature.

6b data block

Latent heat. Number of data lines as given on data line 2, ninth field.

1-15	1st	F	Enter the value of the latent heat.
16-30	2nd	F	Enter the solidus temperature (lower phase change limit).
31-45	3rd	F	Enter the liquidus temperature (upper phase change limit).

Format Fixed	Format Free	Data Type	Entry
7b data block			
Emissivity variation. Number of data lines as given on data line 2, tenth field.			
1-15	1st	F	Enter the value of the emissivity.
16-30	2nd	F	Enter the associated temperature.

Note: In calculating a particular temperature dependent property, the program averages the value of this property at the start and at the end of the increment.

■ CONTROL

Control Option for Fluid Analysis or Fluid-Thermal Analysis

Description

This option allows you to input parameters governing the convergence and the accuracy for fluid analysis.

For coupled fluid-thermal analysis, data block 4 must be used in addition to the 3rd data block.

For nonlinear analysis, the controls are described in *Volume A: Theory and User Information*. They do not appear on the restart file, and so must be re-entered on a restart run.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-7	1st	A	Enter the word CONTROL.
2nd data block			
1-5	1st	I	Maximum number of load steps/increments in this run. Default is 9999. This is a cumulative number and is usually used to stop the run when restart is being used.
6-10	2nd	I	Maximum number of recycles/increments during an increment. Default is 3. If a negative number is entered, then the program will do a maximum of the absolute value entered. If convergence has not been obtained, a warning is given and the program proceeds to the next increment. This is not recommended.
11-15	3rd	I	Minimum number of recycles during an increment. Default is 0. Note that this data field forces this number of recycles to take place at all subsequent increments.

Format Fixed	Format Free	Data Type	Entry
16-20	4th	I	Flag for convergence testing. If set to 0 or left blank, testing is done on residuals. If set to one, testing is done on velocities. Note that testing on relative velocity always requires at least one iteration. If nonlinear analysis is done with the CENTROID parameter, the residuals are not calculated and testing is always done on velocities. Note that fluid analysis with the CENTROID parameter is not recommended.
21-25	5th	I	Flag to specify relative or absolute error testing. If equal to 0, testing is done on relative error. If equal to 1, testing is done on absolute value. If set to 2, testing is done on relative error testing unless reactions or incremental velocities are below minimum value in which case absolute tolerances testing is used.
26-30	6th	I	Iterative procedure flag. 1. Full Newton-Raphson (default). 4. Direct substitution.
31-35	7th	I	Nonpositive definite flag. If set to 1, solution of nonpositive definite system is forced. Default in fluid analysis.
36-40	8th	I	No longer used; enter 0.
41-45	9th	I	To print convergence control messages to log file, enter 1.
3rd data block			
1-10	1st	F	If relative residual checking: Maximum allowable value of maximum residual force divided by maximum reaction force. Default is 0.10. If relative velocity checking: Maximum allowable value of the change in velocity increment divided by the velocity increment. Default is 0.10.

Format Fixed	Format Free	Data Type	Entry
11-20	2nd	F	If relative residual checking: Maximum allowable value of maximum residual moment divided by maximum reaction moment. Default is 0.0, in which case, no checking on residual moment occurs.
			If relative velocity checking: Maximum allowable value of the change in rotational increment divided by the rotational increment. Default is 0.0, in which case, no checking on change in rotational increment occurs.
21-30	3rd	F	If relative residual checking: Minimum reaction force, if reaction force is less than this value, checking will be bypassed or absolute testing will be performed.
			If relative velocity checking: Minimum velocity, if velocity increment is less than this value, checking will be bypassed or absolute testing will be performed.

4th data block

Only necessary for coupled fluid-thermal analysis.

1-10	1st	F	Maximum nodal temperature change allowed. Used to control automatic time step scheme for heat transfer. Default value of 20.
11-20	2nd	F	Maximum nodal temperature change allowed before properties are re-evaluated and matrices reassembled. Default value of 100.
21-30	3rd	F	Maximum error in temperature estimate used for property evaluation. This control provides a recycling capability to improve accuracy in highly nonlinear heat-transfer problems (for example, latent heat, radiation boundary conditions). Default is 0, which bypasses this test. Set to maximum temperature error which is considered acceptable.

■ CONTROL

Control Option for Fluid-Stress or Fluid-Thermal-Stress Analysis

Description

This option allows you to input parameters governing the convergence and the accuracy for fluid-solid or fluid-thermal-solid analysis.

For nonlinear analysis, the controls are described in *Volume A: User Information*. They do not appear on the restart file, and so must be re-entered on a restart run.

In fluid-solid or fluid-thermal-solid analysis, there are two areas of the model which are defined using the REGION option. The data given here governs the convergence behavior in these regions. The 2nd and 3rd data blocks control the behavior in the solid region. The 4th and 5th data blocks control the behavior for thermal analysis in either region. The 6th and 7th data blocks control the fluid behavior.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-7	1st	A	Enter the word CONTROL.
2nd data block			
1-5	1st	I	Maximum number of load steps/increments in this run. Default is 9999. This is a cumulative number and is usually used to stop the run when restart is being used.
6-10	2nd	I	Maximum number of recycles/increments during an increment for solid region. If a negative number is entered, then the program will do a maximum of the absolute value entered. If convergence has not been obtained, a warning is given and the program proceeds to the next increment. This is not recommended.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Minimum number of recycles during an increment for plasticity or other tangent modulus nonlinearities in solid region. Default is 0. Note that this data field forces this number of recycles to take place at all subsequent increments.
16-20	4th	I	Flag for convergence testing. If set to 0 or left blank, testing is done on residuals. If set to one, testing is done on displacements. If set to two, testing is done on strain energy. Note that testing on relative displacements or strain energy always requires at least one iteration. If nonlinear analysis is done with the CENTROID parameter, the residuals are not calculated and testing is always done on displacements. Note that nonlinear analysis with the CENTROID parameter is not recommended.
21-25	5th	I	Flag to specify relative or absolute error testing. If equal to 0, testing is done on relative error. If equal to 1, testing is done on absolute value. If set to 2, testing is done on relative error testing unless reactions or incremental displacements are below minimum value in which case absolute tolerances testing is used.
26-30	6th	I	Iterative procedure flag. 1. Full Newton-Raphson (default). 2. Modified Newton-Raphson (no reassembly during iteration). 3. Newton-Raphson with strain correction modification. 8. Secant method.
31-35	7th	I	Nonpositive definite flag. If set to 1, solution of nonpositive definite system is forced. Note that with use of gap and Herrmann elements, the matrix always is nonpositive definite and this entry has no significance.

Format Fixed	Format Free	Data Type	Entry
36-40	8th	I	No longer used; enter 0.
41-45	9th	I	To print convergence control messages to log file, enter 1.
46-50	10th	I	Control on initial stress stiffness. 0 Normal-full contribution. 1 For Mooney material, reduce contribution of hydrostatic pressure on initial stress stiffness according to: $\sigma^{initial} = \sigma - f_r \cdot p \cdot I$ where $\sigma^{initial}$ is the stress tensor used in the initial stress stiffness matrix, σ is the current stress tensor, f_r is entered through the PARAMETERS option, p is the hydrostatic pressure and I is a unit tensor. 2 No initial stress stiffness. 3 Use stress at beginning of increment, not last iteration. Enter a list of elements to be printed.
3rd data block			
1-10	1st	F	If relative residual checking: Maximum allowable value of maximum residual force divided by maximum reaction force. Default is 0.10. If relative displacement checking: Maximum allowable value of the change in displacement increment divided by the displacement increment. Default is 0.10.

Format Fixed	Format Free	Data Type	Entry
11-20	2nd	F	If relative residual checking: Maximum allowable value of maximum residual moment divided by maximum reaction moment. Default is 0.0, in which case, no checking on residual moment occurs.
			If relative displacement checking: Maximum allowable value of the change in rotational increment divided by the rotational increment. Default is 0.0, in which case, no checking on change in rotational increment occurs.
21-30	3rd	F	If relative residual checking: Minimum reaction force, if reaction force is less than this value, checking will be bypassed or absolute testing will be performed.
			If relative displacement checking: Minimum displacement, if displacement increment is less than this value, checking will be bypassed or absolute testing will be performed.
31-40	4th	F	If relative residual checking: Minimum moment: if moment is less than this value, checking will be bypassed or absolute testing will be performed.
			If relative displacement checking: Minimum rotation: if rotation increment is less than this value, checking will be bypassed.
41-50	5th	F	If absolute residual testing: Maximum value of residual force. Default is 0.0 in which case, no checking on residual force will take place.
			If absolute displacement tasking: Maximum value of displacement increment. Default is 0.0; in which case, no checking on displacements take place.

Format Fixed	Format Free	Data Type	Entry
The 4th and 5th lines are used to control the thermal behavior. In an fluid-structure analysis (no thermal), do not include these blocks.			
4th data block			
1-5	1st	I	Maximum number of load steps in this run. Default is 99999. This is a cumulative number and is usually used to stop the run when RESTART is being used.
6-10	2nd	I	Maximum number of recycles during an increment due to temperature dependent material properties. Default value is 3.
11-15	3rd	I	Minimum number of recycles during an increment. Note that this data field forces this number of recycles to take place in all subsequent increments.
5th data block			
1-10	1st	F	Maximum nodal temperature change allowed. Used to control automatic time step scheme for heat transfer. Default value of 20.
11-20	2nd	F	Maximum nodal temperature change allowed before properties are re-evaluated and matrices reassembled. Default value of 100.
21-30	3rd	F	Maximum error in temperature estimate used for property evaluation. This control provides a recycling capability to improve accuracy in highly nonlinear heat-transfer problems (for example, latent heat, radiation boundary conditions). Default is 0, which bypasses this test. Set to maximum temperature error which is considered acceptable.
6th data block			
1-5	1st	I	Maximum number of load steps/increments in this run. Default is 9999. This is a cumulative number and is usually used to stop the run when restart is being used.

Format		Data	
Fixed	Free	Type	Entry
6-10	2nd	I	Maximum number of recycles/increments during an increment. If a negative number is entered, then the program will do a maximum of the absolute value entered. If convergence has not been obtained, a warning is given and the program proceeds to the next increment. This is not recommended.
11-15	3rd	I	Minimum number of recycles during an increment. Default is 0. Note that this data field forces this number of recycles to take place at all subsequent increments.
16-20	4th	I	Flag for convergence testing. If set to 0 or left blank, testing is done on residuals. If set to one, testing is done on velocities. Note that testing on relative velocity always requires at least one iteration. If nonlinear analysis is done with the CENTROID parameter, the residuals are not calculated and testing is always done on velocities. Note that fluid analysis with the CENTROID parameter is not recommended.
21-25	5th	I	Flag to specify relative or absolute error testing. If equal to 0, testing is done on relative error. If equal to 1, testing is done on absolute value. If set to 2, testing is done on relative error testing unless reactions or incremental velocities are below minimum value in which case absolute tolerances testing is used.
26-30	6th	I	Iterative procedure flag. 1. Full Newton-Raphson (default). 4. Direct substitution.
31-35	7th	I	Nonpositive definite flag. If set to 1, solution of nonpositive definite system is forced. Default in fluid analysis.
36-40	8th	I	No longer used; enter 0.
41-45	9th	I	To print convergence control messages to log file, enter 1.

Format Fixed	Format Free	Data Type	Entry
7th data block			
1-10	1st	F	If relative residual checking: Maximum allowable value of maximum residual force divided by maximum reaction force. Default is 0.10. If relative velocity checking: Maximum allowable value of the change in velocity increment divided by the velocity increment. Default is 0.10.
11-20	2nd	F	If relative residual checking: Maximum allowable value of maximum residual moment divided by maximum reaction moment. Default is 0.0, in which case, no checking on residual moment occurs. If relative velocity checking: Maximum allowable value of the change in rotational increment divided by the rotational increment. Default is 0.0, in which case, no checking on change in rotational increment occurs.
21-30	3rd	F	If relative residual checking: Minimum reaction force, if reaction force is less than this value, checking will be bypassed or absolute testing will be performed. If relative velocity checking: Minimum velocity, if velocity increment is less than this value, checking will be bypassed or absolute testing will be performed.

■ END OPTION

Model Definition Data End

Description

This option is used to signify the end of all model definition data.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10		A	Enter the words END OPTION.

4 List of History Definition Options

Options **Page**

4 *List of History Definition Options*

4 History Definition Options

The END OPTION ends the initial definition of the problem. All data subsequent to the END OPTION is interpreted as history definition or mesh display options. The input data for the history options controls the flow of the program for the next increment or the next series of increments in the case of automatic control of loading.

Facilities exist for changing the boundary conditions and the tying data. This new data is read in when the analysis of the previous series of increments is complete. The data and operations are then already defined for the next series of load increments. This data is always required after the first (zeroth) increment of load. A CONTINUE option terminates the input and initiates the analysis for the series of load increments corresponding to that input. The total number of increments is controlled by the CONTROL option or by conditions set within these options.

Mechanical, Acoustic, or Electromagnetic Analyses

All quantities specified for the previous load increment are also applied in the increment unless it is modified by the optional data described below. By judicious choice of the order of the load incrementation options, you can apply many combinations of displacement, traction, and thermal load vectors. You should note here that (unless the ELASTIC parameter is included) any new loads applied after the END OPTION are incremental and the total load at the end of any increment corresponds to the zero increment load plus all load increments to that point.

Once the repeated linear elastic analysis flag has been set by the ELASTIC parameter, successive load and/or temperature vectors are read in by the DIST LOADS, POINT LOAD, and/or the THERMAL LOAD, CHANGE STATE options.

CAUTION: You must ensure that you zero out load increments that are being switched off since the program, for some options, does not do this automatically.

Heat Transfer Analysis

For heat transfer analysis, total fluxes or temperatures should always be input, and the solution time controlled by the TRANSIENT option.

4

Hydrodynamic Bearing Analysis

In a bearing analysis, the lubricant thickness can be modified or the damping or stiffness behavior can be obtained.

General Controls

This section describes modifications of program controls. The values given here override the values that might have been previously specified either with the model definition options or prior load incrementation data. Note that the CONTROL option is extremely important for specifying the convergence and tolerance controls.

■ COMMENT

Enter Comments

Description

This option allows you to enter informative comments for your own benefit. These data blocks can appear between parameters, model definition, mesh display, and history definition options.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word COMMENT.
11-80	2nd	A	User entered comment.

Alternate Format

1-1	1st	A	Enter the $ character.
2-80	2nd	A	User entered comment.

■ TITLE

Output Title Definition

Description

This option defines the output title. There is no limit to the number of the title data read in as long as the word TITLE appears in the first field. However, only the last TITLE data is used as an output header. Due to the free-format processor, do not place commas within the TITLE data (Columns 11-80).

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word TITLE.
11-80	2nd	A	Enter the title to be output with results.

■ NEW

Use New Format

Description

This option allows the input of data in the K style format. Input is interpreted to be in this format until an OLD option is encountered. This option must not appear embedded inside a model definition, history definition, or mesh display options.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word NEW.
11-15	2nd	I	Enter 1 if the default width of the data fields is used for input. Enter 2 if the double width of the data fields is used for input.

■ OLD

Use Old Format

Description

This option allows the input of data in the old (G, H, or J) style format. Input is interpreted to be in this format until a NEW option is encountered. This option must not appear embedded inside a model definition, history definition or mesh display options.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word OLD.

■ PRINT CHOICE

Define Data to be Printed

Description

This option allows you the control of the output from the program. The data given here remains in control until a subsequent PRINT CHOICE set is inserted – such a set can be included with either the model definition or with history definition set.

The default values print all elements and all nodes. Element quantities are printed at each integration point or at the centroid only, depending on whether the ALL POINTS or CENTROID parameter is used. For shells, only the extreme fibers are output, plus layers where the inelastic strains or state variables are nonzero. For beams, fibers with inelastic strains or nonzero state variables are printed. In addition, section forces are given for these elements. This option also allows debug output of certain items. The default is no debug printout.

All of the above defaults are reset by the PRINT CHOICE option.

The PRINT CHOICE option has no effect on the post processor file (see Chapter 3).

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-12	1st	A	Enter the words PRINT CHOICE.
2nd data block			
1-5	1st	I	Number of sets of first and last element numbers to be printed (maximum 10).
6-10	2nd	I	Number of sets of first and last node numbers to be printed (maximum 10).
11-15	3rd	I	Number of integration points to be printed in each element (not used if ALL POINTS is not flagged).
16-20	4th	I	Number of layers to be printed. This is for beams and shells only, and overrides the default described above.
21-25	5th	I	Increments between printout. Default is print every increment.

Format Fixed	Format Free	Data Type	Entry
26-30	6th	I	Enter 1 for complex nodal quantities to be output as magnitude and phase, otherwise real and imaginary components are given.
31-35	7th	I	Debug print flag. Enter a nonzero value and use data block 7.
36-40	8th	I	Log File Flag: Enter unit number to which log file is to be written.

3rd data block

Include only if the first field of 2nd data block is not zero.

1-5	1st	I	First element in first set.
6-10	2nd	I	Last element in first set.
11-15	3rd	I	First element in second set.
16-20	4th	I	Last element in second set.
Etc.			Etc. in I5 format.

4th data block

Include only if the second field of 2nd data block is not zero.

1-5	1st	I	First node in first set.
6-10	2nd	I	Last node in first set.
11-15	3rd	I	First node in second set.
16-20	4th	I	Last node in second set.
Etc.			Etc. in I5 format.

5th data block

Include only if the third field of 2nd data block is not zero.

		I	Enter the list of integration point to be printed in (16I5) format (number of entries given in the third field of data block 2). This is only used if ALL POINTS is flagged. Be careful with analyses with several different element types.

Format Fixed	Format Free	Data Type	Entry
6th data block			
Include only if the fourth field of 2nd data block is not zero)			
		I	Enter the list of shell or beam fibers to be printed in (16I5) format. This over-rides the program default, so that you should be aware that you do not unintentionally miss plasticity or creep printout.
7th data block			
Include only if the seventh field is not zero.			
1-5	1st	I	Enter debug plot code. See PRINT parameter.

■ PRINT ELEMENT

Define Elements to be Printed

Description

This option allows you to choose which elements, and what quantities associated with an element are to be printed. If you do not specify NODE on the first data block, these values are at the integration points. If you specify the word NODE, these values are the extrapolated nodal values.

Note: This option revokes any NO PRINT that precedes it. Therefore, NO PRINT followed by PRINT ELEM and not followed by PRINT NODE results in the selected element printout and full nodal printout. Use PRINT NODE with a blank node list to suppress node output.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words PRINT ELEMENT.
11-20	2nd	A	Enter the word NODE (optional).
2nd data block			
1-5	1st	I	Enter the number of sets to be given below (optional).
6-10	2nd	I	Increment between printout. Default is print every increment.
11-15	3rd	I	File unit to which output is to be written, default to standard output, unit 6.

Format Fixed	Format Free	Data Type	Entry

Data blocks 3, 4 and if necessary 5 and 6 are repeated for each set.

3rd data block

1-80	1st	A	Enter one or more of the following:

STRAIN	output total strain
STRESS	output total stress
PLASTIC	output plastic strain
CREEP	output creep, swelling and viscoelastic strain
THERMAL	output thermal strain
ENERGY	output of strain energy
CRACK	output of cracking strain
CAUCHY	output Cauchy stress
STATE	output state variables
PREFER	output layered stresses in preferred system
ELECTRIC	output electric field and electric flux
MAGNETIC	output magnetic field and magnetic flux
CURRENT	output current
ALL	output of all of the above.

4th data block

Enter a list of elements to be printed. To suppress all element printout, enter a blank list for the list of elements.

5th data block

Enter a list of nodes to be printed. This node list is a list of internal element node numbers ranging from 1 to the maximum number of nodes per element.

6th data block

Enter a list of layers to be printed. This is only necessary if there are either thin walled beam or shell elements in the mesh.

■ PRINT NODE

Define Nodes and Nodal Quantities to be Printed

Description

This option allows you to choose which nodes and what nodal quantities are to be printed. The average nodal generalized stresses are obtained via an extrapolation and averaging procedure. If there is a geometric or material discontinuity at a node, this value will not be correct unless either double nodes were used and kinematic tying, or you control which elements are to be averaged using the PRINT ELEMENT feature.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words PRINT NODE.
2nd data block			
1-5	1st	I	Enter the number of sets to be given below (optional).
6-10	2nd	I	Increment between printout. Default is print every increment.
11-15	3rd	I	File unit to which output is to be written, default to standard output, unit 6.

Format Fixed	Format Free	Data Type	Entry

Data blocks 3 and 4 are entered as pairs, once for each data set.

3rd data block

1-80	1st	A	Enter one or more of the following:

INCR	output incremental displacement or potentials
TOTA	output total displacement or potentials
VELO	output velocity
ACCE	output acceleration
LOAD	output total applied load
REAC	output reaction/residual force
TEMP	output temperature
FLUX	output flux (Fluxes are only available if the parameter HEAT, 0, 0, 2 is used.)
STRESS	output average generalized stresses at nodes
VOLT	output voltage (Joule analysis)
PRES	output pressure (bearing analysis)
COOR	output coordinates (only for rezoning)
ALL	output all relevant quantities

4th data block

Enter a list of nodes to be printed. To suppress all nodal printout, enter a blank list for the list of nodes.

■ NO PRINT

Suppress Printing

Description

This option suppresses element and nodal output. This option is revoked by using either the PRINT CHOICE, PRINT ELEMENT, or PRINT NODE option.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words NO PRINT.

■ SUMMARY

Create Summary Report

Description

This option produces a summary of the results of the increment and output them in a report format. This option is in effect until a NO SUMMARY option is encountered. The summary consists of the maximum and minimum of temperatures, stresses, strains, plastic strains, creep strains, displacements, velocities, accelerations and reaction forces.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word SUMMARY.
11-15	2nd	I	Enter the unit number to be used for output, default is standard output, unit 6.
16-20	3rd	I	Enter the increment frequency of summary, default is every increment.

■ NO SUMMARY

Suppress Summary

Description

This option turns off the summary feature. The default is off unless the SUMMARY option has been previously invoked.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words NO SUMMARY.

■ ELEMENT SORT

Sort Elements for Report

Description

This option allows various element quantities to be sorted and the output given in report format. This option is in effect until a NO ELEM SORT option is encountered. The ELEMENT SORT option allows you to sort either in ascending or descending order. In addition, you can use either the real numeric value or the absolute value. A range can be given over which to sort.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words ELEM SORT.
2nd data block			
1-5	1st	I	Enter number of sorts to be performed (optional). One data block as given below defines each sort.
6-10	2nd	I	Enter unit number to read sort data.
11-15	3rd	I	Enter the unit number to be used for output; default is standard output, unit 6.
16-20	4th	I	Enter the increment frequency; default is every increment.
3rd data block			
This is given once for each sort.			
1-5	1st	I	Enter code indicating type of quantity to be sorted (see Table 4-1)
6-10	2nd	I	Enter 0 for sort in descending order. Enter 1 for sort in ascending order. Default is 0, sort in descending order.
11-15	34d	I	Enter 1 for sort by real numerical value. Enter 0 for sort by absolute value. Default is 0, sort by absolute value.

Format Fixed	Format Free	Data Type	Entry
16-20	4th	I	Enter number of items to be included in sorted list.
21-25	5th	I	Enter lowest element number of range to be sorted. Defaults to 1.
26-30	6th	I	Enter highest element number of range to be sorted. Defaults to last element in mesh.

Table 4-1 Element Sort Codes

1	first stress	28	fourth plastic strain
2	second stress	29	fifth plastic strain
3	third stress	30	sixth plastic strain
4	fourth stress	31	equivalent plastic strain
5	fifth stress	32	mean plastic strain
6	sixth stress	33	Tresca plastic strain
7	equivalent stress	34	first principal plastic strain
8	mean stress	35	second principal plastic strain
9	Tresca stress	36	third principal plastic strain
10	first principal stress	37	first creep strain
11	second principal stress	38	second creep strain
12	third principal stress	39	third creep strain
13	first strain	40	fourth creep strain
14	second strain	41	fifth creep strain
15	third strain	42	sixth creep strain
16	fourth strain	43	equivalent creep strain
17	fifth strain	44	mean creep strain
18	sixth strain	45	Tresca creep strain
19	equivalent strain	46	first principal creep strain
20	mean strain	47	second principal creep strain
21	Tresca strain	48	third principal creep strain

Table 4-1 Element Sort Codes

22	first principal strain	49	temperature
23	second principal strain	61	voltage
24	third principal strain	73	first gradient
25	first plastic strain	74	second gradient
26	second plastic strain	75	third gradient
27	third plastic strain		

■ NO ELEM SORT

Turn Off Element Sort

Description

This option turns off the ELEMENT SORT feature. The default is off unless the ELEMENT SORT has been previously invoked.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words NO ELEM SORT.

■ NODE SORT

Sort Nodes for Report

Description

This option allows various nodal quantities to be sorted and the output given in report format. This option is in effect until a NO NODE SORT option is encountered. The NODE SORT option allows you to sort either in ascending or descending order. In addition, you can use either the real numeric value or the absolute value. A range can be given over which to sort.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words NODE SORT.
2nd data block			
1-5	1st	I	Enter number of sorts to be performed (optional). One data block as given below defines each sort.
6-10	2nd	I	Enter unit number to read sort data.
11-15	3rd	I	Enter the unit number to be used for output; default is standard output, unit 6.
16-20	4th	I	Enter the increment frequency; default is every increment.
3rd data block			
This is given once for each sort.			
1-5	1st	I	Enter code indicating type of quantity to be sorted (see Table 4-2).
6-10	2nd	I	Enter 0 for sort in descending order. Enter 1 for sort in ascending order. Default is 0, sort in descending value.
11-15	3rd	I	Enter 1 for sort by real numerical value. Enter 0 for sort by absolute value. Default is 0; sort by absolute value.

Format		Data	
Fixed	Free	Type	Entry
16-20	4th	I	Enter number of items to be included in sorted list.
21-25	5th	I	Enter lowest node number of range to be sorted. Default is 1.
26-30	6th	I	Enter highest node number of range to be sorted. Defaults to last node in mesh.

Table 4-2 Node Sort Codes

Column	Code	Result
1-12	sort code I	Results in the Ith component of the incremental displacement to be sorted.
13-34	sort code I+12	Results in the Ith component of the total displacement to be sorted.
25-36	sort code I + 24	Results in the Ith component of the velocity to be sorted.
37-48	sort code I + 36	Results in the Ith component of the acceleration to be sorted.
48-60	sort code I + 48	Results in the nodal temperature to be sorted.
61-72	sort code I + 60	Results in the Ith component of the reaction force to be sorted.

■ NO NODE SORT

Turn Off Node Sort

Description

This option turns off the NODE SORT feature. The default is off unless the NODE SORT option has been previously invoked.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words NO NODE SORT.

■ PRINT VMASS

Print Element Volumes, Masses, Costs, and Strain Energies

Description

This option allows you to obtain printed output of element volumes, masses, costs and strain energies. Options are provided for you to print the total quantities for each group of elements and the quantities for each element in the group or the total quantities for each group of elements only.

In order to have correct mass computations, mass density for each element must be entered through the ISOTROPIC/ORTHOTROPIC option. In order to have the correct cost, the cost per unit mass or the cost per unit volume must be defined through the ISOTROPIC/ORTHOTROPIC option. In order to have the correct cost computation, the cost per unit mass or the cost per unit volume must be defined. The total strain energy and the plastic strain energy, if applicable, are printed. Note that volumes and masses for some special elements (for example, gap element, semi-infinite element, etc.) is not be computed. These quantities can be written on either standard output file unit 6, or your specified unit.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-11	1st	A	Enter the words PRINT VMASS.
2nd data block			
1-5	1st	I	Enter the number of sets to be given below.
6-10	2nd	I	Enter 1 for option to print only total volumes, masses, costs, and strain energy for groups of elements. Default is 0.
11-15	3rd	I	File unit to which output is to be written; default to standard output, unit 6.

Data block 3 is repeated for each set.

3rd data block			
			Enter a list of elements to be printed.

■ CONTROL

Define History Controls

This option allows you to input parameters governing the convergence and solution accuracy for nonlinear stress analysis.

For coupled thermal-stress analysis data block 4 must be used in addition to the 3rd data block.

For nonlinear static analysis, the controls are described in *Volume A: Theory and User Information.* They do not appear on the restart file, and so must be re-entered on a restart run.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-7	1st	A	Enter the word CONTROL.
2nd data block			
1-5	1st	I	Maximum number of load steps in this run. Default is 9999. This is a cumulative number and is usually used to stop the run when restart is being used. If the ELASTIC parameter is included, this field is ignored and all load cases are analyzed.
6-10	2nd	I	Maximum number of recycles during an increment for plasticity, or other tangent modulus nonlinearities. Default is 3. This should usually be increased to 10 for rigid-plastic flow option. If a negative number is entered, the program does a maximum of the absolute value entered. If convergence has not been obtained, a warning is given and the program proceeds to the next increment. This is not recommended.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Minimum number of recycles during an increment for plasticity or other tangent modulus nonlinearities. Default is 0. **Note:** This data field forces this number of recycles to take place at all subsequent increments. **CAUTION:** This value is overwritten by the PROPORTIONAL INCREMENT data block.
16-20	4th	I	Flag for convergence testing. If set to 0 or left blank, testing is done on residuals. If set to 1, testing is done on displacements. If set to 2, testing is done on strain energy. **Note:** Testing on displacements or strain energy always requires at least one iteration. If nonlinear analysis is done with the CENTROID option, the residuals are not calculated and testing is always done on displacements. **Note:** Nonlinear analysis with the CENTROID option is not recommended.
21-25	5th	I	Flag to specify relative or absolute error testing. If equal to 0, testing is done on relative error. If equal to 1, testing is done on absolute value. If set to 2, testing is done on relative error testing unless reactions or incremental displacements are below minimum value in which case absolute tolerances testing is used.
26-30	6th	I	Iterative procedure flag. 1. Full Newton-Raphson. (Default) 2. Modified Newton-Raphson (no reassembly during iteration). 3. Newton-Raphson with strain correction modification (see *Volume A: Theory and User Information*). 8. Secant method.

Format Fixed	Format Free	Data Type	Entry
31-35	7th	I	Nonpositive definite flag. If set to 1, solution of nonpositive definite system is forced. **Note:** With use of gap and Herrmann elements, the matrix always is nonpositive definite, and this entry has no significance.
36-40	8th	I	No longer used; enter 0.
41-45	9th	I	To print convergence control messages to log file, enter 1.
46-50	10th	I	Control on initial stress stiffness 0 Normal-full contribution. 1 For Mooney material, reduce contribution of hydrostatic pressure on initial stress stiffness according to: $\sigma^{initial} = \sigma - f_r \cdot p \cdot I$ where $\sigma^{initial}$ is the stress tensor used in the initial stress stiffness matrix, σ is the current stress tensor, f_r is entered through the PARAMETERS option, p is the hydrostatic pressure and I is a unit tensor. 2 No initial stress stiffness. 3 Use stress at beginning of increment, not last iteration.
3rd data block			
1-10	1st	F	If relative residual checking: Maximum allowable value of maximum residual force divided by maximum reaction force. Default is 0.10. If relative displacement checking: Maximum allowable value of the change in displacement increment divided by the displacement increment. Default is 0.10.

Format Fixed	Format Free	Data Type	Entry
11-20	2nd	F	If relative residual checking: Maximum allowable value of maximum residual moment divided by maximum reaction moment. Default is 0.0, in which case, no checking on residual moment occurs.
			If relative displacement checking: Maximum allowable value of the change in rotational increment divided by the rotational increment. Default is 0.0, in which case, no checking on change in rotational increment occurs.
21-30	3rd		If relative residual checking: Minimum reaction force, if reaction force is less than this value, checking is bypassed or absolute testing is performed.
			If relative displacement checking: Minimum displacement, if displacement increment is less than this value, checking is bypassed or absolute testing is performed.
31-40	4th		If relative residual checking: Minimum moment, if moment is less than this value, checking is bypassed or absolute testing is performed.
			If relative displacement checking: Minimum rotation, if rotation increment is less than this value, checking is bypassed or absolute testing is performed.
41-50	5th	F	If absolute residual testing: Maximum value of residual force. Default is 0.0; in which case, no checking on residual force takes place.
			If absolute displacement testing: maximum value of displacement increment. Default is 0.0; in which case no checking or displacements take place.

Format Fixed	Format Free	Data Type	Entry
51-60	6th	F	If absolute residual testing: Maximum value of residual moment. Default is 0.0; in which case, no checking on residual moments take place. If absolute displacement testing, maximum value of rotation increment. Default is 0.0; in which case, no checking or rotations take place.
61-70	7th	F	$\dot{\bar{\varepsilon}}_o$ initial strain rate. (Rigid-Plastic Analysis only)
71-80	8th	F	$\dot{\bar{\varepsilon}}$ cutoff strain rate. (Rigid-Plastic Analysis only)

4th data block

Only necessary for coupled analysis.

1-10	1st	F	Maximum nodal temperature change allowed. Used to control automatic time step scheme for heat transfer. Default value of 20.
11-20	2nd	F	Maximum nodal temperature change allowed before properties are re-evaluated and matrices reassembled. Default value of 100.
21-30	3rd	F	Maximum error in temperature estimate used for property evaluation. This control provides a recycling capability to improve accuracy in highly nonlinear heat-transfer problems (for example, latent heat, radiation boundary conditions). Default is 0, which bypasses this test. Set to maximum temperature error which is considered acceptable.

■ PARAMETERS

Definition of Parameters used in Numerical Analysis

Description

There are many parameters that are used in the finite element calculations. These parameters can be customized for your particular application. Some of these constants can be entered in other input blocks as well. The last nonzero value is used for the calculation.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word PARAMETERS.
2nd data block			
1-10	1st	E	Enter the scale factor which, when multiplied with the incremental strain, is used to predict the incremental strain in the next increment. Default is 1.0.
11-20	2nd	E	Enter the multiplier used to calculate the penalty used to impose boundary conditions. Default is 1.e9. The penalty used is, hence, 1.e9 times the maximum diagonal of the stiffness matrix. If the APPBC parameter is used, this option is not used.
21-30	3rd	E	Enter the penalty factor used to satisfy incompressibility in rigid plastic analysis for plane strain, axisymmetric, or solid analysis when displacement elements are used. Default is 100.
31-40	4th	E	Enter the penalty factor used to satisfy incompressibility in fluid analysis when displacement elements are used. Default is 1.e6.

Format Fixed	Format Free	Data Type	Entry
41-50	5th	E	Beta parameter used in transient dynamic analysis using Newmark-beta procedure Default is 0.25.
51-60	6th	E	Gamma parameter used in transient dynamic analysis using Newmark-beta procedure. Default is 0.50.
3rd data block			
1-10	1st	E	Enter the angle at which a node separates from a convex corner or becomes stuck in a concave corner for two-dimensional contact. Default is 8.625°.
11-20	2nd	E	Enter the angle at which a node separates from a convex corner or becomes stuck in a concave corner for three-dimensional contact. Default is 20.0°.
21-30	3rd	E	Enter the initial strain rate for rigid plastic analysis. Default is 1.e-4.
31-40	4th	E	Enter the cutoff strain rate for rigid plastic analysis. Default is 1.e-12.
41-50	5th	E	Enter the fraction of the mean strain that is subtracted from the stress tensor in the initial stress calculation. See the tenth field of the CONTROL option. Default is 0.0
51-60	6th	E	Enter the factor used to calculate the drilling mode for shell elements type 22, 75, 138, 139, and 140. Default is 0.0001.

■ SOLVER

Specify Direct or Iterative Solver

Description

This option defines the solver to be used in the analysis. You can specify either the direct or iterative solver. The choice of whether the in-core or out-of-core procedure is used is automatically determined by the program, based upon the amount of workspace required and the number given on the SIZING parameter. You can also select whether the symmetric or nonsymmetric solver is used. At this time, only a direct, nonsymmetric solver is available. Additionally, you can specify if the solution of a nonpositive definite system is to be obtained.

When the iterative solver is chosen, additional parameters must be defined which are used to control the accuracy.

Note: It is not recommended to use the iterative solvers for beam or shell models, because these problems are ill-conditioned, resulting in a large-number of iterations. For a well-conditioned system, the number of iterations should be less than (and possible much less than) the square root of the total number of degrees of freedom in the system.

You control the maximum number of iterations allowed. If this is a positive number, the program stops if this is exceeded. If this is a negative number, the program prints a warning and continues to the next Newton-Raphson iteration or increment.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word SOLVER.
2nd data block			
1-5	1st	I	0 = Profile Direct Solver. 1 = EBE Iterative. 2 = Sparse Iterative. 3 = Hardware Provided Direct Profile Solver 4 = Sparse Direct Solver 5 = Laypack 6 = Hardware Provided Direct Sparse Solver
6-10	2nd	I	Enter 1 if the nonsymmetric solver is to be used.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	Enter 1 if the solution of nonpositive definite system is to be obtained.

3rd data block

Only necessary if EBE iterative solver is used.

1-5	1st	I	Enter maximum number of groups; defaults to 30 times the number of element types.
6-10	2nd	I	Enter 1 if the previous solution is to be used as the initial trial solution.
11-15	3rd	I	Enter maximum number of conjugate-gradient iterations. Default is 1000.
21-25	4th	I	Enter the type of preconditioner: Enter 0 for no preconditioner. Enter 1 for Cholesky preconditioner.

4th data block

Only necessary if EBE iterative solver is to be used.

1-10	1st	F	Enter tolerance on conjugate gradient convergence for stress analysis. Default is 0.01.
11-20	2nd	F	Enter tolerance on conjugate gradient convergence for heat transfer analysis.
21-30	3rd	F	Condition number cut-off. If calculated condition number is less than this value, the analysis is stopped.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
Only necessary if the sparse iterative solver is used.			
1-5	1st	I	Enter maximum number of conjugate-gradient iterations. Default is 1000.
6-10	2nd	I	Enter 1 if the previous solution is to be used as the initial trial solution.
11-15	3rd	I	Enter 3 for diagonal preconditioner. Enter 4 for scaled-diagonal preconditioner. Enter 5 for incomplete Cholesky preconditioner.
4th data block			
Only necessary if the sparse iterative solver is to be used.			
1-10	1st	F	Enter tolerance on conjugate gradient convergence for stress analysis. Default is 0.001.

■ POST INCREMENT

Define Increments between Writing on Post File

Description

This option allows you to alter the increments at which data is written to the post file. This option has the same effect as the data in the ninth field of the POST model definition option.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words POST INCREMENT.
2nd data block			
1-5	1st	I	Enter the number of increments between writing of post data. Defaults to write post file very increment. Enter -1 to turn off all writing of post data until the next POST INCREMENT option.

Notes: Post data is automatically written to the post file at the increment in which the POST INCREMENT option occurs.

This value is not saved upon restart; it must be reset through the POST model definition option or POST INCREMENT option.

Example:

```
POST INCREMENT
2
```

writes every other increment to the post file beginning with the current increment.

■ RESTART INCREMENT

Define Increments between Writing on Restart File

Description

This option allows you to alter the increments at which restart data is written to the restart data file. This option has the same effect as the data in the second field of the RESTART model definition option. This does not effect the RESTART LAST option.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words RESTART INCREMENT.
2nd data block			
1-5	1st	I	Enter the number of increments between writing of the restart data. It defaults to writing of restart data every increment. Enter -1 to stop all writing of restart data until the next RESTART INCREMENT option.

Notes: Restart data is automatically written to the restart file at the increments in which the RESTART INCREMENT option occurs.

This value is not saved upon restart; it must be reset through the RESTART model definition or RESTART INCREMENT option.

Example:

```
RESTART INCREMENT
2
```

writes every other increment to the restart file beginning with the current increment.

History Definition: Static, Dynamic, Creep Analysis

This section describes the application of incremental boundary conditions and/or specify automatic multi-increment load control. These boundary conditions include:

- kinematic constraint of either zero or specified displacement;
- surface, volumetric or nodal loads;
- thermal loads, and
- modification of tying constraint.

These boundary changes are incremental in nature with the following exceptions:

1. If the ELASTIC parameter is used, each load case is an independent analysis and the load value are total values.
2. If user subroutine FORCEM is used with the FOLLOW FOR parameter, the distributed loads given in this routine are total values.
3. If the AUTO INCREMENT option is used, the total load is applied.
4. If the AUTO TIME option is used, the total load at the end of the time period is entered.
5. If the AUTO STEP option is used, the total load at the end of the time period is entered.

There are six possibilities for automatic load control:

1. AUTO LOAD allows you to repeat the same incremental load a prescribed number of times.
2. AUTO INCREMENT divides the incremental mechanical load requested into a series of steps to satisfy the user-prescribed tolerances.
3. AUTO THERM divides the incremental thermal load requested into a series of steps to satisfy the user-prescribed tolerances.
4. AUTO TIME allows automatic time-stepping in dynamic analysis or in coupled thermal-stress analysis.
5. AUTO STEP allows automatic time-stepping in dynamic analysis or in coupled thermal-stress analysis with a choice of error criteria.
6. AUTO THERM CREEP divides the requested incremental thermal load into a series of steps and carries out creep analysis between every two steps to satisfy your prescribed tolerances.

The BUCKLE option activates the calculation of the collapse loads and eigenvectors. Note that eigenvalues can be extracted at any increment of the analysis.

The BACKTOSUBS option allows you to recover the displacements, strains, and stresses from a substructure.

The RECOVER option allows the recovery of stresses and reactions for a specified mode during modal analysis.

■ DISP CHANGE

Define Displacement Boundary Conditions

Description

This option allows new displacement boundary conditions to be specified or old displacement boundary conditions to be changed. The exact numbering sequence of the boundary conditions is used in some applications of this option. This numbering sequence is output after the boundary condition option is used in the input data describing the problem.

This option is used for incrementation of fixed displacement components or for adding or removing displacement constraints. Care should be taken when removing fixed displacement conditions to ensure that the reaction forces are handled properly. The residual load correction should be used to reduce reactions to zero after a constraint has been removed (the LOADCOR parameter might be necessary); however, a constraint force might be too large for the piecewise linear analysis. Note that enough space must be specified on the SIZiNG parameter in the maximum number of boundary condition fields to allow for possible increased storage requirements arising from use of this option. Complex displacement velocity acceleration is more conveniently input by user subroutine FORCDT.

This option implies a proportional increment of 1.0. Any resetting of this factor (for example, the PROPORTIONAL INCREMENT option used before the next CONTINUE option), proportions these displacement increments as well. When used in conjunction with harmonic analysis, this boundary change is used for all excitation frequencies until a new boundary change is invoked. In a coupled thermal-stress analysis, use DISP CHANGE for stress and TEMP CHANGE for thermal analysis.

When used in conjunction with harmonic analysis, this boundary change is used for all excitation frequencies until a new boundary change is invoked.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-11	1st	A	Enter the words DISP CHANGE.

Format Fixed	Format Free	Data Type	Entry
2nd data block			
1-5	1st	I	Set to the number of boundary conditions (specified displacement components to be changed or added). A negative number removes boundary conditions from the end of the boundary condition list. A zero activates the FIXED DISP option; a complete set of necessary boundary conditions are then read, using the blocks for that option except for that key word block.
6-10	2nd	I	Enter 1 if excitation boundary conditions for harmonic analysis.
11-15	3rd	I	Enter 1 if harmonic boundary condition is input as magnitude and phase. If blank, real and imaginary values are given.
3rd data block			
Data block 3 is only entered if the number in columns 1 through 5 in data line 2 is positive and then has the number of data lines required by data block 2.			
1-5	1st	I	Number of the boundary condition being changed. This number is derived from the "Fixed Boundary Condition Summary" table in the input echo of a MARC run. Boundary conditions being added should be given labels which increment the total count of boundary conditions properly. **Note:** A boundary condition in the middle of the list can be removed by specifying that labeled boundary condition as a repeat of some other boundary condition.
6-10	2nd	I	Nodal point to be constrained.
11-15	3rd	I	Degree of freedom to be constrained.
16-30	4th	F	Specified displacement increment (real part).
31-45	5th	F	Specified displacement increment (imaginary part).

■ GAP CHANGE

Redefine Data for Gap Elements

Description

This option allows you to modify the data associated with gap elements. This data includes gap closure distance, gap elastic stiffness, contact coefficient of friction, and momentum ratio.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words GAP CHANGE.
2nd data block			
1-5	1st	I	Number of sets of gap data to be input.
6-10	2nd	I	Unit number for reading data. Defaults to input.
The 3rd and 4th data blocks are entered as pairs, once for each set of gap data.			
3rd data block			
1-10	1st	F	For a fixed direction gap, enter the gap closure distance U_{cl}. For a true distance gap, enter the minimum distance d between end points. **Note:** If d > 0, the two end points are never closer than a distance |d| apart. If d<0, the two end points are never farther apart than |d|.
11-20	2nd	F	μ, the contact coefficient of friction.
21-30	3rd	F	K_{GAP}, the elastic stiffness of the closed gap in the contact direction. Default: Gap is rigid when closed.
31-40	4th	F	$K_{FRICTION}$, the elastic stiffness of the closed gap in the friction direction. Default: Gap is rigid when closed.

Format Fixed	Format Free	Data Type	Entry
41-50	5th	F	User supplied momentum ratio for first gap node. Default: The MARC program calculates this ratio internally.
51-60	6th	F	User supplied momentum ratio for fourth gap node. Default: The MARC program calculates this ratio internally.
61-65	7th	I	Enter 0 for fixed direction gap. Enter 1 for true distance gap. Default is 0.
66-70	8th	I	Enter 0 if gap is open during increment 0. Enter 1 if gap is closed during increment 0. Default is 0.
4th data block			
			Enter a list of gap elements to be associated with this set of gap data.

■ TYING CHANGE

Define Tying Constraints

Description

This allows the number of tying constraints to be modified or a totally new series of tying constraints to be introduced. This option modifies the constraints previously entered on the TYING option. If the number of ties is increased, the TIE parameter is also required.

Note: The use of TYING CHANGE can increase the bandwidth beyond that calculated for the original space allocation and, therefore, the program recalculates the nodal bandwidth and the storage allocation for the assembly and solution part of the program.

To completely remove a set of tying constraints, set column 5 to 1 and column 10 to 0.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-12	1st	A	Enter the words TYING CHANGE.
2nd data block			
1-5	1st	I	Set equal to 1 to reduce the number of tying constraints at this point of the analysis. Set equal to 2 to read in an entire new set of tying constraints. If column 5 is set to 1, the new number of tying constraints has to be less than the originally specified number of ties. The tying constraints are deleted from the end of the list to the desired number of remaining ties. The list is in the same sequence as the list of ties in the input file.
6-10	2nd	I	New number of tying constraints. The data lines required by the tying option are read in next if column 5 of this data line is set to 2, except for the key word block.

■ DIST LOADS

Define Distributed Loads

Description

This option allows pressure (surface and volumetric) loads to be specified.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words DIST LOADS.
11-15	2nd	I	Enter 1 if distributed load is to be applied as excitation loads in a harmonic analysis.
2nd data block			
1-5	1st	I	Enter the number of sets to distributed loads to be entered (optional).
6-10	2nd	I	Enter logical unit number for input of distributed load data, defaults to input.

The 3rd and 4th data blocks are entered as pairs, one for each data set.

3a data block

Use if not harmonic analysis.

1-5	1st	I	Parameter identifying the type of load. See library element description in *Volume B: Element Library.*
6-15	2nd	F	Enter the magnitude of this type of distributed load. For load type 102, enter the magnitude of force per unit mass (gravity acceleration) in first coordinate direction.
16-25	3rd	F	For load type 102, enter the magnitude of force per unit mass (gravity acceleration) in second coordinate direction.

Format Fixed	Format Free	Data Type	Entry
26-35	4th	F	For load type 102, enter the magnitude of force per unit mass (gravity acceleration) in third coordinate direction.
36-40	5th	I	Distributed load index. (Distributed load index is to be used in user subroutine FORCEM).
3b data block			
Use if harmonic analysis.			
1-5	1st	I	Parameter identifying the type of load. See library element description in *Volume B: Element Library*.
6-15	2nd	F	Enter the magnitude of this type of distributed load (real component). For load type 102, enter the magnitude of force per unit mass (gravity acceleration) in first coordinate direction.
16-25	3rd	F	Enter the imaginary magnitude of distributed load if harmonic subincrement. For load type 102, enter the magnitude of force per unit mass (gravity acceleration) in first coordinate direction.
26-35	4th	F	For load type 102, enter the real component of force per unit mass (gravity acceleration) in second coordinate direction.
36-45	5th	F	For load type 102, enter the imaginary component of force per unit mass (gravity acceleration) in second coordinate direction.
46-55	6th	F	For load type 102, enter the real component of force per unit mass (gravity acceleration) in third coordinate direction.
56-65	7th	F	For load type 102, enter the imaginary component of force per unit mass (gravity acceleration) in second coordinate direction.
4th data block			
			Enter a list of elements to which the above distributed loads are applied.

■ POINT LOAD

Define Point Loads

Description

This option allows nodal point loads to be specified. If the number of nodes which have point loads has been changed from the model definition block, you must give an upper bound on the DIST LOADS parameter.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words POINT LOAD.
11-15	2nd	I	Enter 1 to enter real harmonic load. Enter 2 to enter imaginary harmonic load.
2nd data block			
1-5	1st	I	Enter number of sets of point loads to be entered (optional).
6-10	2nd	I	Enter logical unit number for input of point load data; defaults to input.

The 3rd and 4th data blocks are entered as pairs, once for each data set.

3rd data block			
1-10	1st	F	Nodal load associated with first degree of freedom.
11-20	2nd	F	Nodal load associated with second degree of freedom.
21-30	3rd	F	Nodal load associated with third degree of freedom.
31-40	4th	F	Nodal load associated with fourth degree of freedom.
41-50	5th	F	Nodal load associated with fifth degree of freedom.
51-60	6th	F	Nodal load associated with sixth degree of freedom.
61-70	7th	F	Nodal load associated with seventh degree of freedom.
71-80	8th	F	Nodal load associated with eighth degree of freedom.

Note: Continuation data line is necessary and must be in 6E10.3 format. Continuation data lines are needed if there are more than eight degrees of freedom per node in the analysis.

Format Fixed	Format Free	Data Type	Entry
4th data block			
			Enter a list of nodes to which the above point load is applied.

■ AUTO LOAD

Define Equal Load Increments

Description

This option is useful for nonlinear analysis with proportional loads. It generates a specified number of increments; all having the same load increment. This load increment is the net result of the changes and scalings made to the load increment in all the previous increments, plus the effect of any tractions, proportional increment options, etc., in the current increment. These options should come after the AUTO LOAD option. If the proportional increment option is not included, AUTO LOAD sets the proportionality factor to a default of 1.

AUTO LOAD controls only mechanical loads and kinematic boundary conditions – it does not control thermal loads. If a restart is made from one of the increments generated by an AUTO LOAD, the rest of the increments associated with this AUTO LOAD are automatically completed before reading of new input. This can be avoided by using the REAUTO option. The completion of an AUTO LOAD by a restart is done using the control parameters specified by the program used with the RESTART option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-9	1st	A	Enter the words AUTO LOAD.
2nd data block			
1-5	1st	I	Number of times this load increment is to be applied.
6-10	2nd	I	Reassembly interval for stiffness matrices. Defaults to whenever nonlinearity occurs.

■ PROPORTIONAL INCREMENT

Define Proportional Increments

Description

Using this option, the current load increment can be scaled up or down for use in the next load increment. This is most frequently used in elastic-plastic analysis where the first load increment is scaled up to the values that cause first yield. This option governs mechanical loads only; temperature changes are independent of this proportioning.

The option can precede or follow all the other options in this optional series. If it precedes a DIST LOADS, POINT LOAD, or DISP CHANGE option, these options reset the proportionality factor to 1.0. If it follows either of these options, it also scales any nonzero load or displacement increments given in these options.

It is permissible to follow this by a DIST LOADS, POINT LOAD, or CHANGE STATE option to change some or all of the load and temperature vectors.

Note: If the SCALE parameter is used, the load increment that is applied in the first increment is the scaled load multiplied by the value given in the second field.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-22	1st	A	Enter the words PROPORTIONAL INCREMENT.
2nd data block			
1-5	1st	I	Minimum number of cycles for each step of one increment; normally 1 (forces recycling n times). Every recycle might cause an assembly and triangularization of the stiffness matrix. The program automatically recycles if convergence to tolerance is not achieved. The default value of recycles is 0. If this value is set above 1, more cycles are allowed but each increment is forced to cycle at least n times before solution. Use caution that no unnecessary recycling is being forced (for example, in AUTO LOAD). Recycling is usually forced for the first few critical load steps to get convergence and then resume normal condition.

Format		Data	
Fixed	Free	Type	Entry
6-20	2nd	F	Ratio of the next increment of load to the present increment. Only mechanical loads and kinematic boundary conditions are scaled.

■ AUTO INCREMENT

Define Automatic Load Stepping

Description

This option allows automatic load stepping in a quasi-static analysis and is very useful for both geometric (LARGE DISP) and material (elastic-plastic) nonlinear problems. The option is capable of handling elastic/plastic snap-through phenomena; hence, the post-buckling behavior of structures can be analyzed.

You have to specify in the DIST LOADS, POINT LOAD, and/or DISP CHANGE options the total loading for a sequence of load steps, and the program automatically generates the magnitude of each load step based on an initial load step and the amount of nonlinearity occurring during the loading.

The length of the incremental displacement vector ($C = \Delta u^T \Delta u$) is based on the number of recycles in the previous increment. The size of the load increment is controlled by the length of the incremental displacement vector. The analysis is stopped when the total load is reached or when the maximum allowed number of increments is reached. In case of a snap-through problem, the loading can initially increase, decrease after the buckle load has been reached, and increase if the stiffness increases in the post-buckled state. Within the history definition data, the AUTO INCREMENT option can be used as often as desired. For more details, see *Volume A: Theory and User Information.*

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-14	1st	A	Enter the words AUTO INCREMENT.
2nd data block			
1-10	1st	F	Fraction (α) of the total load increment that should be applied in the first cycle of the first increment of this AUTO INCREMENT session resulting in $\Delta u_1 = K^{-1}\alpha P$. $C_{in} = \Delta u_1^T \Delta u_1$.
11-15	2nd	I	Maximum number of increments during this AUTO INCREMENT session.

Format Fixed	Format Free	Data Type	Entry
16-20	3rd	I	Desired number of recycles per increment. Used to increase or decrease load steps during AUTO INCREMENT session. Default is 3. Please allow for more recycles via CONTROL model definition option.
21-30	4th	F	Maximum fraction of the total load that can be applied in any increment of this AUTO INCREMENT session. Default is 1 if no contact is present.
31-40	5th	F	Maximum multiplier of applied arc length in norm of displacement vector to initial arc length. C_{max}/C_{in} Defaults to maximum fraction of load divided by initial fraction of load.
41-50	6th	F	Total time period to be covered; to be used in conjunction with contact analyses. Default is 1.0.
51-60	7th	F	Fraction of the initial arclength to define a minimal arclength. C_{min}/C_{in} Default is 0.01.
61-65	8th	I	Arclength root procedure: 1 = Chrisfield (quadratic constraint) (default) 2 = Riks/Ramm (linear constraint) 3 = Modified Riks/Ramm (linear constraint) 4 = Chrisfield; switch to Modified Riks/Ramm if no real root found

Notes: Upon restart, before reading history definition data, this AUTO INCREMENT session is finished. The maximum number of increments allowed, the desired number of recycles, and the maximum step size for this session can be changed upon restart using the REAUTO model definition option.

The option cannot be used for thermal loading; use the AUTO THERM option instead.

If this option is used for post-buckling analysis, the nonpositive definite flag in the SOLVER model definition option has to be used. This option can be added upon restart.

■ AUTO STEP

Adaptive Load Step Control

Description

This option allows control of the automatic time/load stepping procedure. In this procedure, the time step is adjusted based upon the calculated value of a parameter (strain increment, plastic strain increment, creep strain increment, stress increment, strain rate, strain energy increment, temperature increment, displacement increment, rotation) versus a user-defined maximum. More than one criteria can be specified. If the criteria is not satisfied within an increment, recycling occurs with a reduced time/load applied. After the increment has converged based upon tolerances specified on the CONTROL values, the data given here controls the next increment.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words AUTO STEP.
2nd data block			
1-10	1st	E	Enter the initial time step.
11-20	2nd	E	Enter the total time period.
21-30	3rd	E	Enter the smallest ratio between steps. Default is 0.1.
31-40	4th	E	Enter the largest ratio between steps. Default is 10.0.
41-50	5th	E	Enter the minimum time step. Defaults to total time/maximum number of steps.
51-60	6th	E	Enter the maximum time step. Defaults to total time.
61-65	7th	I	Enter the maximum steps allowed.
66-70	8th	I	Enter the desired number of recycles per increment.

Format Fixed	Format Free	Data Type	Entry
Repeat 3rd and 4th data blocks in pairs for each criteria.			
3rd data block			
1-5	1st	I	Enter the criteria id: Enter 1 for strain increment. Enter 2 for plastic strain increment. Enter 3 for creep strain increment. Enter 4 for normalized creep strain increment. Enter 5 for stress increment. Enter 7 for strain energy increment. Enter 8 for temperature increment. Enter 9 for displacement increment. Enter 10 for rotation increment.
6-80	2nd	I	Enter set name of elements/nodes to which this criteria is to be applied.
4th data block			
1-10	1st	E	ΔYI.
11-20	2nd	E	XMAX1.
21-30	3rd	E	ΔY2.
31-40	4th	E	XMAX2.
41-50	5th	E	ΔY3.
51-60	6th	E	XMAX3.
61-70	7th	E	ΔY4.
71-80	8th	E	XMAX4.

For criteria 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, the time step is adjusted based upon:

if $X \leq XMAX1$ ΔY calculated/$\Delta Y1$

if $XMAX1 < X < XMAX2$ ΔYcalculated/$\Delta Y2$

if $XMAX2 < X < XMAX3$ ΔYcalculated/$\Delta Y3$

if $XMAX3 < X$ ΔYcalculated/$\Delta Y4$

where

Criteria	X	ΔY
1	strain	strain increment
2	plastic strain	plastic strain increment
3	creep strain	creep strain increment
4	creep strain	creep strain increment/elastic strain
5	stress	stress increment
7	strain energy	strain energy increment
8	temperature	temperature increment
9	displacement	displacement increment
10	rotation	rotation increment

■ THERMAL LOADS

Define Thermal Loads

Description

This option allows input of temperature and other state variables (see STATE VARS parameter). Used here, the loads are incremental; in that, they are in addition to any loads previously applied. The loads are total loads only if the ELASTIC parameter is used.

You can specify either a uniform or nonuniform change in temperature (or other state variables). If a nonuniform change is desired, the change of every state variable at every layer of every integration point of every element must be specified. In this case, the program calls user subroutine CREDE for every element in the mesh. (See model definition option THERMAL LOADS for more information.)

If the Fourier decomposition method is being used to analyze an arbitrarily loaded axisymmetric structure, the THERMAL LOADS option must be invoked separately for each Fourier series term that has temperatures (state variables) associated with it. If there is no variation of these variables in the circumferential direction, only the zeroth term of the series should be specified.

Note: On a restart run, any THERMAL LOADS option before the END OPTION reads data.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-13	1st	A	Enter the words THERMAL LOADS.
2nd data block			
1-5	1st	I	Set to 1 if uniform increment temperature (state variable) increment is applied to all elements. Set to 2 if nonuniform incremental total temperature (state variable) is read via user subroutine CREDE. Set to 3 if nonuniform total temperature (state variable) is read via user subroutine CREDE.

Format		Data	
Fixed	Free	Type	Entry
3rd data block			
1-80	1st	F	Include only if the first field of data line 2 is 1; enter the uniform increments in temperature and any additional state variables in (8E10.3) format is applied to all elements.
4th data block			
1-80	1st	F	Include only if the first field of data line 2 is 2 or 3, and using the default subroutine CREDE. Temperature and state variable data to be read in by CREDE. All lines should contain 8 values in (8E10.3) format; do not start a new data line for each element.

■ AUTO THERM

Specify Data for Automatic Thermal Loading

Description

This option is intended to allow automatic, static, elastic-plastic, thermally loaded stress analysis based on a set of temperatures defined throughout the mesh as a function of time. The temperatures are presented to the program through the CHANGE STATE option using any of the input possibilities of that data block, and the program then creates its own set of temperature steps based on a temperature change tolerance provided on this option. You should recall that the elastic-plastic stress analysis is time independent, but that the strain increments should be small to obtain accuracy in the integration of the rate equations of plasticity. As a guideline, the maximum thermal strain should be restricted to 20%–50% of the strain to cause yield, depending on how much free thermal expansion is possible. Based on this argument, a temperature change tolerance of 20%–50% of $\frac{\bar{\sigma}}{E\alpha}$, where $\bar{\sigma}$ is the yield stress, E is Young's modulus and α is the coefficient of thermal expansion, should be set. Given your temperature change tolerance on this option, the program proceeds through your definition of the history of the temperatures provided on the CHANGE STATE set, and linearly subdivides or merges together the user-defined steps so as to conform to the tolerance.

The automatic thermally loaded analysis continues until all steps indicated on the CHANGE STATE option are completed, so that a typical automatic thermally loaded analysis would have as input:

```
AUTO THERM
30., 0, 0, 4.0,
CHANGE STATE
1, 3, 0, 19, 1, 15, 1,
```

In the above case, a temperature change tolerance of 30 is set for the creation of temperature steps by the program; the total transient time in thermal analysis is 4.0. The data in the CHANGE STATE option indicates that the temperatures are stored in a formatted post file (unit 19) and there are 15 sets of temperatures on the file.

If no DIST LOADS, POINT LOAD, or PROPORTIONAL INCREMENT options appear with the AUTO THERM set, all mechanical loads and kinematic boundary conditions are held constant during the AUTO THERM. However, DIST LOADS, POINT LOAD, PROPORTIONAL INCREMENT, or DISP CHANGE can be included in the set – the mechanical loads and kinematic boundary conditions,

which are then defined, are assumed to change in proportion to the time scale of the temperature history defined by the CHANGE STATE option and are applied accordingly, on the basis that the increments of load and displacement correspond to the end of the transient time (TOTIM) of the AUTO THERM input.

Notes: All load options must be specified before AUTO THERM.

You must include the CHANGE STATE option in conjunction with this option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words AUTO THERM.
2nd data block			
1-10	1st	F	Enter the maximum temperature change to be used per step of stress analysis. The program linearly subdivides steps or merges steps together to create increments which are close to, but do not exceed, this tolerance.
11-15	2nd	I	Enter the maximum number of increments to be allowed in this AUTO THERM. If this number of increments is exceeded before the temperature history is completed, the program ends. This is intended as a protection to avoid excessive increments in the case of a data error. Default value is 50 increments if set to 0.
16-20	3rd	I	Reassembly interval for element matrices.

Format		Data	
Fixed	Free	Type	Entry
21-30	4th	F	Total transient time, TOTIM. This is used to proportionally scale the incremental boundary conditions. If TOTIM is equal to zero, mechanical loads given with this group are applied for each increment in this group. If TOTIM is unequal to zero, then the mechanical loads specified in this group are linearly scaled. If the temperatures are obtained from a previous heat transfer analysis/post file, enter the total time period of the heat transfer analysis.
31-40	5th	F	Maximum time step allowed per step of stress analysis. The program linearly subdivides steps or merges steps to create increments which are close to, but do not exceed, this tolerance. Both the maximum temperature change allowed and the maximum time step allowed tolerances must be satisfied.

■ CHANGE STATE

Change State Variables

Description

This option provides various ways of changing the state variables throughout the model. State variables are initialized in the INITIAL STATE model definition set. The number of state variables per point is defined in the STATE VARS parameter. The default is one with temperature always being the first state variable at a point. If more than one state variable per point has been assigned, this option can be used repeatedly to change the values of all state variables. The default value is no change if this option is not used. In this option, the values of the state variable at the end of the current increment are read in. When the temperature is being defined, the following points should be noted:

- For "history following analysis", the thermal strains are based on temperature change during this step.
- For elastic re-analysis (ELASTIC parameter), the thermal strains are always based on temperature change between the initial, stress free temperature field and the values read in here.
- The AUTO THERM option is available for automatic control of a nonlinear (elastic-plastic) temperature loaded stress problem, to be used in conjunction with this option.
- The THERMAL LOADS option can be used as an alternate to input the change of temperature.
- The AUTO THERM CREEP option is available for automatic control of a thermally loaded elastic-plastic-creep problem and is to be used in conjunction with this option.

With this option, state variables can be changed using any of the methods listed below. The state variables input are total state variables.

- Read a range of elements, integration points, and layers, and a corresponding state variable value for the end of the current step.
- Read the state variable values for the end of the current step through user subroutine NEWSV.
- Read the state variable values for the end of the current step from a named step of the post file output from a previous heat transfer analysis with MARC. With this option, the program assumes direct correspondence of the post file elements with the elements in the current analysis. Any spatial interpolation must be provided separately by you.
- Read a list of elements, integration points, and layers, and a corresponding state variable value.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-12	1st	A	Enter the words CHANGE STATE.
2nd data block			
1-5	1st	I	Enter the state variable identifier for the state variable being changed (1,2,3,etc.). 1 = temperature. If more than one state variable is being used, the STATE VARS parameter must be included.
6-10	2nd	I	Enter 1 to change the state variable via the 3rd and 4th data blocks below. In this case, the third field must also be defined, and the sixth field if the AUTO THERM option is in use. Enter 2 to change the state variable via user subroutine NEWSV. This subroutine is now called in a loop on all the elements in the mesh. Enter 3 to read the new values of the state variable from a post file written by a previous heat transfer analysis. In this case, the fourth and fifth fields must be defined, and the sixth field if the AUTO THERM option is in use. Enter 4 to change the state variable via the 5th, 6th, 7th, and 8th data blocks below.
11-15	3rd	I	Only nonzero if the second field is 0 set to 1 or 4. In that case, this entry gives the number of blocks set in data blocks 3 and 4 used to input the new value of the state variable (optional).
16-20	4th	I	Only nonzero if the second field is set to 3. This entry defines the unit number from which the post file information from the previous heat transfer run is read. Defaults to unit 24 for a formatted post file and unit 25 if a binary post file.
21-25	5th	I	Only nonzero if column 10 set to 3. In that case, this entry defines the step number on the heat transfer run post file to be read as the definition of the new value of the state variable at the end of the current step.

Format Fixed	Format Free	Data Type	Entry
26-30	6th	I	Only used if the AUTO THERM option is in use. Give the number of sets of input to be read to define the temperature history.
31-35	7th	I	Enter 1 if formatted post file is used. ***For Cray Only:*** Set to 2 for IEEE binary file Set to 3 for VAX binary file Set to 4 for IBM binary file
36-40	8th	I	Only nonzero if the second field is set to 2. Set to 1 to suppress printout of state variable values that are defined in the user subroutine NEWSV.

3rd data block

Data blocks 3 and 4 are only input if the second field is above set to 1. In that case, the number of sets of blocks is equal to the number given in the third field above.

1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with this value.
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value (11-15 and 16-20 can only be bigger than 1 if ALL POINTS parameter is used).
21-25	5th	I	First layer or cross section point with this value.
26-30	6th	I	Last layer or cross section point with this value (21-25 and 26-30 can only be bigger than 1 for beam or shell elements).

4th data block

1-10	1st	F	New value of this state variable for the above range of points at the end of the current step.

Format Fixed	Format Free	Data Type	Entry

Data blocks 5, 6, 7, and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block

1-10	1st	F	New total value of this state variable for the points given below at the end of the current step.

6th data block

Enter a list of elements to which the above state variable is applied.

7th data block

This data block is not necessary if the CENTROID option is used.

Enter a list of integration points to which the above state variable is applied.

8th data block

This data block is only necessary if there are either beams or shells in the mesh.

Enter a list of layers to which the above state variable is applied.

■ POINT TEMP

Define Point Temperatures

Description

This option defines temperatures at nodal points for an uncoupled thermal stress problems at the end of the increment.

Note: For shell analyses, a uniform temperature is used through the thickness direction.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words POINT TEMP.
2nd data block			
1-5	1st	I	Enter the number of sets of prescribed temperatures (optional).
6-10	2nd	I	Enter file number for input of prescribed temperatures data, defaults to input.
11-15	3rd	I	Flag to indicate that temperatures are read from previously generated post file. Set to 1.
16-20	4th	I	Only nonzero if the third field is set to 1. This entry defines the unit number from which the post file information is read.
21-25	5th	I	Enter step number to be read.
26-30	6th	I	Enter 1 if a formatted post tape is used. ***For Cray Only:*** Set to 2 for IEEE binary file Set to 3 for VAX binary file Set to 4 for IBM binary file

Format Fixed	Format Free	Data Type	Entry
Data blocks 3 and 4 are given in NSET pairs, only if the flag in the third field is not equal to 1.			
3rd data block			
1-10	1st	E	Temperatures at the end of the increment.
4th data block			
			Enter list of nodes for which the above initial temperature is applied.

■ CHANGE PORE

Define Pore Pressures for Uncoupled Soil Analysis

Description

This option provides various ways of changing the pore pressure throughout the model. This option is only used in uncoupled soil analysis.

Given below are four ways of providing the pore pressures.

1. Read a range of elements, integration points and layers, and corresponding pore pressures for the end of the current step.
2. Read the pore pressure values for the end of the current step through user subroutine NEWPO.
3. Read the pore pressure values for the end of the current step from a named step of the post file output from a previous pore pressure analysis with MARC. With this option, the program assumes direct correspondence of the post file elements with the elements in the current analysis. Any spatial interpolation must be provided separately by you.
4. Read a list of elements, integration points and layers, and corresponding pore pressure.

   Note On this option, total pore pressures are input.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-13	1st	A	Enter the words CHANGE PORE.
2nd data block			
1-5	1st	I	Enter 1.
6-10	2nd	I	Enter 1 to change the pore pressure via data block 3 below. In this case, the third field must also be defined. Enter 2 to change the pore pressure via user subroutine NEWPO. This subroutine is then called in a loop on all the elements in the mesh.

Format Fixed	Format Free	Data Type	Entry
			Enter 3 to read the initial values of the pore pressure from the post file written by a previous pore pressure analysis. In this case, the fourth and fifth fields must also be defined. Enter 4 to change the pore pressure via data blocks 5, 6, 7 and 8 below.
11-15	3rd	I	Only nonzero if the second field is set to 1 or 4. In that case, this entry gives the number of blocks set in data blocks 3 and 4 used to input the new value of the pore pressure (optional).
16-20	4th	I	Only nonzero if the second field is set to 3. Then this entry defines the unit number from which the post file information from the previous pore pressure run is read.
21-25	5th	I	Only nonzero if the second field is set to 3. In that case this entry defines the step number on the pore pressure run post file to be read as the definition of the new value of the pore pressure at the end of the current step.
26-30	6th	I	Not used; enter 1.
31-35	7th	I	Enter 1 if a formatted post file is used. ***For Cray Only:*** Set to 2 for IEEE binary file Set to 3 for VAX binary file Set to 4 for IBM binary file
36-40	8th	I	Only nonzero if the second field set to 2. Set to 1 to suppress printout of pore pressure values that are defined in user subroutine NEWPO.

Data blocks 3 and 4 are only input if the second field above is set to 1. In that case, the number of data sets is equal to the number given in the third field above.

3rd data block

1-5	1st	I	First element with this value.
6-10	2nd	I	Last element with this value.

Format Fixed	Format Free	Data Type	Entry
11-15	3rd	I	First integration point with this value.
16-20	4th	I	Last integration point with this value can only be bigger than 1 if ALL POINTS parameter is used.
21-25	5th	I	First layer or cross-section point with this value.
26-30	6th	I	Last layer or cross-section point with this value.
4th data block			
1-10	1st	F	New value of the pore pressure for the above range of points at the end of the current step.

Data blocks 5, 6, 7, and 8 are only input if the second field above is set to 4. In that case, the number of sets is equal to the number given in the third field above.

5th data block

1-10	1st	F	Pore pressure for the points given below at the end of the current increment.

6th data block

Enter a list of elements to which the above pore pressure is to be applied.

7th data block

This data block is not necessary if the CENTROID option is used.

Enter a list of integration points to which the above pore pressure is to be applied.

8th data block

This data block is only necessary if there are either beams or shells in the mesh.

Enter a list of layer points to which the above pore pressure is to be applied.

■ TIME STEP

Define Time Step

Description

This option allows you to enter a time step for static analysis.This option can be used to prescribe the time step in a contact analysis. This time step can be used in conjunction with strain rate effects (CREEP, VISCO ELAS, VISCO PLAS) option. This time step is used for this step or series of steps if AUTO LOAD is used. This time step is not scaled by the proportional increment option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words TIME STEP.
2nd data block			
1-10	1st	F	Enter time step.

■ BUCKLE

Specify Buckling Analysis

Description

This sets a flag for the buckling analysis and solves the eigenvalue problem by the inverse power sweep method or the Lanczos method. The number of modes and the procedure used is specified on the BUCKLE parameter. This option can be exercised after every increment of load. The LARGE DISP parameter should be included for nonlinear collapse analysis. This option can also be used to control perturbation analyses. The perturbation is added to the coordinates in the increment following the eigenvalue extraction.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-6	1st	A	Enter the word BUCKLE.

Option A

2nd data block

1-5	1st	I	Maximum number of iterations allowed. Not used for Lanczos. Enter 0. Default is 40.
6-15	2nd	F	Convergence tolerance. The power sweep terminates when the difference between the eigenvalues in two consecutive sweeps divided by the eigenvalue is less than the tolerance. Not used for Lanczos. Enter 0. Default is 0.0001.
16-20	3rd	I	Enter the harmonic number for Fourier buckling; positive number results in cosine terms, negative number results in sine terms. Default is zero.
21-25	4th	I	Enter 1 for buckling perturbation to occur in the next increment.

Format		Data	
Fixed	Free	Type	Entry
26-30	5th	I	Enter mode number to be used in the perturbation analysis.
31-40	6th	F	Enter the scale factor to be multiplied with the normalized mode and added to the coordinates in the next increment.

■ BACKTOSUBS

Recover Substructure Output

Description

This option allows you to perform a displacement and stress calculation for a given substructure at the current main level increment. It can be followed by output control option, such as PRINT ELEM, PRINT CHOICE, etc. If no other options are given, the program uses the options given in the model definition data of the substructure.

The actual calculation is done after a CONTINUE option has been encountered. To continue the analysis of the main level structure the fourth field should be set to 1.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word BACKTOSUBS.
2nd data block			
1-5	1st	I	Level of substructure.
6-10	2nd	I	Substructure number in the above level.
11-15	3rd	I	Set to 1 to skip the stress analysis of the substructure.
16-20	4th	I	File unit for the auxiliary sequential access substructure data base. (Defaults to value given on SUPER parameter.)
21-25	5th	I	File unit for the direct access substructure data base. (Defaults to value given on SUPER parameter.)

■ ACTIVATE

Activate Elements

Description

This option allows you to activate elements which were deactivated either before the start of the analysis or during the analysis. Elements, which were deactivated before analysis, have zero internal stress upon activation. Elements, which were used earlier and deactivated during analysis, have an internal stress which is equal to the state when they were deactivated. Elements can be activated and deactivated as often as needed. Note that activation of elements results in an increase in the size of the stiffness matrix.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word ACTIVATE.
2nd data block			
1-80	1st	I	Enter the list of elements that are to be activated at this time.

■ DEACTIVATE

Deactivate Elements

Description

This option allows you to deactivate elements during the course of an analysis and, as such, can be useful to model ablation or excavation. After the elements are deactivated, they retain the stress state in effect at the time of deactivation and this state can be postprocessed or printed at any time. At a later stage in the analysis, the elements can again be activated with the ACTIVATE option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word DEACTIVATE.
2nd data block			
1-80	1st	I	Enter the list of elements to be deactivated at this time.

■ FOUNDATION

Define Foundation Spring Force for Elements

Description

This block allows the specification of elements and associated foundation spring force to be used with the elastic foundation option (*Volume A: Theory and User Information*). Nonlinear foundations are available via user subroutine USPRNG (see *Volume D: User Subroutines and Special Routines*).

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word FOUNDATION.
2nd data block			
1-5	1st	I	Number of sets of blocks to be used to input the lists of element and foundation stiffnesses (optional).
6-10	2nd	I	Enter unit number for input of foundation data; default to input.
3rd data block			
Data blocks 3 and 4 are entered as pairs, once for each data set.			
1-5	1st	I	Parameter identifying the type of elastic foundation is the same parameter as used in the DIST LOADS option. See *Volume B: Element Library* for a description of the possible distributed load types for each element type in MARC.
6-15	2nd	F	Spring stiffness per unit surface area (or per unit length for beam elements).
4th data block			
			Enter a list of elements to which the above foundation is applied.

■ CONTACT TABLE

Define Contact Table

Description

This option is useful for deactivating or activating bodies when the CONTACT option is used. To avoid unnecessary detection of contact between bodies, you can control the detection of contact. The default for contact analysis is that every body detects the possibility of contact relative to all other bodies and itself if it is a flexible body. When the CONTACT TABLE option is entered, the default of detection for every body is overridden. Instead, you specify the relationship of detection between bodies for contact. The touching body does not contact itself unless you request it. Whenever the touched body is a flexible one, the capability of double-sided contact is applied between the contacting bodies even if you do not explicitly request it. A positive value of the interference closure implies that there is an overlap between the bodies; a negative value implies that a gap exists.

The following control variables of contact between bodies can be modified throughout the table: contact tolerance, separation force, friction coefficient, interference closure, and contact heat transfer coefficient (for coupled thermal-stress-contact analysis). In addition, you can invoke the glue option. The previous values of those control variables are not overridden unless nonzero values are entered here.

In the glue option, when an node contacts a rigid body, the relative tangential displacement is zero. When a node contacts a deformable body, all of the degrees of freedom are tied.

Note: This option should be placed after the CONTACT option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word CONTACT TABLE.
2nd data block			
1-5	1st	I	Enter the number of sets of bodies to be input.

Format Fixed	Format Free	Data Type	Entry
Data blocks 3 and 4 are entered once for each set of bodies to be input.			
3rd data block			
1-5	1st	I	Enter the touching body number.
6-15	2nd	F	Enter the contact tolerance (ERROR).
16-25	3rd	F	Enter the contact separation force (FNTOL).
26-35	4th	F	Enter the friction coefficient.
36-45	5th	F	Enter the interference closure amount, normal to the contact surface.
46-55	6th	F	Enter the contact heat transfer coefficient (coupled analysis only).
56-60	7th	I	Enter 1 to activate the glue option.
4th data block			
			Enter a list of bodies for which the above body detects contact with the specified parameters.

■ CONTACT NODE

Define Nodes for Surface Contact

Description

This option is used to define which nodes in a body might potentially contact other surfaces. This option can be used to reduce the computational cost if a body has many exterior nodes; yet, it is known for which nodes contact might occur. If this option is not used, all exterior surface nodes are checked for contact.

Note: If this option is used and a node number is not explicitly listed, that node might penetrate other bodies.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-12	1st	A	Enter the words CONTACT NODE.
2nd data block			
1-5	1st	I	Enter the number of bodies for which exterior nodes are defined.
3rd data block			
1-5	1st	I	Body number
4th data block			
			Enter a list of nodes that are potential contact nodes

■ MOTION CHANGE

Define Motion of Rigid Surfaces

Description

This option is useful for prescribing the motion of rigid bodies when the CONTACT option is used. This option can be used to call user subroutine MOTION to change the rigid body speed, position, or coefficient of friction.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word MOTION CHANGE.
2nd data block			
1-5	1st	I	Enter the number of sets of rigid bodies to be input.

Data blocks 3 and 4 are entered as pairs, once for each data set.

2D contact problems

3rd data block			
1-5	1st	I	Enter the rigid body number.
4th data block (If velocity controlled rigid surface.)			
1-10	1st	F	First component of velocity of center of rotation (COR).
11-20	2nd	F	Second component of velocity of COR.
21-30	3rd	F	Angular velocity (in radian/unit time) about COR.
31-40	4th	F	Friction coefficient.

Format Fixed	Format Free	Data Type	Entry
4th data block (If position controlled rigid surface.)			
1-10	1st	F	First component of postiion of COR.
11-20	2nd	F	Second component of position of COR.
21-30	3rd	F	Angular position (in radian/unit time) about COR.
31-40	4th	F	Friction coefficient.

3D contact problems

3rd data block

1-5	1st	I	Enter the rigid body number.
4th data block (If velocity controlled rigid surface.)			
1-10	1st	F	First component of velocity of COR.
11-20	2nd	F	Second component of velocity of COR.
21-30	3rd	F	Third component of velocity of COR.
31-40	4th	F	Angular velocity (in radian/unit time) about local axis through COR.
41-50	5th	F	Friction coefficient.
4th data block (If position controlled rigid surface.)			
1-10	1st	F	First component of position of COR.
11-20	2nd	F	Second component of position of COR.
21-30	3rd	F	Third component of position of COR.
31-40	4th	F	Angular position (in radian/unit time) about local axis through COR.
41-50	5th	F	Friction coefficient.

■ RELEASE

Define Release Data

Description

This option is useful for the analysis of spring-back after bodies contact one another. The body number is entered and then all of the nodes which contact that body are released at the beginning of the increment. The contact force can either be immediately removed or gradually reduced. In addition, the body must either be moving away to avoid nodes recontacting during the same increment or the CONTACT TABLE option should be used to avoid recontracting.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word RELEASE.
11-15	2nd	I	Enter 0 if the contact forces are to be immediately removed (default).
			Enter 1 if the contact forces are to be reduced to zero over the number of increments specified in this load period.
2nd data block			
			Enter a list of bodies for which nodes currently in contact are released.

■ APPROACH

Move Rigid Surfaces into Position

Description

The APPROACH option allows you to move rigid bodies so that they just make contact with deformable bodies. In the case of multistage forging, you usually have a time period where the first set of bodies are released, followed by a new time period where the second set of bodies are brought into contact. This option is used in conjunction with the CONTACT TABLE option to determine which bodies are now applicable, and the MOTION CHANGE option which prescribes the velocity of the new bodies. A time step is associated with this period, but it is only used for output purposes and to coordinate with tables and user subroutines except in coupled analysis where a transient period is performed. MARC moves each rigid body until it comes into contact with a deformable body.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-8	1st	A	Enter the word APPROACH.

■ SYNCHRONIZED

Move Rigid Surfaces into Position

Description

The SYNCHRONIZED option allows you to move rigid bodies so that they just make contact with deformable bodies. (In the case of multistage forging, you usually have a time period where the first set of dies are released, followed by a new time period where the second set of dies are brought into contact. This option is used in conjunction with the CONTACT TABLE option to determine which dies are now applicable, and the MOTION CHANGE option which prescribes the velocity of the new dies. MARC moves each rigid surface until one of the moving rigid surfaces makes contact with a deformable body.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-8	1st	A	Enter the word SYNCHRONIZE.

■ SPLINE

Analytical Surface used to Represent a Deformable Body

Description

In order to improve the accuracy for a deformable-deformable contact analysis, the outer surface of a contacted body can be described based on a spline (2D) or Coons surface (3D) description. The analytical surface is then used to calculate the normal to the deformable body.

In 2D, for a contacted segment, a spline is created based on:

tangent at first and second point of segment

position of first and second point of segment

In 3D, for a contacted segment, a Coons surface is created based on:

tangent vectors at corner points of segment

position of corner points of segment

zero twist vectors

This option should be included in the history definition section if a rezoning step is performed.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-6	1st	A	Enter the word SPLINE.
2nd data block			
1-5	1st	I	Enter the number of deformable bodies for which the spline description must be applied.

Format Fixed	Format Free	Data Type	Entry
The 3rd and 4th data blocks are repeated for each deformable body with a spline description.			
3rd data block			
1-5	1st	I	Body number.
4th data block			
			Enter a list of nodes defining nodes/edges to be excluded from the spline description. **Note:** Typically, this list contains nodes at which the direction of the tangent vector at the outer contour of the structure shows a discontinuity. In 3D, when an element edge should be excluded, the nodes defining the edge must be entered one after another.

■ EXCLUDE

Ignore Contact with Certain Regions

Description

For certain contact problems, you might wish to influence the decision regarding the deformable segment a node contacts. By means of the EXCLUDE option, you can specify a list of nodes defining segments to be excluded from the contacted bodies.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-6	1st	A	Enter the word EXCLUDE.
2 data block			
1-5	1st	I	Enter the number of deformable bodies for which the EXCLUDE option must be applied.

The 3rd and 4th data blocks are repeated for each deformable body with the EXCLUDE option.

3rd data block			
1-5	1st	I	Body number.
4th data block			
			Enter a list of nodes defining segments to be excluded from contacted bodies. **Note:** In 2D, each segment must be defined by two nodes. In 3D, each segment must be defined by four nodes.

■ RELEASE NODE

Define Nodes for Which the Boundary Condition is Gradually Released

Description

This option removes a boundary condition constraint from a node in a gradual manner. This option is similar to changing boundary conditions, but it allows the reaction force to be brought to a zero value over a series of increments. The load is reduced in equal steps if used in conjunction with the AUTO LOAD or DYNAMIC CHANGE option. The load is proportionally reduced to zero if used with the AUTO STEP, AUTO TIME or AUTO INCREMENT option. If the RELEASE NODE and DISP CHANGE are given in the same load incrementation section, the DISP CHANGE option should be given first.

Note: This option should not be applied to nodes in contact with rigid surfaces.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word RELEASE NODE.
2nd data block			
1-5	1st	I	Enter the number of sets of data; this must be given.
6-10	2nd	I	Enter unit number for input of release data, defaults to input.
3rd data block			
			Enter a list of degrees of freedom to be released.
4th data block			
			Enter a list of nodes to be released.

■ ACTUATOR

Define the Length of the Actuator Link

Description

This option can be used in conjunction with the truss element type 9 to simulate an actuator. This is often used in mechanism analyses to allow the prescription of the relative distance between two points. This option should be used with the LARGE DISP parameter whenever large rotations of the actuator or large displacements are anticipated.

The original length of the actuator is given in the fourth field of the GEOMETRY option. The actuator is treated as an elastic link.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word ACTUATOR.
2nd data block			
1-5	1st	I	Enter the number of actuators (optional).
6-10	2nd	I	Enter unit number for input of actuator data, defaults to input.
3rd data block			
Repeat for each actuator to be modified.			
1-5	1st	I	Enter the element number
6-15	2nd	F	Enter the new length of the actuator.

Rate Dependent Analysis

This section describes the control of the transient aspects of rate dependent analysis. There are several ways to specify the time step (TIME STEP) in either creep or viscoelastic analysis. Note that the adaptive options AUTO CREEP and AUTO THERM CREEP are recommended. The ACCUMULATE and EXTRAPOLATE options are techniques to allow extrapolation in time based on the results previously calculated. Note that extrapolation is always a risky procedure.

■ CREEP INCREMENT

Define Creep Increment

Description

This option allows manual control of the creep time step size. This form of control is not recommended.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-15	1st	A	Enter the words CREEP INCREMENT.
2nd data block			
1-15	1st	F	Creep time increment.

■ AUTO CREEP

Control Transient Creep

Description

This option controls the transient creep analysis. You specify a total creep time and a suggested time increment. The program automatically selects the largest possible time increment consistent with the tolerance set on stress and strain increments, which should have been specified earlier in the CREEP model definition option during the description of the problem.

You should make sure that load increments are not left on unintentionally, since this would reduce the time step size severely. Information is also input for limiting the total number of time increments. See *Volume A: Theory and User Information.*

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words AUTO CREEP.
2nd data block			
1-15	1st	F	Enter the suggested time increment for creep. When the automatic creep control is in use, the program iterates for the appropriate increment size to satisfy the tolerances placed on stress and strain increments by the CREEP model definition option.
16-30	2nd	F	Period of creep time to be covered during this AUTO CREEP option. This value can be changed at restart by using the REAUTO option.
31-35	3rd	I	Maximum number of time increments to be allowed during this part of the creep analysis. Default is 50 increments.
36-40	4th	I	Maximum number iterations allowed to modify the time step during an increment. Default is 5.

Format Fixed	Format Free	Data Type	Entry
41-45	5th	I	Number of increments between stiffness matrix updates. This option is used to prevent unnecessary updating of the stiffness matrix during large displacement creep analysis. If left blank, the stiffness matrix is reformed each step if tangent modulus nonlinearities (for example, plasticity) are present.
46-50	6th	I	Not used; enter 0.
51-60	7th	F	Enter stable time step limit, if known. The MARC program uses stresses and strain change tolerances if this is not used. Stable time step limit is needed for viscoplasticity.
3rd data block			
1-10	1st	F	If the fifth field is 0, enter tolerance on the creep strain increment relative to the elastic strain. Default is 0.50. Note that a higher value is likely to cause stability problems. If the fifth field is 1, enter the maximum creep strain increment allowed. Default is .01.
11-20	2nd	F	If the fifth field is 0, enter the tolerance on the stress change per stress during creep. Default is 0.10. If the fifth field is 0, enter the maximum stress increment. Default is 100. This control is included primarily for accuracy purposes. The default value is adequate for creep laws of the type $\varepsilon = A\sigma^n$, where $3<n<6$. For lower values of n, the tolerance can be increased; for higher values, it should be decreased.
21-30	3rd	F	Tolerance on low stress point cut-off. Points with a stress lower than this ratio relative to the maximum stress in the structure are not used in the creep tolerance checking. Default is 0.05.

Format Fixed	Format Free	Data Type	Entry
31-35	4th	I	Number of the element in which the stress change is checked. Leave blank to check all elements for stress change. If a number of elements (but not all elements) are to be checked, enter the number of elements as a negative number, with 14 as the maximum. In this case, the actual elements are entered on the next data block.
36-40	5th	I	Enter 1 if absolute rather than relative testing is to be performed.

4th data block

This series is only required if the entry in the fourth field of the previous block is negative.

1-70	1st	I	Enter the elements to be checked in (14I5) format.

■ ACCUMULATE

Specify Accumulation Option

Description

This flags the start of accumulation of strains and displacements for use with the extrapolate option. If a new accumulation period is to be started immediately after an extrapolation in the same increment, the ACCUMULATE option must be preceded by the EXTRAPOLATE option.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the word ACCUMULATE.

■ EXTRAPOLATE

Specify Extrapolation Option

Description

This option uses the accumulation of strains and displacements during one cycle of loading. It takes a linear extrapolation with respect to time or loading of the accumulated quantities.

Note: In order to use this option, the ACCUMULATE parameter must be included. The extrapolation is made from the period starting with the last entry of the ACCUMULATE option and ending at the current increment. If no ACCUMULATE option was used before, the accumulation period starts at the beginning of the analysis. A regular load increment can be applied simultaneously with the EXTRAPOLATE option.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-9	1st	A	Enter the word EXTRAPOLATE.
2nd data block			
1-10	1st	F	If a positive number is entered: Time to which extrapolation is to be extended. If a negative number is entered: Factor with which the previous accumulation period is multiplied to obtain extrapolation period.

■ AUTO THERM CREEP

Automatic, Thermally-Loaded Elastic-Creep Analysis

Description

This option is intended to allow automatic, thermally loaded elastic-creep/elastic-plastic-creep stress analysis, based on a set of temperatures defined throughout the mesh as a function of time. The temperatures and transient times are presented to the program through the CHANGE STATE option, using input option 3 (post file), and the program then creates its own set of temperature steps (increments) based on a temperature change tolerance provided on this option. The times at all temperature steps are calculated by the program for creep analyses.

At each temperature step (increment), an elastic/elastic-plastic analysis is carried out first to establish stress level in the structure. A creep analysis is performed next on the structure for the time period between current and previous temperature steps (increments). Both the elastic/elastic-plastic stress and the creep analyses are repeated until the total creep time provided on this option is reached. Convergence controls are provided on the CONTROL option for elastic-plastic analysis and on the AUTO THERM CREEP option for creep analysis.

You also specify a suggested time increment for creep analysis. The program automatically selects the largest possible time increment consistent with the tolerance set on stress and strain increments. The analysis can be restarted at temperature steps (increments) or at creep steps (subincrements). The results can be saved on a post file (POST option) for postprocessing.

The automatic thermally loaded elastic-creep/elastic-plastic-creep analysis continues until the total creep time is reached. A typical automatic thermally loaded elastic-creep/elastic-plastic-creep analysis has as input:

```
AUTO THERM CREEP
50.,0,0,4.0,
0.1,2.0,
0.,0.,0.,1,
CHANGE STATE
1,3,0,19,1,4,1,
CONTINUE
```

In the above case, a temperature change tolerance of 50 is set for the creation of temperature steps (increments) by the program; the total transient time in thermal analysis is equal to 4.0; the suggested time increment for creep analysis is 0.1; and the total creep time (time for the termination of this analysis) is 2.0. Note that the total creep time cannot be greater than the total transient time in the thermal analysis.

The data in the CHANGE STATE option indicates that the temperatures are stored in a formatted post file (file 19) and there are four sets of temperatures on the file.

If no DIST LOADS, POINT LOAD or PROPORTIONAL INCREMENT option appears with the AUTO THERM CREEP set, all mechanical loads and kinematic boundary conditions are held constant during the AUTO THERM CREEP. However, DIST LOADS, POINT LOAD, PROPORTIONAL INCREMENT, or DISP CHANGE can be included in the set. The mechanical loads and kinematic boundary conditions, which are then defined, are assumed to change in proportion to the time scale of the temperature history defined by the CHANGE STATE option and are applied accordingly on the basis that the increments of load and displacement correspond to the end of the transient time (TOTIM) of the AUTO THERM CREEP option input.

Note: You must include the CHANGE STATE, option 3 (post file), in conjunction with this option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-16	1st	A	Enter the words AUTO THERM CREEP.
2nd data block			
1-10	1st	F	Enter the maximum temperature change to be used per step of stress analysis. The program linearly subdivides steps, or merges steps together, to create increments which are close to, but do not exceed, this tolerance.
11-15	2nd	I	Enter the maximum number of increments to be allowed in this AUTO THERM CREEP. If this number of increments is exceeded before the temperature history is completed, the program ends. This is intended as a protection to avoid excessive increments in the case of a data error. Default value is 50 increments if set to 0.
16-20	3rd	I	Reassembly internal for element matrices.

Format Fixed	Format Free	Data Type	Entry
21-30	4th	F	Total transient time from heat transfer analysis. This is used to proportionally scale the incremental boundary conditions.
3rd data block			
1-15	1st	F	Enter the suggested time increment for creep. When the automatic creep control is in use, the program iterates for the appropriate increment size to satisfy the tolerances placed on stress and strain increments by the CREEP model definition option.
16-30	2nd	F	Total creep time to be covered during this AUTO THERM CREEP option. This value can be changed at restart by using the REAUTO option.
31-35	3rd	I	Maximum number of subincrements to be allowed during this part of the creep analysis. Default is 50.
36-40	4th	I	Maximum number of iterations allowed to modify the time step during an increment. Default is 5.
41-45	5th	I	Number of increments between stiffness matrix updates. This unnecessary updates of the stiffness matrix during large disp analysis. If left blank, the stiffness matrix is reformed each step if tangent modulus nonlinearities occur.
4th data block			
1-10	1st	F	If the fifth field is a zero, enter tolerance on the creep strain increment relative to the elastic strain. Default is 0.50. Note that a higher value is likely to cause stability problems. If the fifth field is a one, enter the maximum creep strain increment allowed. Default is .01.

Format Fixed	Format Free	Data Type	Entry
11-20	2nd	F	If the fifth field is a zero, enter the tolerance on the stress change per stress during creep. Default is 0.10. If the fifth field is a one, enter the maximum stress increment. Default is 100. This control is included primarily for accuracy purposes. The default value is adequate for creep laws of the type $\varepsilon = A\sigma^n$, where $3<n<6$. For lower values of n, the tolerance can be increased; for higher values, it should be decreased.
21-30	3rd	F	Tolerance on low stress point cut-off. Points with a stress lower than this ratio relative to the maximum stress in the structure is not used in the creep tolerance checking. Default is 0.05.
31-35	4th	I	Number of the element in which the stress change is checked. Leave blank to check all elements for stress change. If a number of elements (but not all elements) are to be checked, enter the number of elements as a negative number, with 14 as the maximum. In this case, the actual elements are entered on the next data line.
36-40	5th	I	Enter 1 if absolute rather than relative testing is to be performed.

5th data block

This data block is only required if the entry in the fourth field of the previous data block is negative.

1-70	1st	I	Enter the elements to be checked in (14I5) format.

Dynamic Analysis

This section describes the options used to control dynamic analysis. The MODAL SHAPE option requests that the program calculate the eigenvalues and eigenvectors of the structure. In linear dynamic analysis, this option immediately follows the END OPTION. It is possible to do a nonlinear static analysis and then perform a modal extraction based on the current deformed state. These modes can then be used to perform a transient analysis. The DYNAMIC CHANGE option is used to specify the incremental time and the total time period to be covered. An automatic time-stepping scheme (AUTO TIME) is also available for dynamic analysis. The SPECTRUM option allows a spectral response analysis be performed based upon the modes previously extracted. In addition, harmonic analysis can be performed at any step in an analysis. The harmonic response is based on the current configuration of the structure.

■ MODAL SHAPE

Define Modal Shape

Description

This option is used when the inverse power sweep method is indicated on the DYNAMIC parameter or during an acoustic analysis to indicate that eigen modes are to be extracted.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-11		A	Enter the words MODAL SHAPE.

Option A

If inverse power sweep method is used (as specified with the DYNAMIC parameter), use this option.

2nd data block

1-5	1st	I	Maximum number of iterations per mode in the power sweep. Default is 40.
6-15	2nd	F	Convergence tolerance. The power sweep terminates when the difference between the eigenvalues in two consecutive sweeps divided by the eigenvalue is less than the tolerance. Default is 1×10^{-5}.
16-25	3rd	F	Initial shift in cycles per time. The power shift is likely to start converging to the eigenvalue closest to this value. Default is 0.

Format Fixed	Format Free	Data Type	Entry
26-35	4th	F	Maximum frequency to be extracted in cycles per time. If this is left blank or zero, the number of modes requested on the DYNAMIC parameter is extracted. If this is nonzero, extraction ends when this frequency is exceeded or when the number of modes requested on the DYNAMIC parameter is reached, whichever occurs first.
36-40	5th	I	Number of modes extracted per shift. This data field determines if auto shifting occurs. If auto shift is not required, set equal to or greater than number of modes requested on the DYNAMIC parameter. Default is 5.
41-50	6th	F	Auto shift parameter. The program determines the new shift point (in frequency squared) as the highest frequency square plus this entry times the difference between the highest and next highest distinct frequency squared. Default is 1.0.

Option B

If the Lanczos method is used (as specified with the DYNAMIC parameter), use the following data blocks.

Format

Format Fixed	Format Free	Data Type	Entry
2nd card series			
1-10	1st	F	Lowest frequency of mode to be extracted (in cycles/time). This is also the initial shift point. This may not be changed upon restart.
11-20	2nd	F	Highest frequency of mode to extract. This may be changed upon restart.

Format		Data	
Fixed	Free	Type	Entry
21-25	3rd	I	Number of requested modes. If zero, all modes between the minimum and maximum specified will be extracted. This may be increased upon restart.
26-30	4th	I	Sturm sequence flag. Set to 1 to perform Sturm sequence check on extracted eigenvalues. Set to 0 to turn off checking.
31-35	5th	I	Restart option flag. Set to 0 to prevent writing of modes to restart file. Set to 1 to write modes onto restart file previously extracted. Set to 2 to read modes from restart tape. Set to 3 to read previously extracted modes and write new modes. The RESTART model definition option must also be included.

■ RECOVER

Recover Option

Description

This option allows for: (1) the storing of eigenvectors on the post file, (2) the recovery of reaction forces, or (3) the recovery of stresses and reactions for a specified number of modes during a modal or a buckling analysis. The option should be used after the modal shapes and frequencies or buckling modes have been extracted (MODAL SHAPE or BUCKLE history definition option), and can be repeated as many times as you wish. Additional input data is required on parameters DYNAMIC or BUCKLE for the activation of this option, and the POST model definition option must be included in the input if the eigenvectors are to be stored on a post file.

In the RECOVER option, the stresses are computed from the modal displacement vector, ϕ (eigenvector without normalization), and the nodal reactions are calculated from $F = K\phi - \omega^2 M\phi$ for the modal analysis or from $F = K\phi$ for buckling analysis.

You can choose your normalization by entering the amplitude value of one particular degree of freedom of one particular node in the mesh. The eigenvector is scaled linearly such that the degree of freedom on this node reaches above amplitude.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-7	1st	A	Enter the word RECOVER.
2nd data block			
1-5	1st	I	Starting mode number. Default is 1.
6-10	2nd	I	Ending mode number. Default is modes specified on parameter (DYNAMIC or BUCKLE).

Format		Data	
Fixed	**Free**	**Type**	**Entry**
11-15	3rd	I	Set to 0 if only the eigenvectors are to be written to post file. Set to 1 if only reactions are to be calculated. Set to 2 if both stresses and reactions are to be calculated. **Note:** If set to 2, 4th field of the DYNAMIC parameter or 3rd field of the BUCKLE parameter must be set equal to 1.
16-20	4th	I	Node number of node with respect to which the eigenvector is scaled. Default is 0.
21-25	5th	I	Degree of freedom of node selected in the fourth field with respect to which the eigenvector is scaled. Default is 0.
26-35	6th	E	Reference amplitude of degree of freedom of node selected above. Default is 0, which uses the eigenvector without user scaling.

■ DYNAMIC CHANGE

Define Integration in Time

Description

This option specifies the parameters required for integration in time. It can be used for either the modal or the direct integration procedure. See *Volume A: User Information* on Dynamic Options and the DYNAMIC, ACOUSTIC, or EL-MA parameter (Chapter 2 of *Volume C: User Input*).

In the case of explicit analysis, `IDYN = 5`, the time step is adjusted each increment to insure that the stability limit is not violated.

In the case of explicit analysis, `IDYN = 4`, the time step is adjusted to ensure that the stability limit is not violated at the start of this DYNAMIC CHANGE.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-14	1st	A	Enter the words DYNAMIC CHANGE.
2nd data block			
1-15	1st	F	Time step size.
16-30	2nd	F	Period of time for this set of boundary conditions.
31-35	3rd	I	Number of time steps in this set of boundary conditions.
36-40	4th	I	This field is not used.
41-45	5th	I	Reassembly interval for mass and stiffness matrices; for linear problems, set equal to value in the third field.
46-50	6th	I	This field is not used; enter 0.
51-60	7th	F	Enter γ for Newmark operator. Default is $\gamma = 0.5$ or what was used in previous DYNAMIC CHANGE option.
61-70	8th	F	Enter β for Newmark operator. Default is $\beta = 0.25$ or what was used in previous DYNAMIC CHANGE option.

■ AUTO TIME

Automatic Time Stepping

Description

This option allows automatic time stepping in dynamic analysis or in coupled thermal-stress analysis (either quasi-static or dynamic). This option is able to handle either linear or nonlinear analysis. When used for dynamic analysis, the Newmark beta operator must be used. This is the only one which allows variable time steps.

You have to specify the total loading for a sequence of load steps in the POINT LOAD, DIST LOADS, and/or FIXED DISP options. The program automatically generates the magnitude of the load step such that equilibrium is satisfied with a minimum number of iterations.

The size of the load vector is controlled by the error in equilibrium of the previous increment/iteration. The analysis is stopped when the total load is reached or when the maximum allowed number of increments is reached.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-9	1st	A	Enter the words AUTO TIME.
2nd data block			
1-10	1st	F	Initial fraction of load/time; must be greater than 10^{-5}.
11-20	2nd	F	Maximum fraction of load/time to apply in an increment, AUTMAX. Default is 1.0.
21-30	3rd	F	Initial time step.
31-40	4th	F	Total time period to be covered (TOTINC).
41-50	5th	F	Smallest reduction factor, FCSML. Default is .5.
51-60	6th	F	Largest increase factor, FCLRG. Default is 2.0.
61-65	7th	I	Maximum number of increments, IOTNUM.

Format		Data	
Fixed	Free	Type	Entry
66-70	8th	I	Desired number of recycles per increment.
71-80	9th	F	Minimum time step allowed. In a static analysis, the default value is TOTINC/1000. In a dynamic analysis (Newmark-beta or Houbolt), the default value is TOTINC/10000.

Note: The increment of time in a step n+1 is always in the range:

$$FCSML*\Delta t^{n} \leq \Delta t^{n+1} \leq FCLRG*\Delta t^{n}$$

and

$$\Delta t^{n+1} \leq AUTMAX*TOTINC.$$

■ SPECTRUM

Initiate Spectral Response Analysis

Description

This option initiates the spectral response analysis.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word SPECTRUM.
2nd data block			
1-5	1st	I	Enter the number of modes to be used. Enter 0 if a range of frequencies are given in the second and third field.
6-15	2nd	F	Enter the lowest frequency to be used in response analysis, in cycles per time unit.
16-25	3rd	F	Enter the highest frequency to be used in response analysis, in cycles per time unit.
3rd data block			
1-10	1st	F	Weighting factor associated with first degree of freedom.
11-20	2nd	F	Weighting factor associated with second degree of freedom.
21-30	3rd	F	Weighting factor associated with third degree of freedom.
			Etc.; for higher degrees of freedom–maximum of 8 factors on a data line.

■ HARMONIC

Define Excitation Frequency

Description

This option is necessary to specify the frequency of excitation implemented.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word HARMONIC.
2nd data block			
1-10	1st	F	Enter the lowest frequency in cycles per time unit.
11-20	2nd	F	Enter the increment in frequency for each sub-increment. If zero, only single frequency is used.
21-30	3rd	F	Enter the highest frequency in cycles per time unit.

■ ACC CHANGE

Define Acceleration Boundary Conditions

Description

This option allows new nodal acceleration conditions to be specified in a dynamic analysis. You specify total accelerations through this option. Complex accelerations are more conveniently defined by user subroutine FORCDT. Note that enough space must be specified on the SIZING parameter in h maximum number of boundary condition field to allow for possible increased storage requirements arising from the user of this option.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words ACC CHANGE.
2nd data block			
1-5	1st	I	Enter the number of sets of boundary condition data lines to be read (optional)

For each set of boundary conditions use the 3rd, 4th, and 5th data blocks.

3d data block			
1-10	1st	E	Prescribed acceleration for first degree of freedom listed in data block 4.
11-20	2nd	E	Prescribed acceleration for second degree of freedom listed in data block 4.
21-30	3rd	E	Prescribed acceleration for third degree of freedom listed in data block 4.
4th data block			
			Enter a list of degrees of freedom to which the above prescribed displacements are prescribed. **Note:** List verbs `EXCEPT` and `INTERSECT` are illegal here.
5th data block			
			Enter a list of nodes to which the above accelerations are applied.

Heat Transfer Analysis

This section describes the control of both steady-state and transient heat transfer analyses. The TRANSIENT option is available to specify the incremental time and the total time in this set. The default is that an adaptive time stepping is performed to cover the complete time period, such that the your tolerances are satisfied. These tolerances are given in the CONTROL option. There are several methods of specifying the fluxes. Note that the total flux is required in a heat transfer analysis.

■ TRANSIENT

Specify Transient or Steady State Heat Transfer Analysis

Description

This option controls the transient heat transfer analysis. The program optionally uses automatic time step controls, based on the maximum nodal temperature change allowed (input on the CONTROL model definition option). You input a suggested initial time step which is adjusted according to the automatic stepping scheme (see *Volume A: User Information*). The transient period can be ended in one of two ways. A time period is input – the transient solution continues until this period is completed. You can also give a finish temperature, and a flag which indicates that the transient solution should end when all nodal point temperatures are below (or above) this finish temperature. This second technique for ending the transient is optional and does not disable the first option, so that an adequately long time period should be allowed if the finish temperature option is to govern the solution. In addition, you should supply a maximum number of steps to be allowed. This is intended to prevent an excessive number of increments in case of a data error. If the program analyzes this number of increments before completing the time period (or reaching the finish temperature), it stops any further analysis.

For steady-state solutions, use the TRANSIENT NON AUTO option with one "infinite" time step (set specific heat to zero if problem is linear). Use the recycling tolerance for property evaluation for nonlinear steady-state solutions. In addition, the STEADY STATE option is also available for steady-state solution.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-9	1st	A	Enter the word TRANSIENT.
11-18	2nd	A	Leave blank to use automatic time stepping. Enter the words NON AUTO to suppress automatic time stepping, and complete total period with a uniform time step.

Format

Format Fixed	Format Free	Data Type	Entry
2nd data block			
1-10	1st	F	Suggested initial time step. This is be adjusted by the automatic time-stepping scheme (see *Volume A: User Information*).
11-20	2nd	F	Time period. The program continues the transient solution until this time period is completed, unless the finish temperature option is flagged.
21-25	3rd	I	Maximum number of steps to be allowed in this transient period. The program stops the analysis if this number of steps is exceeded. This data field is intended to be used to avoid excessive steps in the automatic control. Default (if left blank) is time period divided by suggested step.
26-30	4th	I	Not used; enter 0.
31-35	5th	I	Reassembly interval – number of increments between reassessment of element (material) properties based on temperature or time dependency. For purely linear problems (constant properties), a large number should be given here to avoid reassembly. Default value (if left blank) is automatic reassembly controlled by the temperature change given on the CONTROL option, data line 3, second field. Note that if a property or film coefficient changes, reassembly is needed.
36-40	6th	I	Set to 1 to finish the transient when all nodal temperatures fall below the value given in the seventh field (see below). Set to -1 to finish the transient when all nodal temperatures exceed the value given in the seventh field (see below). Set to 0 to complete transient time period without any check on temperatures reached. Set to 2 to indicate that a steady-state analysis is to be performed in one increment.
41-50	7th	F	Finish temperature value to be used in conjunction with flag set above.

■ STEADY STATE

Specify Steady-State Heat Transfer Analysis

Description

This option allows the solution of the steady-state heat transfer problem. This procedure uses less computation time than using the TRANSIENT NON AUTO option with a large time step. If temperature dependent properties or boundary conditions are included, the recycling tolerance for property evaluation must be used so that iteration is performed.

The program begins execution of the step when a CONTINUE option is encountered. Note that multiple steady-state steps can be performed to solve quasi-steady-state problems.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-10	1st	A	Enter the words STEADY STATE.

■ DIST FLUXES

Define Distributed Fluxes

Description

This option allows distributed (surface and volumetric) fluxes to be specified. Distributed fluxes are converted to consistent nodal fluxes by the program. For a given element type, there is an established convention for the application of surface flux on a particular face. User subroutine FLUX can be used to input and spatial dependent fluxes.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words DIST FLUXES.
2nd data block			
1-5	1st	I	Enter the number of sets of distributed fluxes to be entered (optional).
6-10	2nd	I	Enter tape number for input of distributed flux data, defaults to input.
The 3rd and 4th data blocks are repeated as pairs, once for each set.			
3rd data block			
1-5	1st	I	Parameter identifying the type of distributed flux. See library element description in *Volume B: Element Library.*
6-15	2nd	E	Enter the magnitude of this type of distributed fluxes.
16-20	3rd	I	Flux index. Flux index is to be used in subroutine FLUX.
4th data block			
			Enter a list of elements associated with the above distributed fluxes.

■ POINT FLUX

Define Point Fluxes

Description

This option allows total nodal point fluxes to be specified. User subroutine FORCDT can be used for the time dependent fluxes. If the number of nodes which have point fluxes has been changed from the model definition block, you must give an upper bound on the FLUXES parameter.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	POINT FLUX.
2nd data block			
1-5	1st	I	Enter the number of sets of point fluxes to be entered (optional).
6-10	2nd	I	Enter file number for input of point flux data, defaults to input.

The 3rd and 4th data blocks are entered as pairs, once for each set.

3rd data block			
1-10	1st	F	Magnitude of point flux.
11-20	2nd	F	Magnitude of point flux for second degree of freedom, shell elements only.
21-30	3rd	F	Magnitude of point flux for third degree of freedom, shell elements only.
4th data block			
			Enter a list of nodes to which the above nodal fluxes are applied.

■ CONTROL

Define Controls for Heat Transfer Analysis

Description

This option allows you to input parameters governing the convergence solution and accuracy for heat transfer analysis.

For transient heat transfer, the only data field required to be set is the maximum number of steps, the first field in the second data block. All other fields can, in these cases, be left blank, but notice that the third data block must be included.

For coupled thermal-stress analysis, see CONTROL option for stress analysis.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-7	1st	A	Enter the word CONTROL.
2nd data block			
1-15	1st	I	Maximum number of load steps in this run. Default is 4. This is a cumulative number and is usually used to stop the run when restart is being used.
6-10	2nd		Maximum number of recycles during an increment due to temperature dependent material properties. Default is 3.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
1-10	1st	F	Maximum nodal temperature change allowed. Used to control automatic time step scheme for heat transfer. Default value is 20 degrees.
11-20	2nd	F	Maximum nodal temperature change allowed before properties are re-evaluated and matrices reassembled. Default value is 100 degrees.
21-30	3rd	F	Maximum error in temperature estimate used for property evaluation. This control provides a recycling capability to improve accuracy in highly nonlinear heat-transfer problems (for example, latent heat, radiation boundary conditions). Default is 0, which bypasses this test. Set to maximum temperature error which is considered acceptable.

■ TEMP CHANGE

Specify or Change Fixed Temperatures

Description

This option allows new fixed temperature to be specified or old fixed temperatures to be changed. The exact numbering sequence of the fixed temperatures is used in some applications of this option. This numbering sequence is output after the fixed temperature option is used in the input data describing the problem.

This option is used to change fixed temperatures in heat transfer. Note that enough space must be specified on the SIZING parameter in the maximum number of boundary condition fields to allow for possible increased storage requirements arising from use of this option. Complex temperature histories are more conveniently input by user subroutine FORCDT.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-15	1st	A	Enter the words TEMP CHANGE.
2nd data block			
1-5	1st	I	Set to the number of fixed temperatures to be changed or added. A negative number removes fixed temperatures from the end of the fixed temperature list. A zero activates the FIXED TEMPERATURE option; a complete set of necessary fixed temperatures are then read, using the data blocks for that option except for that key word block.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
Data block 3 is only entered if columns 1 through 5 in data block 2 are a positive number and then has the number of data lines required by data block 2.			
1-5	1st	I	Number of the boundary condition being changed. This number is derived from the "Fixed Boundary Condition Summary" table in the input echo of a MARC run. Boundary conditions being added should be given labels which increment the total count of boundary conditions properly. Note that a boundary condition in the middle of the list can be removed by specifying that labeled boundary condition as a repeat of some other boundary condition.
6-10	2nd	I	Nodal point to be constrained.
11-15	3rd	I	Degree of freedom to be constrained.
16-30	4th	F	Specified temperature.

■ FILMS

Define Film Coefficients and Sink Temperatures

Description

This option allows film coefficients and associated sink temperatures to be input. Nonuniform films or sink temperatures can be obtained via user subroutine FILM, see *Volume D: User Subroutines and Special Routines.*

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-5	1st	A	Enter the word FILMS.
2nd data block			
1-5	1st	I	Number of sets of data used to input film (optional).
6-10	2nd	I	Unit number for input of film data, defaults to input.

The 3rd and 4th data blocks are entered as pairs, once for each data set.

3rd data block			
1-5	1st	I	Face identification. Same as for FLUXES – see *Volume B: Element Library.*
6-15	2nd	F	Reference value of film coefficient.
16-25	3rd	F	Reference value of sink temperature (reference values can be modified by subroutine FILM).
26-30	4th	I	Film coefficient index (optional).
31-35	5th	I	Sink temperature index (optional). Film coefficient and sink temperature indices are to be used in user subroutine FILM.
4th data block			
			Enter a list of elements to which the above film data is applied.

■ VELOCITY CHANGE

Modify Nodal Velocity Components

Description

This option allows the specification of the nodal velocity components in a heat transfer analysis, where the convective terms are to be included. The convective option is specified by placing a 2 in the fifth field of the HEAT parameter. The nodal velocity components are defined by specifying the velocity magnitude of a series of components for sets of nodes. This data can be input from data blocks or from an auxiliary input device. Moreover, the velocity values can be respecified or initialized if no previous data was entered via user subroutine UVELOC. See *Volume D: User Subroutines and Special Routines*.

A summary of nodal velocities appears in the printout. This can be suppressed by specifying a nonzero value for the print-suppress parameter.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word VELOCITY CHANGE.
2nd data block			
1-5	1st	I	Number of sets of data lines used to input nodal velocity components. If a negative value is entered, user subroutine UVELOC is be called for every node.
6-10	2nd	I	Enter the unit number for input of the velocity field. Default to unit 5, unless the INPUT TAPE parameter has been used.
11-15	3rd	I	Set to 1 to suppress printout of the summary of nodal velocity components.

Format Fixed	Format Free	Data Type	Entry
Data blocks 3, 4, and 5 should be repeated for each data set.			
3rd data block			
1-10	1st	F	Enter the magnitude of the velocity in the first coordinate-direction for which the velocity is be given. Additional velocity components in other coordinate directions must be specified on the same data line in F10 format. The number of components must equal the number of directions.
4th data block			
			Enter a list of coordinate directions in which the velocity is specified. **Note:** List verbs `EXCEPT` and `INTERSECT` are illegal here.
5th data block			
			Enter a list of nodes for which the velocity vector, as defined in data blocks 3 and 4, is applied.

Joule Heating Analysis

This section describes the input of data necessary for a Joule heating analysis. All of the options in the previous subsections referring to heat transfer are also applicable. Both the applied currents and the nodal voltages can be specified in this section. In Joule heating analysis, these are total values.

■ DIST CURRENT

Define Distribute Current

Description

This option allows distributed (surface and volumetric) current to be specified. Distributed current is converted to a consistent nodal current by the program. Note that for a given element type, there is an established convention for the application of surface current of a particular face. User subroutine FLUX can be used to input time and spatial dependent current.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the words DIST CURRENT.
2nd data block			
1-5	1st	I	Enter the number of sets of distributed currents to be entered (optional).
6-10	2nd	I	Enter unit number for input of distributed current data, defaults to input.

The 3rd and 4th data blocks are entered as pairs, one for each data set.

3rd data block			
1-5	1st	I	Parameter identifying the type of current. See library element description in *Volume B: Element Library.*
6-15	2nd	F	Enter the magnitude of this type of distributed current.
4th data block			
			Enter a list of elements associated with the above distributed current.

■ POINT CURRENT

Define Nodal Point Current

Description

This option allows total nodal point current to be specified. User subroutine FORCDT can be used for the time dependent current. If the number of nodes which have point currents has been changed from the model definition block, you must give an upper bound on the DIST LOADS parameter.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words POINT CURRENT.
2nd data block			
1-5	1st	I	Enter the number of sets of point currents to be entered (optional).
6-10	2nd	I	Enter unit number of input of point current data, defaults to input.

The 3rd and 4th data blocks are entered as pairs, once for each data set.

3rd data block			
1-10	1st	F	Magnitude of point current.
4th data block			
			Enter a list of nodes to which the above nodal current are applied.

■ VOLTAGE CHANGE

Define or Change Voltage for Joule Heating Analysis

Description

This option allows new voltage conditions to be specified or old voltage conditions to be changed in a Joule heating analysis. The exact numbering sequence of the voltage conditions is used in some applications of this option. This numbering sequence is output after the voltage conditions governing increment zero are read by the program.

This option is used for Joule heating analysis. Enough space must be specified on the SIZING parameter in the maximum number of boundary condition fields to allow for possible increased storage requirements arising from use of this option. Complex time dependent voltage histories are more conveniently input by user subroutine FORCDT.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-15	1st	A	Enter the words VOLTAGE CHANGE.
2nd data block			
1-5	1st	I	Set to the number of specified voltage conditions to be changed or added. A negative number removes voltage conditions from the end of the voltage condition list. A zero activates the voltage option; a complete set of voltages are then read, using the data blocks for that option as described in the model definition section, except for that key word block.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
Data block 3 is only entered if the first field in data block 2 is a positive number determining the number of blocks required in this series.			
1-5	1st	I	Number of the boundary condition being changed. This number is derived from the "Fixed Voltage Condition Summary" table in the input echo of a MARC run. Voltage conditions being added should be given labels which increment the total count of voltage conditions properly. A voltage condition in the middle of the list can be removed by specifying that labeled boundary condition as a repeat of some other boundary condition.
6-10	2nd	I	Nodal point to be constrained.
11-15	3rd	I	Set to 1.
16-30	4th	F	Specified voltage.

Hydrodynamic Bearing Analysis

The purpose of bearing analysis history definition options are mainly to activate the calculation of bearing characteristics like damping and stiffness coefficients for a particular film profile. In addition, it is possible to solve the lubrication problem for a new film profile. The history definition options used are:

- THICKNS CHANGE to specify a variation of the thickness field.
- DAMPING COMPONENTS to activate the calculation of damping coefficients based on the defined thickness change.
- STIFFNS COMPONENTS to activate the calculation of stiffness coefficients based on the defined thickness change.

The calculation of damping and stiffness properties is performed within subincrements.

If only the THICKNS CHANGE option is activated within an increment, the defined thickness change is added to the current film profile. The lubrication problem is then solved for the current film thickness.

■ THICKNS CHANGE

Define Thickness Variations

Description

This option defines the thickness variations of the lubricant film in a bearing analysis. The nodal thickness changes are specified by giving the thickness increments for a list of nodes. This data can be input from data blocks or from a tape. Moreover, the nodal thickness changes can be respecified or initialized, in case no previous data was input, via user subroutine UTHICK. See *Volume D: User Subroutines and Special Routines.*

A summary of nodal thickness changes appears in the printout. This can be suppressed by specifying a nonzero value for the print-suppress parameter.

The THICKNS CHANGE option enables the solution of the lubrication problems for a modified film profile. The defined thickness variation is added to the previously defined film profile. In case this option is combined with either the DAMPING COMPONENTS or STIFFNS COMPONENTS option, no updating is performed and damping or stiffness components are calculated within a subincrement based on the specified thickness variation.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word THICKNS CHANGE.
2nd data block			
1-5	1st	I	Number of sets of data lines used to input nodal thickness changes. If a negative value is entered, user subroutine UTHICK is called for every node.
6-10	2nd	I	Enter the file number for input of film thickness variations. Default to unit 5, unless the INPUT TAPE parameter has been used.
11-15	3rd	I	Set to 1 to suppress printout of the summary of nodal thickness increments.

Format Fixed	Format Free	Data Type	Entry
The 3rd and 4th data blocks are entered as pairs, one for each data set.			
3rd data block			
1-10	1st	F	Enter nodal thickness increment.
4th data block			
			Enter a list of nodes for which the nodal thickness variation as specified in data block 3 applied.

■ DAMPING COMPONENTS

Define Damping Coefficients

Description

This option activates the calculation of damping coefficients for bearing analysis. Based on the thickness variation specified in the THICKNS CHANGE option, an incremental pressure distribution due to the thickness variation per unit time is calculated for the current bearing configuration within a subincrement. The resulting bearing force represents the damping properties pertaining to the specified rate of thickness change for the current film profile.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words DAMPING COMPONENTS.

■ STIFFNS COMPONENTS

Define Stiffness Coefficients

Description

This option activates the calculation of stiffness coefficients for bearing analysis. Based on the thickness variation specified in the THICKNS CHANGE option, an incremental pressure distribution due to the thickness variation is calculated for the current bearing configuration within a subincrement. The resulting bearing force represents the stiffness properties pertaining to the specified thickness change for the current film profile.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word STIFFNS COMPONENTS.

Acoustic Analysis

This section describes options for the control an acoustic analysis in a cavity with rigid boundaries. In an acoustic analysis, you would first extract the eigenmodes of the cavity using the MODAL SHAPE option. The time step during the transient option would be controlled by the DYNAMIC CHANGE option. These two options are previously discussed in this chapter. The DIST SOURCES, POINT SOURCE, and PRESS CHANGE options are available to specify the data to allow the program to calculate the fundamental frequencies of the cavity as well as the pressure distribution in the cavity. Incremental load information is provided here.

■ PRESS CHANGE

Define Fixed Pressures

Description

This option allows new fixed pressures to be specified or old fixed pressures to be changed. The exact numbering sequence of the fixed pressures is used in some applications of this option. This numbering sequence is output after the fixed pressure option is used in the input data describing the problem.

This option is used to change fixed pressures in acoustic transfer. Enough space must be specified on the SIZING parameter in the maximum number of boundary condition fields to allow for possible increased storage requirements arising from use of this option. Complex pressure histories are more conveniently input by user subroutine FORCDT.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-15	1st	A	Enter the words PRESS CHANGE.
2nd data block			
1-5	1st	I	Set to the number of fixed pressure to be changed or added. A negative number removes fixed pressures from the end of the fixed pressure list. A zero activates the FIXED PRESSURE option; a complete set of necessary fixed pressures are then read, using the blocks for that option except for that key word block.

Format Fixed	Format Free	Data Type	Entry
3rd data block			
Data block 3 is only entered if columns 1 through 5 in data block 2 is a positive number and then has the number of data lines required by data block 2.			
1-5	1st	I	Number of the boundary condition being changed. This number is derived from the "Fixed Boundary Condition Summary" table in the input echo of a MARC run. Boundary conditions being added should be given labels which increment the total count of boundary conditions properly. A boundary condition in the middle of the list can be removed by specifying that labeled boundary condition as a repeat of some other boundary condition.
6-10	2nd	I	Nodal point to be constrained.
11-15	3rd	I	Degree of freedom to be constrained.
16-30	4th	F	Specified pressure.

Electrostatic Analysis

In an electrostatic analysis, the boundary conditions are charges given in the POINT CHARGE and DIST CHARGE option, and the potential given in the FIXED POTENTIAL option. The history definition consists of the STEADY STATE option and the CONTINUE option which initiates the analysis. The analysis is linear, so no iterations will occur.

■ STEADY STATE

Specify Steady-State Electrostatic Analysis

Description

This option allows the solution of the steady-state electrostatic problem.

The program begins execution of the step when a CONTINUE option is encountered.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-10	1st	A	Enter the words STEADY STATE.

Magnetostatic Analysis

In a magnetostatic, the boundary conditions are currents given to the POINT CURRENT and DIST CURRENT options and the potential given in the FIXED POTENTIAL option. The history definition consists of the STEADY STATE and the CONTINUE options which initiate the analysis. If a nonlinear material is present, defined through the B-H RELATION option, iteration is required as specified in the CONTROL option; otherwise, no iterations occur.

■ STEADY STATE

Specify Steady-State Magnetostatic Analysis

Description

This option allows the solution of the steady-state magnetostatic problem. If nonlinear magnetic properties are included, the recycling tolerance for property evaluation must be used so that iteration is performed.

The program begins execution of the step when a CONTINUE option is encountered.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1-10	1st	A	Enter the words STEADY STATE.

Electromagnetic Analysis

This section describes options used to control both transient and harmonic electromagnetic analysis. The DYNAMIC CHANGE option is used to specify the time step, while the HARMONIC option is used to control the excitation frequency. Note that all loads are incremental in nature.

■ HARMONIC

Define Excitation Frequency

Description

This option is necessary to specify the frequency of excitation implemented. This can be used if the harmonic flag (1) has been set on the EL-MA parameter.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word HARMONIC.
2nd data block			
1-10	1st	F	Enter the lowest frequency in cycles per time unit.
11-20	2nd	F	Enter the increment in frequency for each subincrement. If 0, only single frequency is used.
21-30	3rd	F	Enter the highest frequency in cycles per time unit.

■ DYNAMIC CHANGE

Define Dynamic Change

Description

This option specifies the parameters required for integration in time. This can be used if the transient flag (0) has been set on the EL-MA parameter. The Newmark-beta procedure with a fixed time step is used.

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-14	1st	A	Enter the words DYNAMIC CHANGE.
2nd data block			
1-15	1st	F	Time step size.
16-30	2nd	F	Period of time for this set of boundary conditions.
31-35	3rd	I	Number of time steps in this set of boundary conditions.
36-40	4th	I	This field is not used.
41-45	5th	I	Reassembly interval for mass and stiffness matrices; for linear problems, set equal to value in the seventh field.

■ POTENTIAL CHANGE

Define or Redefine Potential Boundary Conditions

Description

This option allows new potential boundary conditions to be specified or old potential boundary conditions to be changed. The exact numbering sequence of the boundary conditions is used in some applications of this option. This numbering sequence is output after the boundary condition option is used in the input data describing the problem.

This option is used for incrementation of fixed potential components or for adding or removing potential constraints. Care should be taken, when removing fixed potential conditions, to ensure that the reaction forces are handled properly. Note that enough space must be specified on the SIZING parameter in the maximum number of boundary condition fields to allow for possible increased storage requirements arising from use of this option. Time dependent potentials are more conveniently input by user subroutine FORCDT.

This option implies a proportional increment of 1.0. Any resetting of this factor (for example, the PROPORTIONAL INCREMENT option used before the next CONTINUE option) proportions these potential increments as well.

When used in conjunction with harmonic analysis, this boundary change is used for all excitation frequencies until a new boundary change is invoked.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-11	1st	A	Enter the words POTENTIAL CHANGE.

Format Fixed	Format Free	Data Type	Entry
2nd data block			
1-5	1st	I	Set to the number of boundary conditions (specified potential components to be changed or added). A negative number removes boundary conditions from the end of the boundary condition list. A zero activates the FIXED POTENTIAL option; a complete set of necessary boundary conditions are then read, using the blocks for that option except for that key word block.
6-10	2nd	I	Enter 1 if excitation boundary conditions for harmonic analysis.
11-15	3rd	I	Enter 1 if harmonic boundary condition is input as magnitude and phase. If blank, real and imaginary values are given.
3rd data block			
Data block 3 is only entered if the number in columns 1 through 5 in data block 2 is positive and then has the number of data lines required by data block 2.			
1-5	1st	I	Number of the boundary condition being changed. This number is derived from the "Fixed Boundary Condition Summary" table in the input echo of a MARC run. Boundary conditions being added should be given labels which increment the total count of boundary conditions properly.
6-10	2nd	I	Nodal point to be constrained.
11-15	3rd	I	Degree of freedom to be constrained. Note that a boundary condition in the middle of the list can be removed by specifying that labeled boundary condition as a repeat of some other boundary condition.
16-30	4th	F	Specified potential increment (real part).
31-45	5th	F	Specified potential increment (imaginary part).

■ POINT CURRENT

Define Point Current and/or Charge

Description

This option allows nodal point currents and point charges to be specified. User subroutine FORCDT can be used for nonuniform loading conditions.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words POINT CURRENT.
11-15	2nd	I	Enter 1 to enter real harmonic current and charge. Enter 2 to enter imaginary harmonic current charge.
2nd data block			
1-5	1st	I	Enter number of sets of point current and charge to be entered (optional).
6-10	2nd	I	Enter file number for input of point current and charge data. Defaults to input.

The following 3rd and 4th data blocks are entered as pairs, once for each data set.

3rd data block			
1-10	1st	F	Magnitude of point current for first degree of freedom.
11-20	2nd	F	Magnitude of point current for second degree of freedom.
21-30	3rd	F	Magnitude of point current for third degree of freedom.
31-40	4th	F	Magnitude of point charge.
4th data block			
			Enter a list of nodes to which the above point current-charge applies.

■ DIST CURRENT

Define Distributed Current

Description

This option allows distributed (surface and volumetric) currents to be specified. Distributed currents are converted to consistent nodal currents by the program. For a given element type, there is an established convention for the application of surface current of a particular face. User subroutine FORCEM can be used to input spatially dependent current.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words DIST CURRENT.
11-15	2nd	I	Enter 1 if distributed current is to be applied as excitation load in a harmonic analysis.
2nd data block			
1-5	1st	I	Enter the number of sets of distributed currents to be entered (optional).
6-10	2nd	I	Enter file number for input of distributed current data, defaults to input.

The following 3rd and 4th data blocks are given in pairs, once for each data set.

3a data block

Use if not harmonic analysis.

1-5	1st	I	Parameter identifying the type of current. See library element description in *Volume B: Element Library.*
6-15	2nd	F	Enter the magnitude of this type of distributed current.
16-20	3rd	I	Current index. Current index is to be used in subroutine FORCEM.

Format Fixed	Format Free	Data Type	Entry
3b data block			
Use if harmonic analysis.			
1-5	1st	I	Parameter identifying the type of current. See library element description in *Volume B: Element Library.*
6-15	2nd	F	Enter the real component of distributed current.
16-25	3rd	F	Enter the imaginary component of distributed current.
26-30	4th	I	Current index. Current index is to be used in subroutine FLUX.
4th data block			
			Enter a list of elements associated with the above distributed current.

■ DIST CHARGE

Define Distributed Charges

Description

This option allows distributed (surface and volumetric) charges to be specified. Distributed charges are converted to consistent nodal charges by the program. Note that for a given element type, there is an established convention for the application of surface charge on a particular face. User subroutine FORCEM can be used to input spatially-dependent charges.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words DIST CHARGE.
11-15	2nd	I	Enter 1 if distributed charge is to be applied as excitation load in a harmonic analysis.
2nd data block			
1-5	1st	I	Enter the number of sets of distributed charges to be entered (optional).
6-10	2nd	I	Enter file number for input of distributed charge data. Defaults to input.

The following 3rd and 4th data blocks are given as pairs, once for each data set.

3a data block

Use if not harmonic analysis.

1-5	1st	I	Parameter identifying the type of distributed charge. See description of library element in *Volume B: Element Library.*
6-15	2nd	E	Enter the magnitude of this type of distributed charges.
16-20	3rd	I	Charge index (optional). Charge index is to be used in subroutine FORCEM.

Format Fixed	Format Free	Data Type	Entry
3b data block			
Use if harmonic analysis.			
1-5	1st	I	Parameter identifying the type of charge. See library element description in *Volume B: Element Library*.
6-15	2nd	F	Enter the real component of distributed charge.
16-25	3rd	F	Enter the imaginary component of distributed charge.
26-30	4th	I	Charge index. Charge index is to be used in subroutine FORCEM.
4th data block			
			Enter a list of elements associated with the above distributed charge.

■ CONTINUE

End Loadcase

Description

The CONTINUE option is necessary to indicate that all data for this increment or series of increments has been read in. The analysis is then initiated.

Format

Format		Data	
Fixed	Free	Type	Entry
1-8	1st	A	Enter the word CONTINUE.

5 List of Rezoning Options

5

5 Rezoning

For the analysis of metal forming problems the MARC program utilizes the updated Lagrange approach. In this approach, the state at the beginning of an increment serves as the reference state for the calculation of the incremental values. At each subsequent increment, the reference state is updated.

This analysis method has several advantages, but it has a limit on the maximum deformation attainable. Due to the large deformations, the element mesh can degenerate strongly, and, in the updated approach, this means that the analysis of subsequent increments is carried out with a very poor mesh. This effect can even be so serious that elements locally turn inside out, which makes further analysis impossible.

In order to continue the analysis with sufficient accuracy, it is necessary to use a new mesh. The state in the old mesh must be transferred to the new mesh. Such a transfer is, of course, only possible if the state in the old mesh is defined with respect to the current configuration. Hence, if you use the updated Lagrange approach, you might be required to rezone the mesh to successfully complete a given analysis.

The MARC procedure to define the new mesh and to transfer the state of the old mesh to the new mesh is called rezoning. One increment is required to perform this definition of the new mesh using a complete set of input options, as defined in the following sections of this chapter.

The rezoning capability is available for the following element types:

- Continuum 2D and 3D displacement elements (except semi-infinite).
- Shell elements 22 and 75.
- Herrmann elements 80-84 using techniques defined under the SECTIONING option.
- Heat transfer continuum elements.

5

Rezoning Options

When you insert the REZONE option into a typical data setup of a problem, the program is able to read a selection of the rezoning option format to control the rezoning steps. These rezoning option formats are described on the following pages. These options must follow immediately after a load incrementation CONTINUE option. If you do not want any rezoning steps, the program reinterprets the input as load incrementation data.

You can select as many rezoning steps in one increment as are needed. Every rezoning step is defined by the data starting with the REZONE option and ending with the CONTINUE option. The complete set of rezoning steps that form a complete rezoning increment is terminated by the END REZONE option. The rezoning input is followed by normal load incrementation data or again by rezoning data.

■ REZONE

Specify Rezoning Input

Description

This option starts the rezoning input and should be present in the rezoning input data. If this option is not encountered, no rezoning step is performed and the option read in is interpreted as a history definition option.

This option must be repeated for every rezoning step; if the rezoning increment consists of more than one step, this option must be repeated.

Format

Format		Data	
Fixed	Free	Type	Entry
1-6	1st	A	Enter the word REZONE.
11-15	2nd	I	Enter 1 if the total displacements in subsequent increments are to be printed with respect to the original configuration; otherwise, they are with respect to the updated coordinate position defined in this rezoning step. For the regions modeled using Mooney, Ogden, or powder materials, this should be set to 1.

■ SECTIONING

Define Sections for Rezoning

Description

This option allows rezoning of parts of the mesh. Three input definitions are allowed for specifications of the mesh part to be rezoned. Only one of the following sectioning options can be used during a rezoning step definition.

A. Selection by element numbers – a list of element numbers must be supplied.

B. Selection by element type – the element type must be supplied.

C. Selection by material type – the material type must be supplied.

If Herrmann elements are to be used in a rubber rezoning, they should be selected using method B described above. Only Herrmann element types 80-84 and 118-120 can be rezoned.

Note: Only one of the above sectioning options can be used during a rezoning step definition.

If the sectioning option is not used, the complete mesh is taken into account during the current rezoning step which can cause problems. If discontinuities in material type or element thickness are present, not all element variables are continuous across element boundaries. In that case, SECTIONING should be used to divide the regions. If continuum and shell elements are to be rezoned, two rezoning steps should be taken and the SECTIONING option used to separate the element types.

CAUTION: If an analysis has both Herrmann elements and displacement elements and a rezoning step is to be performed, it is necessary to have double nodes at the interface and to tie all the displacement degrees of freedom. See note on COORDINATE CHANGE option on definition of coordinate points

Format

Format		Data	
Fixed	**Free**	**Type**	**Entry**
1st data block			
1-10	1st	A	Enter the word SECTIONING.

Format Fixed	Format Free	Data Type	Entry
2nd data block			
1-5	1st	I	Enter the number of lists of elements to be given in data block 3. If either option B or C are used, enter 0.
6-10	2nd	I	If a particular element type is selected enter the element type number to be selected.
11-15	3rd	I	If a particular material type is selected enter the material type to be selected.
3rd data block			
The 3rd data block is repeated once for each list of elements.			
			Enter a list of elements to be taken into account during this rezoning step.

■ CONNECTIVITY CHANGE

Define or Change Connectivity

Description

This option provides the possibility of changing the connectivity of a number of elements or to delete and/or add some elements. If elements are added, you must ensure that all the other element quantities (for example, GEOMETRY) are initialized for this element number in the model definition data and that the element number to be added is smaller than the maximum number of elements given on the SIZING parameter. If elements are deleted, you must enter element type 0 in this option.

CAUTION: If this option is used, the bandwidth of the system matrix might change and this makes a renewed calculation of the bandwidth necessary. This is automatically done by the program.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-19	1st	A	Enter the words CONNECTIVITY CHANGE.
2nd data block			
1-5	1st	I	Enter the number of elements to be change with this option (optional).
6-10	2nd	I	Enter the file number for input of connectivity. Defaults to input.
11-15	3rd	I	Set to 1 to suppress printing of element connectivity list during this option.
3rd data block			
One data block per element.			
1-5	1st	I	Element number.
6-10	2nd	I	Element type. Enter 0 if this element is to be deleted. In the latter case, no nodes have to be read in for this option.
11-15	3rd	I	First nodal point.
16-20	4th	I	Second nodal point.
etc.		etc.	Repeat until all nodes for this element type are read in. Continuation, if necessary, is in format 16I5.

■ GEOMETRY CHANGE

Specify New Geometry

Description

This option can be used to specify the GEOMETRY model definition data for changed or added elements.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-15	1st	A	Enter the words GEOMETRY CHANGE.
2nd data block			
1-5	1st	I	Number of distinct sets of element geometries to be input (optional).
6-10	2nd	I	Enter file number for input of geometry data. Defaults to input.
3rd data block			
Element geometries. The third and fourth data blocks should be entered as pairs; one for each element set.			
1-10	1st	E	EGEOM1
11-20	2nd	E	EGEOM2
21-30	3rd	E	EGEOM3
31-40	4th	E	EGEOM4
41-50	5th	E	EGEOM5
51-60	6th	E	EGEOM6
			See *Volume B: Element Library* for the meaning of EGEOM1, etc., for each element type.
4th data block			
			Enter a list of elements to which the above geometry is applied.

■ ORIENTATION CHANGE

Redefine Orientation

Description

This option allows redefinition of the internal material orientation data for the new rezoned mesh. See the discussion of the ORIENTATION model definition option in Chapter 3.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words ORIENTATION CHANGE.
2nd data block			
1-5	1st	I	Enter the number of orientation angle data sets to follow.
6-10	2nd	I	Unit number for input.
Data blocks 3 and 4 are repeated as pairs, once for each data set.			
3rd data block			
1-10	1st	A	Enter one of the following to specify orientation angle type: EDGE 1-2 EDGE 2-3 EDGE 3-4 EDGE 3-1 EDGE 4-1 XY PLANE YZ PLANE ZX PLANE XU PLANE YU PLANE ZU PLANE UU PLANE UORIENT 3D ANISO
11-20	2nd		Enter orientation angle.

Format Fixed	Format Free	Data Type	Entry	
For XU PLANE, YU PLANE, ZU PLANE, UU PLANE, and 3D ANISO, complete the following:				
21-30	3rd	F	1	components of user vector 1 w.r.t. global coords.
31-40	4th	F	2	
41-50	5th	F	3	
For UU PLANE, 3D ANISO, complete the following:				
51-60	6th	F	1	components of user vector 2 w.r.t. global coords.
61-70	7th	F	2	
71-80	8th	F	3	

4th data block

Enter a list of elements associated with this orientation angle.

■ ISOTROPIC CHANGE

Redefine Properties for Isotropic Materials

Description

This option allows redefinition of isotropic material properties for the new rezoned mesh. Note that the material id used must not be greater than the maximum material number used in the original mesh.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the word ISOTROPIC CHANGE.
2nd data block			
1-5	1st	I	Enter the number of sets of isotropic material data to follow
6-10	2nd	I	Enter the unit number for input. Defaults to input format.

Data blocks 3, 4, 5, and 6 are entered as a set; repeat once for each set of isotropic material data.

3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing TEMPERATURE EFFECTS and WORK HARD options data.
6-15	2nd	A	Enter one of the following yield criteria: VON MISES – Von Mises (Default) LIN MOHRC – Linear Mohr-Coulomb PBL MOHRC– Parabolic Mohr-Coulomb BUY MOHRC– Buyukozturk Concrete Model NORM ORNL– Normal ORNL CRMO ORNL– 2-1/4 Cr-Mo ORNL

Format Fixed	Format Free	Data Type	Entry
			REVP ORNL – Reversed Plasticity ORNL
			ARST ORNL – Full alpha reset ORNL
			GEN-PLAST – Generalized Plasticity Model
			VISCO PLAS– Viscoplastic model.
16-25	3rd	A	Enter one of the following hardening rules:
		I	ISOTROPIC – Isotropic hardening (Default)
			KINEMATIC – Kinematic hardening
			COMBINED – Combined (isotropic/kinematic) hardening
26-30	4th		Not used; enter 0.
31-35	5th	I	Enter 1 to turn on concrete cracking.

4th data block

The data entered in the following data blocks should be the values at the lowest temperature expected during an analysis, not necessarily at the stress-free temperature.

1-10	1st	F	Young's modulus.
11-20	2nd	F	Poisson's ratio.
21-30	3rd	F	Mass density (stress analysis).
31-40	4th	F	Coefficient of thermal expansion.
41-50	5th	F	Equivalent (Von Mises) tensile yield stress. (For Mohr-Coulomb behavior, this is at zero hydrostatic stress.)
51-60	6th	F	For ORNL yield criteria, equivalent 10th cycle tensile yield stress. (For Mohr-Coulomb yield criteria, α-β parameter.)
61-70	7th	F	Cost of material per unit volume (optional).
71-80	8th	F	Cost of material per unit mass (optional).

Format Fixed	Format Free	Data Type	Entry
5th data block			
Necessary only in a coupled thermal-stress analysis.			
1-10	1st	F	Thermal conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis).
31-40	4th	F	Not used; enter 0.
41-50	5th	F	Emissivity for radiating cavities.
6th data block			
			Enter a list of elements associated with this material. (Do not enter composite elements which use this material in its layers.)

■ ORTHOTROPIC CHANGE

Redefine Properties for Orthotropic Materials

Description

This option allows redefinition of orthotropic material properties for the new rezoned mesh. Note that the material id used must not be greater than the maximum material number used in the original mesh.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words ORTHOTROPIC CHANGE.
2nd data block			
1-5	1st	I	Enter the number of sets of orthotropic material data to follow.
6-10	2nd	I	Enter the unit number for input. Defaults to input format.

Data blocks 3, 4, 5, 6, 7, and 8 are entered as a set; repeat once for each set of isotropic material data.

3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing ORTHO TEMP, WORK HARD options data and user subroutines.
6-15	2nd	A	Enter one of the following yield criteria: VON MISES – Von Mises (Default) NORM ORNL– Normal ORNL CRMO ORNL– 2-1/4 Cr-Mo ORNL REVP ORNL – Reversed Plasticity ORNL ARST ORNL – Full alpha reset ORNL GEN-PLAST – Generalized Plasticity Model VISCO PLAS– Viscoplastic model.

Format Fixed	Format Free	Data Type	Entry
16-25	3rd	A	Enter one of the following hardening rules:
		I	ISOTROPIC – Isotropic hardening (Default)
			KINEMATIC – Kinematic hardening
			COMBINED – Combined (isotropic/kinematic) hardening
26-30	4th	I	Enter 1 to call user subroutines ANELAS, HOOKLW, ANPLAS, ANEXP, ANKOND, and ORIENT.

4th data block

The data entered in the following blocks should be the values at the lowest temperature expected during an analysis, not necessarily at the stress free temperature.

These values are with respect to the user coordinate (1, 2, 3) system.

1-10	1st	F	E_{11} Young's modulus.
11-20	2nd	F	E_{22} Young's modulus.
21-30	3rd	F	E_{33} Young's modulus.
31-40	4th	F	ν_{12} Poisson's ratio.
41-50	5th	F	ν_{23} Poisson's ratio.
51-60	6th	F	ν_{31} Poisson's ratio.
61-70	7th	F	ρ Mass density (stress analysis.)

5th data block

1-10	1st	F	G_{12} Shear modulus.
11-20	2nd	F	G_{23} Shear modulus.
21-30	3rd	F	G_{31} Shear modulus.
31-40	4th	F	α_{11} Coefficients of thermal expansion.
41-50	5th	F	α_{22} Coefficients of thermal expansion.
51-60	6th	F	α_{33} Coefficients of thermal expansion.
61-70	7th	F	Cost of material per unit volume (optional).
71-80	8th	F	Cost of material per unit mass (optional).

Format Fixed	Format Free	Data Type	Entry
6th data block			
1-10	1st	F	Equivalent (von Mises) yield stress. Default: 10^{20}
11-20	2nd	F	For ORNL, 10th cycle equivalent yield stress.
21-30	3rd	F	YRDIR1 Direct stress anisotropic yield ratios (Hill).
31-40	4th	F	YRDIR2 Default: YRDIR1=YRDIR2=YRDIR3=1.0
41-50	5th	F	YRDIR3
51-60	6th	F	YRSHR1 Shear stress anisotropic yield ratios (Hill).
61-70	7th	F	YRSHR2 Default: YRSHR1=YRSHR2=YRSHR3=1.0
71-80	8th	F	YRSHR3
7th data block			
Necessary only in a coupled thermal-stress analysis.			
1-10	1st	F	K_{11} Thermal conductive.
11-20	2nd	F	K_{22} Thermal conductive.
21-30	3rd	F	K_{33} Thermal conductive.
31-40	4th	F	ρ Mass density.
41-50	5th	F	Specific heat.
51-60	6th	F	R_{11} If Joule heating analysis, resistive.
61-70	7th	F	R_{22} If Joule heating analysis, resistive.
71-80	8th	F	R_{11} If Joule heating analysis, resistive.
8th data block			
Only required in a coupled analysis with radiating cavities.			
1-10	1st	F	Emissivity.
9th data block			
			Enter a list of elements associated with this material. (Do not enter composite elements which use this material in their layers.)

■ HYPOELASTIC CHANGE

Redefine Properties for Hypoelastic Materials

Description

This option allows redefinition of hypoelastic material properties for the new rezoned mesh. Note that the material id used must not be greater than the maximum material number used in the original mesh.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words HYPOELASTIC CHANGE.
2nd data block			
1-5	1st	I	Enter the number of hypoelastic material data sets to follow.
6-10	2nd	I	Enter the unit number for input. Defaults to input format.

Data blocks 3, 4, and 5 are entered as a set; once for each hypoelastic material data set.

3rd data block			
1-5	1st	I	Material identification number (1, 2, 3, etc.) for cross-referencing TEMPERATURE EFFECTS option data and user subroutines.
6-10	2nd	I	Enter 1 to call user subroutines ANEXP and ORIENT.
11-15	3rd	I	Flag to use user subroutine HYPELA2. Enter 1 to pass in deformation gradient (F) and rotation (R). Enter 2 to pass in deformation gradient (F) and stretch ratios (λ). Enter 3 to pass in F, R, and λ.

Format Fixed	Format Free	Data Type	Entry
4th data block			
1-10	1st	F	Mass density (stress analysis).
11-20	2nd	F	Coefficient of thermal expansion.
21-30	3rd	F	Conductivity.
31-40	4th	F	Specific heat.
41-50	5th	F	Resistivity.
51-60	6th	F	Mass density (heat transfer analysis).

In a coupled thermal-stress analysis, the thermal material model defaults to isotropic heat transfer behavior.

5th data block

Enter a list of elements using this material model. (Do not enter composite elements using this material in a layer.)

■ MOONEY CHANGE

Redefine Data for Mooney-Rivlin Materials

Description

This option allows redefinition of the Mooney-Rivlin material data for the new rezoned mesh. Note that the material id used must not be greater than the maximum material numbers used in the original mesh.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words MOONEY CHANGE.
2nd data block			
1-5	1st	I	Enter the number of data sets to follow.
6-10	2nd	I	Unit number for data input. Defaults to input format.

Data blocks 3, 4, and 5 are entered as a set, once for each set of Mooney-Rivlin materials.

3rd data block			
1-5	1st	I	Material identification number.
6-10	2nd	I	Not used; enter 0.
4th data block			
1-10	1st	F	C_{10}, Mooney-Rivlin constants.
11-20	2nd	F	C_{01}
21-30	3rd	F	ρ, mass density (stress analysis).
31-40	4th	F	α, coefficient of thermal expansion.
41-50	5th	F	C_{11}, higher order constants.
51-60	6th	F	C_{20}
61-70	7th	F	C_{30}

Format Fixed	Format Free	Data Type	Entry
5th data block			
Only necessary in a coupled thermal-stress analysis.			
1-10	1st	F	Conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis).
In a coupled thermal-stress analysis, this material model defaults to isotropic heat transfer behavior.			
6th data block			
			Enter a list of element numbers associated with this particular elastomeric material.

■ OGDEN CHANGE

Redefine Data for Ogden Material Model

Description

This option allows redefinition of the data associated with the Ogden model for incompressible and nearly incompressible rubber material.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the words OGDEN CHANGE.
2nd data block			
1-5	1st	I	Enter the number of sets of Ogden material data to follow (optional).
6-10	2nd	I	Enter the logical unit number for input. Defaults to input format.

Data blocks 3 and 4 are repeated for each data set.

3rd data block			
1-5	1st	I	Enter the material identification.
6-10	2nd	I	Enter the number of terms that defines the strain energy function.
4th data block			
1-10	1st	F	Enter the bulk modulus, default is such that material is incompressible.
11-20	2nd	F	Enter the mass density.
21-30	3rd	F	Enter the coefficient of thermal expansion.

Format Fixed	Format Free	Data Type	Entry
5th data block			
Only necessary in a coupled thermal-stress analysis.			
1-10	1st	F	Conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis). **Note:** In a coupled thermal-stress analysis, the thermal material model defaults to isotropic heat transfer behavior.
Data block 6 is repeated once for each term specified in the 3rd data block.			
6th data block			
1-10	1st	F	Enter the modulus.
11-15	2nd	F	Enter the power.
7th data block			
			Enter a list of element numbers associated with this particular elastomeric material.

■ FOAM CHANGE

Redefine Data for Foam Material Model

Description

This option allows redefinition of the data associated with the Foam model for highly compressible rubber material.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words FOAM CHANGE.
2nd data block			
1-5	1st	I	Enter the number of sets of foam material data to follow (optional).
6-10	2nd	I	Enter the logical unit number for input. Defaults to input format.

Data blocks 3, 4, 5, 6, and 7 are repeated for each data set.

3rd data block			
1-5	1st	I	Enter the material identification.
6-10	2nd	I	Enter the number of terms that defines the strain energy function.
4th data block			
1-10	1st	F	Not used; enter 0.
11-20	2nd	F	Enter the mass density.
21-30	3rd	F	Enter the coefficient of thermal expansion.

Format Fixed	Format Free	Data Type	Entry

Data block 5 is repeated once for each term specified in the 3rd data block.

5th data block

Only necessary is a coupled thermal-stress analysis.

1-10	1st	F	Conductivity.
11-20	2nd	F	Specific heat.
21-30	3rd	F	Mass density (heat transfer analysis).

Note: In a coupled thermal-stress analysis, the thermal material model defaults to isotropic heat transfer behavior.

6th data block

1-10	1st	F	Enter the modulus (μ_n).
11-15	2nd	F	Enter the power for deviatoric behavior (α_n).
21-30	3rd	F	Enter the power for volumetric behavior (β_n).

7th data block

Enter a list of element numbers associated with the particular elastomeric material.

■ GAP DATA CHANGE

Redefine Gap Data

Description

This option allows redefinition of gap data for the new rezoned mesh.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-10	1st	A	Enter the words GAP DATA CHANGE.
2nd data block			
1-5	1st	I	Number of sets of gap data to be input.
6-10	2nd	I	Unit number for reading data. Defaults to input format.

Data blocks 3, 4, and 5 are entered as a set; once for each set of gap data materials.

3rd data block			
1-10	1st	F	For a fixed direction gap, enter the gap closure distance U_{cl}. For a true distance gap, enter the minimum distance between end points |d|.

Note: If d>0, the two end points are never closer than a distance |d| apart.
If d<0, the two end points are never farther apart than |d|.

11-20	2nd	F	μ, the contact coefficient of friction.
21-30	3rd	F	$K_{GAP'}$, the elastic stiffness of the closed gap in the contact direction. Default: gap is rigid when closed.
31-40	4th	F	$K_{FRICTION'}$, the elastic stiffness of the closed gap in the friction direction. Default: gap is rigid when closed.

Format		Data	
Fixed	**Free**	**Type**	**Entry**
41-50	5th	F	User-supplied momentum ratio for first gap node. Default: MARC calculates this ratio internally.
51-60	6th	F	User-supplied momentum ratio for fourth gap node. Default: MARC calculates this ratio internally.
61-65	7th	I	Enter 1 for true distance gap. Default is 0: fixed direction gap.
66-70	8th	I	Enter 1 if gap is closed during increment 0. Default is 0: gap is open during increment 0.
4th data block			
			Enter a list of gap elements associated with this set of gap data.

■ COORDINATE CHANGE

Redefine Node Coordinates

Description

This option allows redefinition of the coordinates of a number of nodes to redefine the mesh.

If the UPDATE parameter or a rubber rezoning analysis using element types 80–84 is used, the new coordinates to be read in by this option are the updated coordinates. The program itself divides this into new coordinates and new total displacements. If UPDATE is not used, the coordinates to be read in are the actual new coordinates without considering the displacements.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-17	1st	A	Enter the words COORDINATE CHANGE.
2nd data block			
1-5	1st	I	Enter the maximum number of coordinate directions to be read in per node; Defaults to the number of coordinates per node.
6-10	2nd	I	Enter the number of nodal points for which the new coordinates are read in this block (optional).
11-15	3rd	I	Enter the unit number to read coordinates. Defaults to input.
16-20	4th	I	Enter 1 to suppress printout of nodal coordinate list.
3rd data block			
One data block per nodal point.			
1-5	1st	I	Nodal point number.
6-15	2nd	F	Coordinate 1.
16-25	3rd	F	Coordinate 2.
26-35	4th	F	Coordinate 3.
etc.		etc.	Number of coordinates per node to read in.

Input six coordinates per data block; CONTINUE options in format 6E10.

■ UFRORD

Use Subroutine UFRORD

Description

This option calls the user subroutine UFRORD to generate or modify nodal coordinates. See *Volume D: User Subroutines and Special Routines*. The option can be repeated as often as necessary. If coordinates must be modified, this option should follow the COORDINATE CHANGE option. If coordinates must be defined, this option can be used alone.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-6	1st	A	Enter the word UFRORD.
2nd data block			
			Enter a list of nodes for which subroutine UFRORD is to be called.

■ MOVE

Redefine Node Coordinates

Description

This option allows you to apply a uniform translation to a list of elements, redefining the coordinates of all nodes in the elements. This translation is applied to the last value of the coordinates, excluding any displacement. This option can be repeated as often as necessary.

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-10	1st	A	Enter the word MOVE.
2nd data block			
1-5	1st	I	Enter the number of lists to be entered.
3rd data block			
1-10	1st	F	The amount of translation in the first coordinate direction.
11-20	2nd	F	The amount of translation in the second coordinate direction.
21-30	3rd	F	The amount of translation in the third coordinate direction.
4th data block			
This data block is entered once for each list.			
			Enter a list of element numbers to which the above translation is applied.

■ CONTACT CHANGE

Change Surface Contact after Rezoning

Description

This option allows changes to a deformable surface definition (rezoned mesh) in 2D and 3D contact problems after rezoning occurs. The rigid surface definition cannot be changed. It also allows changes to friction type, choice of Coulomb friction calculation, maximum number of separations in each increment, suppression of splitting of increment. In addition, relative sliding velocity for sticking condition, contact tolerance, separation force, as well as average and cut-off strain rates in rigid-plastic analysis, can also be changed.

Format

Format Fixed	Format Free	Data Type	Entry
1st data block			
1-14	1st	A	Enter the words CONTACT CHANGE.
2nd data block			
1-5	1st	I	Number of surfaces to be defined. (Must be same value as before rezoning.)
6-10	2nd	I	Maximum number of entities to be created for any surface. (Same value as before rezoning.)
11-15	3rd	I	Upper bound to the number of nodes that lie on the periphery of any deformable surface. (Same value as before rezoning.)
16-20	4th	I	Friction type 0: No Friction 1: Shear Friction 2: Coulomb 3: Shear Friction for Rolling 4: Coulomb Friction for Rolling 5: Stick-slip Coulomb Friction (Same value as before rezoning.)
21-25	5th	I	Enter 1 for the calculation of Coulomb friction based on nodal force instead of nodal stress. Default is 0. (Same value as before rezoning.)

Format Fixed	Format Free	Data Type	Entry
26-30	6th	I	Maximum of separations allowed in each increment. Default is 9999.
31-35	7th	I	Enter 1 for the suppression of the splitting of an increment in fixed time step procedure. Enter 2 for adaptive time step procedure. Default is 0. Enter 3 to use contact procedure which does not require increment splitting.
36-40	8th	I	Enter 1 for the interference kinematic check. Enter 2 to suppress bounding box checking (this might eliminate penetration, but will slow down the solution). Enter 3 to not reset NCYCLE=0; this speeds up the solution but might result in instabilities. Enter 4 for analytical surfaces only; check for separation only when convergent solution, similar to PWL approach.
41-45	9th	I	Control separations within an increment. When 0 is entered, if the force on a node is greater than the separation force, the node separates and an iteration occurs. When 1 is entered, if a node which was in contact at the end of the previous increment has a force greater than the separation force, the node does not separate in this increment, but separates at the beginning of the next increment. When 2 is entered, if a new node comes into contact during this increment, it is not allowed to separate during this increment (prevents chattering). When 3 is entered, both (1) and (2) above are in effect.
46-50	10th	I	Parameter governing normal direction/thickness contribution of shell (ISH). Enter 0 – Check Node Contact with top and bottom surface Enter 1 – Nodes only come into contact with bottom layer

Format Fixed	Format Free	Data Type	Entry
			Enter 2 – Nodes only come into contact with bottom layer and ignore shell thickness
			Enter -1 – Nodes only come into contact with top layer
			Enter -2 – Nodes only come into contact with top layer and ignore shell thickness
51-55	11th	I	Enter 1 to reduce printout of surface definition.
56-60	12th	I	Enter 1 to have separation based upon stresses not forces.
3rd data block			
1-10	1st	F	For friction types 1, 2, 3, or 4 enter the relative sliding velocity between surfaces below which sticking is simulated (RVCNST). Default = 1.0.
			For friction type 5, enter the slip-to-stick transition region (β); Default is 1.e-6.
11-20	2nd	F	Distance below which a node is considered touching a surface (ERROR). Leave blank if you want MARC to calculate it. This number is also used to divide splines. If splines are used, this must be defined.
21-30	3rd	F	Average strain rate. Used in rigid-plastic analysis only to start a problem. Default is 1.e-4
31-40	4th	F	Cutoff strain rate below which flow stresses drop linearly to zero. Used in rigid-plastic analysis only. Default is 1.e-12.
41-50	5th	F	Separation force above which a node separates from a surface (FNTOL). Default is the maximum residual force.
			If the 12th field of the second line is 1, enter the separation stress.
51-60	6th	F	Contact tolerance BIAS factor. (0-1)

Format		Data	
Fixed	**Free**	**Type**	**Entry**
61-70	7ht	F	For stick-slip model, enter the friction coefficient multiplier (α). Defaults to 1.05
71-80	8th	F	For stick-slip model, enter the friction force tolerance (e). Defaults to 0.05.

For two- and three-dimensional contact problems

The data blocks 4 and 5 are repeated once for each data set.

4th data block

1-5	1st	I	Surface number.
6-10	2nd	I	Number of sets of geometrical data to be input for this surface. Enter 0 if flexible surface.

The 5th data block is only necessary if the surface is deformable; if the surface is rigid, no additional data is required.

5th data block

Enter a list of elements of which the surface is comprised.

■ PRINT CHOICE

Select Print Settings

Description

This option allows control of the output from the program. The data given here remains in control until a subsequent PRINT CHOICE set is inserted – such a set can be included with either the model definition or the history definition data blocks.

The default values print all elements and all nodes. Element quantities are printed at each integration point or at the centroid only, depending on whether ALL POINTS parameter was used. For shells, only the extreme fibers are output, plus layers where the inelastic strains or state variables are nonzero. For beams, fibers with inelastic strains or nonzero state variables are printed. In addition, section forces are given for these elements. This option also allows debug output of certain items. The default is no debug printout.

All of the above defaults are reset by the PRINT CHOICE option.

The PRINT CHOICE option has no effect on the post processor file (see Chapter 3).

Format

Format		Data	
Fixed	Free	Type	Entry
1st data block			
1-12	1st	A	Enter the words PRINT CHOICE.
2nd data block			
1-5	1st	I	Number of sets of first and last element numbers to be printed (maximum 10).
6-10	2nd	I	Number of sets of first and last node numbers to be printed (maximum 10).
11-15	3rd	I	Number of integration points to be printed in each element (not used if ALL POINTS parameter not flagged).
16-20	4th	I	Number of layers to be printed. This is for beams and shells only, and overrides the default described above.

Format Fixed	Format Free	Data Type	Entry
21-25	5th	I	Increments between printout. Default is print every increment.
26-30	6th	I	Enter 1 for complex nodal quantities to be output as magnitude and phase; otherwise, real and imaginary components are given.
31-35	7th	I	Debug print flag. Enter a nonzero value and use data block 7.
36-40	8th	I	Log File Flag: Enter unit number to which log file is to be written.

3rd data block

Include only of the first field of 2nd data block is not zero.

1-5	1st	I	First element in first set.
6-10	2nd	I	Last element in first set.
11-15	3rd	I	First element in second set.
16-20	4th	I	Last element in second set.
Etc.			Etc. in I5 format.

4th data block

Include only if the second field of 2nd data block is not zero.

1-5	1st	I	First node in first set.
6-10	2nd	I	Last node in first set.
11-15	3rd	I	First node in second set.
16-20	4th	I	Last node in second set.
Etc.			Etc. in I5 format.

5th data block

Include only if the third field of 2nd data block is not zero.

Enter the list of integration points to be printed in (16I5) format (number of entries given in the third field of cards series 2). This is only used if ALL POINTS parameter is flagged. Be careful with analyses with several different element types.

Format Fixed	Format Free	Data Type	Entry
6th data block			
Include only if the fourth field of the 2nd data block is not zero.			
			Enter the list of shell or beam fibers to be printed in (16I5) format. This over-rides the program default, so that you should be aware that you do not unintentionally miss plasticity or creep printout.
7th data block			
Include only if the seventh field of the 2nd data block is not zero			
1-5			Enter debug plot code. See the PRINT parameter.

■ CONTINUE

End Rezoning Input

Description

This option closes the input for this rezoning step and gives control back to the executing routine.

Format

Format		Data	
Fixed	Free	Type	Entry
1-8	1st	A	Enter the word CONTINUE.

■ END REZONE

End Input for Rezoning Increment

Description

This option closes the input for the rezoning increment. It must follow the CONTINUE option of the last defined rezoning step of the increment.

If this option is read, control is given back to the main control routine of the program, and the next input is interpreted as either mesh plot data or as history definition data.

Format

Format		Data	
Fixed	Free	Type	Entry
1-10	1st	A	Enter the words END REZONE.

A Program Messages

MARC Exits

The MARC program provides an exit number when execution terminates, unless a system abort interrupts execution first. These exits are grouped as follows:

Exit Number	Classification
1-1000	Simple data errors detected during initial data input (before END OPTION).
1001-2000	Errors detected during stiffness assembly or load distribution.
2001-3000	Errors detected during solution of stiffness matrix or boundary condition or constraint application.
3001-4000	Exits during load incrementation control and output. Most normal exits are in this range.
4001-5000	System I/O Errors.
5001-6000	Errors detected during adaptive meshing.

The following subsections list the current exit numbers with their corresponding description and probable cause.

Exit Numbers 1-1000

Exit Number	Explanation
1	The number of elements associated with a distributed load exceeds the maximum number given on the SIZING, DIST LOADS, or FLUXES parameters. This usually happens in the history definition when a new distributed load is added, or the number of elements is increased. Increase the value on the parameter. This cannot be done in a restarted analysis, it must be done beginning with increment zero.
2	A line being read as a parameter line is unidentifiable. This can be caused by illegal data or a mistyped line.

Exit Number	Explanation
6	With the options chosen, the analysis cannot be run in the memory space available. Either increase work space on the SIZING parameter or modify the input. Adding the ELSTO parameter stores element data onto disk, reducing the amount of memory necessary. Reducing the number of layers associated with beam and shell elements reduces the amount of memory necessary. Switching to reduced integration elements reduces the amount of memory necessary. Be sure that the best solver has been chosen for your problem and that the bandwidth has been minimized.
7	Normal finish caused by setting STOP parameter. Try to have work space requested on the SIZING parameter to be larger than work space mentioned in the output file, remove STOP parameter, and resubmit the analysis.
9	Too many points (more than 60) used in the BEAM SECT parameter. Either the number of branches in a beam section is large, or the number of points per branch is large. Reduce the relevant number(s).
11	Not enough space to store stiffness matrix. Add bandwidth optimizer if this has not already been done. Increase memory available on SIZING option. Change to sparse solver or iterative solver if possible.
12	Normal exit when mesh display run is completed and no subsequent analysis has been requested.
13	Data errors have been detected in data input for model definition, history definition, MESH2D, or REZONE option. Please refer to output for location of error. Likely causes are misspelled keywords or mistyped lines.
23	The wave front optimization scheme has lost continuity. This indicates that the mesh is not continuous. Two or more distinct bodies are defined in the same problem, and the wave front renumbering scheme cannot handle this case. Switch to the Sloan optimizer.
24	No elements, springs, or ties exist; the analysis cannot be meaningful. Check your input. It is possible that the input file is empty.

Exit Number	Explanation
25	Internal renumbering scheme for Lagrange multiplier does not converge. Check if there are Herrmann elements where all displacement degrees of freedom are prescribed. Try a different optimization scheme. Otherwise MARC system error; consult MARC analyst.
26	No element type chosen. Include ELEMENTS or SIZING parameter.
30	Selected surface entity not allowable in CONTACT option. Check input data.
31	A surface has more entities than the maximum declared. Increase the maximum number of entities on the CONTACT option.
32	Circle segment data not consistent. Likely cause is that radii calculated are not consistent with the given input. Check input.
33	There are more boundary nodes than the upper bound declared. Increase the upper bound to the number of boundary nodes on the CONTACT option.
34	Error in CONTACT option; either no deformable body has been defined, or there are rigid surfaces defined before deformable bodies. There must be at least one deformable body on the CONTACT model definition block. All deformable bodies should be specified before rigid bodies.
35	During initial surface approach, velocities were set to zero before initial contact occurred.
36	Data error in supplying the surface profile for a 2D deformable body. Check that elements specified for the body form a continuous path. Make sure that there are no inside out or upside down elements.
37	Friction calculations with the CONTACT option are being attempted without the relative sliding velocity below which sticking is simulated being defined. Input the relative sliding velocity below which sticking is simulated on the CONTACT option.
38	The CONTACT option is being exercised without a time increment being defined. Enter the time period using TIME STEP, AUTO TIME, AUTO STEP, AUTO INCREMENT, DYNAMIC CHANGE, or TRANSIENT history definition option.
39	Attempt to change contact data during a REZONE increment. Only the elements associated with a deformable body can be changed during rezoning. Other changes must be done through the MOTION CHANGE or CONTACT TABLE history definition option.

Exit Number	Explanation
40	During the approach stage of a contact analysis, the rigid surfaces did not contact the deformable body within 1000 iterations with a time step of 1.0. Your initial surface velocity is too small or has a wrong direction. Change the initial surface velocity on the CONTACT option. Reduce the velocity, if necessary, using the MOTION CHANGE option.
41	Error in determining normal direction to contact surface; potential conflict with boundary condition. Try to remove boundary conditions on nodes that come into contact. It is better to use symmetry surfaces.
42	The number of subdivisions of a 3D arc must be greater than 1. Change your input on the CONTACT option.
43	The number of data points of a 3D curve must be less than or equal to the number of subdivisions. Change your input on the CONTACT option.
44	The number of data points on a polyline must be equal to the number of subdivisions minus 1. Change your input on the CONTACT option.
45	The number of spline data points should be greater than 3 and less than 30. If the number is less than 3, convert the spline segment into a line. If the number of points is greater than 30, consider breaking the spline segment into multiple spline segments.
46	The number of Bezier data points should be greater than 3. If the number of points is 2, treat entity as a 4-node patch (type 7).
47	The input data points are either too close or colinear. Could not convert input to proposed surface type. Check input on the CONTACT option.
48	The current patch is too small to find its normal vector. Check input of rigid contact surfaces. It is also possible that deformation has caused a surface of a deformable body to become very small.
51	END parameter is missing. This must be included in input file.
52	Upper bound of the number of boundary nodes is too small for higher order elements. Increase the upper bound to the number of boundary nodes on the CONTACT option.
53	Upper bound of the number of entities per body is too small for a deformable body.
54	Upper bound of the number of entities per body is too small for the body.
55	The number of nodes on the SIZING parameter must be increased to accommodate beam-beam contact.

Exit Number	Explanation
56	The number of elements on the SIZING parameter must be increased to accommodate beam-beam contact.
57	The upper bound on the number of boundary nodes must be increased to accommodate beam-beam contact.
58	The upper bound on the number of entities must be increased to accommodate beam-beam contact.
67	MARC password security has determined that you are not allowed to run on this machine; possible reasons are: **1.** The choice of machine or current OS level is not compliant with the passwords. **2.** The file `license.dat` in the MARC subdirectory `marcdir` contains wrong information. **3.** The MARC subdirectory `marcdir` has wrong access permissions.
68	The MARC password security has determined that you are not allowed to run on this machine; possible reasons are: **1.** The license has expired. **2.** The file `license.dat` in the MARC subdirectory `marcdir` contains wrong information.
69	The MARC password security has determined that you are not allowed to run on this machine; the most likely reason is that there are too many jobs running simultaneously.
76	Incompatible view factor data has been read in. The number of segments read in is not consistent with the number required.
77	Error in restart run. There is an illegal change in parameter or model definition options in the input file of the restart run, such that the core allocation has changed.
100	Viewpoint chosen for plotting causes excessive distortion. The probable cause is that the viewpoint chosen is either within the body or too close to it.
666	The Powder model data input is incomplete. Check input on POWDER option.

Exit Numbers 1001-2000

Exit Number	Explanation
1001	Connectivity exceeded at the node given in the message: `MORE THAN MAXNP JOINED TO NODE...` during in-core assembly of elements. This aborts the program at that point. If this occurs during a contact analysis, try to activate single sided (K3 style) contact on the CONTACT option. Otherwise MARC system error; consult MARC analyst.
1002	Connectivity exceeded at the node given in the message: `MORE THAN MAXNP JOINED TO NODE...` during out-of-core assembly of elements. This aborts the program at that point. If this occurs during a contact analysis, try to activate single sided (K3 style) contact on the CONTACT option. Otherwise MARC system error; consult MARC analyst.
1003	Too many nodes joined to node in forming fluid coupling matrix. MARC system error; consult MARC analyst.
1005	Errors during stiffness or mass matrix generation. The output reveals which element has a particular problem. If this occurs during the first assembly, it is due to input errors associated with the COORDINATES, GEOMETRY, or CONNECTIVITY options. If this occurs during a later increment, it is due to excessive deformation in the element. This can occur during the iterative process, so that it is not always possible to visualize the excessive deformation. Check the material behavior and the magnitude of the incremental loads.
1006	Elastic reanalysis attempted with nonzero displacement boundary conditions and boundary condition enforcement by row/column elimination. This is not possible. Remove the APPBC parameter.
1009	Error encountered in stress recovery. The output reveals which element has a particular problem. The error is usually due to excessive deformation in the element. This can occur during the iterative process, so that it is not always possible to visualize the excessive deformation. Check the material behavior and the magnitude of the incremental loads.
1021	Error in adding fluid mass to node. MARC system error; consult MARC analyst.

Exit Number	Explanation
1030	Reference vector has zero length; cannot determine preferred orientation. Check input on ORIENTATION option.
1031	Error while determining view factors in cavity. Make sure that node numbers defining cavity are in consecutive order in the input file.
1040	Maximum number of element groups exceeded while using EBE iterative solver. Increase the maximum number of groups allowed on the SOLVER option.

Exit Numbers 2001-3000

2004	The determinant of the stiffness matrix becomes zero or negative when the indicated node has been reached during Gaussian elimination during the solution process. This means that the stiffness matrix is nonpositive definite. If this happens at the start of the analysis, this condition is usually caused by failure to remove all rigid body modes. It can also be caused by incorrect material properties (for example, Poisson's ratio greater than 0.5; note that these situations can arise through temperature dependence of properties). In nonlinear cases, the structure can buckle or reach a plastic limit load. In a rubber analysis, it can also be due to the strain state being in a region where the input data for the strain energy function is invalid. In a contact analysis with friction, lack of normal forces can cause friction to be absent. If desired, the program can be forced to continue by using the PRINT parameter or the model or history definition CONTROL option. Either one of these procedures can be done upon restart. Whenever a nonpositive definite situation occurs, you must exercise caution as the resultant numerical solution can be infeasible.
2007	System error in sparse conjugate gradient solver. MARC system error; consult MARC analyst.
2008	Maximum connectivity has been exceeded during application of tying constraints. If tyings are included, check the TYING and TYING CHANGE options. This can also occur because of inconsistencies in deformable-deformable contact; switch to single sided (K3 style) contact. MARC system error; consult MARC analyst.
2009	Not enough space to convert system from nodal blocks to row format. MARC system error; consult MARC analyst.
2011	Errors encountered during application of TYING equations. Printout indicates specific problem.

Exit Number	Explanation
2012	Should not occur anymore.
2014	Search vector for eigen extraction is zero. Caused by inadequate guess vector, or asking for more eigenvalues than the system contains. Do not attempt to extract more modes than exist in the system. The number of modes is the total number of degrees of freedom minus the number of boundary conditions. Remember that contact to rigid bodies effectively apply boundary conditions to the system, hence removing potential modes.
2015	Not enough work space to do fluid/solid interface calculations. Increase the space specified on the SIZING parameter.
2016	Not enough work space to do fluid/solid interface calculations. Increase the space specified on the SIZING parameter.
2017	Not enough work space to do fluid/solid interface calculations. Increase the space specified on the SIZING parameter.
2020	Conjugate gradient iterative solver fails to converge in required number of iterations. This can be due to the fact that there are rigid body modes in the system, or that the system is numerically ill-conditioned, such as with shell or membrane structures. Possible things to do: **1.** Add preconditioner; incomplete Cholesky preferred. **2.** Increase the number of iterations. **3.** Set the number of iterations to a negative number. This forces the program to continue. If the solution is truly poor, MARC does not converge in the outer Newton-Raphson iterations.
2030	Unable to perform dynamic memory allocation for hardware provided direct sparse solver. Decrease the amount of memory requested on the SIZING option to be only the number printed on the output. If you still get this message, either increase the amount of virtual / real memory on your computer or switch to the sparse iterative solver.
2031	Error occurred during factorization process with hardware provided direct sparse solver. Switch to one of the MARC solvers. MARC system error; consult MARC analyst.

Exit Number	Explanation
2109	Mode with zero energy is found during transient modal response analysis. Probable cause is failure to ask for MODAL SHAPE option before DYNAMIC CHANGE is used. Check input file.
2201	Attempt to recover a substructure, before solving the one level higher superstructure.
2400	A node on the boundary of a deformable body tried to slide out of surface definition in a contact analysis. Either the segment of a rigid body is not large enough, in which case increase the length, or the nodal points in the area of contact have a large incremental displacement due to possible instability.

Exit Numbers 3001-4000

3001 The maximum number of increments specified on the CONTROL option has been reached. Increment zero has to be considered as an increment.

3002 Analysis has failed to converge to required convergence tolerances. One of several error conditions has been detected and the run aborted. The output specifies additional messages.

Following are the more common messages:

```
NO CONVERGENCE TO TOLERANCE IN SPECIFIED CYCLES
```

This means the stiffness approach fails to find a convergent increment. The tolerance and the maximum number of recycles are set on the CONTROL option.

This condition is most usually caused by:

1. Relative residual checking has been selected, and there are no reaction forces and/or reaction moments.
2. Relative displacement checking has been selected, and there are no incremental displacements and/or incremental rotations.
3. A too large load step.
4. A too tight tolerance.
5. The solution is reaching a buckling state.
6. A limit load or another material instability, such as cracking or material damage, is reached.

Exit Number	Explanation
	7. The solution is oscillating between two values due to contact separation. 8. The solution is oscillating between two values due to nodes sliding, 9. Friction behavior is dominant. You can do a combination of the following to improve convergence: 1. Switch to displacement checking or include cut-off force and/or moment on the CONTROL option. 2. Switch to residual checking or include cut-off displacement and/or rotation on the CONTROL option. 3. Decrease the load step or time step if using the AUTO LOAD, DYNAMIC CHANGE, or TRANSIENT options. 4. Specify tighter parameters if using the AUTO INCREMENT, AUTO TIME, or AUTO STEP options. 5. Increase the tolerances specified on the CONTROL option, or increase the maximum number of cycles. 6. If an elastic-plastic analysis, make sure that the stress-strain data includes the proper amount of work hardening. If a rubber analysis, make sure that the data for the strain energy function is appropriate for the current strain levels. 7. Increase the contact separation force on the CONTACT or CONTACT TABLE option. If using the CONTACT option, do not allow a node which comes into contact to separate during this increment. 8. Reduce the magnitude of the incremental load. 9. Reduce, if possible, the coefficient of friction on the CONTACT or CONTACT TABLE option, or increase the relative sliding velocity below which sticking is simulated on the CONTACT option.

Exit Number	Explanation

AUTO CREEP FAILS TO FIND CONVERGENT TIME STEP IN MAXIMUM CYCLES ALLOWED

This means the creep time selection has not found a step which brings stress change and strain change within their respective tolerances. The stress and strain change tolerances are specified on the CREEP model definition option or the AUTO CREEP history definition option.

This condition is most usually caused by:

1. Severe lack of equilibrium remaining from the previous load history.
2. A too large guess for initial time increment in AUTO CREEP.
3. A too large exponent when using a Norton Creep model.
4. An incorrect user subroutine CRPLAW.

You can do a combination of the following to improve convergence:

1. Introduce a zero load step before beginning the creep stage.
2. Decrease the initial time step.
3. Loosen the tolerances on the AUTO CREEP option.
4. Check programming of user subroutine CRPLAW.

MORE THAN MAXIMUM STEPS ALLOWED IN AUTO CONTROL

The total time has not been reached when using either the adaptive loading procedures AUTO CREEP, AUTO INCREMENT, AUTO TIME, or AUTO STEP. The program has been forced to take smaller time/load steps than you had anticipated to satisfy the required convergence tolerances. This can be due to too tight tolerances.

You should do one of the following:

1. Loosen the tolerances,
2. Increase the number of increments allowed during the automatic loading procedure.

3004 This is a normal exit to a MARC analysis, indicating that no additional incremental data was found and the analysis is complete.

3005 This is a normal exit to a MARC analysis when the CASE COMBIN option is used.

3006 This is a normal exit to a MARC analysis when you have requested that the restart file is being read and to either print results or create a post file.

Exit Number	Explanation
3007	A restart analysis has been requested at an increment that is not on the restart file. The previous output gave the message `RESTART DATA AT INCREMENT XX` The new input file should specify XX on the RESTART model definition option.
3008	Contact caused the restarting of the increment an excessive number of times. This may be due to separation or increment splitting. Increase the separation force/stress or reduce the size of the step. An alternative is to increase the number of iterations allowed.
3010	Successful completion of design sensitivity analysis.
3011	Successful completion of design optimization analysis.
3012	No constraints activated at vertex of design space. Enter bounds on design variables.
3013	User element is not supported in design sensitivity analysis or design optimization analysis.
3015	Unable to reduce the time step when using the AUTO STEP option to satisfy both the user criteria and the minimum time step constraint. It is necessary to change the minimum time step in this option or change the required criteria.
3300	The eigenvalue extraction did not converge within the maximum number of iterations allowed. Increase maximum number of cycles or tolerance on the MODAL SHAPE, MODAL INCREMENT, BUCKLE, or BUCKLE INCREMENT options or, if applicable, change from the inverse power sweep to the Lanczos method using the BUCKLE parameter.
3301	Eigenvalue analysis requested, but either mass is zero or initial stress stiffness is zero. For a dynamic modal analysis, make sure that there is mass in the system. This can be from either entering a mass density or mass points. In a buckling analysis, make sure that load has been previously applied to the structure, such that stresses are present.
3302	Nonpositive definite system occurred during Lanczos eigenvalue extraction for buckling analysis. Decrease the applied load before the buckling analysis is performed or switch to the inverse power sweep method using the BUCKLE parameter.

Exit Numbers 4001-5000

Exit Number	Explanation
4001	Sequential I/O error; problems with opening, reading, or writing files. Check file access permissions and available disk space. If on restart or post file, attempt to read increment which is not on file. Otherwise, MARC system error; consult MARC analyst.
4004	Sequential I/O error; problems with opening, reading, or writing files. Check file access permissions and available disk space. If on restart or post file, attempt to read increment which is not on file.
4009	Sequential I/O error, unable to open unit. Either a file requested is not found, or there is a system protection against opening file. Make sure that you have read/write permission in the current directory.
4210	Restart requested, but no restart file was provided. Add `-rid` parameter when submitting the job.
4211	Substructure analysis requested, but no data base found. Add `-sid` parameter when submitting the job.
4031	Error in dynamic memory request, reduce the value on the SIZING parameter, and/or reduce the value of MAXSIZE in the include file.

Exit Numbers 5001-6000

5001	The number of nodes created during adaptive meshing is greater than the number specified on the ADAPTIVE parameter. Either increase the number on the ADAPTIVE parameter or reduce the number of levels allowed on the ADAPTIVE model definition option. The analysis can be forced to continue using the ADAPTIVE parameter.
5002	The number of elements created during adaptive meshing is greater than the number specified on the ADAPTIVE parameter. Either increase the number on the ADAPTIVE parameter or reduce the number of levels allowed on the ADAPTIVE model definition option. The analysis can be forced to continue using the ADAPTIVE parameter.
5003	Adaptive meshing is not available for element type requested. Either use a valid element type, remove ADAPTIVE option or use ADAPTIVE option with a set specifying elements which are of a valid type.
5004	Adaptive meshing is not available for element type requested. Either use a valid element type, remove ADAPTIVE option or use ADAPTIVE option with a set specifying elements which are of a valid type.

Exit Number	Explanation
5005	Number of boundary conditions created during adaptive meshing exceeds the number specified on SIZING parameter. Increase the number of boundary conditions specified on the SIZING parameter.
5007	Error in performing adaptive meshing with elastic foundations or films; MARC system error; consult MARC analyst.
5008	Number of elastic foundations or films created during adaptive meshing exceeds the number specified on the parameter. Add FOUNDATION of FILMS option.
5009	Error in memory allocation while performing adaptive meshing. MARC system error; consult MARC analyst.
5010	System error during adaptive meshing. MARC system error; consult MARC analyst.
5011	Unable to find surface for adaptive meshing with shells if the ATTACH NODE option is used. Check definition of surfaces.
5012	Unable to find surface normal when using the ATTACH NODE option with third adaptive meshing. Check definition of surfaces.
5023	Not enough space on SIZING parameter to create new mesh. Increase workspace available.
5024	Not enough space on SIZING parameter to perform rezoning. Increase workspace available.

B Workspace Definition and the Sizing Option

Finite element analysis requires the generation of a large amount of data including element quantities, nodal quantities, input data and the stiffness matrix. Under the most optimal circumstances, all of this data can be stored in core.

MARC K7 uses dynamic memory management in performing the analysis. The amount of memory that may be allocated for an analysis is controlled through three parameters. The SIZING parameter specifies the amount of memory to obtain initially. If the value on the SIZING parameter is zero, MARC uses the value NORMAL specified in the include file in the tools directory. MARC then requests additional memory as it requires until either the value of MAXSIZE is reached or until there is no additional memory to allocate. In such a case, MARC cannot place all variables in memory, it activates either the ELSTO parameter, or the out-of-core solver options. The ELSTO parameter places the element quantities onto the disk. This option may substantially reduce the memory requirements with only a small penalty to the system performance. The out-of-core solvers allow solution of almost any size problem. Note that the I/O is dependent upon the number of nodes that can be placed within memory, so performance is improved with more memory. Note that the value of NORMAL is 5,000,000 real * 4 words (real * 8 on CRAY). The value of MAXSIZE is 60,000,000. Both of these values can be changed in the include file.

On some machines the maximum amount of data that can be requested on the SIZING parameter is limited by the actual physical size of computer memory; these are nonvirtual memory machines. Recently these machines have increased in size and often you can request as much as 100 million words.

On other machines the maximum amount of data that can be requested on the SIZING parameter is limited only by the operating system; these are virtual memory machines. The limit is usually due to the design of the address translation tables or requirements imposed by the job scheduler. Because of the almost limitless size that you can request the MARC internal data bases, ELSTO and out-of-core solver have, to some extent, been superseded. In the last few years, the real memory of these machines has substantially increased and hence, there is very little swapping of data between real memory and virtual memory.

Regardless of whether a virtual or nonvirtual machine is used, the optimal sizing selected by you is one such that all data fits in memory but does not exceed the machine limitations. One can use the STOP parameter to determine this optimal number. This optimal sizing number appears in the MARC output file with the message

```
TOTAL WORKSPACE NEEDED WITH IN-CORE MATRIX STORAGE =
```

Of course, the bandwidth should be optimized to reduce the size of the stiffness matrix.

In addition to the specification of the data space on the SIZING parameter, you might need to define the size of the common block /SPACE/. This common block on many machines is declared in the main program. The declaration of this common block must be greater than that requested on the SIZING parameter. On UNIX-based machines, the shell script ensures that this occurs.

Estimating Workspace Sizes MARC Programs

Unfortunately, there is no easy method for estimating how much workspace will be needed by MARC, as the space computation is a complicated function of many variables. The most efficient method is to select a workspace to handle as large a variety of runs as possible, without at the same time sacrificing efficiency or wasted core space.

The MARC-STRESS program presents some complications to estimating sizing as both in-core and out-of-core solutions are possible. It is also possible to have the storage of elements out-of-core (through the use of the ELSTO parameter). The solution type is automatically chosen by MARC, so that you have no direct control over this feature. For problems of less than 1000 degrees of freedom, a workspace of 50,000 words is suggested. When the problem size is between 1000 and 3000 degrees of freedom, increase the workspace to 100000 words in the MARC parameter. Use the ELSTO parameter, if necessary, to use out-of-core storage for element variables. When a problem of more than 3000 degrees of freedom is to be analyzed, flag ELSTO and allocate between 100,000 and 1,000,000.

I/O With MARC

In this section we will discuss, in general, the input and output (I/O) that the program performs during an analysis. The machine dependent specifics are discussed in subsequent sections. In many cases, it is your responsibility to preserve one or more of these files for subsequent analysis.

The execution of MARC is performed in a noninteractive manner; that is the data is placed in a file and is not modified during the execution of the problem. During the execution of this job, this file and others are either read, written or both written and read.

The primary data file is called the input file and is read from FORTRAN unit 5. This file is a so called card image file. It is necessary for all analyses.

The resultant of the finite element analysis containing the stresses, strains, displacements, reactions, etc. are written to FORTRAN unit 6. This is called the output file. This file is always created by MARC.

If you request a restart file, this is written by default to unit 8. Upon continuing (restarting) the analysis, this previously generated file is read from unit 9. These are binary sequential files containing the results of one or more increments. You can control the frequency at which the restart information is written. The information written for each increment contains control information, element and nodal data.

If you request, the triangular stiffness matrix is written to the restart file. This considerably increases the size of the restart file and is of primary interest only in Elastic or Creep analysis.

You can request that a subset of the results be written to an auxiliary file labeled the post file. This information is typically used for post processing by Mentat. Mentat uses this information to produce either displacement, contour, x-y, or time history plots. The PLDUMP program, as discussed in *Volume D: User Subroutines and Special Routines*, can also be used to examine this information. In addition, this file is used to transfer temperature data from a heat transfer analysis to a stress analysis.

You can select the post file to be either a binary or formatted (ASCII) file. Either type of a file is a sequential file. By default, the binary file is written to unit 16 and the formatted file is written to unit 19. The advantage of generating a binary file is that the I/O time is faster and the resultant file is smaller. The advantage of generating a formatted file is that the results can be transferred to another machine for post processing. The PLDUMP program can be used to translate the post file from one format to the other. One or more increments of analysis can be written to this file.

Upon restart, you can request that the program make a new post file that contains either the information from the previous analysis (continuous) or data only from this analysis. If the RESTART option appears before the POST option, a continuous post file is written. In such an analysis, the old post file is read from units 17 or 20 depending on whether a binary or formatted file is made, respectively. Of course, the creation of a continuous post file results in a much larger file.

In a thermal stress analysis, if the CHANGE STATE option is used, the post files previously generated in a heat transfer analysis can be read from unit 24 if formatted or unit 25 if binary.

The use of substructures/superelements, requires I/O on multiple files. The primary data base is a direct access binary file written to unit 31. Optionally, you can place the bulk of the information required in substructure analysis to auxiliary sequential binary files. In this case, each formed superelement is placed in its own file. The direct access database is still required.

MARC also uses additional files for scratch purposes depending on the size of the analysis or options selected, as discussed below.

If you include the ELSTO parameter, the information associated with element quantities is stored on a binary direct access file on unit 3. These element quantities are stresses, strains, plastic strains, etc. It is possible for the program to automatically switch on the ELSTO parameter if the workspace specified on the SIZING parameter is not adequate to store the information in core.

The global stiffness matrix typically requires the largest amount of data. If the amount of data exceeds the available workspace, the program automatically utilizes the out-of-core solver. The out-of-core assembly and solver utilizes as many as five files. Some of these files are used for multiple purposes during this process. Files 11, 12, 13 and 15 are binary sequential files, while file 14 is a binary direct access file.

The program also uses FORTRAN units 12 and 13 during the iteration process of nonlinear analysis.

If the Lanczos eigenvalue extraction procedure is used, the eigenvectors are stored in a binary sequential file on unit 22.

Estimating File Sizes

In an out-of-core solution, units 11 and 12 have approximately the same storage space requirements. The number of single-precision words needed is:

```
NUMNP*(MAXBW*NDEG*2+3)*NDEG
```

where

NUMNP = number of nodes in the mesh
MAXBW = average nodal half-bandwidth
NDEG = number of degrees of freedom per node

The other two out-of-core solution files, units 13 and 15, also are approximately the same for total disk space usage. The number of single- precision words needed is:

$$MAXBW^2 * NDEG^2$$

where

MAXBW and NDEG are defined as above.

Also important is the need to have some idea of how large the files created by the POST or RESTART options are going to be. For the POST option, the approximate number of words written per increment is:

```
(NUMV*INTEL*NUMEL)+(NUMNP*IDYN*NDEG)
```

where

NUMV = number of element variables per integration point
INTEL = maximum number of integration points per element
NUMEL = number of elements
NUMNP = number of nodes
IDYN = 2 for heat transfer, = 3 for statics, = 5 for dynamics
NDEG = number of degrees of freedom per node

The format of a restart file is much more complicated and as such, an accurate estimate of space requirements is very difficult. The amount of data output depends, to some extent, on the options within MARC that have been selected.

However, the largest amount of data on the file is formed from three parts:

1. Space for element variables (connectivity, stresses, etc.).
2. Space for the storage of the displacement, load and coordinate vectors.
3. Space for the stiffness matrix (if written to disk).

The space for the first two can be found by examining the output of a MARC run. About one-third of the way down the page entitled "Key to Strain, Stress, and Displacement Output", there is a section "Internal Core Allocation Parameters". In this section, the total space required in words for element and vector storage is printed.

The size of the stiffness matrix is the same as that for file 12, described above. However, note that the stiffness matrix is only written to disk if this option has been flagged on the RESTART model definition option (see RESTART on page 3-116 for more details on this option).

When these three numbers are added together, this will give the user a rough idea of how many words per increment are written to the restart file. When a large analysis of many increments is run, the total number of words output is quite large.

Note that since all these files contain unformatted variable-length records, some extra space must be added to the above figures to determine the total disk space. Also, MARC requires some additional space for internal control variables, thereby increasing the total space even more. Thus, any estimates should be at least doubled to provide a sufficient safety margin.

Running MARC

This section describes the MARC usage on UNIX based machines applicable to either BSD4 or System V machines except where noted. MARC is mainly controlled by a shell script program called `run_marc` which is stored in the `marck71` subdirectory `tools`. If you have used the option to creating a link during the installation, this shell script is also known system wide as `marck71` (for example, `marck71`). It is designed to handle practically all possible options.

The shell script will submit a job and automatically take care of the file assignments providing that use is made of the default FORTRAN file units as specified in Table B-1. Note that the program automatically opens file units 1-36, excluding 26-30. The shell script must be executed in the directory where all relevant input and output files concerning the job are available. To use the shell script, each MARC job should have a unique name qualifier and all MARC output files connected to that job use this same qualifier.

Table B-1 FORTRAN File Units Used by the UNIX Version of MARC

File name	Unit	Description	Comments
`jidname.log`	0	Error message output unit	Except HP
`jidname.t01`	1	Formatted data file	Usually contains mesh
`jidname.t02`	2	OOC* solver scratch file	random access binary file
`jidname.t03`	3	ELSTO file	sequential access binary file
`jidname.t04`	4	Neutral plot file	sequential access binary file
`jidname.dat`	5	Formatted data input file	formatted FORTRAN file
`jidname.out`	6	Printed output file	formatted FORTRAN file
`jidname.log`	7	Error message output file	HP only
`jidname.t08`	8	New restart file	sequential access binary file
`ridname.t08`	9	Old restart file	sequential access binary file
`jidname.t11`	11	OOC* solver scratch file	sequential access binary file
*OOC denotes Out-Of-Core solution.			

Table B-1 FORTRAN File Units Used by the UNIX Version of MARC (Continued)

jidname.t12	12	OOC* solver scratch file	sequential access binary file
jidname.t13	13	OOC* solver scratch file	sequential access binary file
jidname.t14	14	OOC* solver scratch file	random access binary file
jidname.t15	15	OOC* solver scratch file	sequential access binary file
jidname.t16	16	New post file (FORTRAN file)	sequential access binary file
ridname.t16	17	Old post file (FORTRAN file)	sequential access binary file
jidname.t18	18	Formatted data file, optimization table	formatted FORTRAN file
jidname.t19	19	New post file	formatted FORTRAN file
ridname.t19	20	Old post file	formatted FORTRAN file
jidname.t22	22	Subspace iteration scratch file	sequential access binary file
jidname.t23	23	Fluid-solid interaction file	sequential access binary file
pidname.t19	24	Heat data input file	formatted FORTRAN file
pidname.t16	25	Heat data input file (FORTRAN file)	sequential access binary file
sidname.t31	31	Substructure master data file	random access binary file
jidname.t32	32	Secant method file	sequential access binary file
jidname.t34	34	Neutral plot file	formatted FORTRAN file
sidname.t35	35	Substructure file	sequential access binary file
sidname.t36	36	Substructure file	sequential access binary file
jidname.t41	41	New Post file – Domain Decomposition	sequential access binary
ridname.t42	42	Post file – Domain Decomposition	formatted Fortran file
jidname.t45	45	Optimization data file	formatted Fortran file
jidname.t46	46	Optimization scratch file	formatted Fortran file
USRDEF	98	Global default file	formatted Fortran file
EXITMSG	99	Exit messages	formatted Fortran file
*OOC denotes Out-Of-Core solution.			

MARC input files should always be named `job_name.dat`, whereby the prefix `job_name` is the name qualifier which you are free to choose. The suffix `.dat` is obligatory.

To actually submit a MARC job, the following command should be used. The single input line is split over multiple lines for clarity:

```
run_marc      -jid       job_name    (required as minimum)
              -rid       restart_name
              -pid       post_name
              -sid       substructure_name
              -prog      program_name
              -user      user_subroutine_name
              -save      save_user_executable
              -queue     queue_name
              -ver       verification_flag
              -def       data_name
              -vf        viewfactor
              -nprocd    domains
              -nthread   threads
```

Table B-2 describes the meaning of these input options and Table B-3 gives examples.

Table B-2 `run_marc` Input Options*

Keyword	Options	Description
`-jid (-j)`	**job_name**	Job and input file name identification. Requires `job_name.dat` for all programs except the curve fit and neutral plot programs.
`-prog (-pr)`	marc pldump progname	**Run marc** with or **without user subroutine**. Run the postfile conversion program pldump. Run saved executable *progname.marc* from a previous job.
`-user (-u)`	user_name	User subroutine `user_name.f` is used to generate a new executable program called `user_name.marc`.
`-save (-sa)`	**no** yes	Do *not* save the new executable program `user_name.marc`. Save the executable program `user_name.marc` for a next time.
`-rid -(r)`	restart_name	For marc or progname: identification of previous job that created restart file.
`-pid (-p)`	post_name	For marc or progname: identification of previous job that created postfile containing temperature data.
`-sid (-si)`	substructure	Substructure jobs only: name of the substructuring file `substructure.t31`.
`-queue (-q)`	**background** foreground queue name	Run the program in the background. Run the program in the foreground. Submit to batch queue the queue name. Only available for machines with batch queue; for example, Convex, Cray. Queue names and submit command syntax can differ from site to site, adjust `run_marc` if necessary.
`-back (-b)`	**yes** no	Alternative for -queue: run the program in the background. Run the program in the foreground.
`-ver (-v)`	**yes** no	Ask for confirmation before starting the job. Will start the job immediately.
`-def (-de)`	data_name	File name containing user defined default data.
`-vf`	viewfactor	Name of file containing viewfactors for radiation viewfactor.vfs.
`-nprocd`	number	Number of domains for parallel processing.
`-nthread`	number	Number of threads per task.
`-pq`	0,1,2,etc	Batch queue only: queue priority.
`-at (-a)`	date/time	Batch queue only: delay time for start of job. Syntax: January,1,1994,12:30 or: today,5pm
`-cpu`	sec	Batch queue only: CPU time limit.
*Default options are shown in **bold**.		

Table B-3 Examples of Running MARC Jobs

Examples of Running MARC Jobs	Description:
`run_marc -jid e2x1`	Runs the job `e2x1` in the background. The input file `e2x1.dat` resides in the current working directory.
`run_marc -jid e2x14 -user u2x14 -save yes`	Runs the job `e2x14` in the background, using the user subroutine `u2x14.f` and the input file `e2x14.dat`. An executable program named `u2x14.marc` is saved after completion of the job.
`run_marc -jid e2x14a -prog u2x14`	Runs the job `e2x14a` using the executable produced by job `e2x14`.
`run_marc -jid e3x2a -ver no -back no`	Runs the job `e3x2a` in the foreground. The job runs immediately without verifying interactively.
`run_marc -jid e3x2b -rid e3x2a`	Performs a restart job using the results of the previous job `e3x2a`.

MARC Sizing

A single routine in MARC controls the data storage (workspace) size by means of a common block called SPACE. The size is set in the file `mrmain.f` which can be found in the `marck71` subdirectory `main`. The default is currently set to 6,000, 000 words. When MARC is started, it makes an estimate of how much data storage it actually needs for the analysis job. You are required to define a SIZING parameter in the input file. This is an estimate for the amount of data storage your analysis needs. Two situations can arise:

- If the actual required amount is less than SIZING parameter, the problem can be stored internally and is executed in the most efficient way.
- If the actual required amount exceeds SIZING parameter, MARC splits the analysis problem during the solver phase into parts and store these parts on temporary files. It is done automatically and is called the out-of-core solution.

If you have a FORTRAN compiler installed the shell script `run_marc`, which starts the MARC execution, scans the input file for the value of SIZING parameter and, depending on its findings, does the following:

- If the SIZING parameter is less than the default of the MARC workspace, the job starts immediately.

- If the SIZING parameter is greater than the default but less than the upper limit, a new executable is created using a modified main routine with a workspace of SIZING. It is created in a similar way as a job with a user subroutine. After the job finishes, the executable is removed.
- If the SIZING parameter exceeds both the default and the upper limit, a new executable is created using the upper limit as workspace size. After the job finishes, the executable is removed.

You may choose to modify the default size permanently by means of the `maintain` shell script:

```
cd "parent_directory"/marck71/tools
./maintain
```

Choose option 4 from the main menu to get you into the maintenance submenu. Here, choose option 4.4 to redefine the default workspace size and execute option 4.3 to regenerate the MARC executable with the modified workspace size.

Note that apart from the default value of the workspace you are also able to set an upper limit to the workspace: option 4.5 of the maintenance submenu. This upper limit should reflect the maximum permissible program size which your computer can handle. Both default and upper limit are given in single precision words; that is, four bytes per word.

Changes in the File Assignments

The file assignments in MARC have been set up to be convenient for you if they are used in conjunction with the shell script used to run the programs. Hence, it is not recommended that you change them. However, you can modify the file assignments for MARC if necessary. To that end, the three routines having to do with file assignments have been included in the directory/main. These are:

`fsetup.f:` Here the file names are defined based on your input.

`intint.f:` Here all ASCII files and all files whose names are based on restart or post files, are opened.

`quit.f:` Here all files are closed and, if appropriate, removed.

MARC: Permanently Replace Routine(s)

You must have a FORTRAN compiler to replace MARC routines.

The `maintain` shell script can be used to replace routines in MARC. These updates are usually shipped as separate FORTRAN routines. Copy these files into the `marck71` subdirectory `update`. Next, compile these routines and regenerate the MARC executable:

```
cd "parent_directory"/marck71/tools
./maintain
```

Choose option 4 from the main menu. This will bring you into the maintenance submenu. Here, choose option 4.2 to compile the updated routines. You will be prompted to supply a list of file names. Next, execute option 4.3 to generate a new MARC executable. Repeatedly using the exit option gets you out of the `maintain` script.

C Default File

MARC allows commonly used options to be stored in a file, so that it is not necessary to include them into the input file. This can be done in two ways. First, an organization could have a file that has options that are commonly used and shared by all users. The file name is defined by an environment variable named `USRDEF`. This default file is read first. Second, you can specify a file when you are running an analysis. This is done when submitting a job using the `-def` option. This file is read second.

Values of data previously read can be overwritten. Your actual model data specified by `-jid` is read last. It can also overwrite previously defined data. The form of the file containing the defaults is the same as the standard MARC input file (it must include an END parameter and an END OPTION model definition line), but is restricted to selective options. These options include:

Parameters

All parameters can be put in the default file.

Model Definition Options

ADAPTIVE
CONTROL
CONVERT
NO PRINT
OPTIMIZE
POST
RESTART
RESTART LAST
SOLVER
SUMMARY
END OPTION

D Control File

MARC allows you to change the controls of the program while the analysis is being performed. This is done by creating a file named `jid.cnt` where `jid` is the job id name.

This file supports the following options.

STOP	Stops the job as soon as the current operation is complete; goes through the MARC quit procedure to correctly close files.
STOP NEXT	Stops the job after the completion of the current increment. As you can change the restart frequency or post frequency, you can make sure that his last increment is saved.
CONTROL	Same as normal CONTROL history definition option; can be used to change tolerances, etc.
POST INCREMENT	Same as normal POST INCREMENT option; can be used to change post tape frequency.
RESTART IN	Same as normal RESTART INCREMENT option, can be used to change restart file frequency.

E Environment Variables

MARC has introduced user-controlled environment variables. They can be put into your login shell script or the `run_marc` shell script. These environment variables include:

`EXITMSG`	Name of file containing centralized exit messages.
`USRDEF`	Name of file containing default input options/data.
`AFMATDAT`	Name of file containing material data base.
`IBIG`	Set to 1 for reading integers in I5 format, reals in F10 format.
	Set to 2 for reading integers in I10 format, reals in F20 format.

For UNIX environments, you typically use:

```
setenv USRDEF myfile.dat
```

For PC environments, you typically use:

```
set USRDEF=myfile.dat
```

F Parameters

Parameters

G Options

MARC
Analysis
Research
Corporation

Document Title: **MARC Volume C: Program Input K7**
Part Number: RF-3003-07
Revision Date: August, 1997

CALIFORNIA - Palo Alto
MARC Corporate Headquarters
260 Sheridan Avenue, Suite 309
Palo Alto, CA 94306, USA
Tel: 1 650 329 6800
Fax: 1 650 323 5892
Email: support@marc.com

CALIFORNIA - San Diego
MARC Analysis Research Corp.
4330 LaJolla Village Drive, Suite 320
San Diego, CA 92122
Tel: 1 619 658 9588
Fax: 1 619 587 8710

CONNECTICUT - E. Berlin
MARC Analysis Research Corp.
1224 Mill Street
E. Berlin, CT 06023
Tel: 1 860 828-2055
Fax: 1 860 828 2031

MICHIGAN - Ann Arbor
MARC Analysis Research Corp.
24 Frank Lloyd Wright Drive
Ann Arbor, MI 48106-0523
Tel: 1 313 998 0540
Fax: 1 313 998 0542

CZECH REPUBLIC
MARC Overseas, Inc.
Podolska 50
147 00 Praha 4
Czech Republic
Tel: 011 420 (2) 61 21 41 23
Mobil: 011 420 (602) 21 23 26
Fax: 011 420 (2) 61 21 41 23

FRANCE
MARC France S.A.
30, Rue de Rocroy
94100 Saint-Maur Des Fossés (near Paris)
France
Tel: 011 33 (1) 41 81 1 0 94
Fax: 011 33 (1) 41 81 1 0 95

GERMANY - Munich
MARC Software Deutschland GmbH
Ismaninger Str. 9
85609 Aschheim (bie Munich)
Germany
Tel: 011 49 (89) 904 50 33
Fax: 011 49 (89) 903 06 76
Email: system@marc.co.de

GERMANY - Hannover
MARC Software Deutschland Gmbh
Alte Dohrener Str. 66
30173 Hannover, Germany
Tel: 011 49 (511) 980 5182
Fax: 011 49 (511) 980 5187

ITALY
Espri-MARC s.r.l.
16129 Viale Brigata Bisagno 2/10
16129 Genova
Italy
Tel: 011 39 (10) 585 949
Fax: 011 39 (10) 585 949

THE NETHERLANDS
MARC Analysis Research Corporation
Dublinstraat 32
2713 HS Zoetermeer
The Netherlands
Tel: 011 31 (79) 3510 411
Fax: 011 31 (79) 3517 560
Email: support@marc.nl

UNITED KINGDOM
MARC UK Ltd.
35, Shenley Pavilions
Chalkdell Drive
Shenley Wood
Milton Keynes, MK56LB
Tel: 011 44 (1908) 506 505
Fax: 011 44 (1908) 506 522

CHINA
MARC Overseas, Inc.
Bright Chang An Building, Suite 321
7# Jian Guo Men Nei Street
Beijing, China 100 005
Tel: 011 86 (10) 6510 2056
011 86 (10) 6510 2057
011 86 (10) 6510 2058
Fax: 011 86 (10) 6510 2053

JAPAN - Nagoya
Nippon MARC Co., Ltd.
Nagoya Nishiki Daiichi Seimei Bldg.
2F, 6-5 Nishiki 1-chome
Naka-ku, Nagoya, Aichi 460
Tel: 011 81 (52) 202 5661
Fax: 011 81 (52) 202-5665

JAPAN - Osaka
Nippon MARC Co., Ltd.
4F 2nd Kimi Building
2-11 Toyotucho Suita-shi
Osaka 564
Tel: 011 81 (6) 385 1101
Fax: 011 81 (6) 385-4343

JAPAN - Tokyo
(Asian Operations Headquarters)
Nippon MARC Co., Ltd.
Shinjuku Daiichi Seimei Building
P.O. Box 5056
2-7-1 Nishi-Shinjuku, Shinjuku-ku
Tokyo 163
Tel: 011 81 (3) 3345 0181
Fax: 011 81 (3) 3345 1529
Email: system@marc.co.jp

KOREA
Dong Kyung Bldg. 7FL
824-19 Yuksam-Dong
Kangnam-Ku
Seoul, Korea
Tel: 011 82 (2) 561 7543
Fax: 011 82 (2) 561 7767