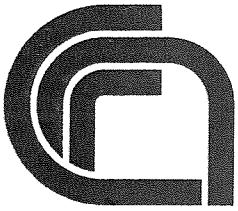

Consiglio Nazionale delle Ricerche

The finite element code NOSA
User's manual

Technical Report CNUCE-B4-2001-014

September 2001

Compiled by the Mechanics of Materials and Strucures Team.
S. Degl'Innocenti, M. Lucchesi, C. Padovani,
A. Pagni, G. Pasquinelli, N. Zani

Pisa

The finite element code NOSA

User's manual

Technical Report CNUCE-B4-2001-014

September 2001

Compiled by the Mechanics of Materials and Structures Team.
S. Degl'Innocenti, M. Lucchesi, C. Padovani,
A. Pagni, G. Pasquinelli, N. Zani

PRESENTATION

The finite element code NOSA (Non-linear Structural Analysis) has been developed by the team of Mechanics of Materials and Structures of the CNUCE. The work's primary goal has been the development of a suitable set of tools for experimentation on engineering problems involving material constitutive equations and algorithms for integrating the equation of motion as well as other numerical techniques of research interest to the group.

A first version of the code, written in the early 1980's, included plane, three-dimensional and axisymmetric isoparametric elements [1] and permitted elastic-plastic analysis in the presence of infinitesimal strains with the hardening model described in [2]. Subsequently, the possible applications of NOSA were extended to include cases of finite strains based on the studies carried out on both constitutive equations [3, 4, 5, 6] and methods of numerical integration of the equation of motion [7, 8, 9] in the presence of follower forces and contact problems. At the same time, the element library was broadened by the adding of the shell elements (thin and thick) [10].

More recently, elastic material, with low resistance to traction, has been added in order to permit static analysis of masonry solids [11, 12, 13, 14, 15].

Finally, the capability of performing heat transfer and dynamic analysis has been added.

Over these years many engineering students of the University of Pisa have collaborated in the development of the NOSA code as part of their thesis research.

The authors

INDEX

A. INTRODUCTION

Preparation of input data and display of numerical results may be performed using the pre and post-processing code MENTAT II[1].

PRE-PROCESSING	MENTAT II
F.E.M. ANALYSIS	NOSA
POST-PROCESSING	MENTAT II

The code MENTAT II can process two- and three-dimensional meshes; the main operations that can be carried out are:

- automatic mesh generation,
- definition of nodal coordinates,
- definition of element connectivities,
- definition of material properties of elements,
- definition of geometrical properties of elements,
- definition of loads and boundary conditions,
- plotting NOSA output .

The F.E.M. code NOSA can be used for solving linear and non-linear equilibrium problems; the output of NOSA is the stress, strain and displacement field, as well as the reactions forces. Two kinds of non-linearities can be taken into account: material non-linearities due to the material's constitutive equation and geometric non-linearities due to finite strains or contact conditions.

[1] MENTAT is a product of the MARC Analysis Research Corporation.

A1. MATERIAL MODELS

ELASTICITY: static and/or dynamic analysis is performed for structures made up of a linear elastic material, using all elements of the library.

COMPOSITE: the shell elements (type 5, 10 and 17) and the beam element (type 9) allow modeling of the structures consisting of sections of different mechanical and/or thermal properties, such as composite materials.

ELASTO-PLASTICITY: static and/or dynamic analysis is performed for structures made up of elastic-plastic materials under the assumption of infinitesimal strains and finite strains in the presence of isotropic, cinematic and combined hardening, using all elements except shells and beams (the finite strain elasto-plasticity is available only for the elements with linear interpolating functions).

MASONRY-LIKE static and/or dynamic analysis is performed, using all element types, for structures made up of masonry-like materials non-resistant to traction and infinitely resistant to compression, as well as for masonry-like materials with bounded tensile and compressive strength.

HEAT TRANSFER non-linear heat transfer analysis is performed for plane, axisymmetric, shells and 3d elements.

The elements available in the NOSA code are listed in the following Table.

Structural type	Element identifier number	Interpolating functions	Remarks
three-dimensional element	1	quadratic	20 nodes isoparametric element
plane stress element	2	quadratic	8 nodes isoparametric element
plane strain element	3	quadratic	8 nodes isoparametric element
axisymmetric element	4	quadratic	8 nodes isoparametric element
thin shell element	5	linear for displacements, quadratic for rotations	8 nodes isoparametric element
plane strain element	6	linear	4 nodes isoparametric element
axisymmetric element	7	linear	4 nodes isoparametric element
three-dimensional element	8	linear	8 nodes isoparametric element
straight beam element	9	linear	2 nodes isoparametric element
thick shell element	10	linear for displacements and rotations	4 nodes isoparametric element
plane heat transfer element	11	quadratic	8 nodes isoparametric element
plane heat transfer element	12	linear	4 nodes isoparametric element
axisymmetric heat tranfer element	13	quadratic	8 nodes isoparametric element
axisymmetric heat tranfer element	14	linear	4 nodes isoparametric element
3D heat transfer element	15	linear	8 nodes isoparametric element
3D heat transfer element	16	quadratic	20 nodes isoparametric element
heat transfer shell element	17	linear	4 nodes isoparametric element

Table A1.

B. BIBLIOGRAPHIC REFERENCES

For a complete and detailed description of the theories and algorithms used in NOSA, the interested reader is referred to the following:

[1] HINTON E., OWEN D. R. J., Finite Element Programming, Academic Press, 1977.

[2] GUIDOTTI P, LUCCHESI M, PAGNI A., PASQUINELLI G., Elastic-Plastic Behavior with Work Hardening: an Appropriate Model for Structural Software, *Meccanica* **19**, 1984.

[3] LUCCHESI M., PODIO GUIDUGLI P., Materials with Elastic Range: a Theory with a view toward Applications. Part I, *Arch. Rat. Mech. Anal.*

[4] LUCCHESI M., PODIO GUIDUGLI P., Materials with Elastic Range: a Theory with a view toward Applications. Part II, *Arch. Rat. Mech. Anal.*

[5] LUCCHESI M., PODIO GUIDUGLI P., Materials with Elastic Range: a Theory with a view toward Applications. Part III, *Arch. Rat. Mech. Anal.*

[6] LUCCHESI M., PODIO GUIDUGLI P., Materials with Elastic Range and the Possibility of Stress Oscillations in Pure Shear , Proc. Int. Conf. on Comp. Plasticity, Model, Software and Applications, Barcelona, 6-10 aprile 1987.

[7] GUIDOTTI P., LUCCHESI M., A Numerical Method for Solving Boundary-Value problems in Finite Plasticity, *Meccanica*, **28**, 1988.

[8] DEGL'INNOCENTI S., PADOVANI C., PASQUINELLI G., An improved numerical method to integrate the equation of motion in finite elastoplasticity problems, Complas II, Second International Conference on Computational Plasticity, Barcelona, Settembre 1989.

[9] PASQUINELLI G., Simulation of Metal-Forming Processes by the Finite Element Method, *Int J. Plasticity*, Vol. **11**, No. 5 (1995), pp. 623-651.

[10] GUIDOTTI P., LUCCHESI M., PAGNI A., PASQUINELLI G., Application of Shell Theory to Structural Problem Using the Finite Element Method , Quaderni de "La Ricerca Scientifica", **115**, 1986.

[11] LUCCHESI M., PADOVANI C. and PAGNI A., A numerical method for solving equilibrium problems of masonry-like solids. *Meccanica*, **24** (1994), pp. 175-193.

[12] LUCCHESI M., PADOVANI C. and PASQUINELLI G., On the numerical solution of equilibrium problems of elastic solids with bounded tensile strength. *Comput. Methods Appl. Mech. Engrg.* **127** (1995), pp. 37-56

[13] LUCCHESI M., PADOVANI C. and ZANI N., Masonry-like materials with bounded compressive strength. *Int. J. Solids Structures* 33 (1996) pp. 1961-1994

[14] LUCCHESI M., PADOVANI C., PAGNI A. and ZANI N., Un metodo numerico per lo studio degli archi in muratura. Atti del VII Convegno Italiano di Meccanica Computazionale, Trieste 1-3 Giugno 1993, pp. 239-244.

[15] LUCCHESI M., PADOVANI C., PASQUINELLI G. and ZANI N., Un metodo numerico per le volte in muratura. Atti del VIII Convegno Italiano di Meccanica Computazionale, Torino 15-17 Giugno 1994, pp. 44-49.

[16] PADOVANI C., PAGNI A. and PASQUINELLI G., Glie elementi guscio nel codice agli elementi finiti NOSA. Internal report CNUCE-B4-1998-012, Luglio 1998.

[17] PASQUINELLI G., Un elemento trave introdotto nel codice agli elementi finiti NOSA. Internal report CNUCE B4-2000-029, Dicembre 2000.

[18] PASQUINELLI G., Conduzione non lineare del calore nel codice agli elementi finiti NOSA. Internal report CNUCE B4-2000-030, Dicembre 2000.

C. NOSA ELEMENT LIBRARY

At present, the elements available in NOSA are seventeen and they are described in the next sections.

C1. Element 1 (Three-dimensional 20-node brick)

Twenty-node isoparametric element with quadratic interpolating functions. This element is a rapidly converging element for three-dimensional analysis. For thick-shell situations, one element through the thickness will usually provide an acceptable solution for both displacement and stress. Nonetheless, it is advisable that the length to thickness ratio not exceeds a value of 20.

Connectivity

Twenty nodes numbered as shown in Figure 1.1.

Integration

The element is integrated numerically using twenty-seven points (Gaussian quadrature). Integration points are shown in Figure 1.2.

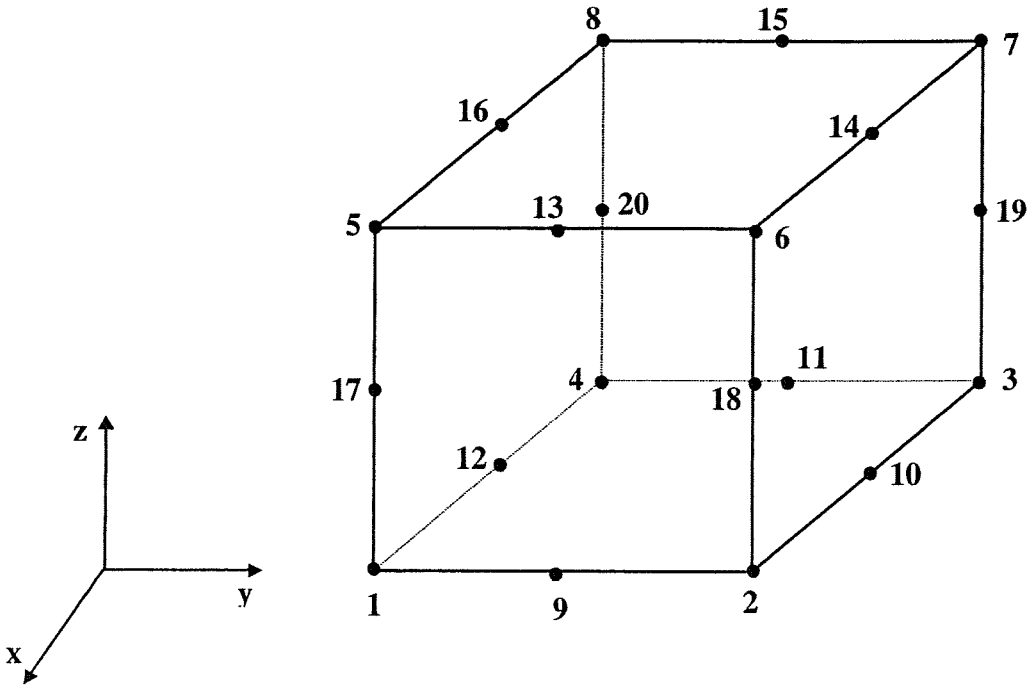

Figure 1.1 Element 1.

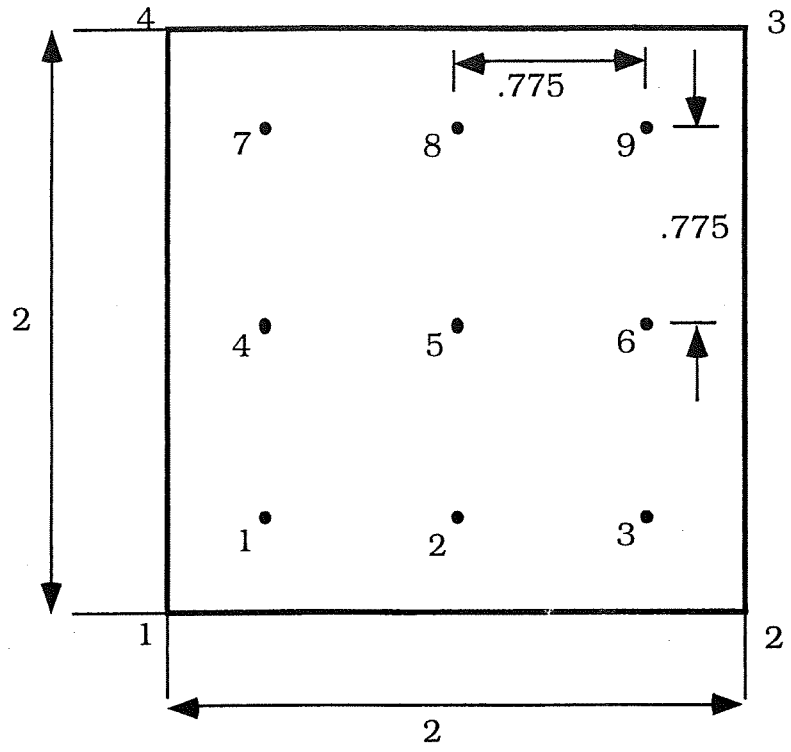

Fig. 1.2 Integration points of element 1.

Distributed loads

Distributed loads by IBODY value are as follows:

IBODY	DESCRIPTION
1	Body force. The three components of the force per unit volume in the global reference system must be assigned.
2	Body force. The three global components of the force per unit volume are calculated in the user subroutine FORCEM.
3	Centrifugal force. The rotation axis must be defined in the card ROTATION AXIS and the angular velocity must be specified in the field reserved for the force magnitudes.
11	Pressure on the 1-2-3-4 face (force per unit area) in the global reference system.
12	Pressure on the 1-2-3-4 face in the local reference system. The first component of the force per unit area is tangential to the face in the 1-2 direction; the third component is orthogonal to the face and is positive if directed towards the interior of the element, the second component has the direction given by the vector product of the first and third directions.

13 Pressure on the 1-2-3-4 face in the global reference system. The three components of the force per unit area on the eight nodes of the face are to be specified in the user subroutine FORCEM.

14 Pressure on the 1-2-3-4 face in the local reference system. The three components of the force per unit area on the eight nodes of the face are to be specified in the user subroutine FORCEM.

21 Pressure on the 5-8-7-6 face (force per unit area) in the global reference system.

22 Pressure on the 5-8-7-6 face in the local reference system. The first component of the force per unit area is tangential to the face in the 5-8 direction; the third component is orthogonal to the face and is positive if directed towards the interior of the element. The second component has the direction given by the vector product of the first and third directions.

23 Pressure on the 5-8-7-6 face in the global reference system. The three components of the force per unit area on the eight nodes of the face are to be specified in the user subroutine FORCEM.

24 Pressure on the 5-8-7-6 face in the local reference system. The three components of the force per unit area on the eight nodes of the face are to be specified in the user subroutine FORCEM.

31 Pressure on the 1-5-6-2 face (force per unit area) in the global reference system.

32 Pressure on the 1-5-6-2 face in the local reference system. The first component of the force per unit area is tangential to the face in the 1-5 direction, the third component is orthogonal to the face and is positive if directed towards the element's interior; the second component has the direction given by the vector product of the first and third directions.

33 Pressure on the 1-5-6-2 face in the global reference system. The three components of the force per unit area on the eight nodes of the face are to be specified in the user subroutine FORCEM.

34 Pressure on the 1-5-6-2 face in the local reference system. The three components of the force per unit area on the eight nodes of the face are to be specified in the user subroutine FORCEM.

41 Pressure on the 2-6-7-3 face (force per unit area) in the global reference system.

42 Pressure on the 2-6-7-3 face in the local reference system. The first component of the force per unit area is tangential to the face in the 2-6 direction, the third component is orthogonal to the face and is positive if directed towards the element's interior; the second component has the direction given by the vector product of the first and third directions.

43 Pressure on the 2-6-7-3 face in the global reference system. The three components of the force per unit area on the eight nodes of the face are to be specified in the user subroutine FORCEM.

44 Pressure on the 2-6-7-3 face in the local reference system. The three components of the force per unit area on the eight nodes of the face are to be specified in the user subroutine FORCEM.

51 Pressure on the 3-7-8-4 face (force per unit area) in the global reference system.

52 Pressure on the 3-7-8-4 face in the local reference system. The first component of the force per unit area is tangential to the face in the 3-7 direction, the third component is orthogonal to the face and is positive if directed towards the element's interior; the second component has the direction given by the vector product of the first and third directions.

53 Pressure on the 3-7-8-4 face in the global reference system. The three components of the force per unit area on the eight nodes of the face are to be specified in the user subroutine FORCEM.

54 Pressure on the 3-7-8-4 face in the local reference system. The three components of the force per unit area on the eight nodes of the face are to be specified in the user subroutine FORCEM.

61 Pressure on the 4-8-5-1 face (force per unit area) in the global reference system.

62 Pressure on the 4-8-5-1 face in the local reference system. The first component of the force per unit area is tangential to the face in the 4-8 direction, the third component is orthogonal to the face and is positive if directed towards the element's interior; the second component has the direction given by the vector product of the first and third directions.

63 Pressure on the 4-8-5-1 face in the global reference system. The three components of the force per unit area on the eight nodes of the face are to be specified in the user subroutine FORCEM.

64 Pressure on the 4-8-5-1 face in the local reference system. The three components of the force per unit area on the eight nodes of the face are to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point loads on nodes and to thermal dilatation loads.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangular prism or a tetrahedron.

Coordinates

Three global coordinates in the x, y and z directions.

Degrees of freedom

Three degrees of freedom u, v, w (the displacements along the global coordinate directions).

Output of strains and stresses

1 = xx, 2 = yy, 3 = zz, 4 = xy, 5 = yz, 6 = xz.

Analysis types

- Infinitesimal elasto-plasticity.

- Masonry-like materials.

C2. Element 2 (plane stress)

Eight node isoparametric element with quadratic interpolating functions.

Connectivity

Eight nodes, numbered as shown in Figure 2.1.

Integration

The element is integrated numerically using nine points (Gaussian quadrature). Integration points are shown in Figure 2.2.

Geometry

The thickness of the element can be specified (default 1).

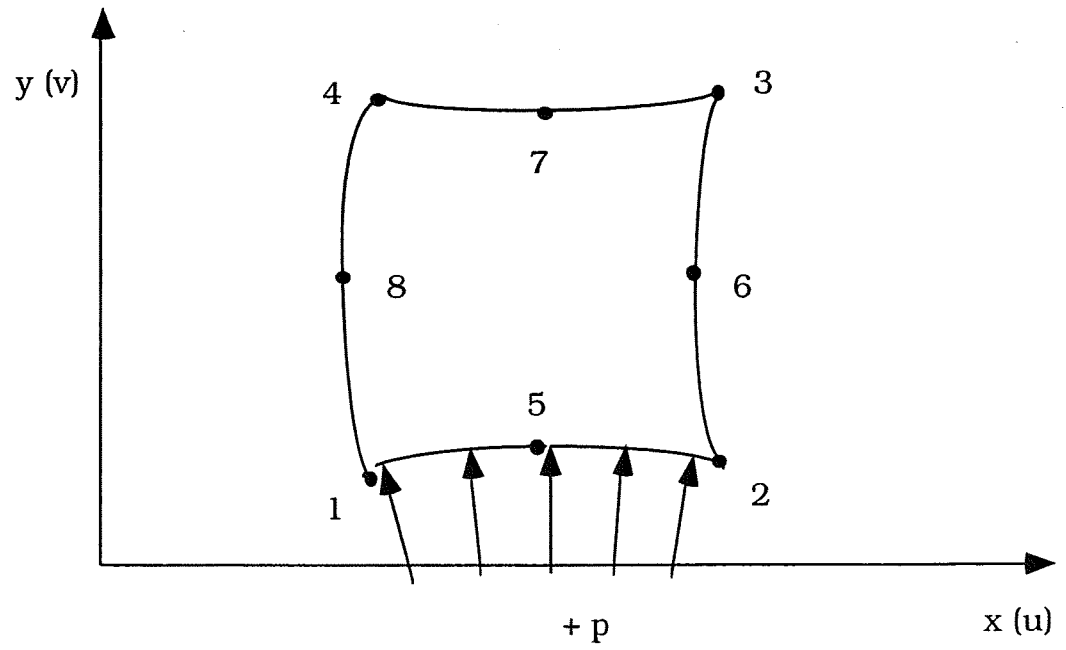

Figure 2.1 Element 2.

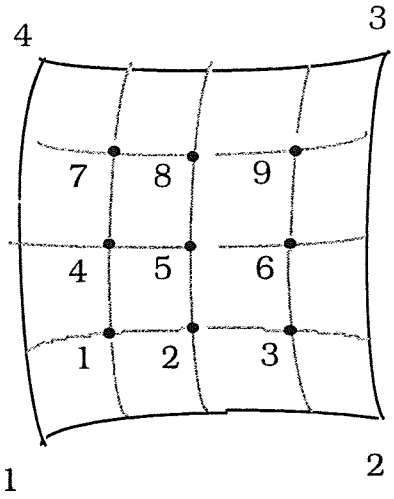

Figure 2.2 Integration points of element 2.

Distributed loads

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Body force. The two components of the force per unit volume in the global reference system must be assigned.
2	Body force. The two global components of the force per unit volume are calculated in the user subroutine FORCEM.
3	Centrifugal force. The rotation axis is orthogonal to the plane of the element; the angular velocity must be specified in the field reserved for the force magnitudes.
11	Pressure on the 1-2-5 edge (force per unit area) in the global reference system.
12	Pressure on the 1-2-5 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 1-2 direction.
13	Pressure on the 1-2-5 edge in the global reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

14 Pressure on the 1-2-5 edge in the local reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

21 Pressure on the 2-3-6 edge (force per unit area) in the global reference system.

22 Pressure on the 2-3-6 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 2-3 direction.

23 Pressure on the 2-3-6 edge in the global reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

24 Pressure on the 2-3-6 edge in the local reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

31 Pressure on the 3-4-7 edge (force per unit area) in the global reference system.

32 Pressure on the 3-4-7 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 3-4 direction.

33 Pressure on the 3-4-7 edge in the global reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

34 Pressure on the 3-4-7 edge in the local reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

41 Pressure on the 4-1-8 edge (force per unit area) in the global reference system.

42 Pressure on the 4-1-8 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 4-1 direction.

43 Pressure on the 4-1-8 edge in the global reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

44 Pressure on the 4-1-8 edge in the local reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point loads on nodes and to thermal dilatation loads.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangle.

Nodal coordinates

Two global coordinates in the x and y directions.

Nodal degrees of freedom

Two degrees of freedom u, v (the displacements along the global coordinate directions).

Output of strains and stresses (at integration points)

1 = xx, 2 = yy, 3 = xy.

Analysis types

- Infinitesimal elasto-plasticity.

- Masonry-like materials.

C3. Element 3 (Plane strain)

Eight-node isoparametric element with quadratic interpolating functions.

Connectivity

Eight nodes numbered as shown in Figure 3.1.

Integration

The element is integrated numerically using nine points (Gaussian quadrature). Integration points are shown in Figure 3.2.

Geometry

The thickness of the element can be specified (default 1).

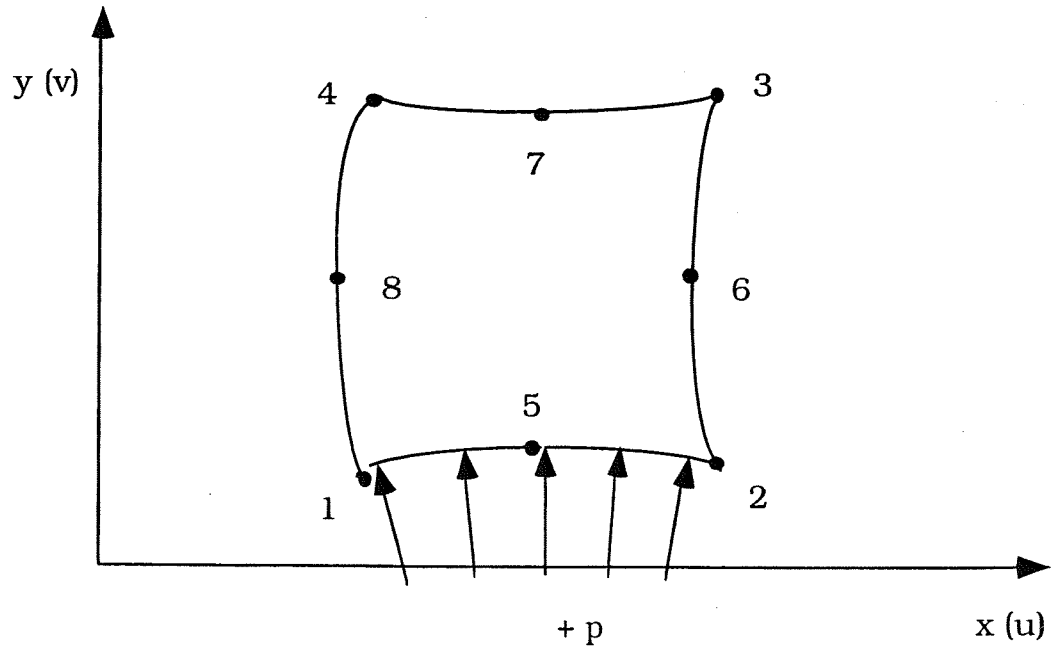

Figure 3.1 Element 3.

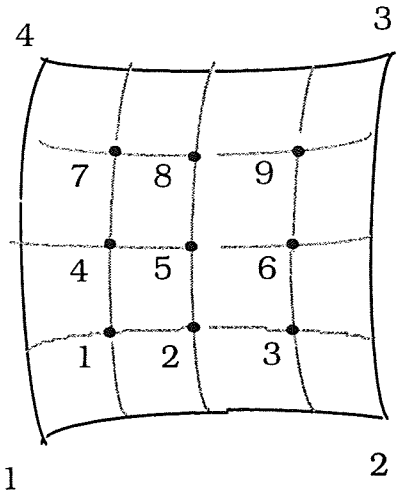

Figure 3.2 Integration points of element 3.

Distributed loads

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Body force. The two components of the force per unit volume in the global reference system must be assigned.
2	Body force. The two global components of the force per unit volume are calculated in the user subroutine FORCEM.
3	Centrifugal force. The rotation axis is orthogonal to the plane of the element; the angular velocity must be specified in the field reserved for the force magnitudes.
11	Pressure on the 1-2-5 edge (force per unit area) in the global reference system.
12	Pressure on the 1-2-5 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 1-2 direction.
13	Pressure on the 1-2-5 edge in the global reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

14 Pressure on the 1-2-5 edge in the local reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

21 Pressure on the 2-3-6 edge (force per unit area) in the global reference system.

22 Pressure on the 2-3-6 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 2-3 direction.

23 Pressure on the 2-3-6 edge in the global reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

24 Pressure on the 2-3-6 edge in the local reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

31 Pressure on the 3-4-7 edge (force per unit area) in the global reference system.

32 Pressure on the 3-4-7 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 3-4 direction.

33 Pressure on the 3-4-7 edge in the global reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

34 Pressure on the 3-4-7 edge in the local reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

41 Pressure on the 4-1-8 edge (force per unit area) in the global reference system.

42 Pressure on the 4-1-8 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 4-1 direction.

43 Pressure on the 4-1-8 edge in the global reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

44 Pressure on the 4-1-8 edge in the local reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point loads on nodes and to thermal dilatation loads.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangle.

Nodal coordinates

Two global coordinates in the x and y directions.

Nodal degrees of freedom

Two degrees of freedom u, v (the displacements along the global coordinate directions).

Output of strains and stresses (at the integration points)

1 = xx, 2 = yy, 3 = xy.

Analysis types

- Infinitesimal elasto-plasticity.

- Masonry-like materials.

C4. Element 4 (Axisymmetric element)

Eight-node isoparametric element with quadratic interpolating functions.

Connectivity

Eight nodes numbered as shown in Figure 4.1.

Integration

The element is integrated numerically using nine points (Gaussian quadrature). Integration points are shown in Figure 4.2.

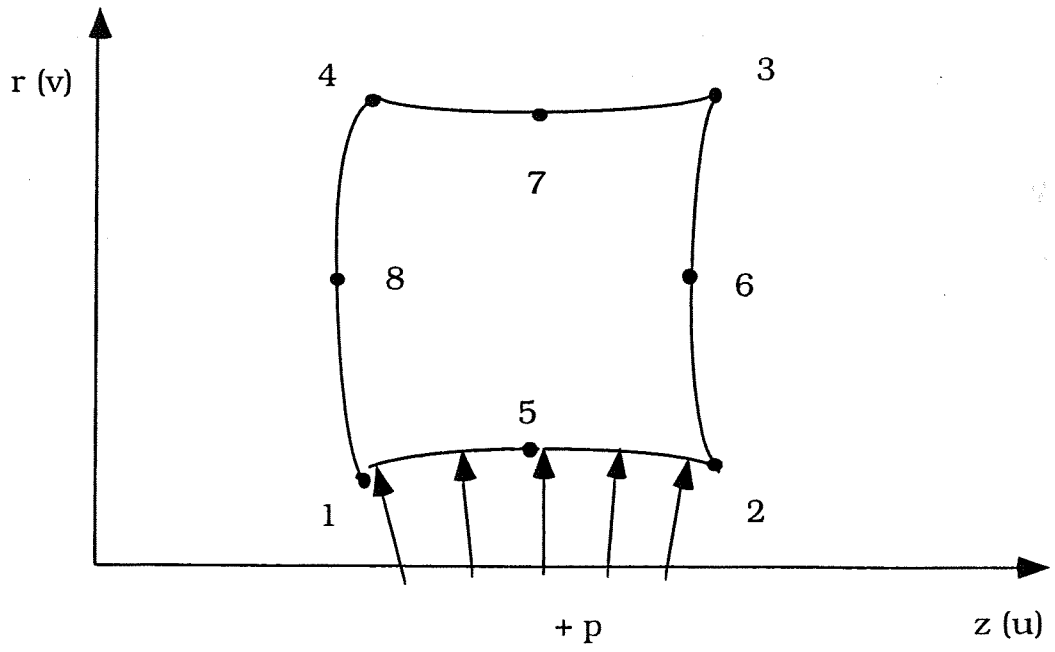

Figure 4.1 Element 4.

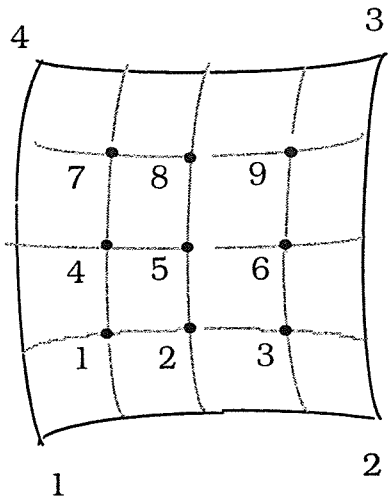

Figure 4.2 Integration points of elements 4.

Distributed loads

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Body force. The two components of the force per unit volume in the global reference system must be assigned.
2	Body force. The two global components of the force per unit volume are calculated in the user subroutine FORCEM.
3	Centrifugal force. The rotation axis coincides with the axis of symmetry, the angular velocity must be specified in the field reserved for the force magnitudes.
11	Pressure on the 1-2-5 edge (force per unit area) in the global reference system.
12	Pressure on the 1-2-5 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 1-2 direction.
13	Pressure on the 1-2-5 edge in the global reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

14 Pressure on the 1-2-5 edge in the local reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

21 Pressure on the 2-3-6 edge (force per unit area) in the global reference system.

22 Pressure on the 2-3-6 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 2-3 direction.

23 Pressure on the 2-3-6 edge in the global reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

24 Pressure on the 2-3-6 edge in the local reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

31 Pressure on the 3-4-7 edge (force per unit area) in the global reference system.

32 Pressure on the 3-4-7 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 3-4 direction.

33 Pressure on the 3-4-7 edge in the global reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

34 Pressure on the 3-4-7 edge in the local reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

41 Pressure on the 4-1-8 edge (force per unit area) in the global reference system.

42 Pressure on the 4-1-8 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 4-1 direction.

43 Pressure on the 4-1-8 edge in the global reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

44 Pressure on the 4-1-8 edge in the local reference system. The two components of the force per unit length on the three nodes of the edge are to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point loads on nodes and to thermal dilatation loads.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as.a triangle.

Nodal coordinates

Two global coordinates in the z and r directions.

Nodal degrees of freedom

Two degrees of freedom u, v (the displacements along the global coordinate directions).

Output of strains and stresses (at the integration points)

1 = zz, 2 = rr, 3 = $\vartheta\vartheta$, 4 = zr.

Analysis types

- Infinitesimal elasto-plasticity.

- Masonry-like materials.

C5. Element 5 (Thin shell)

Eight-node isoparametric element, the interpolating functions are linear for displacement and quadratic for rotations.

Connectivity

Eight nodes numbered as shown in Figure 5.1.

Integration

Integrating on the element surface is performed numerically using four-point Gaussian quadrature. The relative integration points are shown in Figure 5.2. Integration within the thickness is also performed numerically using instead the Sympson method, the number of sections into which it is divided being defined in card SHELL SECT (the default value is 1, the maximum number of sections is 99).

Geometry

For homogeneous shells the shell thickness must be specified; for non-homogeneous shells the thickness of each section must be specified (COMPOSITE option). The thickness of the whole shell or of each layer can be specified at each corner node by using the user routine UGEOM.

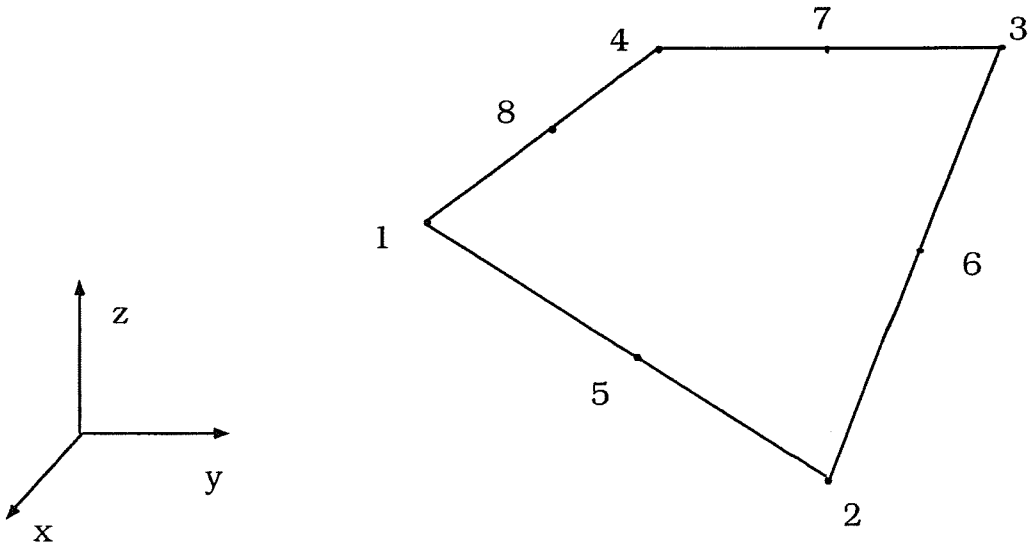

Figure 5.1 Element 5.

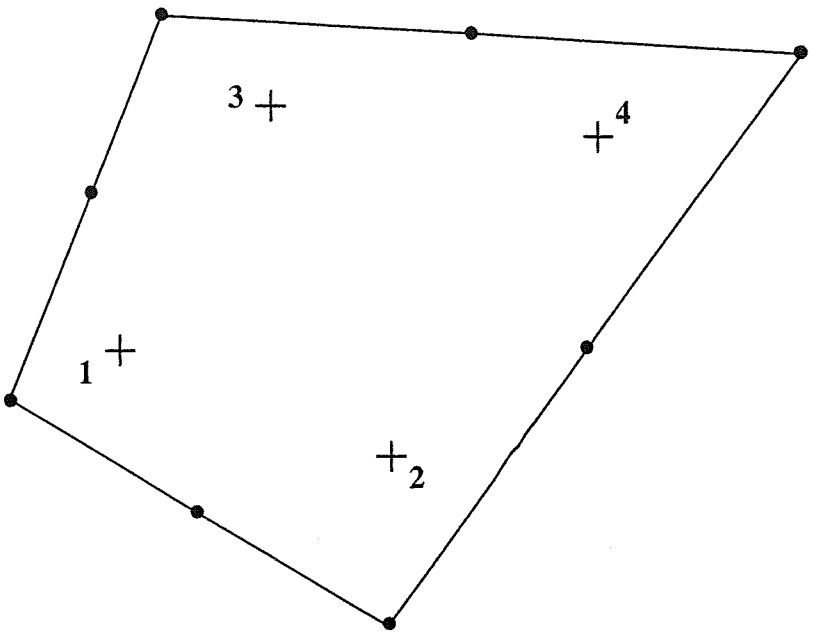

Figure 5.2 Integration points of element 5.

Distributed loads

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Body force. The three components of the force per unit volume in the global reference system must be assigned.
2	Body force. The three global components of the force per unit volume for each section are calculated in the user subroutine FORCEM.
11	Pressure on the 1-2 edge (force per unit length) in the global reference system.
13	Pressure on the 1-2 edge in the global reference system. The three components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.
21	Pressure on the 2-3 edge (force per unit length) in the global reference system.
23	Pressure on the 2-3 edge in the global reference system. The three components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM

31 Pressure on the 3-4 edge (force per unit length) in the global reference system.

33 Pressure on the 3-4 edge in the global reference system. The three components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM

41 Pressure on the 4-1 edge (force per unit length) in the global reference system.

43 Pressure on the 4-1 edge in the global reference system. The three components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM

51 Pressure on the surface of the element (force per unit area) in the global reference system.

52 Pressure on the surface of the element in the local reference system. The first component of the force per unit area is directed tangential to the surface in the 1-2 direction; the second component is directed tangential in the 2-3 direction; the third is directed perpendicular to the element's surface in the positive direction determined by the right-hand rule.

53 Pressure on the surface of the element in the global reference system. The three components of the force per unit area on the four nodes of the element are to be specified in the user subroutine FORCEM.

54 Pressure on the surface of the element in the local reference system. The three components of the force per unit area on the four nodes of the element are to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point loads on corner nodes, concentrated moments on mid-side nodes and to thermal dilatation loads.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangle. In this case the collapsed edge has zero stiffness.

Nodal coordinates

Three global coordinates in the x, y and z directions. It is not necessary to specify the coordinates of mid-side nodes.

Nodal degrees of freedom

The corner nodes have three degrees of freedom u, v and w (the displacements along the global coordinate directions); the degree of freedom of the mid-side nodes is the rotation q about the corresponding edge (positive if counterclockwise).

Output of strains and stresses (at the integration points)

Strains are printed for the four Gauss points of the mean surface of the shell. The components of the strain are given in the local orthonormal reference system s, t, n in the following order:

1 = ss, 2 = tt, 3 = st.

The three values of stress are printed for the four integration points for each section of the element. The stress components are given in the local orthonormal reference system s, t, n in the following order:

1 = ss, 2 = tt, 3 = st.

Analysis types

- Linear elasticity.

- Masonry-like materials.

C6. Element 6 (Plane strain)

Four-node isoparametric element with bilinear interpolating functions.

Connectivity

Four nodes numbered as shown in Figure 6.1.

Integration

The element is integrated numerically using four points (Gaussian quadrature). Integration points are shown in Figure 6.1. If selective reduced integration is required, the hydrostatic part of the strain is calculated using one integration point, namely the centroid of the element.

Geometry

The thickness of the element must be specified (default 1).

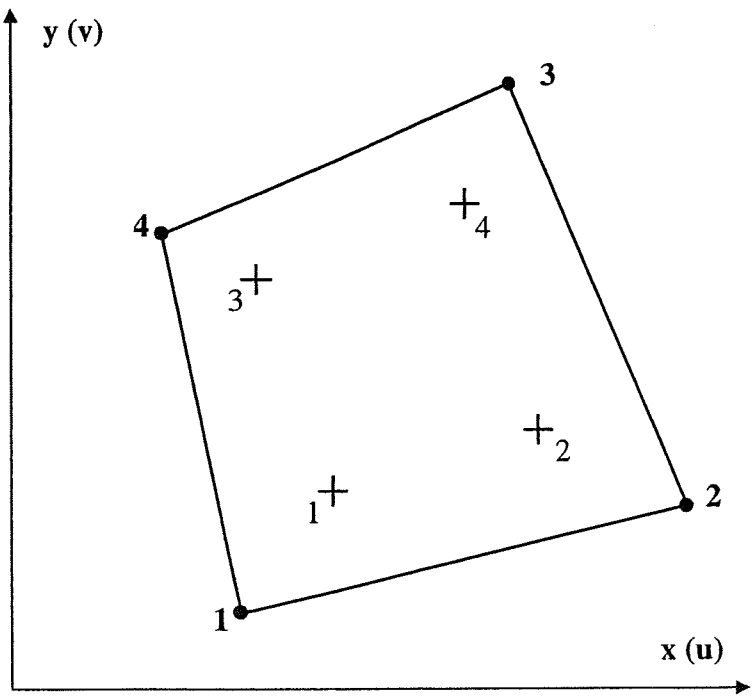

Fig. 6.1 Element 6.

Distributed loads

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Body force. The two components of the force per unit volume in the global reference system must be assigned.
2	Body force. The two global components of the force per unit volume are calculated in the user subroutine FORCEM.
3	Centrifugal force. The rotation axis is orthogonal to the plane of the element; the angular velocity must be specified in the field reserved for the force magnitudes.
11	Pressure on the 1-2 edge (force per unit area) in the global reference system.
12	Pressure on the 1-2 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 1-2 direction.
13	Pressure on the 1-2 edge in the global reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.
14	Pressure on the 1-2 edge in the local reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.
21	Pressure on the 2-3 edge (force per unit area) in the global reference system.
22	Pressure on the 2-3 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 2-3 direction.
23	Pressure on the 2-3 edge in the global reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.
24	Pressure on the 2-3 edge in the local reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.
31	Pressure on the 3-4 edge (force per unit area) in the global reference system.

32 Pressure on the 3-4 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 3-4 direction.

33 Pressure on the 3-4 edge in the global reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.

34 Pressure on the 3-4 edge in the local reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.

41 Pressure on the 4-1 edge (force per unit area) in the global reference system.

42 Pressure on the 4-1 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 4-1 direction.

43 Pressure on the 4-1 edge in the global reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.

44 Pressure on the 4-1 edge in the local reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point loads on nodes and to thermal dilatation loads.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangle.

Nodal coordinates

Two global coordinates in the x and y directions.

Nodal degrees of freedom

Two degrees of freedom u, v, the displacements along the global coordinate directions.

Output of strains and stresses (at the integration points)

1 = xx, 2 = yy, 3 = xy.

Analysis types

- Infinitesimal elasto-plasticity.

- Finite elasto-plasticity.

- Masonry-like materials.

C7. Element 7 (Axisymmetric quadrilateral)

Four-node isoparametric element with bilinear interpolating function.

Connectivity

Four nodes numbered as shown in Figure 7.1.

Integration

The element is integrated numerically using four points (Gaussian quadrature). Integration points are shown in Figure 7.1. If selective reduced integration is required, the hydrostatic part of the strain is calculated using one integration point, namely the centroid of the element.

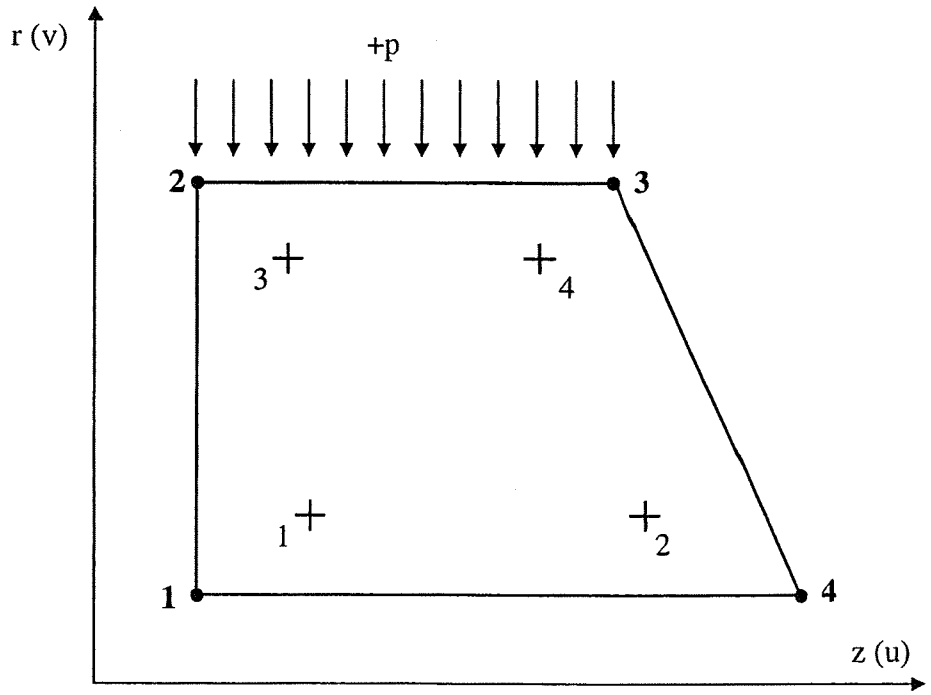

Fig. 7.1 Element 7

Distributed loads

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Body force. The two components of the force per unit volume in the global reference system must be assigned.

2 Body force. The two global components of the force per unit volume are calculated in the user subroutine FORCEM.

3 Centrifugal force. The rotation axis is the axis of symmetry, the angular velocity must be specified in the field reserved for the force magnitudes.

11 Pressure on the 1-2 edge (force per unit area) in the global reference system.

12 Pressure on the 1-2 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 1-2 direction.

13 Pressure on the 1-2 edge in the global reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.

14 Pressure on the 1-2 edge in the local reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.

21 Pressure on the 2-3 edge (force per unit area) in the global reference system.

22 Pressure on the 2-3 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 2-3 direction.

23 Pressure on the 2-3 edge in the global reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.

24 Pressure on the 2-3 edge in the local reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.

31 Pressure on the 3-4 edge (force per unit area) in the global reference system.

32 Pressure on the 3-4 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 3-4 direction.

33 Pressure on the 3-4 edge in the global reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.

34 Pressure on the 3-4 edge in the local reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.

41 Pressure on the 4-1 edge (force per unit area) in the global reference system.

42 Pressure on the 4-1 edge in the local reference system. The first component of the force per unit length is orthogonal to the edge and is positive if directed towards the element's interior; the second component is tangential to the edge in the 4-1 direction.

43 Pressure on the 4-1 edge in the global reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.

44 Pressure on the 4-1 edge in the local reference system. The two components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point loads on nodes and to thermal dilatation loads.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangle.

Nodal coordinates

Two global coordinates in the z and r directions.

Nodal degrees of freedom

Two degrees of freedom u, v, the displacements along the global coordinate directions.

Output of strains and stresses (at the integration points)

1 = zz, 2 = rr, 3 = $\vartheta\vartheta$, 4 = zr.

Analysis types

- Infinitesimal elasto-plasticity.

- Finite elasto-plasticity.

- Masonry-like materials.

C8. Element 8 (three-dimensional element)

Eight-node isoparametric element with bilinear interpolating functions. This element is suitable for three-dimensional analysis with finite strains.

Connectivity

Eight nodes numbered as shown in Figure 8.1.

Integration

The element is integrated numerically using eight points (Gaussian quadrature). Integration points are shown in Figure 8.2. If selective reduced integration is required, the hydrostatic part of the strain is calculated using one integration point, namely the centroid of the element.

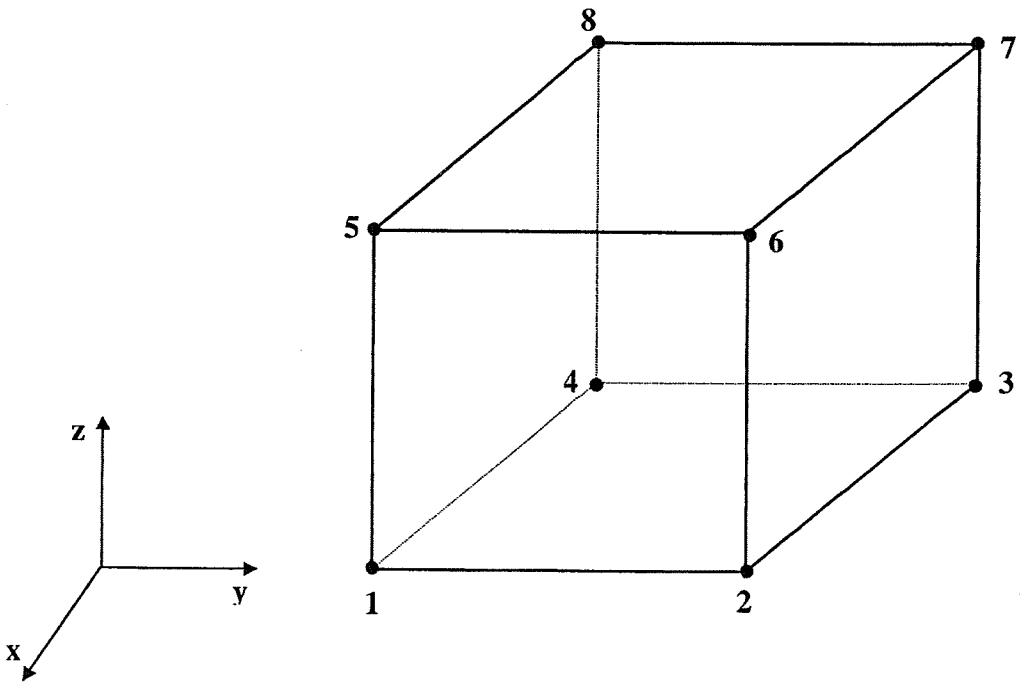

Fig. 8.1 Element 8.

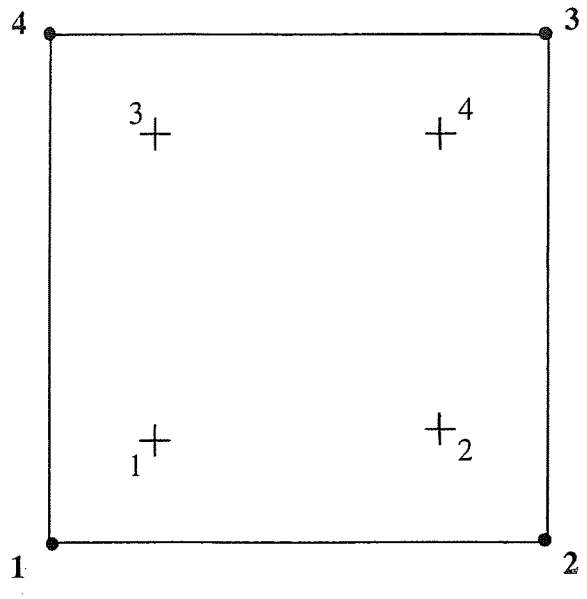

Figure 8.2 Integration points for element 8.

Distributed loads

Distributed loads chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Body force. The three components of the force per unit volume in the global reference system must be assigned.
2	Body force. The three global components of the force per unit volume are calculated in the user subroutine FORCEM.
3	Centrifugal force. The rotation axis must be defined in the card ROTATION AXIS, the angular velocity must be specified in the field reserved for the force magnitudes.
11	Pressure on the 1-2-3-4 face (load per unit area) in the global reference system.
12	Pressure on the 1-2-3-4 face in the local reference system. The first component of the force per unit area is tangential to the face in the 1-2 direction, the third component is orthogonal to the face and is positive if directed towards the element's interior; the second component has the direction given by the vector product of the first and third directions.
13	Pressure on the 1-2-3-4 face in the global reference system. The three components of the force per unit area on the four nodes of the face are to be specified in the user subroutine FORCEM.

14 Pressure on the 1-2-3-4 face in the local reference system. The three components of the force per unit area on the four nodes of the face are to be specified in the user subroutine FORCEM.

21 Pressure on the 5-8-7-6 face (load per unit area) in the global reference system.

22 Pressure on the 5-8-7-6 face in the local reference system. The first component of the force per unit area is tangential to the face in the 5-8 direction, the third component is orthogonal to the face and is positive if directed towards the element's interior; the second component has the direction given by the vector product of the first and third directions.

23 Pressure on the 5-8-7-6 face in the global reference system. The three components of the force per unit area on the four nodes of the face are to be specified in the user subroutine FORCEM.

24 Pressure on the 5-8-7-6 face in the local reference system. The three components of the force per unit area on the four nodes of the face are to be specified in the user subroutine FORCEM.

31 Pressure on the 1-5-6-2 face (load per unit area) in the global reference system.

32 Pressure on the 1-5-6-2 face in the local reference system. The first component of the force per unit area is tangential to the face in the 1-5 direction, the third component is orthogonal to the face and is positive if directed towards the element's interior; the second component has the direction given by the vector product of the first and third directions.

33 Pressure on the 1-5-6-2 face in the global reference system. The three components of the force per unit area on the four nodes of the face are to be specified in the user subroutine FORCEM.

34 Pressure on the 1-5-6-2 face in the local reference system. The three components of the force per unit area on the four nodes of the face are to be specified in the user subroutine FORCEM.

41 Pressure on the 2-6-7-3 face (load per unit area) in the global reference system.

42 Pressure on the 2-6-7-3 face in the local reference system. The first component of the force per unit area is tangential to the face in the 2-6 direction, the third component is orthogonal to the face and is positive if directed towards the element's interior; the second component has the direction given by the vector product of the first and third directions.

43 Pressure on the 2-6-7-3 face in the global reference system. The three components of the force per unit area on the four nodes of the face are to be specified in the user subroutine FORCEM.

44 Pressure on the 2-6-7-3 face in the local reference system. The three components of the force per unit area on the four nodes of the face are to be specified in the user subroutine FORCEM.

51 Pressure on the 3-7-8-4 face (load per unit area) in the global reference system.

52 Pressure on the 3-7-8-4 face in the local reference system. The first component of the force per unit area is tangential to the face in the 3-7 direction, the third component is orthogonal to the face and is positive if directed towards the element's interior; the second component has the direction given by the vector product of the first and third directions.

53 Pressure on the 3-7-8-4 face in the global reference system. The three components of the force per unit area on the four nodes of the face are to be specified in the user subroutine FORCEM.

54 Pressure on the 3-7-8-4 face in the local reference system. The three components of the force per unit area on the four nodes of the face are to be specified in the user subroutine FORCEM.

61 Pressure on the 4-8-5-1 face (load per unit area) in the global reference system.

62 Pressure on the 4-8-5-1 face in the local reference system. The first component of the force per unit area is tangential to the face in the 4-8 direction, the third component is orthogonal to the face and is positive if directed towards the element's interior; the second component has the direction given by the vector product of the first and third directions.

63 Pressure on the 4-8-5-1 face in the global reference system. The three components of the force per unit area on the four nodes of the face are to be specified in the user subroutine FORCEM.

64 Pressure on the 4-8-5-1 face in the local reference system. The three components of the force per unit area on the four nodes of the face are to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point loads on nodes and to thermal dilatation loads.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a tetrahedron.

Nodal coordinates

Three global coordinates in the x, y and z directions.

Nodal degrees of freedom

Three degrees of freedom u, v, w, the displacements along the global coordinate directions.

Output of strains and stresses (at the integration points)

1 = xx, 2 = yy, 3 = zz, 4 = xy, 5 = yz, 6 = xz.

Analysis types

- Infinitesimal elasto-plasticity.
- Finite elasto-plasticity.
- Masonry-like materials.

C9. Element 9 (Beam)

Two-node isoparametric element, the interpolating functions are linear for displacements and rotations.

Connectivity

Two nodes numbered as shown in Figure 9.1. Moreover, a local reference system e_1, e_2, e_3, have to be specified, by using the AXIS option (e_1 and e_2 define the principal directions on the cross section and e_3 is in the axial direction, from node 1 to node 2.

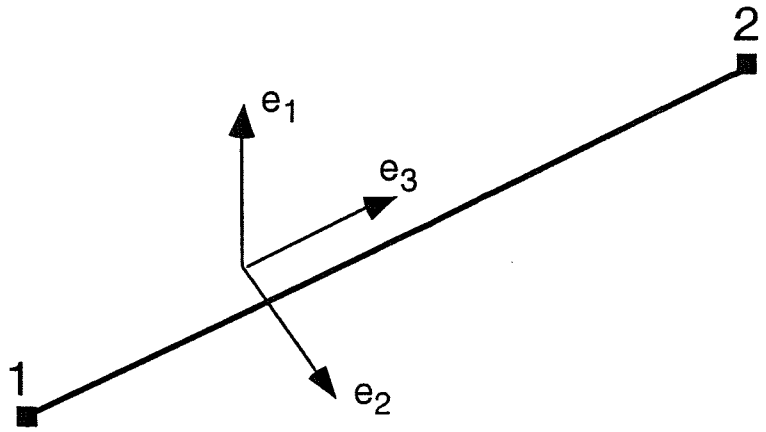

Fig. 9.1 Layout of the beam element type 9.

Integration

The integration along the element axis is performed numerically using two-point Gaussian quadrature. Integration on the cross section is also performed numerically using instead the Sympson method, the number of fibers into which it is divided being defined in card BEAM SECT (the default value is 3x3, the maximum number of fibers is 21x21).

Geometry

By default, the beam is supposed to have a rectangular cross section, so for homogeneous beams the thickness, along e_1 and e_2 directions, must be specified; for non-homogeneous beams the thickness of each fiber must be specified following the order shown in Fig. 9.2. Is

to be remarked that it is possible to give the fiber thicknesses on a nodal basis by using the user routine UGEOM.

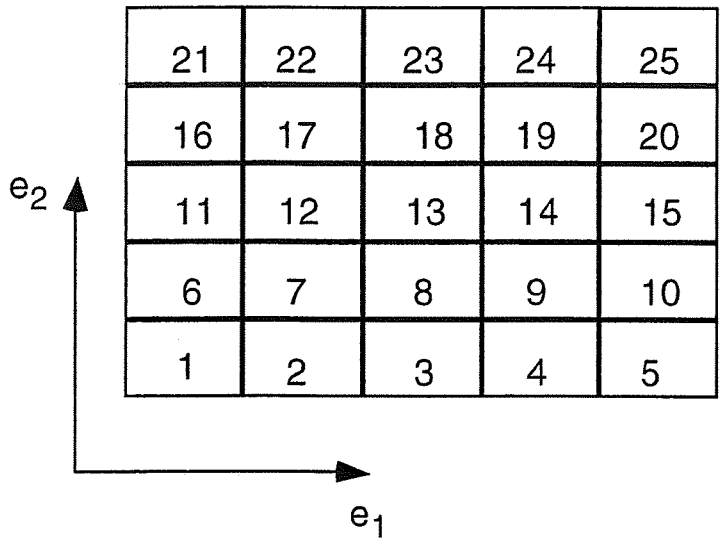

Fig. 9.2. Numbering scheme for the fibers of the beam on a 5x5 example.

Distributed loads

Distributed loads chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Body force and/or moment. The three components of the force and moment per unit volume in the global reference system must be assigned.
2	Body force and/or moment. The three global components of the force and moment per unit volume are calculated in the user subroutine FORCEM.
3	Centrifugal force. The rotation axis must be defined in the card ROTATION AXIS, the angular velocity must be specified in the field reserved for the force magnitudes.
11	Pressure on beam axis (load per unit length) in the global reference system.
12	Pressure on the beam axis in the local reference system e_1, e_2, e_3.
13	Pressure on the beam axis in the global reference system. The components of the load per unit length at the two nodes of the beam are to be specified in the user subroutine FORCEM.

14 Pressure on the beam axis in the local reference system. The components of the load per unit length at the two nodes of the beam are to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point loads and/or concentrated moments on nodes. Thermal loads are not yet available.

Nodal coordinates

Three global coordinates in the x, y and z directions.

Nodal degrees of freedom

Six degrees of freedom u, v, w (the displacements along the global coordinate directions) and q_X, q_Y, q_Z (the rotations, counterclockwise positive, around the global coordinate directions).

Output of strains and stresses characteristics (at the integration points)

Strains and stress characteristics are printed for the two Gauss points. The components of the strain characteristics are given in the local reference system e_1, e_2, e_3 in the following order :

1) ε axial stretch,
2) k_1 curvature change around e_1 axis,
3) k_2 curvature change around e_2 axis,
4) ψ twist around e_3 axis,
5) γ_1 shear deformation on the e_1-e_3 plane,
6) γ_2 shear deformation on the e_2-e_3 plane.

Analogously, for the stress characteristics we have:

1) N axial force,
2) M_1 moment around e_1 axis,
3) M_2 moment around e_2 axis,
4) M_3 torque around e_3 axis,
5) Q_1 shear force on the e_1-e_3 plane,
6) Q_2 shear force on the e_2-e_3 plane.

Analysis types

- Linear elasticity.

- Masonry-like materials.

C10. Element 10 (Thick shell)

Four-node isoparametric element, the interpolating functions are linear for displacements and rotations.

Connectivity

Four nodes numbered as shown in Figure 10.1.

Integration

Integrating on the element surface is performed numerically using four-point Gaussian quadrature. The relative integration points are shown in Figure 10.2. Integration within the thickness is also performed numerically using instead the Sympson method, the number of sections into which it is divided being defined in card SHELL SECT (the default value is 3, the maximum number of sections is 99).

Geometry

For homogeneous shells the shell thickness must be specified; for non-homogeneous shells the thickness of each section must be specified (COMPOSITE option). The thickness of the whole shell or of each layer can be specified at each node by using the user routine UGEOM.

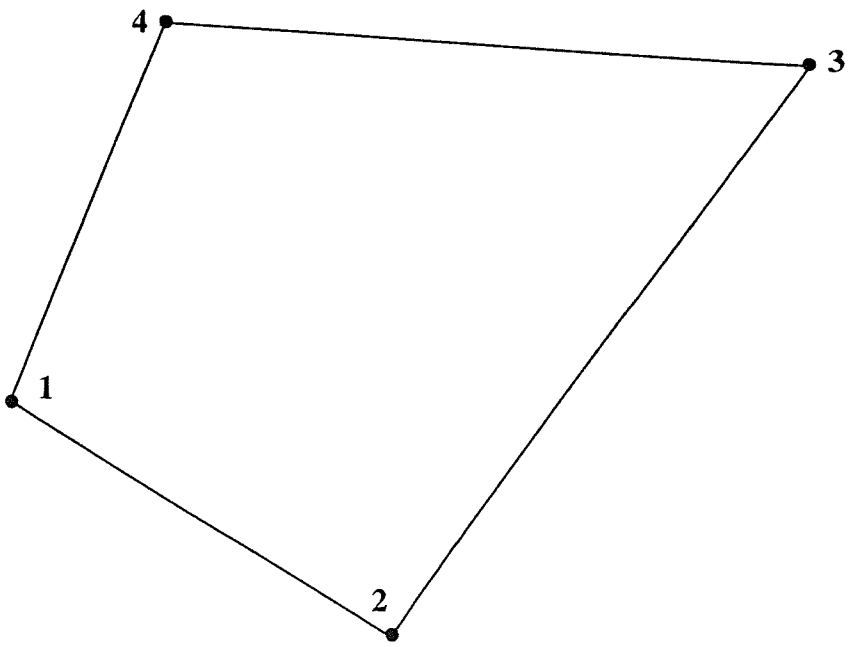

Figure 10.1 Element 10.

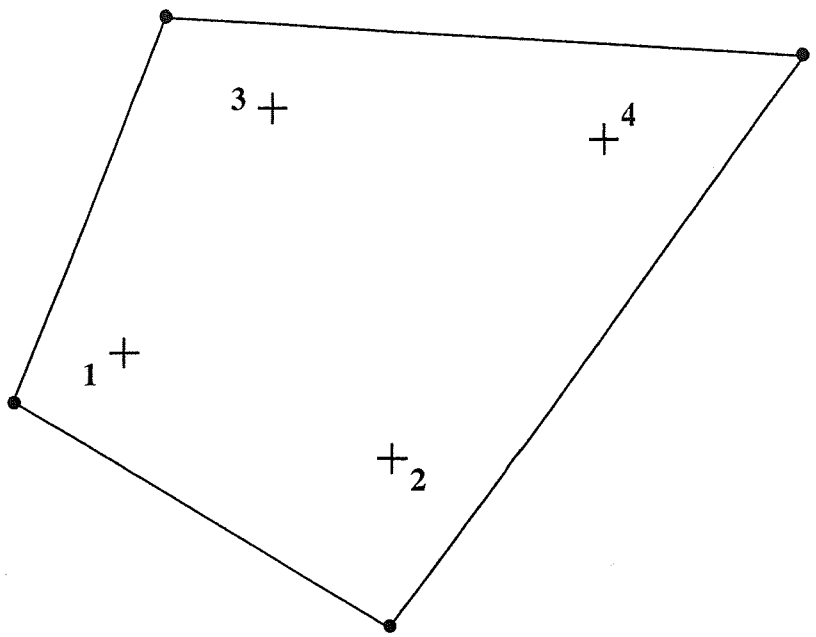

Figure 10.2 Integration points of element 10.

Distributed loads

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Body force. The three components of the force per unit volume in the global reference system must be assigned.
2	Body force. The three global components of the force per unit volume for each section are calculated in the user subroutine FORCEM.
11	Pressure on the 1-2 edge (force per unit length) in the global reference system.
13	Pressure on the 1-2 edge in the global reference system. The three components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM.
21	Pressure on the 2-3 edge (force per unit length) in the global reference system.
23	Pressure on the 2-3 edge in the global reference system. The three components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM

31 Pressure on the 3-4 edge (force per unit length) in the global reference system.

33 Pressure on the 3-4 edge in the global reference system. The three components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM

41 Pressure on the 4-1 edge (force per unit length) in the global reference system.

43 Pressure on the 4-1 edge in the global reference system. The three components of the force per unit length on the two nodes of the edge are to be specified in the user subroutine FORCEM

51 Pressure on the surface of the element (force per unit area) in the global reference system.

52 Pressure on the surface of the element in the local reference system. The first component of the force per unit area is directed tangential to the surface in the 1-2 direction; the second component is directed tangential in the 2-3 direction; the third is directed perpendicular to the element's surface in the positive direction determined by the right-hand rule.

53 Pressure on the surface of the element in the global reference system. The three components of the force per unit area on the four nodes of the element are to be specified in the user subroutine FORCEM.

54 Pressure on the surface of the element in the local reference system. The three components of the force per unit area on the four nodes of the element are to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point loads and/or concentrated moments on nodes, and to thermal dilatation loads.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangle.

Nodal coordinates

Three global coordinates in the x, y and z directions.

Nodal degrees of freedom

Six degrees of freedom u, v, w (the displacements along the global coordinate directions) and q_X, q_Y, q_Z (the rotations, counterclockwise positive, around the global coordinate directions).

Output of strains and stresses (at the integration points)

Strains are printed for the four Gauss points of the mean surface of the shell. The components of the strain are given in the local orthonormal reference system s, t, n in the following order :

1 = ss, 2 = tt, 3 = st, 4 = tn, 5 = sn.

The three values of stress are printed for the four integration points for each section of the element. The stress components are given in the local orthonormal reference system s, t, n in the following order :

1 = ss, 2 = tt, 3 = st, 4 = tn, 5 = sn.

Analysis types

- Linear elasticity.

C11. Element 11 (Plane heat transfer)

Eight-node isoparametric element with quadratic interpolating functions (geometrically it is identical to elements 2 and 3).

Connectivity

Eight nodes numbered in anticlockwise order.

Integration

The element is integrated numerically using nine integration points (Gaussian quadrature).

Distributed fluxes

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Surface flux. The heat quantity per unit area and unit time must be assigned.
2	Surface flux. The heat quantity per unit area and unit time is calculated in the user subroutine FORCEM.
11	Edge flux on the 1-2-5 edge (heat per unit length and unit time).
13	Edge flux on the 1-2-5 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.
21	Edge flux on the 2-3-6 edge (heat per unit length and unit time).
23	Edge flux on the 2-3-6 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.
31	Edge flux on the 3-4-7 edge (heat per unit length and unit time).
33	Edge flux on the 3-4-7 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.
41	Edge flux on the 4-1-8 edge (heat per unit length and unit time).
43	Edge flux on the 4-1-8 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point fluxes on nodes and convective fluxes by specifying the film coefficients and sink temperatures for each edge.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangle.

Nodal coordinates

Two global coordinates in the x and y directions.

Nodal degrees of freedom

One degree of freedom T (the nodal temperature).

Output of temperature gradient (at integration points)

1 = x, 2 = y.

Analysis types

- Linear and non-linear transient heat transfer analysis.

C12. Element 12 (Plane heat transfer)

Four-node isoparametric element with linear interpolating functions (geometrically it is identical to element 6).

Connectivity

Four nodes numbered in anticlockwise order.

Integration

The element is integrated numerically using four integration points (Gaussian quadrature).

Distributed fluxes

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Surface flux. The heat quantity per unit area and unit time must be assigned.
2	Surface flux. The heat quantity per unit area and unit time is calculated in the user subroutine FORCEM.
11	Edge flux on the 1-2 edge (heat per unit length and unit time).
13	Edge flux on the 1-2 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.
21	Edge flux on the 2-3 edge (heat per unit length and unit time).
23	Edge flux on the 2-3 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.
31	Edge flux on the 3-4 edge (heat per unit length and unit time).
33	Edge flux on the 3-4 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.
41	Edge flux on the 4-1 edge (heat per unit length and unit time).
43	Edge flux on the 4-1 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point fluxes on nodes and convective fluxes by specifying the film coefficients and sink temperatures for each edge.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangle.

Nodal coordinates

Two global co-ordinates in the x and y directions.

Nodal degrees of freedom

One degree of freedom T (the nodal temperature).

Output of temperature gradient (at integration points)

1 = x, 2 = y.

Analysis types

- Linear and non-linear transient heat transfer analysis.

C13. Element 13 (Axisymmetric heat transfer)

Eight-node isoparametric element with quadratic interpolating functions (geometrically it is identical to element 4).

Connectivity

Eight nodes numbered in anticlockwise order.

Integration

The element is integrated numerically using nine integration points (Gaussian quadrature).

Distributed fluxes

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Surface flux. The heat quantity per unit area and unit time must be assigned.
2	Surface flux. The heat quantity per unit area and unit time is calculated in the user subroutine FORCEM.
11	Edge flux on the 1-2-5 edge (heat per unit length and unit time).
13	Edge flux on the 1-2-5 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.
21	Edge flux on the 2-3-6 edge (heat per unit length and unit time).
23	Edge flux on the 2-3-6 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.
31	Edge flux on the 3-4-7 edge (heat per unit length and unit time).
33	Edge flux on the 3-4-7 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.
41	Edge flux on the 4-1-8 edge (heat per unit length and unit time).
43	Edge flux on the 4-1-8 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point fluxes on nodes and convective fluxes by specifying the film coefficients and sink temperatures for each edge.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangle.

Nodal coordinates

Two global coordinates in the x and y directions.

Nodal degrees of freedom

One degree of freedom T (the nodal temperature).

Output of temperature gradient (at integration points)

1 = x, 2 = y.

Analysis types

- Linear and non-linear transient heat transfer analysis.

C14. Element 14 (Axisymmetric heat transfer)

Four-node isoparametric element with linear interpolating functions (geometrically it is identical to element 7).

Connectivity

Four nodes numbered in anticlockwise order.

Integration

The element is integrated numerically using four integration points (Gaussian quadrature).

Distributed fluxes

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Surface flux. The heat quantity per unit area and unit time must be assigned.
2	Surface flux. The heat quantity per unit area and unit time is calculated in the user subroutine FORCEM.
11	Edge flux on the 1-2 edge (heat per unit length and unit time).
13	Edge flux on the 1-2edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.
21	Edge flux on the 2-3 edge (heat per unit length and unit time).
23	Edge flux on the 2-3 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.
31	Edge flux on the 3-4 edge (heat per unit length and unit time).
33	Edge flux on the 3-4 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.
41	Edge flux on the 4-1 edge (heat per unit length and unit time).
43	Edge flux on the 4-1 edge. The heat per unit length and unit time on the three nodes of the edge is to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point fluxes on nodes and convective fluxes by specifying the film coefficients and sink temperatures for each edge.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangle.

Nodal coordinates

Two global coordinates in the x and y directions.

Nodal degrees of freedom

One degree of freedom T (the nodal temperature).

Output of temperature gradient (at integration points)

1 = x, 2 = y.

Analysis types

- Linear and non-linear transient heat transfer analysis.

C15. Element 15 (3D heat transfer)

Eigth-node isoparametric element with linear interpolating functions (geometrically it is identical to element 8).

Connectivity

8 nodes numbered as in Fig. 8.1.

Integration

The element is integrated numerically using 8 integration points (Gaussian quadrature).

Distributed fluxes

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Volumetric flux. The heat quantity per unit volume and unit time must be assigned.
2	Volumetric flux. The heat quantity per unit volume and unit time is calculated in the user subroutine FORCEM.
11	Surface flux on the 1-2-3-4 face (heat per unit area and unit time).
13	Surface flux on the 1-2-3-4 face. The heat per unit area and unit time on the four nodes of the face is to be specified in the user subroutine FORCEM.
21	Surface flux on the 5-6-7-8 face (heat per unit area and unit time).
23	Surface flux on the 5-6-7-8 face. The heat per unit area and unit time on the four nodes of the face is to be specified in the user subroutine FORCEM.
31	Surface flux on the 1-5-6-2 face (heat per unit area and unit time).
33	Surface flux on the 1-5-6-2 face. The heat per unit area and unit time on the four nodes of the face is to be specified in the user subroutine FORCEM.
41	Surface flux on the 2-6-7-3 face (heat per unit area and unit time).
43	Surface flux on the 2-6-7-3 face. The heat per unit area and unit time on the four nodes of the face is to be specified in the user subroutine FORCEM.
51	Surface flux on the 3-7-8-4 face (heat per unit area and unit time).

53 Surface flux on the 3-7-8-4 face. The heat per unit area and unit time on the four nodes of the face is to be specified in the user subroutine FORCEM.

61 Surface flux on the 4-8-5-1 face (heat per unit area and unit time).

63 Surface flux on the 4-8-5-1 face. The heat per unit area and unit time on the four nodes of the face is to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point fluxes on nodes and convective fluxes by specifying the film coefficients and sink temperatures for each edge.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangular prism or to a tetrahedron.

Nodal coordinates

Three global coordinates in the x, y and z directions.

Nodal degrees of freedom

One degree of freedom T (the nodal temperature).

Output of temperature gradient (at integration points)

1 = x, 2 = y, 3 = z.

Analysis types

- Linear and non-linear transient heat transfer analysis.

C16. Element 16 (3D heat transfer)

Twenty-node isoparametric element with quadratic interpolating functions (geometrically it is identical to element 1).

Connectivity

Twenty nodes numbered as in Fig. 1.1.

Integration

The element is integrated numerically using twenty-seven integration points (Gaussian quadrature).

Distributed fluxes

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Volumetric flux. The heat quantity per unit volume and unit time must be assigned.
2	Volumetric flux. The heat quantity per unit volume and unit time is calculated in the user subroutine FORCEM.
11	Surface flux on the 1-2-3-4 face (heat per unit area and unit time).
14	Surface flux on the 1-2-3-4 face. The heat per unit area and unit time on the four nodes of the face is to be specified in the user subroutine FORCEM.
21	Surface flux on the 5-6-7-8 face (heat per unit area and unit time).
23	Surface flux on the 5-6-7-8 face. The heat per unit area and unit time on the four nodes of the face is to be specified in the user subroutine FORCEM.
31	Surface flux on the 1-5-6-2 face (heat per unit area and unit time).
33	Surface flux on the 1-5-6-2 face. The heat per unit area and unit time on the four nodes of the face is to be specified in the user subroutine FORCEM.
41	Surface flux on the 2-6-7-3 face (heat per unit area and unit time).
43	Surface flux on the 2-6-7-3 face. The heat per unit area and unit time on the four nodes of the face is to be specified in the user subroutine FORCEM.
51	Surface flux on the 3-7-8-4 face (heat per unit area and unit time).

53 Surface flux on the 3-7-8-4 face. The heat per unit area and unit time on the four nodes of the face is to be specified in the user subroutine FORCEM.

61 Surface flux on the 4-8-5-1 face (heat per unit area and unit time).

63 Surface flux on the 4-8-5-1 face. The heat per unit area and unit time on the four nodes of the face is to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point fluxes on nodes and convective fluxes by specifying the film coefficients and sink temperatures for each edge.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangular prism or to a tetrahedron.

Nodal coordinates

Three global coordinates in the x, y and z directions.

Nodal degrees of freedom

One degree of freedom T (the nodal temperature).

Output of temperature gradient (at integration points)

1 = x, 2 = y, 3 = z.

Analysis types

- Linear and non-linear transient heat transfer analysis.

C17. Element 17 (shell heat transfer)

Four-node isoparametric element with linear interpolating functions (geometrically it is identical to element 10).

Connectivity

Four nodes numbered as in Fig. 10.1.

Integration

Integrating on the element surface is performed numerically using four-point Gaussian quadrature. The relative integration points are shown in Figure 10.2. Integration within the thickness is also performed numerically using instead the Sympson method, the number of sections into which it is divided being defined in card SHELL SECT (the default value is 3, the maximum number of sections is 99).

Geometry

For homogeneous shells the shell thickness must be specified; for non-homogeneous shells the thickness of each section must be specified (COMPOSITE option). The thickness of the whole shell or of each layer can be specified at each node by using the user routine UGEOM.

Distributed fluxes

Distributed loads are chosen by value of IBODY, as follows:

IBODY	DESCRIPTION
1	Volumetric flux. The heat quantity per unit volume and unit time must be assigned.
2	Volumetric flux. The heat quantity per unit volume and unit time is calculated in the user subroutine FORCEM.
11	Edge flux on the 1-2 edge (heat per unit length and unit time).
13	Edge flux on the 1-2 edge. The heat per unit length and unit time on the two nodes of the edge is to be specified in the user subroutine FORCEM.
21	Edge flux on the 2-3 edge (heat per unit length and unit time).
23	Edge flux on the 2-3 edge. The heat per unit length and unit time on the two nodes of the edge is to be specified in the user subroutine FORCEM.

31 Edge flux on the 3-4 edge (heat per unit length and unit time).

33 Edge flux on the 3-4 edge. The heat per unit length and unit time on the two nodes of the edge is to be specified in the user subroutine FORCEM.

41 Edge flux on the 4-1 edge (heat per unit length and unit time).

43 Edge flux on the 4-1 edge. The heat per unit length and unit time on the two nodes of the edge is to be specified in the user subroutine FORCEM.

51 Surface flux (heat per unit area and unit time).

53 Surface flux. The heat per unit area and unit time on the four nodes is to be specified in the user subroutine FORCEM.

Moreover, the element may be subjected to point fluxes on nodes and convective fluxes by specifying the film coefficients and sink temperatures for each edge.

Collapse

By suitably repeating node numbers in the connectivity, the element may be reduced as far as a triangle.

Nodal coordinates

Three global coordinates in the x, y and z directions.

Nodal degrees of freedom

ITYRD degrees of freedom (temperatures) per node (the value of the ITYRD parameter is specified into the HEAT TRANSFER option)

ITYRD=2 (linear distribution trough the thickness)
1 Top surface temperature
2 Bottom surface temperature

ITYRD=3 (parabolic distribution trough the thickness)
1 Top surface temperature
2 Bottom surface temperature
3 Mid surface temperature.

Output of temperature gradient (at integration points)

The components of the temperature gradient are give in the local reference system of the shell. Namely, the first two components are along the local tangent direction, whereas the third component is along the shell normal unit vector.

Analysis types

- Linear and non-linear transient heat transfer analysis.

D. DESCRIPTION OF INPUT DATA

The input cards can be logically grouped into three sections:

I) Control cards. These control the program flow and define the general features of the analysis.

II) Model cards. These describe the geometrical model and material properties, and provide information concerning the algorithm used and the accuracy of analysis.

III) Load cards. They describe the load increments and allow re-definition of the boundary conditions.

In an input stream, the cards of each set follow one another in the order I, II, III. Within each section, the cards or groups of cards may have any order. The cards have a free format and fields are separated by blanks and/or commas. Integer or real fields in a card can be constituted by a number with a maximum length of ten characters; sign, point and exponent (E or D) included. If a card requires one or more extensions, the character '=' must be written after the last field in the card, thus the next card is interpreted as a continuation of the previous card. The number of continuation cards cannot exceed a maximum of 999. Comment cards, on the other hand, cannot be continued; if other comment rows are necessary, extra comment cards must be added.

D1. CONTROL CARDS

Index of keywords

ALIAS

Element aliases.

Field	Format	Content	Variable
1	A	ALIAS	
2*I	I	Alias number identifier of an element type.	JALIA(1, I)
2*I+1	I	The corresponding actual element type.	JALIA(2, I)

BEAM SECT

Definition of the number of integration points on the cross section of the beam elements.

Field	Format	Content	Variable
1	A	BEAM SECT	
2	I	Number of integration points along the first local direction on the cross section of the beam. This number must be an odd number between 3 and 21 (including the extreme values).	MBINT(1)
3	I	Number of integration points along the second local direction on the cross section of the beam. This number must be an odd number between 3 and 21 (including the extreme values).	MBINT(2)

COMMENT or $

Comment card.

Field	Format	Content	Variable
1	A	COMMENT or $	
2	A	Comment text.	

Note: Comment cards may be inserted anywhere.

COMPOSITE

Shell elements with sections made up of different materials and/or with different thickness will be used.

Field	Format	Content	Variable
1	A	COMPOSITE	

CONTACT

Rigid surfaces for contact problems will be present. In this version of NOSA contact problems are limited to plane strain and axisymmetric situations.

Field	Format	Content	Variable
1	A	CONTACT	
2	I	Maximum number of rigid surfaces used.	MSURF
3	I	maximum number of parts (segments or circumference arcs) constituting each rigid surface.	MPART

DAMPING

Viscous and/or numerical damping will be present in dynamic analysis.

Field	Format	Content	Variable
1	A	DAMPING	IDAMP

DEFORMABLE

Deformable bodies for contact problems will be present. In the current version of NOSA only one deformable body can be considered.

Field	Format	Content	Variable
1	A	DEFORMABLE	
2	I	Maximum number of deformable bodies (1).	MDEFO

DIST LOADS

Card for the definition of the maximum number of distributed loads that will be used in the analysis.

Field	Format	Content	Variable
1	A	DIST LOADS	
2	I	Maximum number of distributed loads (default 3).	MDIST

DYNAMIC

Dynamic analysis using the Newmark method of integration will be performed.

Field	Format	Content	Variable
1	A	DYNAMIC	IDYNA

ELASTIC

Linear elastic analysis with multiple loads will be performed.
Each load defined after this card will be interpreted as total load. If the card ELASTIC is not filled in, the load will be interpreted as incremental.

Field	Format	Content	Variable
1	A	ELASTIC	IELAS

ELEMENTS

Card for declaring the identification codes of the elements which will be used in the analysis.

Field	Format	Content	Variable
1	A	ELEMENTS	
2	I	Identification code of the i-th type of element. The maximum number of element types is 30	NTYPE(I)

Note: when the card ELEMENT is present, the element types declared in this card substitute the elements declared in the SIZING card. At least one element type must be declared.

END

End of the control cards.

Field	Format	Content	Variable
1	A	END	

Note: this card is obligatory.

ENERGY

Setting up the calculation of mechanical energy (kinetic and potential), as well as of the work done by the applied external forces, for dynamic and static stress analyses.

Field	Format	Content	Variable
1	A	ENERGY	LENER

FILM

Film coefficients of convective heating/cooling will be read in.

Field	Format	Content	Variable
1	A	FILM	IIFILM

FINITE STRAIN

Finite deformations will be considered in the analysis.

Field	Format	Content	Variable
1	A	FINITE STRAIN	IFINI

FOLLOWER FORCES

Follower forces will be used in the analysis. This option is active for element types 6, 7 and 8. The loads declared in the DIST LOADS cards will be calculated with respect to the current configuration, not to the initial one.

Field	Format	Content	Variable
1	A	FOLLOWER FORCES	IFOLL

FRICTION

In contact problems, friction forces will be taken into account.

Field	Format	Content	Variable
1	A	FRICTION	
2	I	= 0, frictionless contact; = 1, contact with friction.	IFRIC

HARDENING

Hardening laws of the material will be assigned through cards.

Field	Format	Content	Variable
1	A	HARDENING	
2	I	Maximum number of data pairs that constitute every hardening curve (default 0).	MHARD

HEAT TRANSFER

A transient heat transfer analysis will be performed.

Field	Format	Content	Variable
1	A	HEAT TRANSFER	IHEAT
2	I	Used to set the number of nodal temperatures and the order of interpolation trough the thickness for the heat transfer shell element type 17.	ITYRD
		=1, default value, valid for all element types except type 17,	
		=2, linear interpolation, each node will have 2 degrees of freedom, i.e. the temperature at the top and bottom surface of the shell,	
		=3, parabolic interpolation, each node will have 3 degrees of freedom, i.e. the temperature at the top, bottom and mid surface of the shell.	

ITRESS

The initial stress field will be read from a suitable file (FORTRAN unit 22).

Field	Format	Content	Variable
1	A	ITRESS	ITRES

LUMPED MASS

A lumped mass matrix will be used in carrying out dynamic analyses.

Field	Format	Content	Variable
1	A	LUMPED	LUMP

MASONRY

Masonry-like materials will be used.

Field	Format	Content	Variable
1	A	MASONRY	IMASO

MATERIALS

Card for declaring the maximum number of different materials that will be used in the analysis.

Field	Format	Content	Variable
1	A	MATERIALS	
2	I	Maximum number of different materials (default 1)	NMATS

POTENTIAL

Card for declaring the maximum number of different materials with anisotropic yield (Hill's potential theory).

Field	Format	Content	Variable
1	A	POTENTIAL	
2	I	Maximum number of different materials with anisotropic yield (default NMATS).	NMAPH

Note: the HARDENING card is required.

PRINT LEVEL

Card for setting the print output level.

Field	Format	Content	Variable
1	A	PRINT	
2	I	Print output level (default 1); = 1; displacements, reactions, stresses and strains at the end of the increment will be printed; > 1; displacements, reactions, stresses, strains and incremental nodal forces for each element at the end of the increment will be printed.	LVPRT
3	I	Print frequency; the data will be printed every IFREQP increments (default 1).	IFREQP
4	I	= 0; the data of all elements will be printed; = 1; the data of a selected set of elements will be printed.	NELPR
5	I	= 0; the data of all nodes will be printed; = 1; the data of a selected set of nodes will be printed.	NODPR

RESTART

The analysis is the continuation of a previous one. All information relative to the problem must be read from a suitable file (FORTRAN unit n.99).

Field	Format	Content	Variable
1	A	RESTART	IREST

Note: if this card is present, the other control and model cards need not be declared. After the END card, the cards for definition of incremental loads must be assigned.

SCALE

Card for scaling the results of the problem at the first incremental load in such a way that the points nearest the first yield are brought to that limit.

Field	Format	Content	Variable
1	A	SCALE	ISCAL

Note: This option works only in the case of infinitesimal elastic-plastic behavior.

SETNAME

Sets of elements and/or nodes will be used.

Field	Format	Content	Variable
1	A	SETNAME	
2	I	Maximum number of element sets.	MSETE
3	I	Maximum number of node sets.	MSETN
4	I	Maximum number of items in each set.	MSET
5	I	Maximum number of items in each compound set (item list formed by applying logical operators to existing sets).	MSETEN

SHELL SECT

Card for defining the number of integration points within the thickness for shell elements.

Field	Format	Content	Variable
1	A	SHELL SECT	
2	I	Number of integration points (default 3, maximum value 99).	MSHEL

Note: the number of integration points must be odd. The Simpson integration points are located on the mean surface of each section for composite shells, whereas for homogeneous shells they are located on the boundary of layers.

SIZING

Card for the defining the analysis dimension.

Field	Format	Content	Variable
1	A	SIZING	
2	I	Dimension of the work vector VEC containing real variables. This dimension must be less than or equal to the dimension declared in the main program MNOSA.	MSIZR
3	I	Dimension of the work vector IVE containing integer variables. This dimension must be less than or equal to the dimension declared in the main program MNOSA.	MSIZI
4	I	Total number of elements.	NELEM
5	I	Total number of nodes.	NPOIN
6...15	I	Identification code of the element used.	NTYPE(I)

Note: this card is obligatory.

STOP

If this card is present, the analysis must end after the reading of control and model cards and when memory allocation has been performed, with no further calculations.

Field	Format	Content	Variable
1	A	STOP	ISTOP

TEMP TABLES

This card declares that the mechanical properties (Young modulus, Poisson ratio and coefficient of thermal expansion) can be variable with the temperature.

Field	Format	Content	Variable
1	A	TEMP TABLES	
2	I	Maximum number of data tables.	MTTEMP
3	I	Maximum number of data pairs in each table.	MTTAB

THERMAL LOADS

Thermal loads due to temperature changes will be taken into account. This card declares the way to read the temperature field.

Field	Format	Content	Variable
1	A	THERMAL LOADS	IRDTMP
2	I	=1, nodal temperatures will be read (this is the value to be used with all element type, except with shells. =2, (for shell elements only) the temperature inside the thickness will be calculated by linearly interpolating the temperatures of the lower and upper layers. =3, (for shell elements only) the temperature inside the thickness will be calculated by parabolic interpolation of the temperatures of the lower, middle and upper layers.	ITYRD

TIE

Tying option will be used in the analysis.

Field	Format	Content	Variable
1	A	TIE	
2	I	Maximum number of tied nodes.	MTIE
3	I	Maximum number of retained nodes for each tied node.	MRET

TITLE

Card for assigning a title to the analysis.

Field	Format	Content	Variable
1	A	TITLE	
2	A	Title of the analysis (max 70 characters including blanks).	TITLE

Note: the presence of this card is obligatory. The first field must be aligned to the left.

REMARK

The control cards TITLE, SIZING and END are obligatory. Other cards must be declared when the user wishes to use a particular option or when input values different from the default values are required.

D2. MODEL CARDS

Index of keywords

REMARK

In the input cards it may be necessary to define lists of nodes and/or elements. A list may be specified as follows:

- a sequence of integers (1 10 33.....);

- in the form "n TO m <BY k>" (if k is not specified, it is assumed to be equal to 1);

- a sequence of names of sets connected by logic operators.

The logical operators are:

AND	insert into the current list those entities belonging to the set declared on the right of this operator,
EXCEPT	exclude from the current list all entities in the set declared on the right of this operator;
INTERSECT	select the entities contemporaneously belonging to the connected sets.

A series of logical operations on sets are always resolved from left to right. Thus, each operator always acts upon the list resulting from the operations performed up to that point (i.e. the result of all operations on its left) and the set declared on its immediate right.

BODY

Cards for defining deformable bodies present in the structure.

Field	Format	Content	Variable
First card			
1	A	BODY	
Second card			
1	I	Number of deformable bodies.	NDEFO

The following cards must be repeated NDEFO times.

Third card			
1...M	I	List of elements belonging to the i-th deformable body.	IDEFO

Note: In the current version of NOSA only one deformable body, constituted by all elements of the mesh, can be taken into consideration.

BOUNDARY CONDITIONS

Card for definition of boundary conditions.

Field	Format	Content	Variable
First card			
1	A	BOUNDARY CONDITIONS	
Second card			
1	I	Number of groups of boundary conditions.	NBOUN

The following cards must be repeated NBOUN times.

Third card			
1...N	R	Value of the i-th prescribed degree of freedom.	PRESC
Fourth card			
1...N	I	List of prescribed degrees of freedom.	IFPRE
Fifth card			
1...M	I	List of constrained nodes.	

Note: At least one group of this card type is required and, in the case of static analysis, constraints must be such as to avoid at least rigid body motions.

Note: If negative values are specified in the degrees of freedom list, the user routine UBND will manage the boundary conditions relative to the degree of freedom given by the absolute value of the indicate negative numbers.

COMMENT o $

Comment card.

Field	Format	Content	Variable
1	A	COMMENT o $	
2	A	Comment text.	

Note: the comment cards may be inserted at any point.

COMPOSITION

Cards for defining the composition of elements.

Field	Format	Content	Variable
First card			
1	A	COMPOSITION	
Second card			
1	I	Number of groups of elements having the same composition.	NCOMGR

The following cards must be repeated NCOMGR times.

Third card			
1...KLAYR	I	The identification number of the material of the element. If the COMPOSITE option has been set, it is the identification number of the material of a single layer (if KLAYR<MSHEL, the layers from KLAYR+1 to MSHEL are constituted of the KLAYR-th material).	MATNO
Fourth card			
1	I	List of elements having the composition defined in the previous card.	

CONNECTIVITY

Cards for defining the mesh connectivity.

Field	Format	Content	Variable
First card			
1	A	CONNECTIVITY	
Second card			
1	I	Number of elements for which connectivity is assigned. If zero or blank, then NEL=NELEM.	NEL
Third card (1 card for each element)			
1	I	Number of the element.	
2	I	Element type.	
3.NEL+2	I	Identifier numbers of nodes of the element.	LNODS

Note: at least one group of these cards is obligatory.

CONTROL

Cards for defining the parameters, which control the accuracy of numerical, results the analysis.

Field	Format	Content	Variable
First card			
1	A	CONTROL	
Second card			
1	I	Maximum number of increments (default 4).	MAXINC
2	I	Maximum number of iterations for each increment (default 3).	MITER
3	I	Type of algorithm used for the solution of the system. = 1, initial stiffness matrix, = 2; modified Newton-Raphson method, = 3, Newton-Raphson method. The default value is 3.	NALGO
Third card			
1	R	Ratio in % between the norm of residual force and the norm of total force, including reactions, in order to an increment is considered as converged. For shell and beam elements, the moments are divided by the corresponding thickness to be dimensionally homogeneous to forces.	TOLER
2	R	Minimum value of the norm of total force. If the norm is less than FUZTOL, the convergence control is skipped (default 1.D-08).	FUZTOL
3	R	For dynamic analyses it is the minimum displacements change. If the norm of displacement is less than FUZDIS, the convergence control is skipped (default 1.D-12).	FUZDIS
		For heat transfer analyses it is the minimum temperature change. If the maximum temperature change is less than FUZTMP, the convergence control is skipped (default 1D.-08).	FUZTMP

4	R	For heat transfer analyses it is the maximum temperature change allowed in automatic time stepping mode (default 20.D0). If the maximum temperature change is greater than TMPFIN, the increment is repeated with reduced time step in order that the maximum temperature change be equal to TMPFIN (this can be obtained with precision in linear cases).	TMPFIN

COORDINATES

Cards for the definition of nodal coordinates.

Field	Format	Content	Variable
First card			
1	A	COORDINATES	
Second card			
1	I	Maximum number of coordinates per node. If zero or blank then NC=MCORD, where MCORD is the maximum number of nodal coordinates in the mesh.	NC
2	I	Number of nodes for which the coordinates are assigned. If zero or blank, then NP = NPOIN.	NP
Third card (1 card for each node)			
1	I	Node number.	IPOIN
2...NC+1	R	Coordinates of node IPOIN	COORD

Note: at least one group of these cards is obligatory.

DAMPING

Card for definition of damping coefficients in dynamic analysis.

Field	Format	Content	Variable
First card			
1	A	DAMPING	
Second card			
1	I	Number of groups of conditions.	NDAMP

The following cards must be repeated NDAMP times.

Third card			
1	R	Value of the multiplier of mass matrix for viscous damping (default 0.).	DAMP(1)
2	R	Value of the multiplier of stiffness matrix for viscous damping (default 0.)	DAMP(2)
3	R	Value of the multiplier of the stiffness matrix for numerical damping (default 0.).	DAMP(3)
Fifth card			
1...M	I	List of elements with these damping conditions.	

DEFINE

Cards for defining sets of elements and/or nodes.

Field	Format	Content	Variable
First card			
1	A	DEFINE	
2	A	ELEMENT or ELEMENT SET, for an element set; NODE or NODE SET or NDSQ or NDSQ SET for a node set.	
3	A	Name of the set (maximum 8 characters)	CSETE or CSETN
Second card			
1	I	List of elements or nodes constituting the set.	NSETE or NSETN

Note: The names of other sets cannot be included in the lists of set definition

Note: A set can not be referenced in the input data before being defined.

DYNAMIC
Card for definition of integration coefficients in dynamic analysis.

Field	Format	Content	Variable
First card			
1	A	DYNAMIC	
Second card			
1	R	Value of the "gamma" coefficient in Newmark method (default 0.5).	GAMNEW
2	R	Value of the "beta" coefficient in Newmark method (default 0.25).	BETNEW

END OPTION

Card for declaring of the end of model cards.

Field	Format	Content	Variable
1	A	END OPTION	

Note: this card is obligatory.

FILM COEFFICIENT

Card for definition of film coefficients in heat transfer analysis.

Field	Format	Content	Variable
First card			
1	A	FILM COEFFICIENT	
Second card			
1	I	Number of groups of conditions.	NFILM

The following cards must be repeated NFILM times.

Third card			
1	I	Identifier of the element face.	
2	R	Value of the film coefficient (default 0.).	FILM
3	R	Value of the sink temperature (default 0.)	SINK
4	I	If different from zero, indicates that the film coefficient value will be calculated by the user routine UFILM.	INDFIL
4	I	If different from zero, indicates that the sink temperature value will be calculated by the user routine UFILM.	INDSIN
Fifth card			
1...M	I	List of elements with these film conditions.	

FIXED ACCELERATION

Card for definition of acceleration boundary conditions in dynamic analysis.

Field	Format	Content	Variable
First card			
1	A	FIXED ACCELERATION	
Second card			
1	I	Number of groups of boundary conditions.	NFIXA

The following cards must be repeated NFIXA times.

Third card			
1...N	R	Value of the i-th prescribed acceleration.	PRSCA
Fourth card			
1...N	I	List of prescribed acceleration degrees of freedom.	IPREA
Fifth card			
1...M	I	List of constrained nodes.	

Note: If negative values are specified in the degrees of freedom list, the user routine UBND will manage the boundary conditions relative to the degree of freedom given by the absolute value of the indicate negative numbers.

FIXED DISPLACEMENT

Card for definition of displacement boundary conditions.

Field	Format	Content	Variable
First card			
1	A	FIXED DISPLACEMENT	
Second card			
1	I	Number of groups of boundary conditions.	NFIXD
The following cards must be repeated NFIXD times.			
Third card			
1...N	R	Value of the i-th prescribed degree of freedom.	PRESC
Fourth card			
1...N	I	List of prescribed degrees of freedom.	IFPRE
Fifth card			
1...M	I	List of constrained nodes.	

Note: At least one group of this card type is required and, in the case of static analysis, constraints must be such as to avoid at least rigid body motions.

Note: If negative values are specified in the degrees of freedom list, the user routine UBND will manage the boundary conditions relative to the degree of freedom given by the absolute value of the indicate negative numbers.

FIXED TEMPERATURE

Card for definition of heat transfer boundary conditions.

Field	Format	Content	Variable
First card			
1	A	FIXED TEMPERATURE	
Second card			
1	I	Number of groups of boundary conditions.	NFIXT

The following cards must be repeated NFIXT times.

Third card			
1..ITYRD	R	Values of the prescribed temperature.	PRESC
Fourth card			
1..ITYRD	I	List of prescribed degrees of freedom.	IFPRE
Fifth card			
1...M	I	List of constrained nodes.	

Note: At least one group of this card type is.

Note: If negative values are specified in IFPRE, the user routine UBND will manage these temperature boundary conditions.

FRICTION

Cards for the defining the friction coefficients for contact problems.

Field	Format	Content	Variable
First card			
1	A	FRICTION	
Second card			
1	I	Number of friction coefficients to be defined.	NFRIC

The following cards must be repeated NFRIC times.

Third card			
1	R	Value of friction coefficient. If the value is negative, the friction coefficient will be defined in the user routine UFRI.	CFRICT
Fourth card			
1...L	I	List of deformable bodies.	
Fifth card			
1...M	I	List of rigid surfaces.	
Sixth card			
1...N	I	List of the parts of rigid surfaces.	

Note: The friction coefficient CFRICT(I,J,K) is relative to the interface between the deformable body K-th and the I-th part of the J-th rigid surface.

GEOMETRY

Cards for defining the thickness for plane stress elements (type 2), plane strain elements (types 3 and 6) or the thickness of layers for shell elements (type 5 and 10).
For other element types this group of cards is unnecessary.

Field	Format	Content	Variable
First card			
1	A	GEOMETRY	
Second card			
1	I	Number of groups of elements with the same geometry.	NGEOM
The following cards must be repeated NGEOM times.			
Third card			
1...KLAYR	R	Thickness of elements or layers (default 1.0). If the COMPOSITE option has been set it is the thickness of a single layer and, if KLAYR<MSHEL, the layers from KLAYR+1 to MSHEL have the thickness of the KLAYR-th layer. A negative value of GEOM for shell element type means that the thickness will be evaluated on a nodal basis by the user routine UGEOM.	GEOM
Fourth card			
1...N	I	List of elements having the previously defined geometry.	

HARDENING

Card for definition of materials hardening curves.

Field	Format	Content	Variable
First card			
1	A	HARDENING	
Second card			
1	I	Number of curves to be defined	KHARD

The following cards must be repeated KHARD times.

Third card			
1	I	Order number, as defined in the PROPERTY cards of the material which the hardening curve to be defined refers to.	JMATS
2	I	Number of data pairs constituting the hardening curve to be defined.	NHARD
3	R	Fraction of hardening to be considered isotropic. If HAISO = 0.0, the hardening is purely kinematic; if HAISO = 1.0, the hardening is purely isotropic.	HAISO

The following card must be repeated NHARD(JMATS) times.

Fourth card			
1	R	Value of the equivalent accumulated plastic strain (=0.0 for the first data pair).	HARD(1)
2	R	Value of the von-Mises equivalent stress corresponding to the plastic strain HARD(1).	HARD(2)

INITIAL DISPLACEMENT

Card for definition of the initial nodal displacement.

Field	Format	Content	Variable
First card			
1	A	INITIAL	
2	A	DISPLACEMENT	
Second card			
1	I	Number of groups of conditions.	NDISPI

The following cards must be repeated NDISPI times.

Third card			
1...N	R	Values of the components of the initial displacement (default 0.).	DISPI
Fourth card			
1...N	I	List of the degrees of freedom with the previously defined initial displacement.	
Fifth card			
1...M	I	List of nodes with initial displacement DISPI.	

Note: If negative values are specified in the degrees of freedom list, the user routine UDSPI will manage the displacement components corresponding to the absolute value of the indicate negative numbers.

INITIAL TEMPERATURE

Card for definition of the initial or reference nodal temperature.

Field	Format	Content	Variable
First card			
1	A	INITIAL	
2	A	TEMPERATURE	
Second card			
1	I	Number of groups of conditions.	NTINI

The following cards must be repeated NTINI times.

Third card			
1..ITYRD	R	Values of the prescribed temperature (default 0.).	TEMPI
Fourth card			
1	I	1 or –1 (index of the temperature degree of freedom).	IUSER
Fifth card			
1...M	I	List of nodes with initial temperature TEMPI.	

Note: If negative values are specified in IUSER, the user routine UTEMPI will manage that initial temperature values.

INITIAL VELOCITY

Card for definition of the initial nodal velocity.

Field	Format	Content	Variable
First card			
1	A	INITIAL	
2	A	VELOCITY	
Second card			
1	I	Number of groups of conditions.	NVELI

The following cards must be repeated NVELI times.

Third card			
1...N	R	Values of the components of the initial velocity (default 0.).	VELOI
Fourth card			
1...N	I	List of the degrees of freedom with the previously defined initial velocity.	
Fifth card			
1...M	I	List of nodes with initial velocity VELOI.	

Note: If negative values are specified in the degrees of freedom list, the user routine UVELI will manage the velocity components with index given by the absolute value of the indicate negative numbers.

LOCAL AXIS

Card for definition of the local reference system for beam elements.

Field	Format	Content	Variable
First card			
1	A	LOCAL	
2	A	AXIS	
Second card			
1	I	Number of different local systems.	NLAX
The following cards must be repeated NLAX times.			
Third card			
1	I	If different from zero, indicates that the local system is to be define by the user routine ULAXIS. (default 0)	IUSER
2, 3, 4	R	Values of the components of the unit vector v_1 along the first local direction in the plane of the beam cross section. The third local direction v_3 is defined by the beam axis and the second local direction v_2 is found by the cross product $v_2=v_3 \times v_1$.	AXIS
Fourth card			
1...M	I	List of elements with this local reference system.	

LUMPED MASS

Card for declaring the use of lumping in the calculation of the mass matrix.

Field	Format	Content	Variable
First card			
1	A	LUMPED	
2	A	MASS	
Second card			
1	I	Number of different sets.	NLUMP

The following cards must be repeated NLUMP times.

Third card			
1	I	If different from zero, indicates that the mass matrix of the indicated elements will be lumped by summing a row into the diagonal element. (default 0)	
Fourth card			
1...M	I	List of elements having the previously defined properties.	

MASONRY

Cards for definition of masonry-like materials.

Field	Format	Content	Variable
First card			
1	A	MASONRY	
Second card			
1	I	Number of groups of elements having the same material behavior.	NMASO

The following cards must be repeated NMASO times.

Third card			
1...KLAYR	I	=0, elastic-plastic material; =1, masonry-like material with low tensile strength and infinite resistance to compression; =2, masonry-like material weakly resistant to tension, with bounded compressive strength.	MASO
Fourth card			
1	I	List of elements having the previously defined properties.	

Note: in this version of NOSA, the shell elements cannot have layers with different material behaviors.

MASSES

Cards for definition of concentrated masses in dynamic analysis.

Field	Format	Content	Variable
First card			
1	A	MASSES	
Second card			
1	I	Number of groups of nodes with equal concentrated masses applied.	NMASS

The following cards must be repeated NMASS times.

Third card			
1...N	R	Value of the concentrated mass for the degrees of freedom listed below (default 0.).	PMASS
Fourth card			
1...N	I	List of the inertia component indexes.	
Fifth card			
1...M	I	List of nodes with concentrated mass PMASS.	

4

Note: If negative values are specified in the degrees of freedom list, the user routine UPMAS will manage the inertia components with index given by the absolute value of the indicate negative number.

POST

Card for defining the structure and content of the post-processing file.

Field	Format	Content	Variable
First card			
1	A	POST	
Second card			
1	I	Number of variables to be written to the post-processing file (default 0, maximum 200).	MVAR
2	I	Write frequency of the increment results on the post-processing file. The output will be written corresponding to the NPOST-th increment after the last written (default value 0, i.e. no post data writing).	NPOST
3	I	=0, the post-processing file is written using the character format (FORTRAN unit 20); =1, the post-processing file is written using binary format (FORTRAN unit 21).	IBINI
4	I	Write frequency of the results of the different iterations on the post-processing file (default 0, i.e. no iteration results writing)	INCSB
5	I	=0, if a restart is required, the initial part of the post-processing file, including connectivity and nodal coordinates is not printed; =1, if a restart is required, the initial part of the post-processing file is printed.	ICONT
6	R	Alternative method to write on the post-processing file. Time interval between the writings of results on the post-processing file (default 0.d0, i.e. no time writing chosen).	WRTIM

The following card must be repeated MVAR times.

Third card			
1	I	Code number of the post variable.	INDVAR

2	A	Name of the post variable (maximum number of characters, 24).	CHVAR

The post variable codes have the following meanings:

Negative number	The variable must be defined in the user routine PLOTV.
1-6	Components of the total strain.
9	Total temperature.
11-16	Components of the Cauchy stress.
17	Von Mises equivalent stress.
21-26	Components of plastic strain. For masonry-like materials, components of tensile inelastic strain.
27	Equivalent plastic strain for elastic-plastic materials. For masonry-like materials, norm of the tensile inelastic strain.
31-36	Components of compression inelastic strain for masonry-like materials with bounded compressive strength.
37	For masonry-like material, norm of compressive inelastic strain.
41-48	Components of strain characteristics for shell elements.
51-58	Components of stress characteristics for shell elements.
71	Total enthalpy (kinetic plus deformation energy minus the work done by the external forces).
72	Total kinetic energy.
73	Total deformation energy.
74	Total work done by the external forces (including the reactions).
75	Total mechanical energy (kinetic plus deformation energy).
81	Density of the enthalpy.
82	Density of the kinetic energy.
83	Density of the deformation energy.

84	Density of the external work.
85	Density of the mechanical energy.
181-183	Components of temperature gradient in heat transfer analysis.

If a variable relative to a particular layer of a shell element must be written to tape, the variable code must be in the form 1000*I+J, where I indicates the layer and J is the variable code as previously defined (or -(1000*I+J) when the user routine PLOTV is required)

POTENTIAL HILL

Cards for defining anisotropy ratios of the yield points following the Hill's theory. Moreover, the initial rotation matrix is read, which defines the anisotropy principal axes.

Field	Format	Content	Variable
First card			
1	A	POTENTIAL	
2	A	HILL	
Second card			
1	I	Number of materials with anisotropic yield	KMAPH

The following cards must be repeated KMAPH times.

Third card			
1	I	identifier of the material to which the data refer	JMATS
2,..,7	R	the ratios (r_x, r_y, r_z, r_{xy}, r_{yz}, r_{xz}) between the yield stress values along the different directions and the reference value	RAPH
Fourth card			
1,..,9	R	The initial rotation matrix (given by rows) between the global reference system and the anisotropy principal system	RSP0

PROPERTY

Cards for defining the properties of the material constituting the elements and/or sections of the elements. The properties are different between static and/or dynamic analyses and heat transfer cases.

Field	Format	Content	Variable
First card			
1	A	PROPERTY	
Second card			
1	I	Number of different materials.	KMATS
The following cards must be repeated KMATS times.			
Third card (static or dynamic stress analysis)			
1	I	Material identification number	JMATS
2	R	Young's modulus (default 0).	PROPS(1)
3	R	Poisson's ratio (default 0).	PROPS(2)
4	R	Density (default 0).	PROPS(3)
5	R	Coefficient of thermal expansion (default 0).	PROPS(4)
6	R	Equivalent (von Mises) tensile yield stress (default 1.E+20). For masonry-like material, maximum tensile strength.	PROPS(5)
7	R	For masonry-like materials, maximum compressive strength (default 1. E+20).	PROPS(6)
8	R	Coefficient BETA for isotropic hardening, infinitesimal plasticity (default 0.).	PROPS(7)
9	R	Coefficient ALFA for kinematic hardening, infinitesimal plasticity (default 0.).	PROPS(8)
10	R	Coefficient E1 for kinematic hardening, infinitesimal plasticity (default 0.).	PROPS(9)

11	R	Coefficient E2 for kinematic hardening, infinitesimal plasticity (default 0.).	PROPS(10)

Third card (heat transfer stress analysis)

1	I	Material identification number	JMATS
2	R	Heat conductivity (default 0).	PROPS(1)
3	R	Specific heat per unit mass (default 0).	PROPS(2)
4	R	Density (default 0).	PROPS(3)
5	R	Emissivity (default 0).	PROPS(4)

Note: at least one group of these card is obligatory.

Note: usually the isotropic, kinematic and combined hardening are defined through the user routines CPHI and CPSI and the cards HARDENING.

ROTATION AXIS

Cards for defining the direction cosine (components of a unit vector) of the rotation axis and the coordinates of a point belonging to this axis when centrifugal loads are assigned.

Field	Format	Content	Variable
First card			
1	A	ROTATION AXIS	
Second card			
1	R	First direction cosine of the rotation axis.	ROTAX(1)
2	R	Second direction cosine of the rotation axis.	ROTAX(2)
3	R	Third direction cosine of the rotation axis.	ROTAX(3)
Third card			
1	R	First global coordinate x of a point belonging to the rotation axis.	ROTAX(4)
2	R	Second global coordinate y of a point belonging to the rotation axis.	ROTAX(5)
3	R	Third global coordinate z of a point belonging to the rotation axis.	ROTAX(6)

Note: for two-dimensional problems the rotation axis is orthogonal to the plane containing the structure and only the values ROTAX(4), ROTAX(5) need to be specified. For axisymmetric problems, the rotation axis coincides with the axis of symmetry, and the ROTATION AXIS card may therefore be omitted.

SELECTIVE INTEGRATION

Card for reduced selective integration. This feature can be used only for element types 6, 7, 8. When used, the option implies the reduced integration (1 integration point) of the volumetric part of the deformation, and its use is recommended in finite strain analyses..

Field	Format	Content	Variable
First card			
1	A	SELECTIVE INTEGRATION	
Second card			
1	I	Number of different kinds of integration required.	NSELE

The following cards must be repeated NSELE times.

Third card			
1	I	Identification code of the type of integration: =0, complete integration, =1, reduced integration.	ISELE
Fourth card			
1	I	List of elements.	

SURFACE

Cards for defining rigid surfaces for two-dimensional contact problems.

Field	Format	Content	Variable
First card			
1	A	SURFACE	
Second card			
1	I	Number of rigid surfaces to be defined.	NSURF

The following cards must be repeated NSURF times.

Third card			
1	I	Number of parts constituting the rigid surface.	NPART
2	R	Global coordinate x of the guide point of the surface.	SURF(1,1)
3	R	Global coordinate y of the guide point of the surface.	SURF(2,1)
4	R	First velocity component of the guide point of the surface.	SURF(3,1)
5	R	Second velocity component of the guide point of the surface.	SURF(4,1)
6	R	Angular velocity of the guide point of the rigid surface (rad/sec, positive if counterclockwise).	SURF(5,1)

The following card must be repeated NPART times.

Fourth card			
1	I	Type of the part to be defined, =1, segment; =2, circumference arc (1st approach); =3, circumference arc (2nd approach).	ISURF

The following fields have different meanings according to the type of part to be defined.

1) Segment.

2	R	Coordinate x of the first point.	SURF(1, N)
3	R	Coordinate y of the first point.	SURF(2, N)
4	R	Coordinate x of the end point.	SURF(3, N)
5	R	Coordinate y of the end point.	SURF(4 ,N)

2) Circumference arc (1st approach).

2	R	Coordinate x of the first point.	SURF(1, N)
3	R	Coordinate y of the first point.	SURF(2, N)
4	R	Coordinate x of the center.	SURF(5, N)
5	R	Coordinate y of the center.	SURF(6, N)
6	R	Angle at center (°, positive if in counterclockwise direction).	SURF(7, N)

2) Circumference arc (2nd approach).

2	R	Coordinate x of the first point.	SURF(1, N)
3	R	Coordinate y of the first point.	SURF(2, N)
4	R	Coordinate x of the end point.	SURF(3, N)
5	R	Coordinate y of the end point.	SURF(4, N)
6	R	radius, >0 if the center is within the rigid surface, <0 otherwise.	SURF(8, N)

Note: A rigid surface must be bounded by a plane closed curve. The parts constituting the curve must be defined in such a way as to cover the curve in the counter clockwise direction.

TEMP TABLE

Cards for the input of the tables, which define the variation of the material properties with respect to the temperature.

Field	Format	Content	Variable
First card			
1	A	TEMP	
2	A	TABLE	
Second card			
1	I	Table length (number of data pairs).	ITTEMP(3)
2	I	Identifier number of the material to which the table refers.	ITTEMP(1)
3	I	Identifier of the property to which the table refers. Stress analysis: =1, Young modulus; =2, Poisson ratio; =3, coefficient of thermal expansion (linear).	ITTEMP(2)
		Heat transfer analysis: =1, heat conductivity; =2, specific heat, per unit mass; =3, emissivity.	ITTEMP(2)

The following card must be repeated ITTEMP(3) times, so the values of the I-th pair of the J-th table have to be specified as follows:

Third card.			
1	R	I-th temperature in the J-th table.	RTTEMP(1, I, J)
2	R	The value corresponding to the I-th temperature in the J-th table.	RTTEMP(2, I, J)

TYING

Cards for definition of the tying relations between nodal degrees of freedom. This option allows tying unpaired nodes during (non-gradual) refining of the mesh, as well as tying the degrees of freedom of different element types and setting the kinematic constraints along directions not aligned with the Cartesian reference axes.

Field	Format	Content	Variable
First card			
1	A	TYING	
Second card			
1	I	Number of tying relations to be defined.	NGTIE

The following card must be repeated NGTIE times.

Third card			
1	I	Identifier of the tying relation.	ITIE(1)
2	I	Identifier number of the tied node of the tying relation or the name of the node set containing the tied nodes of the tying relation.	ITIE(2)
3...N	I	Identifier numbers of retained nodes of the tying relation or the names of node sets containing the retained nodes of the tying relation.	ITIE(3...N)

Note: The management of the tying relations is carried out by the user routine UTIE.

Note: If the tied and retained nodes are specified by sets, so these sets must be in one to one correspondence in the sense that the i-th items of the sets containing the retained nodes must be the identifiers of the retained nodes pertaining to the tied node identified by the i-th item of the set of tied nodes.

D3. LOAD CARDS

Index of keywords

AUTO LOAD

Cards for the definition of the number of automatic load or heat flux increments.

Field	Format	Content	Variable
First card			
1	A	AUTO LOAD	
Second card			
1	I	number of times the load increment is to be applied.	NAUTO
2	I	= 0, load increments are equal to the first load increment of the list. = 1, each load increment of the list will be calculated in order to allow utilization of the user routines FORCEM and UPLOAD.	NAUT1

BOUNDARY CHANGE

Cards for redefining the prescribed incremental values of degrees of freedom.

Field	Format	Content	Variable
First card			
1	A	BOUNDARY CHANGE	
Second card			
1	I	Number of groups of different boundary conditions.	NBOUN

The following cards must be repeated NBOUN times.

Third card			
1...N	R	Value of the i-th prescribed degree of freedom.	PRESC
Fourth card			
1...N	I	List of the prescribed degrees of freedom.	IFPRE
Fifth card			
1...M	I	List of constrained nodes.	NOFIX

Note: If the card BOUNDARY CHANGE is present, the boundary conditions are completely redefined; in which case the boundary conditions which do not change with respect to the previous increment must be repeated as well.

Note: The values assigned to the degrees of freedom through the card BOUNDARY CHANGE are displacement or temperature increments; therefore, if prescribed degrees of freedom are added or subtracted, a new linear system must be solved (simple back-substitution is not sufficient). Thus, the CONTROL option must be activated and the use of the Newton-Raphson algorithm specified in it.

COMMENT or $

Comment card.

Field	Format	Content	Variable
1	A	COMMENT or $	
2	A	Comment text.	

Note: comment cards may be inserted anywhere.

CONTROL

Cards for defining parameters, which control the accuracy of numerical results for non-linear analysis.

Field	Format	Content	Variable
First card			
1	A	CONTROL	
Second card			
1	I	Maximum number of load increments (default 4).	MAXINC
2	I	Maximum number of iterations for each load increment (default 3).	MITER
3	I	Type of algorithm used for the solution of the system. 1 = initial stiffness matrix, 2 = modified Newton-Raphson method, 3 = Newton-Raphson method. The default value is 1.	NALGO
Third card			
1	R	Ratio in % between the norm of residual force and the norm of total force, including reactions, in order to an increment is considered as converged. For shell elements, the moments are divided by the thickness to be dimensionally homogeneous to forces.	TOLER
2	R	Minimum value of the norm of total force. If the norm is less than FUZTOL, the convergence control is skipped (default 1.D-08).	FUZTOL
3	R	For dynamic analyses it is the minimum displacements change. If the norm of displacement is less than FUZDIS, the convergence control is skipped (default 1.D-12).	FUZDIS
		For heat transfer analyses it is the minimum temperature change. If the maximum temperature change is less than FUZTMP, the convergence control is skipped (default 1D.-08).	FUZTMP

4	R	For heat transfer analyses it is the maximum temperature change allowed in automatic time stepping mode (default 20.D0). If the maximum temperature change is greater than TMPFIN, the increment is repeated with reduced time step in order that the maximum temperature change be equal to TMPFIN (this can be obtained with precision in linear cases).	TMPFIN

DISTRIBUTED FLUXES

Cards for the definition of distributed heat fluxes on faces or edges of elements.

Field	Format	Content	Variable
First card			
1	A	DISTRIBUTED FLUXES	
Second card			
1	I	Number of different types of fluxes.	NDIST
The following cards must be repeated NDIST times.			
Third card			
1	I	Identifier code of the flux type (see the library of elements).	IDIST(1)
2	I	Order number of the type of flux.	IDIST(2)
Fourth card			
1...N	R	Values of the heat flux applied on the elements (see the library of elements).	RDIST(I, 1)
Fifth card			
1	I	List of elements on which the flux is applied.	

DISTRIBUTED LOADS

Cards for the definition of distributed loads on faces or edges of elements in the global or local reference system.

Field	Format	Content	Variable
First card			
1	A	DISTRIBUTED LOADS	
Second card			
1	I	Number of different types of loads.	NDIST
The following cards must be repeated NDIST times.			
Third card			
1	I	Identifier code of the load type (see the library of elements).	IDIST(1)
2	I	Order number of the type of load.	IDIST(2)
Fourth card			
1...N	R	Values of components, in local or global coordinates, of the force applied on the elements (see the library of elements).	RDIST(I, 1)
Fifth card			
1	I	List of elements on which the load is applied.	

ELPRINT

Cards for the selective print of elemental data (strains, stresses *etc.*).

Field	Format	Content	Variable
First card			
1	A	ELPRINT	
Second card			
1	I	Number of groups of elements whose data are to be printed.	NELGR

The following card must be repeated NELGR times.

Third card			
1	I	List of elements to be printed.	IELPR

END INCREMENT

This card signals the end of definition of an increment load. The presence of the card END INCREMENT is obligatory at the end of the definition of each load increment.

Field	Format	Content	Variable
Fist card			
1	A	END INCREMENT	

FIXED ACCELERATION

Card for definition of acceleration boundary conditions in dynamic analysis.

Field	Format	Content	Variable
First card			
1	A	FIXED ACCELERATION	
Second card			
1	I	Number of groups of boundary conditions.	NFIXA

The following cards must be repeated NFIXA times.

Third card			
1...N	R	Value of the i-th prescribed acceleration.	PRSCA
Fourth card			
1...N	I	List of prescribed acceleration degrees of freedom.	IPREA
Fifth card			
1...M	I	List of constrained nodes.	

Note: If negative values are specified in the degrees of freedom list, the user routine UBND will manage the boundary conditions relative to the degree of freedom given by the absolute value of the indicate negative numbers.

Note: If the card FIXED ACCELERATION is present, the acceleration boundary conditions are completely redefined; in which case the boundary conditions which do not change with respect to the previous increment must be repeated as well.

Note: The values assigned to the degrees of freedom through the card FIXED ACCELERATION are total accelerations; therefore, if prescribed degrees of freedom are added or subtracted, a new linear system must be solved (simple back-substitution is not sufficient). Thus, the CONTROL option must be activated and the use of the Newton-Raphson algorithm specified in it.

FIXED DISPLACEMENT

Card for definition of displacement boundary conditions.

Field	Format	Content	Variable
First card			
1	A	FIXED DISPLACEMENT	
Second card			
1	I	Number of groups of boundary conditions.	NFIXD

The following cards must be repeated NFIXD times.

Third card			
1...N	R	Value of the i-th prescribed degree of freedom.	PRESC
Fourth card			
1...N	I	List of prescribed degrees of freedom.	IFPRE
Fifth card			
1...M	I	List of constrained nodes.	

Note: If negative values are specified in the degrees of freedom list, the user routine UBND will manage the boundary conditions relative to the degree of freedom given by the absolute value of the indicate negative numbers.

Note: If the card FIXED DISPLACEMENT is present, the displacement boundary conditions are completely redefined; in which case the boundary conditions which do not change with respect to the previous increment must be repeated as well.

Note: The values assigned to the degrees of freedom through the card BOUNDARY CHANGE are displacement increments; therefore, if prescribed degrees of freedom are added or subtracted, a new linear system must be solved (simple back-substitution is not sufficient). Thus, the CONTROL option must be activated and the use of the Newton-Raphson algorithm specified in it.

FIXED TEMPERATURE

Card for definition of heat transfer boundary conditions.

Field	Format	Content	Variable
First card			
1	A	FIXED TEMPERATURE	
Second card			
1	I	Number of groups of boundary conditions.	NFIXT

The following cards must be repeated NFIXT times.

Third card			
1..ITYRD	R	Values of the prescribed temperature.	PRESC
Fourth card			
1..ITYRD	I	List of prescribed degrees of freedom.	IFPRE
Fifth card			
1...M	I	List of constrained nodes.	

Note: If negative values are specified in IFPRE, the user routine UBND will manage these temperature boundary conditions.

Note: If the card FIXED TEMPERATURE is present, the boundary conditions are completely redefined; in which case the heat transfer boundary conditions which do not change with respect to the previous increment must be repeated as well.

Note: The values assigned to the degrees of freedom through the card FIXED TEMPERATURE are temperature increments; therefore, if prescribed degrees of freedom are added or subtracted, a new linear system must be solved (simple back-substitution is not sufficient). Thus, the CONTROL option must be activated and the use of the Newton-Raphson algorithm specified in it.

NODPRINT

Cards for the selective print of nodal data (displacements, nodal reactions, *etc.*).

Field	**Format**	**Content**	**Variable**
First card			
1	A	NODPRINT	
Second card			
1	I	Number of groups of nodes whose data are to be printed.	NNOGR

The following card must be repeated NNOGR times.

Third card			
1	I	List of nodes to be printed.	INODPR

POINT FLUXES

Cards for defining concentrated heat fluxes applied on the nodes.

Field	Format	Content	Variable
First card			
1	A	POINT FLUXES	
Second card			
1	I	Number of different types of fluxes.	NDIST

The following cards must be repeated NDIST times.

Third card			
1	I	If different from zero, it means that the concentrated flux value will be calculated by the user routine UPLOAD	IUPLOA
2...N	R	Values of the concentrated heat flux.	POINT
Fourth card			
1	I	List of nodes to which the flux is applied.	

POINT LOADS

Cards for defining point loads applied on the nodes of the structure in the global reference system.

Field	Format	Content	Variable
First card			
1	A	POINT LOADS	
Second card			
1	I	Number of different types of loads.	NDIST

The following cards must be repeated NDIST times.

Third card			
1	I	If different from zero, it means that the concentrated load values will be calculated by the user routine UPLOAD	IUPLOA
2...N	R	Values of components of the point load.	POINT
Fourth card			
1	I	List of nodes to which the point load is applied.	

POST

Cards for redefining the frequency of writing the incremental data on the post-processing file.

Field	Format	Content	Variable
First card			
1	A	POST	
1	I	Frequency of writing incremental data on the post-processing file. The data will be written for the first increment and every NPOST increment.	NPOST
2	I	Write frequency of the results of the different iterations on the post-processing file (default 0, i.e. no iteration results writing).	INCSB
3	I	= 0, if a restart is required, the initial part of the post-processing file, including the connectivity and the nodal coordinates, is not printed; = 1, if a restart is required, the initial part of the post-processing file is printed.	ICONT
4	R	Alternative method to write on the post-processing file. Time interval between the writings of results on the post-processing file (default 0.d0, i.e. no time writing chosen).	WRTIM

PRINT CHOICE

Cards for control of output printing.

Field	Format	Content	Variable
First card			
1	A	PRINT CHOICE	
Second card			
1	I	Code for the control of output printing (default 1). = 0 no print, = 1 print results when convergence has been reached, = 2 print results at every iteration.	IOUTP

PROPORTIONAL INCREMENT

Cards for defining the proportionality factor between the next load and the initial load for analyses containing more than one load increment.

Field	Format	Content	Variable
First card			
1	A	PROPORTIONAL INCREMENT	
Second card			
1	R	Multiplicative factor of the last defined increment load.	FACTO

Note: a defined load is a load determined by the set of cards: DISTRIBUTED LOADS, POINT LOADS, FIXED ACCELERATION, FIXED DISPLACEMENT, BOUNDARY CONDITIONS, BOUNDARY CHANGE, SURFACE, SURFACE CHANGE.

SAVE INCREMENT

Cards for saving the incremental results on a suitable file (unit FORTRAN n.99).

Field	**Format**	**Content**	**Variable**
First card			
1	A	SAVE INCREMENT	
Second card			
1	I	Write code. (default 0). = 0, the incremental results are unsaved, > 0, the results have been saved for every ISAVE increments.	ISAVE

Note: Since the file is overwritten, it will contain only data from the last increment saved.

STEPSIZE
Cards for defining magnitude and number of steps in heat transfer or dynamic analysis.

Field	Format	Content	Variable
First card			
1	A	STEPSIZE	
Second card			
1	I	If different from zero sets the automatic calculation of step size in heat transfer analysis.	AUTTIM
2	I	Maximum number of steps in this series.	NUMSTP
3	R	Initial time step.	DELTII
4	R	Time interval to be covered by this series of steps.	PERIOD
5	I	If different from zero, requires the calculation of the initial acceleration in dynamic analysis.	IACCI

Note: In case of heat transfer analysis with automatic step size calculation, denoting by DTMAX the absolute value of the maximum temperature change we have to compare it with the maximum temperature change allowed TMPFIN.
If DTMAX>TMPFIN, the step will be repeated with a smaller step size DELTIM=DELTIM*TMPFIN/DTMAX., whereas if DTMAX<TMPFIN the step size for the next step will be increased to DELTIM=0.9*TMPFIN/DTMAX.

SURFACE CHANGE

Cards for redefining the velocity of rigid surfaces used in two-dimensional contact problems.

Field	Format	Content	Variable
First card			
1	A	SURFACE CHANGE	
Second card			
1	I	Number of rigid surfaces whose velocity must be redefined.	NSURF

The following card must be repeated NSURF times.

Third card			
1	I	Ordinal number of the rigid surface.	INDSUR
2	R	First component of the rigid surface's guide node velocity.	SURF(3, 1)
3	R	Second component of the rigid surface's guide node velocity.	SURF(4, 1)
4	R	Angular velocity of the rigid surface's guide node (rad/sec, positive if counter-clockwise).	SURF(5, 1)

THERMAL LOADS

Cards for defining the nodal increments of temperature for calculation of loads due to thermal dilatation.

Field	Format	Content	Variable
First card			
1	A	THERMAL LOADS	
Second card			
1	I	=1, nodal temperature values are to be read from cards; =2, nodal temperature values are to be read from binary post file (Fortran unit 26); =3, nodal temperature values are to be read from formatted post file (Fortran unit 25); =4; nodal temperature values are to be calculated by the user routine URDTEM.	IRDTMP

The following cards need to be inserted only when IRDTMP=1.

1	I	Number of nodes group to which the temperature increments are applied.	NDIST

The following cards must be repeated NDIST times.

Fourth card			
1..ITYRD	R	Values of the temperature increment on the nodes declared in the following list.	TEMPE
Fifth card			
1	I	List of nodes subjected to the temperature increment defined in the previous card.	IPOIN(1)

The following cards need to be inserted only when IRDTMP>1.

Third card

1	I	The number of the increment of the post file from which the temperature data are to be read.	INCTMP
2	I	Number of thermal load steps necessary to achieve the thermal state read from increment INCTMP.	NAUTO

Note: for shell elements, temperature data are to be indicated at the bottom and top layer when ITYRD=2 and at the bottom, middle and top layer when ITYRD=3.

TIME STEP

Cards for defining time increments for calculation of the motion of rigid surfaces in two-dimensional contact problems.

Field	Format	Content	Variable
First card			
1	A	TIME STEP	
Second card			
1	I	If different from zero, the user routine UMOTIO is called for the definition of the rigid surface's velocity.	IUSURF
2	R	Time increment.	DELTII

TYING CHANGE

Cards for redefining tying relations between the nodal degrees of freedom.

Field	Format	Content	Variable
First card			
1	A	TYING CHANGE	
Second card			
1	I	Number of tying relations to be re-defined.	NGTIE

The following card must be repeated NGTIE times.

Third card			
1	I	Identifier of the tying relation.	ITIE(1)
2	I	Identifier number of the tied node of the tying relation or the name of the node set containing the tied nodes of the tying relation.	ITIE(2)
3...N	I	Identifier numbers of retained nodes of the tying relation or the names of node sets containing the retained nodes of the tying relation.	ITIE(3...N)

Note: The management of the tying relations is carried out by the user routine UTIE.

Note: If the tied and retained nodes are specified by sets, so these sets must be in one to one correspondence in the sense that the i-th items of the sets containing the retained nodes must be the identifiers of the retained nodes pertaining to the tied node identified by the i-th item of the set of tied nodes.

Note: If the card TYING CHANGE is present, the table of tying relations must be completely re-defined, even for those parts which have not changed with respect to the previous situation.

E. USER'S ROUTINES

CPHI

The subroutine CPHI allows one to define the parameters which control the isotropic hardening for elastic-plastic materials; more precisely the radius ρ of the elastic range and its derivative ρ' with respect to the Odqvist parameter ζ are assigned (the Odqvist parameter is defined as the time integral of the norm of the plastic deformation rate). In general, the dependence on the Odqvist parameter is specified through a table containing points belonging to an experimental hardening curve.
This routine is called by the subroutines for integration of the constitutive law.

Scheme of the subroutine

```
      SUBROUTINE CPHI (PHI, PHI1, ZETA, NHARD, HAISO, HARD)
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
      INCLUDE 'LOCAL'
C
      DIMENSION HARD (2, MHARD)
      .
      .user code
      .
      RETURN
      END
```

Arguments of the call

PHI Radius of the elastic radius $\rho(\zeta)$ (output).

PHI1 Derivative of the radius $\rho(\zeta)$ with respect to ζ (output).

ZETA Odqvist parameter ζ (input).

NHARD Number of data pairs in the table HARD, which defines the hardening curve (input).

HAISO Percent of isotropic hardening present in the hardening curve (input).

HARD Table, which defines the hardening, curve (input).
HARD(1, I) accumulated equivalent plastic strain;
HARD(2, I) von Mises equivalent stress.

MHARD Max number of data pairs in a hardening table (input from COMMON/CNTR/).

CPSI

The CPSI allows definition of the tensor M which controls kinematic hardening for elastic-plastic materials; more precisely tensor M may depend on the Odqvist parameter ζ, the plastic strain, the tensor orthogonal to the elastic range and, finally the center of the elastic range. Usually the dependence on the Odqvist parameter is specified by a table containing points belonging to an experimental hardening curve.
This routine is called by those routines, which integrate the constitutive equation.

Scheme of the subroutine

```
      SUBROUTINE CPSI (ZETA, TC, TEP, TN, TM, NHARD, HAKIN, HARD)
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
      INCLUDE 'LOCAL'
C
      DIMENSION TC(3, 3), TEP(3, 3), TN(3, 3), TM(3, 3).
      DIMENSION HARD (2, MHARD)

      .user code

      RETURN
      END
```

Arguments of the call

ZETA	Odqvist parameter ζ (input).
TC	Center of the elastic range (input).
TEP	Plastic strain tensor (input).
TN	Outward unit normal to the elastic range (input).
TM	Tensor, which controls the kinematic hardening (output).
NHARD	Number of data pairs in the table HARD, which defines the hardening curve (input).
HAKIN	Percent of kinematic hardening present in the hardening curve (input).
HARD	Table defining the hardening curve (input). HARD(1, I) accumulated equivalent plastic strain; HARD(2, I) von Mises equivalent stress.
MHARD	Max number of data pairs in a hardening table (input from COMMON/CNTR/).

FORCEM

The routine FORCEM allows definition of non-uniform distributed loads or fluxes.
The library of elements contains the list of the identifier numbers to be declared in the cards DIST LOADS or DIST FLUXES, relative to the definition of loads through this routine.
If required, the routine FORCEM is called at the beginning of each increment by the subroutine LOADIN that manages the load increments.

Scheme of the routine

```
      SUBROUTINE FORCEM ( LNODS, COORD, MATNO, PROPS, GEOM,
     * DELTAH, ZETSH, TDISP, IELEM, LOACOD, LOANUM, BFORCE,
     * NNODE, NINT )
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
      INCLUDE 'CONVRG'
      INCLUDE 'FILES'
      INCLUDE 'LOCAL'
      INCLUDE 'MOTION'
C
      DIMENSION
     * BFORCE (LDOFN, NNODE, NINT),
     * COORD (MCORD, NPOIN),
     * DELTAH (LDELTA, LGASP),
     * GEOM (MDT, MGEOM),
     * LNODS (MNODE),
     * MATNO (MTHICK),
     * PROPS (MPROP, NMATS),
     * TDISP (MDOFN,NPOIN),
     * ZETSH(LINT,LGASP)
      .
      .user code
      .
      RETURN
      END
```

Arguments of the call

LNODS Numbers of nodes constituting the element (input).

COORD Coordinates of all points of the mesh (input).

MATNO Array containing the identifier numbers of the materials making up the element (input).

The material making up the element is always unique, except for shell and beam elements where it may vary from layer to layer or from fiber to fiber.

PROPS Array of the properties of the materials present in the mesh (input).

GEOM Element thickness. For homogeneous shells and beams GEOM(1,I) is the thickness of the first layer or fiber at the I-th node. For non-homogeneous shell GEOM(J, I) is the thickness of the J-th layer or fiber at the I-th node. (input).

DELTAH Shell or beam thickness. DELTAH(J, I) is the thickness of the J-th layer or fiber at the I-the Gauss integration point (input).

ZETSH ZETSH(J, I) is the distance from the shell mean surface, or from the beam axis, of the J-th Simpson integration point at the I-th Gauss integration point.

TDISP Array of the total displacements or temperatures of all degrees of freedom in the mesh (input).

IELEM Ordinal number of the element (input).

LOACOD Code of the load or flux to be defined, as declared in the cards DIST LOADS or DIST FLUXES (input).

LOANUM Ordinal number of the load or flux to be defined, as declared in the cards DIST LOADS or DIST FLUXES (input).

BFORCE Magnitude of the load or flux to be defined (output).
In the case of body forces or volumetric fluxes, BFORCE must be defined for all global degrees of freedom of all element nodes. In the case of pressures or surface fluxes, BFORCE must be defined for all degrees of freedom (global or local) of the nodes belonging to the face or the edge indicated by the load code in LOACOD.

NNODE Number of nodes for which BFORCE must be defined (input).
In the case of body forces or volumetric fluxes, NNODE is the number of element nodes. In the case of pressures or surface fluxes, NNODE is the number of nodes of the loaded face or edge.

NINT Its value is 1 except for shell elements, in which case it indicates the number of integration points along the thickness (input).

LDOFN Number of degrees of freedom of the element nodes (input from COMMON/LOCAL/).

LGASP Number of Gauss integration points od the element (input from COMMON/LOCAL/).

MCORD Maximum number of coordinates per node (input from COMMON/CNTR/).

NPOIN Total number of nodes in the mesh (input from COMMON/CNTR/).

MDT For shell or beam elements, it is the number of layers or fibers of the element; for the other types of elements, its value is 1 (input from COMMON/CNTR/).

LDELTA Same as MDT (input from COMMON/CNTR/).

MTHICK For shell or beam elements, it is the number of integration points on the cross section; for the other types of elements, its value is 1 (input from COMMON/CNTR/).

LINT Same as MTHICK (input from COMMON/CNTR/).

MNODE Maximum number of element nodes (input from COMMON/CNTR/).

MPROP Number of mechanical properties of a material (input from COMMON/CNTR/).

NMATS Number of different materials contained in the mesh (input from COMMON/CNTR/).

PLOTV

The routine PLOTV allows the user to define quantities, on the basis of elemental data, to be written on the post-processing file. The routine PLOTV is called for each layer or fiber (shell and beam elements only) of each integration point of each element.

Scheme of the routine

```
      SUBROUTINE PLOTV ( VAR, T, E, EAN, EANC, TEMP, GPCOD,
     * DISPG, IELEM, IGASP, ILAY, INDVA, LNODS, COORD,
     * TDISP,GRADT, IFBE, SECT, GEI, DELTAH, ZETSH)
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
      INCLUDE 'LOCAL'
C
      DIMENSION
     *COORD(MCORD,NPOIN),
     *DELTAH(MBTHCK),
     *DISPG(LCORD)
     *E(6),
     *EAN(6),
     *EANC(6),
     *GEI(2, LGASP),
     *GPCOD(LCORD),
     *GRADT(LCORD, MCASI, LGASP),
     *IFBE(MBTHCK),
     *LNODS(MNODE,NELEM),
     *SECT(LGASP),
     *TDISP(MDOFN,NPOIN),
     *T(6),
     *ZETSH(MINT)

C
      REAL VAR
C
      .
      .user code
      .
      RETURN
      END
```

Arguments of the call

VAR value of the variable to be defined (output).

T array of the stress components in the order x-x, y-y, z-z, x-y, y-z, x-z (input).

E	array of the total strain components (input).
EAN	array of the total plastic strain or, for masonry-like materials, total crack strain(input).
EANC	array of the total crushing strain for masonry-like materials with bounded compressive strength (input).
TEMP	Value of the total temperature (input).
GPCOD	Global coordinates of the integration point (input).
DISPG	Displacement at the integration points (input).
IELEM	Number of the element (input).
IGASP	Number of the integration point (input).
ILAY	Number of the layer or fiber(input).
INDVA	Identifier number of the post processing variable to be defined (input).
LNODS	Array of the connectivities of all elements (input).
COORD	Array of the coordinates of all nodes (input).
TDISP	Array of the total displacement for all degrees of freedom (input).
GRADT	Values of the temperature gradient (input).
IFBE	If IFBE(I) is zero indicates that the I-th fiber of the current beam element is dummy (input).
SECT	For beam elements is the area of the cross section at the current Gauss point (input).
GEI	For beam elements is the array of the moment of inertia, around the first two local axes, of the cross section at the current Gauss point (input).
DELTAH	Shell or beam thickness. DELTAH is the thickness array of the layers or fiber sat the current Gauss integration point (input).
ZETSH	ZETSH(I) is the distance from the shell mean surface, or from the beam axis, of the I-th Simpson integration point at the current Gauss integration point.
LCORD	Number of direction coordinates of the integration point (input from COMMON/LOCAL/).

LGASP	Number of Gauss integration points for the current element (input from COMMON/LOCAL/).
MCORD	Maximum number of direction coordinates (input from COMMON/CNTR/).
NPOIN	Maximum number of nodes in the mesh (input from COMMON/CNTR/).
MCASI	Number of the cross section integration points for shell or beam elements (input from COMMON/CNTR/).
MINT	Same as MCASI (input from COMMON/CNTR/).
MBTHCK	Same as MCASI, for beam elements only (input from COMMON/CNTR/).
MNODE	Maximum number of nodes in an element (input from COMMON/CNTR/).
NELEM	Maximum number of elements in the mesh (input from COMMON/CNTR/).

UBND

The routine UBND allows definition of kinematic constraints (displacements, temperatures, accelerations), which vary with position and time; this routine is activated by inserting the character "-" before the degrees of freedom set in the cards BOUNDARY CONDITION or BOUNDARY CHANGE. The routine UBND is called as needed at the beginning of each iteration.

Scheme of the routine

```
      SUBROUTINE UBND ( IFFIX, KFIX, IDOFN, COORD, TDISP,
     *TREAC, FIXED, ICODE)
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
      INCLUDE 'MOTION'
C
      DIMENSION
     *COORD (MCORD),
     *TDISP (MDOFN),
     *TREAC (MDOFN)
      .
      .user code
      .
      RETURN
      END
```

Arguments of the call

IFFIX	Fixity code. It is different from zero if the degree of freedom considered has been constrained in the cards BOUNDARY CONDITIONS, BOUNDARY CHANGE, FIXED DISPLACEMENT, FIXED ACCELERATION or FIXED TEMPERATURE (input/output). If the user wants to release the degree of freedom considered, IFFIX must be set to zero (0).
KFIX	Number of the node (input).
IDOFN	Ordinal number of the degree of freedom of the node KFIX whose constraint condition is managed by the routine (input).
COORD	Array of the KFIX node coordinates (input).
TDISP	Array of total current displacements of the node KFIX (input).
TREAC	Array of the total current reactions at the node KFIX (input).

FIXED — If IFFIX is non-zero, it represents the displacement or temperature value prescribed for the degree of freedom IDOFN of the node KFIX (input/output).

ICODE — If set to zero (0), indicates that a displacement or temperature change is to be calculated. If set to one (1), indicates that a total acceleration is to be calculated.

MCORD — Maximum number of coordinate directions (input from COMMON/CNTR/).

MDOFN — Maximum number of degrees of freedom per node (input from COMMON/CNTR/).

UDSPI

The routine UDSPI allows definition of initial displacement values. It is called once at the beginning of the analysis.

Scheme of the routine

```
      SUBROUTINE UDSPI ( DISPI, COORD, IUSER )
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
C
      DIMENSION
     *COORD (MCORD)
      .
      .user code
      .
      RETURN
      END
```

Arguments of the call

DISPI	Value of the component of initial displacement (output).
COORD	Coordinates of the node considered (input).
IUSER	Its absolute value indicates which degree of freedom must be initialized (input).
MCORD	Maximum number of coordinates of a node (input, from COMMON/CNTR/).

UFILM

The routine UFILM allows definition of the film coefficients in heat transfer analysis. It is called once at the beginning of each time step.

Scheme of the routine

```
      SUBROUTINE UFILM ( IFAC, IFILM, LNODS, COORD, TTEMP,
     * FILM, SINK, FILDOT )
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
      INCLUDE 'LOCAL'
C
      DIMENSION
     *COORD (MDIME,NPOIN),
     *FILDOT(MSIDE),
     *FILM(MSIDE),
     *IFILM(2),
     *LNODS(MNODE),
     *SINK(MSIDE),
     *TTEMP(NPOIN)
      .
      .user code
      .
      RETURN
      END
```

Arguments of the call

COORD	Array of the nodal coordinates (input).
FILDOT	Derivative of film coefficient with respect to the temperature at the nodes on the element face (output).
FILM	Film coefficient at the nodes of the element face (output).
IFILM	If IFILM(1)≠0, the film coefficient is to be calculated; if IFILM(2)≠0, the sink temperature is to be calculated (input).
LNODS	Connectivity of the element (input).
SINK	Sink temperature at the nodes of the element face (output).
TTEMP	Array of the total nodal temperatures.
MDIME	Maximum number of coordinates for each node (input, from COMMON/CNTR/)

NPOIN	Total number of nodes in the mesh (input, form COMMON/CNTR/)
MNODE	Maximum number of nodes for each element (input, from COMMON/CNTR/)
MSIDE	Maximum number of nodes on an element face or edge (input, from COMMON/CNTR/)

UFRI

The routine UFRI allows definition of the friction coefficient as a function of position, time and conditions at the interface between the deformable body and a rigid surface. This routine is activated by inserting the character "-" before the value of the coefficient declared in the card FRICTION. The routine UFRI is called as needed at the beginning of each iteration.

Scheme of the routine

```
      SUBROUTINE UFRI ( THRS, REACN, REACT, KFRICT, CFRICT,
     *COORD, VERSN, VERST )
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
      INCLUDE 'MOTION'
C
      DIMENSION
     *COORD (MCORD),
     *VERSN (MDIME),
     *VERST (MDIME)
      .
      .user code
      .
      RETURN
      END
```

Arguments of the call

THRS	Limit value of the friction force (input/output).
REACN	Value of the contact force along the vector orthogonal to the interface between the deformable body and rigid surface (input).
REACT	Value of the force along the vector tangential to the interface between the deformable body and rigid surface (input).
KFRICT	Code indicating the current condition of the friction (input). = 1, no friction; = 2, blockage condition; = 3, slipping condition at constant force.
CFRICT	Friction coefficient (input/output).
COORD	Array of coordinates of the node considered (input).
VERSN	Unit vector orthogonal to the interface oriented towards the interior of the deformable body (input).

VERST Unit vector tangential to the interface (VERST can be obtained by rotating VERSN 90° in the counter-clockwise direction) (input).

MCORD Maximum number of coordinates per node (input, from COMMON/CNTR/).

MDIME Dimensions of the structure (2 in this version of NOSA) (input, from COMMON/CNTR/).

UGEOM

The routine UGEOM allows definition of thickness of the layers of a shell element, or the thickness of the fibers of a beam element, on a nodal basis.

Scheme of the routine

```
      SUBROUTINE UGEOM ( COORD, LNODS, GEOM, KTYPE, LCORN )
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
      INCLUDE 'ELEM'
C
     DIMENSION
    *COORD(MCORD, NPOIN),
    *GEOM(MDT, LCORN),
    *LNODS(MNODE)
      .
      .user code
      .
     RETURN
     END
```

Arguments of the call

COORD	Array of coordinates of the node considered (input).
LNODS	Connectivity of the element (input).
GEOM	Array of the nodal thickness (output)
KTYPE	Identifier of the element type (input).
LCORN	Number of the corner nodes of the shell element (input).
MCORD	Maximum number of coordinates for a node (input from COMMON/CNTR/).
NPOIN	Maximum number of nodes in the mesh (input from COMMON/CNTR/).
MDT	Number of layers/fibers of a shell/beam element (input from COMMON/CNTR/).
MNODE	Maximum number of nodes in an element (input from COMMON/CNTR/).

ULAXIS

The routine ULAXIS allows definition of the local reference system on the cross section of the beam elements.

Scheme of the routine

```
      SUBROUTINE ULAXIS ( IELEM, LNODS, COORD, AXIS )
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
C
     DIMENSION
     *AXIS(MCORD),
     *COORD(MCORD, NPOIN),
     *LNODS(MNODE)
      .
      .user code
      .
     RETURN
     END
```

Arguments of the call

AXIS	Components of the unit vector defining the first direction of the local reference system (output).
COORD	Array of coordinates of the node considered (input).
LNODS	Connectivity of the element (input).
MCORD	Maximum number of coordinates for a node (input from COMMON/CNTR/).
NPOIN	Maximum number of nodes in the mesh (input from COMMON/CNTR/).
MNODE	Maximum number of nodes in an element (input from COMMON/CNTR/).

UMOTIO

The routine UMOTIO allows definition of the velocity of a rigid surface as a function of the time and the position of the surface.

Scheme of the routine

```
      SUBROUTINE UMOTIO ( SURF, TIM, TIMSTP, JSURF )
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
C
      DIMENSION SURF (8, MPART+1, MSURF)
      .
      .user code
      .
      RETURN
      END
```

Arguments of the call

SURF	Position and velocity of the rigid surface considered (input/output).
TIM	Total time since the start of the analysis (input).
TIMSTP	Time increment (input/output).
JSURF	Ordinal number of the rigid surface under consideration (input).
MPART	Maximum number of parts making up the rigid surface under consideration (input from COMMON/CNTR/).
MSURF	Maximum number of rigid surfaces (input, from COMMON/CNTR/).

The data relative to the J-th part of the I-th rigid surface are contained in the array SURF as follows:

SURF(1, 1, I)	Current x coordinate of the surface guide node.
SURF(2, 1, I)	Current y coordinate of the surface guide node.
SURF(3, 1, I)	Component x of the current velocity of the surface guide node.
SURF(4, 1, I)	Component y of the current velocity of the surface guide node.
SURF(5, 1, I)	Current angular velocity of the surface guide node.

SURF(6, 1, I) Initial coordinate x of the surface guide node.

SURF(7, 1, I) Initial coordinate y of the surface guide node.

SURF(8, 1, I) Current rotation angle of the surface guide node.

If the J-th part is a segment, we have:

SURF(1, J+1, I) Current coordinate x of the segment's starting point

SURF(2, J+1, I) Current coordinate y of the segment's starting point.

SURF(3, J+1, I) Current coordinate x of the segment's end point.

SURF(4, J+1, I) Current coordinate y of the segment's end point.

If the J-th part is a circumference arc, we have:

SURF(1, J+1, I) Current coordinate x of the arc's starting point.

SURF(2, J+1, I) Current coordinate y of the arc's starting point

SURF(3, J+1, I) Current coordinate x of the arc's end point.

SURF(4, J+1, I) Current coordinate y of the arc's end point.

SURF(5, J+1, I) Current coordinate x of the center of the circumference.

SURF(6, J+1, I) Current coordinate y of the center of the circumference.

SURF(7, J+1, I) Center angle subtended by the arc.

SURF(8, J+1, I) Radius of the circumference.

UPLOAD

Routine for management of the point loads or fluxes as functions of position and time.

Scheme of the routine

```
      SUBROUTINE UPLOAD ( IPOIN, COORD, DISP, PLOAD, SELDIS,
     *LNODS, ICKFR )
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
      INCLUDE 'MOTION'
C
      DIMENSION
     *COORD(MCORD),
     *DISP(MDOFN),
     *ICKFR(2, MDOFN),
     *LNODS(MNODE, NELEM),
     *PLOAD(MDOFN),
     *SELDIS(MDOFN, MNODE, NELEM)
      .
      .
       .user code
      .
      .
      RETURN
      END
```

Arguments of the call

IPOIN	Ordinal number of the loaded node (input).
COORD	Array of the initial coordinates of the loaded node (input).
DISP	Array of the total current displacements or temperature change of the loaded node (input).
ICKFR	ICKFR(1, I) indicates the element where the I-th d.o.f of the node IPOIN appears for the first time. ICKFR(2, I) indicates the element where the I-th d.o.f. of the node IPOIN can be eliminated from the solution front (input).
LNODS	Array of the connectivities for all elements in the mesh (input).
PLOAD	Array of the point loads or fluxes applied at the node IPOIN (output).
MCORD	Maximum number of coordinates of a node node (input, from COMMON/CNTR/).

MDOFN Maximum number of degrees of freedom of a node (input, from COMMON/CNTR/).

UPMAS

UPMAS calculates the concentrated masses as a function of the position of the nodes.

Scheme of the routine

```
      SUBROUTINE UPMAS ( PMASS, COORD, IUSER.)
C
       IMPLICIT REAL*8 (A-H,O-Z)
C
       INCLUDE 'CNTR'
C
       DIMENSION
     *COORD(MCORD),
     *PMASS(MDOFN)
      .
       .user code
      .
     RETURN
     END
```

Arguments of the call

COORD	Array of the initial coordinates of the loaded node (input).
PMASS	Array of the concentrated masses on a node (output). It should be noted that these are real masses only for translational degrees of freedom, whereas inertia moments are connected to rotational degrees of freedom.
IUSER	Fixed to 1 (input).
MCORD	Maximum number of coordinates of a node (input, from COMMON/CNTR/).
MDOFN	Maximum number of degrees of freedom of a node (input, from COMMON/CNTR/)

URDTEM

Routine for calculating the temperature values at the nodes.

Scheme of the routine

```
      SUBROUTINE URDTEM(COORD, DTTMP, DTEMP, TEMPE, NAUTO,
     *ICODE )
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
      INCLUDE 'CREAD'
      INCLUDE 'TMPPST'
      INCLUDE 'FILES'
      INCLUDE 'ELEM'
C
      DIMENSION
     *COORD(MCORD,NPOIN),
     *DTTMP(ITYRD,NPOIN),
     *DTEMP(ITYRD,NPOIN),
     *TEMPE ITYRD,NPOIN)
      .
       .user code
      .
      RETURN
      END
```

Arguments of the call

COORD	Array of the initial coordinates of the loaded node (input).
DTTMP	Array of the variation of the nodal temperature at the end of the series of NAUTO increments (output).
DTEMP	Array of the nodal temperature increment (output).
TEMPE	Array of the current nodal total temperature (input).
ITYRD	Indicator of the interpolation type of the temperature across the thickness of a shell element (input, from COMMON/CNTR/).
MCORD	Maximum number of coordinates of a node (input, from COMMON/CNTR/).
NPOIN	Maximum number of nodes in the mesh (input, from COMMON/CNTR/).

UTEMPI

The routine UTEMPI allows definition of initial temperature values. It is called once at the beginning of the analysis.

Scheme of the routine

```
      SUBROUTINE UTEMPI ( TEMPI, COORD, ID, ICODE )
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
C
      DIMENSION
     *COORD (MCORD)
      .
      .user code
      .
      RETURN
      END
```

Arguments of the call

TEMPI	Value of the component of initial temperature (output).
COORD	Coordinates of the node considered (input).
ID	Used for shell elements. Its value varies from 1 to ITYRD (input).
ICODE	Fixed to 1.
MCORD	Maximum number of coordinates of a node (input, from COMMON/CNTR/).

UTIE

Routine for management of the tying relations among degrees of freedom as functions of position and time. A tying relation means that a degree of freedom (tied) is constrained to assume the value of a linear combination of the values of other degrees of freedom (retained). The routine UTIE is called at the beginning of each load increment.

Scheme of the routine

```
      SUBROUTINE UTIE ( ITIE, RTIE, IRTIE, COORD, TDISP, TREAC, M,
     *ITYCH )
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
      INCLUDE 'FILES'
C
      DIMENSION
     *COORD(MDOFN, NPOIN),
     *ITIE(M+2),
     *IRTIE(MDOFN, MRET, MDOFN),
     *RTIE(MDOFN, MRET, MDOFN),
     *TDISP(MDOFN, NPOIN),
     *TREAC(MDOFN, NPOIN)

      LOGICAL ITYCH
        .
      user code
        .
      RETURN
      END
```

Arguments of the call

ITIE(1) Identifier number of the tying relation (input).

ITIE(2) Ordinal number of the tied node (input).

ITIE(3) Ordinal numbers of the retained nodes (input).

.

.

ITIE(M+2)

RTIE(I, K, J) Array of the coefficients of the linear relations among the degrees of freedom of the tied node and the degrees of freedom of the retained node. RTIE(I,K,J) links the J-th degree of freedom of the tied node to the I-th degree of freedom of the K-th retained node (output).

IRTIE(I, K, J) Map of the non-zero coefficients contained in the array RTIE.
If IRTIE(I, K, J) is equal to one (1) then the corresponding coefficient RTIE(I, K, J) is different from zero (output).

COORD Array of coordinates of all nodes in the mesh (input).

TDISP Array of the total displacements of all nodes in the mesh (input).

TREAC Array of the total reactions of all nodes in the mesh.

M Number of retained nodes relative to the tied node under consideration (input).

ITYCH Indicator of a change in the overall structure of the tyings (input/output).

MDOFN Maximum number of degrees of freedom per node (input from COMMON/CNTR/).

NPOIN Total umber of nodes in the mesh (input from COMMON/CNTR/).

MRET Maximum number of retained nodes per tied node (input from CPMMON/CNTR/).

UVELI

The routine UVELI allows definition of initial velocities values. It is called once at the beginning of the analysis.

Scheme of the routine

```
      SUBROUTINE UVELI ( VELOI, COORD, IUSER, NODE )
C
      IMPLICIT REAL*8 (A-H,O-Z)
C
      INCLUDE 'CNTR'
C
      DIMENSION
     *COORD (MCORD)
      .
      .user code
      .
      RETURN
      END
```

Arguments of the call

VELOI	Value of the component of initial velocity (output).
COORD	Coordinates of the node considered (input).
IUSER	Its absolute value is the identifier of the displacement component (input).
NODE	Identifier number of the node (input).
MCORD	Maximum number of coordinates of a node (input, from COMMON/CNTR/).

F. DESCRIPTION OF THE COMMON VARIABLES

In this section a brief description of the variables contained in the various COMMON of the NOSA will be given. The COMMON/BLNK/, which contains the working areas, will be described in the next section.
A routine may access the data contained in the COMMON through an instruction INCLUDE (followed by the name of the COMMON)

VARIABLES OF THE COMMON/CNTR/

This COMMON contains most of the control parameters.

```
 common/cntr/icard, iincs, iiter, ioutp, ityrd, isave, kearea,
*            kresl, mbcasi, lvprt, mbdt, mbuf1, mbuf2, mbuf3,
*            mbpint, mbint(2), mbthck, mbshel(2), mcasi, mchan,
*            mchan1, mcord, mdefo, mdist, mdime, mdofn, mdt,
*            mint, mpint, mearea, mevab, mfac, mfron, mgaus,
*            mgeom, mhard, miter, mnode, mpart, mpart0, mprop,
*            mret, mset, msete, mseten, msetn, mshel, mside,
*            mstif, msurf, mtab, mthick, mtie, mttemp, mtype,
*            mvsete, mvsetn, nalgo, nelem, nmaph, nmats, npoin,
*            nraph,
*            ialia, ibeam, icomp, icore, idamp, idyna, ielas,
*            ielsh, iifilm, ifini, ifoll, ifric, iheat, imaso,
*            irest, irsdis, iscal, istop, itres, first, lump,
*            lener
 logical    ialia, ibeam, icomp, icore, idamp, idyna, ielas,
*           ielsh, iifilm, ifini, ifoll, ifric, iheat, imaso,
*           irest, irsdis, iscal, istop, itres, first, lump,
*           lener
```

Variable	Description
FIRST	If .TRUE., the analysis is at the beginning (the first iteration of first increment).
KEAREA	Number of double words making up the working area which contains the elemental data. When the in-core solution is possible, KEAREA is the sum of the dimensions JEAREA for all elements; otherwise it is the maximum value of the elemental JEAREA (calculated in the routine CORE).
KRESL	Identifier of the solution method used: = 1, resolution with re-assembly of the stiffness matrix; = 2, resolution via a simple back substitution, without any assembly of the stiffness matrix. (calculated in the routine ALGOR).
IALIA	If .TRUE., indicates the presence of element aliases (default .FALSE.).
IBEAM	If .TRUE., indicates the presence of beam elements (default .FALSE.).
ICARD	FORTRAN unit for reading the input data (default JINP = 5).
ICOMP	If .TRUE., indicates that the sections of the shell or beam element are of non-homogeneous thickness and/or possess non-homogeneous mechanical

properties. Activation is required from the card COMPOSITE (default .FALSE.).

ICORE — If .TRUE., indicates that an out-of-core analysis will be performed (default .FALSE., calculated in the routine CORE).

IDAMP — If .TRUE., indicates that the damping terms have to be calculated in a dynamic analysis (default .FALSE.).

IDYNA — If .TRUE., indicates a dynamic analysis (default .FALSE.).

IELAS — If .TRUE., indicates selection of the option ELASTIC and that a linear elastic analysis will therefore be performed (default .FALSE.).

IELSH — If .TRUE., indicates the presence of shell elements in the mesh (default .FALSE.).

IFINI — If .TRUE., indicates selection of the option FINITE STRAIN and that finite deformation will therefore be considered (default .FALSE.).

IFOLL — If .TRUE., indicates selection of the option FOLLOWER FORCES for calculation of the local distributed loads in the current configuration (default .FALSE.).

IFRIC — If .TRUE., indicates the use of friction coefficients in a contact analysis (default .FALSE.).

IHEAT — If .TRUE., indicates a heat transfer analysis (default .FALSE.).

IIFILM — If .TRUE. indicates the use of film coefficients in a heat transfer analysis (default .FALSE.).

IINCS — Number of the current load increment (calculated in the routine PSTRES).

IITER — Number of the current iteration (calculated in the routine PSTRES).

IMASO — If .TRUE., indicates selection of the option MASONRY in the control cards (default .FALSE.).

IOUTP — Indicates the print level of the results on the output file.
= 0, no print;
= 1, results are printed when the incremental convergence is reached;
> 1, results are printed at the end of each iteration.
This values is read from the routine LOADIN in the card PRINT CHOICE, of the load deck (default 1).

IREST — If .TRUE., indicates selection of the option RESTART (default .FALSE.).

IRSDIS — If .TRUE., indicates that, in a dynamic analysis, the iteration convergence criterion is based on the residual displacements instead on the residual forces (default .FALSE.).

ISAVE — If different from zero, it indicates selection of the option SAVE INCREMENT in the load cards and its value represents the saving frequency (default 0, *i.e.* no saving).

ISCAL — If .TRUE., indicates selection of the option SCALE in the control cards (default .FALSE.).

ISTOP — If .TRUE., it indicates selection of the option STOP in the control cards (default .FALSE.).

ITRES — If .TRUE., it indicates selection of the option ITRESS in the control cards (default .FALSE.).

ITYRD — Method for interpolating temperatures across the thickness in shell elements.
= 1, no interpolation (this is the value to be used with all elements except shells);
=2, linear interpolation across the thickness (the temperature values on the top and bottom layers must be present in the post-processing file);
= 3, parabolic interpolation across the thickness (the temperature values on the top, bottom and middle layers must be present in the post-processing file).

LENER — If .TRUE., it indicates selection of the option ENERGY in the control cards (default .FALSE.).

LUMP — If .TRUE., it indicates selection of the option LUMPED MASS in the control cards (default .FALSE.).

LVPRT — Print level in output.
= 1, displacements, nodal reactions, strains and stresses at the integration points of the elements are printed;
>1, displacements, nodal reactions, equivalent nodal forces, strains and stresses at the integration points of the elements are printed.
This value is specified in the card PRINT LEVEL of the control cards (default 1).

MBCASI — If beam elements are present, it takes the value MBINT(1) + MBINT(2), otherwise it is set to 1 (default value).

MBDT — If beam elements are present, it takes the value MBSHEL(1) + MBSHEL(2), otherwise it is set to 1 (default value).

MBINT — If beam elements are present, MBINT(1) is the number of section integration points along the first local direction and MBINT(2) is the number of section

integration points along the second local direction, otherwise they are set to 1 (default values).

MBPINT — If beam elements are present, it takes the value MBSHEL(1) * MBSHEL(2) (*i.e.* the total number of integration points on the cross section of the beam) , otherwise it is set to 1 (default value).

MBSHEL — If beam elements are present, MBSHEL(1) is the number of section fibers along the first local direction and MBINT(2) is the number of section fibers along the second local direction, otherwise they are set to 1 (default values).

MBTHCK — If beam elements are present, it takes the value MBSHEL(1) * MBSHEL(2) (i.e. the total number of fibers on the cross section of the beam), otherwise it is set to 1 (default value).

MBUF1 — Dimension of the arrays for the definition of load increments. Its value is NELEM in case of in-core solution, whereas it is calculated in the routine CORE in case of out-of-core solution.

MBUF2 — Dimensions of the arrays used for resolution; it is equal to MDOFN*NPOIN if an in-core solution is performed, whereas it is calculated from the routine CORE for the out-of-core solution.

MBUF3 — Indicates how many elemental work-areas are contained in the EAREA array. Its value is NELEM in case of in-core solution, whereas it is calculated in the routine CORE in case of out-of-core solution.

MCASI — Maximum number of integration points trough the thickness for shell elements.

MCHAN — Reserved for future development.

MCHAN1 — Reserved for future development.

MCORD — Maximum number of coordinates per node.

MDEFO — Maximum number of deformable bodies.

MDIST — Maximum number of different (by type or element) distributed loads or fluxes (default 3).

MDIME — Maximum number of dimensions of the structure.

MDOFN — Maximum number of degrees of freedom per node.

MDT — The maximum between MSHEL and MBDT.

MEAREA — Maximum number of double words in an elemental work-area.

MEVAB	Maximum number of degrees of freedom per element.
MFAC	Maximum number of faces per element.
MFRON	Maximum length of the resolution front (calculated in the routine CHECK).
MGAUS	Maximum number of Gauss integration points for each local direction.
MGEOM	Number of nodes per element with variable thickness.
MHARD	Maximum number of hardening curves which are present. This value is read from the card HARDENING in the control cards (default 0).
MINT	The maximum between MCASI and MBCASI.
MITER	Maximum number of iterations permitted for each load increment (default 3).
MNODE	Maximum number of nodes per element.
MPART	Maximum number of parts making up a rigid surface (default 0).
MPART0	Maximum number of post-processing parts making up a rigid surface.
MPINT	The maximum between MCASI and MBPINT.
MPROP	Maximum number of properties for each material (default 10).
MRET	Maximum number of retained nodes for each tied node in the tying relations (default 0).
MSET	Maximum number of entity (nodes or elements) making up a set (default 0).
MSETE	Maximum number of sets of elements present in the structure (default 0).
MSETEN	Maximum number of entities in a compound set (set to MSET as default value).
MSETN	Maximum number of sets of nodes present in the structure (default 0).
MSHEL	Maximum number of layers for a shell element (default 3).
MSIDE	Maximum number of nodes belonging to an element face.
MSTIF	= MFRON*(MFRON + 1)/2, if the stiffness matrix is symmetric; = MFRON*MFRON, if the stiffness matrix is non-symmetric.

MSURF	Maximum number of rigid surfaces present in the structure (default 0).
MTAB	Maximum number of tables of temperature dependent properties (default 0).
MTHICK	The maximum between MSHEL and MBTHCK.
MTIE	Maximum number of tied nodes in the tying relations (default 0).
MTTEMP	Maximum number of data pairs in each table of temperature dependent properties.
MTYPE	Maximum number of different element types present in the structure (default 1).
MVSETE	Maximum number of items in an element set.
MVSETN	Maximum number of items in a node set.
NALGO	Identifier of the type of algorithm used at each iteration = 1, initial stiffness matrix; = 2, modified Newton-Raphson method, = 3, Newton-Raphson method.
NELEM	Maximum number of elements.
NMAPH	Maximum number of materials with anisotropic yield stress.
NMATS	Maximum number of materials present in the structure.
NPOIN	Maximum number of nodes.
NRAPH	Maximum number of values for the anisotropic yield stress .

VARIABLES OF THE COMMON/CONVRG/

This COMMON contains the parameters that control the convergence of increments.

```
 common/convrg/presd ,dtmax, pscmax,facto ,toler, fuztol,
*              fuzdis,fuztmp,tmpfin,ncheck,nchkou,
*              maxinc,numstp
 logical ncheck,nchkou
```

Variable	Description
DTMAX	Maximum value of the temperature change found in a heat transfer increment.
FACTO	Proportionality factor which multiplies the load increment, as declared in the cards PROPORTIONAL INCREMENT (default 1.).
FUZDIS	The minimum value of the norm of the incremental displacements in order the displacement convergence check be done in dynamic analysis. The displacement convergence is used in dynamic analysis if the norm of total forces, including inertia and damping forces, is less than FUZTOL or if this option is explicitly set in the cards CONTROL.
FUZTMP	The minimum value of DTMAX in order to allow the step size variation in heat transfer analysis.
FUZTOL	The minimum value of the norm of total forces in order the convergence check be done.
MAXINC	Maximum number of increments allowed in an analysis.
NCHECK	If .TRUE. the increment satisfies the convergence criteria, and the next increment can be processed.
NCHKOU	Output parameter. If .TRUE. the output of the current increment can be done.
NUMSTP	Maximum number of steps in a series, for heat transfer analysis.
PRESD	Ratio in % between the norm of residual forces/displacements and the norm of the total forces or incremental displacements.
PSCMAX	Factor by which the values of elemental quantities must be divided so that the most stressed integration point is brought to the first yield limit.
TMPFIN	The maximum value allowed for the temperature change in an increment of the heat transfer analysis.

TOLER The maximum ratio, in percent, between the norm of residual forces/displacements and the norm of the total external forces or the incremental displacements must be less than TOLER, in order that an increment satisfy the convergence criterion (default 0.01%).

VARIABLES OF THE COMMON/CREAD/

This COMMON contains the variables for decoding the input data.

```
      common /cread/ flout(8000),card(80000),iout(16000),
     *               char1(10,16000),lchar1(16000)
      character*1 card,char1
```

Variable	Description
CARD	Array containing an input data card and its continuations, if any, in formatted characters.
CHAR1	Array containing alphanumeric fields (each fields has a maximum number of ten characters) present in an input data card and its continuations, if any.
FLOUT	Array containing the decoding (from formatted characters to double precision real number) of fields present in an input data card and its continuations, if any.
IOUT	Array containing the decoding (from formatted characters to integer numbers) of fields present in an input data card and its continuations, if any.
LCHAR1	Array containing the lengths, in characters, of alphanumeric fields present in CHAR1.

VARIABLES OF THE COMMON/DYNA/

This COMMON contains the parameters characteristic Newmark integration method for the dynamic analysis.

```
common/dyna/gamnew, betnew
```

Variable	Description
BETNEW	Weighting factor of the acceleration in the interpolation of the displacement in the Newmark method (default .25).
GAMNEW	Weighting factor of the acceleration in the interpolation of the velocity in the Newmark method (default 0.5).

VARIABLES OF THE COMMON/ELEM/

This COMMON contains the parameters characteristic of the different element types (maximum thirty) used in the analysis.

```
 common /elem/ ncord(30) ,ncorn(30) ,ndime(30) ,ndofn(20,30),
*              nevab(30) ,nfac(30)  ,ngasp(30) ,ngaus(30) ,
*              nnode(30) ,nprinc(30),nside(30) ,nstre(30) ,
*              nstr1(30) ,ntype(30) ,nalia(30)
```

Variable	Description
NALIA(I)	If different from 0, indicates the alias identifier of the I-th element type.
NCORD(I)	Number of coordinates for each node of the I-th element type.
NCORN(I)	Number of corner nodes of the I-th element type.
NDIME(I)	Number of dimensions of the I-th element type.
NDOFN(J, I)	Number of degrees of freedom of the J-th node of the I-th element type.
NEVAB(I)	Maximum number of degrees of freedom of the I-th element type.
NFAC(I)	Number of faces of the I-th element type.
NGASP(I)	Total number of Gauss integration points of the I-th element type.
NGAUS(I)	Number of Gauss integration points for each local direction of the I-th element type.
NNODE(I)	Total number of nodes of the I-th element type.
NPRINC(I)	Number of principal stress components in each integration point of the I-th element type.
NSIDE(I)	Number of nodes belonging to a face of the I-th element type.
NSTRE(I)	Total number of stress components in each integration point of the I-th element type.
NSTR1(I)	Total number of components of the deviatoric part of the stress in each integration point of the I-th element type.
NTYPE(I)	Identifier of the I-th element type.

VARIABLES OF THE COMMON/ENERG/

This COMMON contains the values of the energy terms in the mesh.

```
common/energ/enerm, enerc, enerp, workex
```

Variable	Description
ENERC	The value of the total kinetic energy.
ENERM	ENERC+ENERP-WORKEX.
ENERP	The value of the total deformation energy.
WORKEX	The value of the total work done by the externally applied forces, including the reaction forces.

VARIABLES OF THE COMMON/ERROD/

This COMMON contains the parameters for management of accidental errors in the input data.

```
common /errod/ neror(200), ifatl, inftl
```

Variable	Description
IFATL	Number of the errors encountered which prevent the analysis from continuing.
INFTL	Number of the errors encountered which do not prevent the analysis from continuing.
NEROR(I)	Number of type I errors found in the reading and decoding of the input data.

VARIABLES OF THE COMMON/FILES/

This COMMON contains the numbers of the I/O FORTRAN units.

```
 common /files/ jinp ,jout, jfil1, jfil2, jfil3, jfil4, iposb,
*               iposf, jrest, jpsttf, jpsttb, nfil2, krec2,
*               nrec2
```

Variable	Description
IPOSB	= 21 FORTRAN unit for writing the post-processing data in binary format.
IPOSF	= 20 FORTRAN unit for writing the post-processing data in character format.
JFIL1	= 1 FORTRAN unit for writing the definition data of distributed loads in case of out-of-core resolution.
JFIL2	= 2 FORTRAN unit for writing the reduced equation in the case of out-of-core resolution, if the required space is less than 2 Gbytes. When more than 2 Gbytes are required, the space is subdivided into NFIL2 files, each one smaller than 2 Gbytes, so we shall have JFIL2=30+I, with I ranging from 1 to NFIL2.
JFIL3	= 3 direct access FORTRAN unit for writing the data contained in the elemental work areas in the case of out-of-core solution.
JFIL4	= 4 FORTRAN unit for writing the right-hand members of the reduced equations for out-of core back-substitution.
JINP	= 5 FORTRAN unit for reading the input data.
JOUT	= 8 FORTRAN unit for printing the results of the analysis.
JPSTTB	= 25 FORTRAN unit for reading the binary post-processing file containing temperature values for the thermal loads evaluation.
JPSTTF	= 26 FORTRAN unit for reading the formatted post-processing file containing temperature values for the thermal loads evaluation.
JREST	= 99 FORTRAN unit for writing and reading analysis data in the case of saving the load increment or restart of an analysis.
KREC2	It is the record index into the current file used to store the reduced equations in the case of out-of-core solution. It varies from 1 to NREC2.

NFIL2	Number of files, each one smaller than 2 Gbytes, required for writing the reduced equations in the case of out-of-core solution.
NREC2	It is the maximum number of records in each file used to store the reduced equations in the case of out-of-core solution.

VARIABLES OF THE COMMON/FREQPR/

This COMMON contains the parameters which manage output data printing.

```
common /freqpr/ifreqp, nelpr, nodpr
```

Variable	Description
IFREQP	Printing frequency of incremental data. If printouts are required, the results will be printed every IFREQP increments (default 1).
NELPR	If different from zero, it indicates that a printout of results is required only for a selected set of elements (default 0).
NODPR	If different from zero, it indicates that a printout of results is required only for a selected set of nodes (default 0).

VARIABLES OF THE COMMON/LOADS/

This COMMON contains the parameters for definition of centrifugal loads.

```
common /loads/rotax(6), icent
```

Variable	Description
ICENT	If different from zero, it indicates that a rotation axis has been defined (default 0).
ROTAX(1..3)	Components of the unit vector corresponding to the angular velocity (default 0., 0., 1.).
ROTAX(4...6)	Coordinates of a point belonging to the rotation axis (default 0., 0., 0.).

VARIABLES OF THE COMMON/LOCAL/

This COMMON contains the information relative to the element under consideration. The presence of different values corresponding to different section integration points (up to 441 layers for shells or 21×21 fibers for beams) has meaning only for non-homogeneous shell/beam elements.

```
 Common /local/young(441), poiss(441), shear(441), alph1(441),
*              bulk(441) , yeld(441) , yelc(441) , alfa(441) ,
*              beta(441) , cke1(441) , cke2(441) , rho0(441) ,
*              cond(441) , sphe(441) , dens(441) , emis(441) ,
*              alpha ,itype ,ktype ,lcord ,lcorn ,ldime ,
*              ldofn ,levab ,lfac  ,lgasp ,lgaus ,lnode ,
*              lprinc,lprop(441)   ,lside ,lstre ,lstr1 ,
*              ldelta,lins  ,lint  ,lpint ,lthck
```

Variable	Description
ALFA(I)	Value of the parameter which intervenes in the kinematic hardening model for infinitesimal elasto-plasticity, corresponding to the I-th section integration point of the element[1].
ALPHA	Reserved for future developments.
ALPH1(I)	Value of the coefficient of thermal dilatation corresponding to the I-th section integration point of the element.
BETA(I)	Value of the parameter b which intervenes in the isotropic hardening model for infinitesimal elasto-plasticity, corresponding to the I-th section integration point of the element[1].
BULK(I)	Value of the bulk modulus corresponding to the I-th section integration point of the element.
CKE1(I)	Value of the parameter e_1 which intervenes in the kinematic hardening model for infinitesimal elasto-plasticity, corresponding to the I-th section integration point of the element[1].
CKE2(I)	Value of the parameter e_2 which intervenes in the kinematic hardening model for infinitesimal elasto-plasticity, corresponding to the I-th section integration point of the element[1].
COND(I)	Value of the heat conductivity corresponding to the I-th section integration point of the element.

[1] These parameters are valid only for 2d and 3d elements, so only the value in the position 1 is meaningful.

DENS(I) Value of the density corresponding to the I-th section integration point of the element

EMIS(I) Value of the emissivity corresponding to the I-th section integration point of the element.

ITYPE Ordinal number of the element identifier as specified in cards SIZING and/or ELEMENTS.

KTYPE Identifier of the element type.

LCORD Number of coordinates for each node of the element.

LCORN Number of corner nodes in the element.

LDELTA = MSHEL for shell elements,
= MBDT for beam element,
= 0 in all other cases.

LDIME Number of element dimensions.

LDOFN Maximum number of degrees of freedom for each node of the element.

LEVAB Total maximum number of degrees of freedom of the element.

LFAC Number of faces of the element.

LGASP Total number of Gauss integration points of the element.

LGAUS Number of Gauss integration points of the element for each local direction.

LINS = MCASI for shell elements,
= 1 for beam element,
= 1 in all other cases.

LINT = MCASI for shell elements,
= MBCASI for beam element,
= 0 in all other cases.

LNODE Total number of element nodes.

LPINT = MCASI for shell elements,
= MBPINT for beam element,
= 1 in all other cases.

LPRINC Number of principal stress components for each integration point of the element.

LPROP(I) Identifier of the material at the I-th section integration point of the element.

LSIDE Number of nodes belonging to an element face.

LSTRE Number of stress components for each integration point of the element.

LSTR1 Number of components in the deviatoric part of the stress for each integration point of the element.

LTHCK = MSHEL for shell elements,
= MBTHCK for beam element,
= 0 in all other cases.

POISS(I) Value of the Poisson ratio corresponding to the I-th section integration point of the element.

RHO0(I) CKE1(I) + CKE2(I).

SHEAR(I) Value of the shear stiffness module corresponding to the I-th section integration point of the element.

SPHE(I) Value of the specific heat per unit mass corresponding to the I-th section integration point of the element.

YELC(I) For masonry-like materials, the value of the compressive strength for the I-th section integration point of the element.

YELD(I) For elastic-plastic materials, the limit value of first shear yield for the I-th section integration point of the element. For masonry-like materials the value of the tensile strength for the I-th section integration point of the element.

YOUNG(I) Value of the Young modulus corresponding to the I-th section integration point of the element.

VARIABLES OF THE COMMON/LSIZE/

This COMMON contains the lengths in double words of the arrays making up the work areas containing the elemental data.

```
 common /lsize/kgpcod(30),ktstre(30),ktstra(30),kstrap(30),
*              kstrup(30),kcentr(3) ,kacca(30) ,kzeta(30) ,
*              kdmatg(30),kcderv(30),kdvolu(30),kdstre(30),
*              kdstra(30),ksdh(30)  ,ksdc(30)  ,ksdepl(30),
*              ksdupl(30),ksdz(30)  ,kther(30) ,kttther(30),
*              krspti(30),krsptf(30),kcostr(30),kgpsph(30),
*              kgpcon(30),ktthei(30),kgradt(30),kcder0(30),
*              kemass(30),kshap5(30),kbmat5(30),kbmat9(30),
*              kbma10(30),kcbase(30),kgpbas(30),kthick(30),
*              kzetsh(30),kifbe(30) ,ksect(30) ,kgei(30)  ,
*              kenerg(30),kcmat(30)
```

Variable	Description
KACCA(I)	Dimensions of the array ACCA defining the exponential part of the kinematic hardening for each integration point of the I-th type of element.
KBMAT5	Dimensions of the array BMAT5 containing the strain-displacement matrix for each integration point of the thin shell element.
KBMAT9	Dimensions of the array BMAT9 containing the strain-displacement matrix for each integration point of the beam element.
KBMA10	Dimensions of the array BMA10 containing the strain-displacement matrix for each integration point of the thick shell element.
KCBASE	Dimensions of the array CBASE containing the centroidal base vectors of the shell/beam elements.
KCDERV(I)	Dimensions of array CDERV containing the Cartesian derivatives of the shape functions for each integration point of the I-th type of element.
KCDER0(I)	Dimensions of array CDER0 containing the Cartesian derivatives of the shape functions calculated at the centroid of the I-th type of element.
KCENTR(I)	Dimensions of array CENTR containing the current values of components of the center of elastic range for each integration point of the I-th type of element.
KCMAT	Dimensions of the array CMAT containing the damping matrix used in dynamic analysis.

KCOSTR(I) Dimensions of array COSTRE containing the corrective term to the stiffness matrix due to the presence of anisotropic yield materials for each integration point of the I-th type of element.

KDMATG(I) Dimensions of array DMATG containing the constitutive matrix for each integration point of the I-th type of element.

KDSTRA(I) Dimensions of array DSTRA containing the incremental values of the total deformation components for each integration point of the I-th type of element.

KDSTRE(I) Dimensions of array SDSTRE containing the incremental values of the Cauchy stress components for each integration point of the I-th type of element.

KDVOLU(I) Dimensions of array DVOLU containing the elements of volume or surface used in the Gauss quadrature for each integration point of the I-th type of element.

KEMASS(I) Dimensions of the array EMASS containing the mass matrix terms of the I-th type of element.

KENERG Dimensions of the array ENERG containing the densities of energy and external work.

KGEI Dimensions of the array GEI containing the cross section moments of inertia for beam elements.

KGPBAS Dimensions of the array GPBASE containing the base vectors for each integration point of the shell elements.

KGPCOD(I) Dimensions of the array GPCOD containing the values of the Cartesian coordinates of the Gauss integration points of the I-th type of element.

KGPCON(I) Dimensions of the array GPCON containing the heat conductivity values for each integration point of the I-th type of element.

KGPSPH(I) Dimensions of the array GPSPH containing the specific heat values for each integration point of the I-th type of element.

KGRADT(I) Dimensions of the array GRADT containing the temperature gradient values for each integration point of the I-th type of element.

KIFBE Dimensions of the array IFBE containing, in case of beam element, the map of the actual fibers of the cross section.

KRSPTF(I) Dimensions of array RSPTF containing the current values of the rotation matrix from global to principal reference system for each integration point of the I-th type of element.

KRSPTI(I) Dimensions of array RSPTI containing the values, at the beginning of the increment, of the rotation matrix from global to principal reference system for each integration point of the I-th type of element.

KSDC(I) Dimensions of array SDC containing the incremental value of the deformation tensor center of the elastic range for each integration point of the I-th type of element.

KSDEPL(I) Dimensions of array SDEP containing the incremental values of the plastic deformation for each integration point of the I-th type of element.

KSDH(I) Dimensions of array SDH containing the incremental values of tensor which defines the exponential part of the kinematic hardening for each integration point of the I-th type of element.

KSDUPL(I) Dimensions of array SDUP containing the incremental values of the crushing deformation for each integration point of the I-th type of element.

KSDZ(I) Dimensions of array SDZ containing the incremental values of the accumulated equivalent plastic strain (Odquist parameter) for each integration point of the I-th type of element.

KSECT Dimensions of the array SECT containing the cross section areas of beam elements.

KSHAP5 Dimensions of the array SHAP5 containing the shape function values for each integration point of the thin shell element.

KSTRAP(I) Dimensions of array TSTRAP containing the total values of the plastic strain components for each integration point of the I-th type of element.

KSTRUP(I) Dimensions of array TSTRUP containing the total values of the crushing strain components for each integration point of the I-th type of element.

KTHER(I) Dimensions of array THERM containing the values of temperature increments for each integration point of the I-th type of element.

KTHICK Dimensions of the array THICK containing the thickness of the layers/fibers of shell/beam elements.

KTSTRA(I) Dimensions of array TSTRA containing the values of the total deformation components for each integration point of the I-th type of element.

KTSTRE(I) Dimensions of array TSTRE containing the total values of the Cauchy stress components for each integration point of the I-th type of element.

KTTHEI(I) Dimensions of the array TTHERI containing the initial temperature values for each integration point of the I-th type of element.

KTTHER(I) Dimensions of array TTHERM containing the values of the total temperature for each integration point of the I-th type of element.

KZETA(I) Dimensions of array ZETA containing the total values of the accumulated equivalent plastic strain (Odquist parameter) for each integration point of the I-th type of element.

KZETSH Dimensions of the array ZETSH containing the position of the integration point along the thickness of shell/beam element types.

VARIABLES OF THE COMMON/MOTION/

This COMMON contains the parameters for time stepping in contact cases, dynamic analyses or heat transfer problems.

```
 common/motion/tottim, deltim, tottii, deltii, period, diag ,
*              auttim, irdinc,iudloa ,iuploa ,iutloa ,ityich,
*              iusurf,iuflux ,iubnd  ,idynch , iacci ,itrans,
*              nauto ,incini
 logical auttim, irdinc, iudloa, iuploa, iutloa, ityich,
*        iusurf, iuflux,iubnd  ,idynch ,iacci  ,itrans
```

Variable	Description
AUTTIM	If .TRUE., it indicates that the automatic time stepping procedure will be used in heat transfer analysis.
DELTII	Initial time step of a series of increments.
DELTIM	Time increment corresponding to the current load increment.
DIAG	Estimate of the total mesh dimensions.
IACCI	If .TRUE., it indicate that the initial acceleration will be calculated in a dynamic analysis.
INCINI	Not used.
IDYNCH	Not used.
IRDINC	Not used.
ITRANS	Not used.
ITYICH	If set to .TRUE. in the user routine UTIE, it indicates that the structure of the tying relations is changed and a recalculation of the front-width will be necessary.
IUBND	Not used.
IUDLOA	Not used.
IUFLUX	Not used.
IUPLOA	If .TRUE., it indicates that the user routine UPLOAD will be called to calculate the concentrated loads or fluxes.

IUSURF	If .TRUE., it indicates that the user routine UMOTIO will be called to redefine the velocity of rigid surfaces in a contact analysis.
IUTLOA	Not used.
NAUTO	It indicates how many increments remain to complete a series.
PERIOD	Duration of a series of increments.
TOTTII	Initial total time at the beginning of a series of increments.
TOTTIM	Total time elapsed from the beginning of the analysis

VARIABLES OF THE COMMON/POST/

This COMMON contains the parameters managing the post-processing data file.

```
 common/post/ wrtim, ibini, icont, npost, mvar, indvar(200),
*             incsb, lstinc, chvar(24,200)
 character*1 chvar
 logical ibini,icont
```

Variable	Description
CHVAR(I)	Name (maximum 24 characters) of the I-th elemental variable to be written to the post-processing file.
IBINI	If .TRUE., the post-processing file will be written in binary, without any format control, otherwise the file will be written in alphanumeric characters in card format.
ICONT	If .TRUE., in the case of restart, the initial part of the post-processing file containing the connectivity and nodal coordinates is written.
INCSB	Write frequency of iteration data on the post-processing file (Default 0, i.e. only the end-of-increment data will be written).
INDVAR(I)	Identifier (as specified in the card POST) of the I-th elemental variable to be written to the post-processing file.
LSTINC	Number of the lastly written increment.
MVAR	Number of the elemental variables which will be written to the post-processing file for each integration point.
NPOST	Write frequency for the post-processing file. This file will be written for the first increment and every NPOST increment starting from the increment LSTINC.
WRTIM	Write frequency in time, i.e. data will be written if the time increment is a integer multiple of WRTIM (Default 0.0 and the write frequency is controlled by NPOST).

VARIABLES OF THE COMMON/SHFUN/

This COMMON contains the shape function and their derivatives for each integration of the element types present in the mesh.

```
 common /shfun/gp1d2(2), gp2d4(2,4), gp2d8(2,9), gp3d8(3,8) ,
*               gp3d20(3,27), gpf2d4(2,2), gpf2d8(2,3),
*               gpf3d8(3,4), gf3d20(3,9), sh1d2(2,2),
*               sh2d4(4,5), sh2d8(8,9), sh3d8(8,9),
*               sh3d20(20,27),shsh5(9,4), shf2d4(4,2),
*               shf2d8(8,3),shf3d8(8,4), sf3d20(20,9),
*               shfsh5(4,2), dr1d2(2,2), dr2d4(2,4,5),
*               dr2d8(2,8,9),dr3d8(3,8,9), dr3d20(3,20,27),
*               drsh5(2,9,4), drf2d4(2,4,2), drf2d8(2,8,3),
*               drf3d8(3,8,4), df3d20(3,20,9), drfsh5(2,4,2),
*               wg1d2(2), wg2d4(4), wg2d8(9), wg3d8(8),
*               wg3d20(27),wg1d3(3)
```

Variable **Description**

DF3D20(I, J, K) Is the value of the derivative along the I-th local direction of the J-th shape function calculated at the K-th integration point of the face of the 20-nodes 3D element.

DRFSH5(I, J, K) Is the value of the derivative along the I-th local direction of the J-th shape function calculated at the K-th integration point of the edge of the thin shell element.

DRF2D4(I, J, K) Is the value of the derivative along the I-th local direction of the J-th shape function calculated at the K-th integration point of the edge of the 4-nodes 2D element.

DRF2D8(I, J, K) Is the value of the derivative along the I-th local direction of the J-th shape function calculated at the K-th integration point of the edge of the 8-nodes 2D element.

DRF3D8(I, J, K) Is the value of the derivative along the I-th local direction of the J-th shape function calculated at the K-th integration point of the face of the 8-nodes 3D element.

DRSH5(I, J, K) Is the value of the derivative along the I-th local direction of the J-th shape function calculated at the K-th integration point of the thin shell element.

DR1D2(I, J) Is the value of the derivative of the I-th shape function calculated at the J-th integration point of the beam element.

DR2D4(I, J, K) Is the value of the derivative along the I-th local direction of the J-th shape function calculated at the K-th integration point of the 4-nodes 2D element (the case K=5 indicates the element centroid).

DR2D8(I, J, K) Is the value of the derivative along the I-th local direction of the J-th shape function calculated at the K-th integration point of the 8-nodes 2D element.

DR3D20(I, J, K) Is the value of the derivative along the I-th local direction of the J-th shape function calculated at the K-th integration point of the 20-nodes 3D element.

DR3D8(I, J, K) Is the value of the derivative along the I-th local direction of the J-th shape function calculated at the K-th integration point of the 8-nodes 3D element (the case K=9 indicates the element centroid).

GPF2D4(I, J) Is the I-th coordinate of the J-th integration point on the edge of the 4-node 2D element.

GPF2D8(I, J) Is the I-th coordinate of the J-th integration point on the edge of the 8-node 2D element.

GPF3D20(I, J) Is the I-th coordinate of the J-th integration point on the face of the 20-node 3D element.

GPF3D8(I, J) Is the I-th coordinate of the J-th integration point on the face of the 8-node 3D element.

GP1D2(I) Is the coordinate of the I-th integration point for the beam element.

GP2D4(I, J) Is the I-th coordinate of the J-th integration point for the 4-nodes 2D elements.

GP2D8(I, J) Is the I-th coordinate of the J-th integration point for the 8-nodes 2D elements.

GP3D20(I, J) Is the I-th coordinate of the J-th integration point for the 20-nodes 3D elements.

GP3D8(I, J) Is the I-th coordinate of the J-th integration point for the 8-nodes 3D elements.

SF3D20(I, J) Is the value of the I-th shape function calculated at the J-th integration point of the face of the 20-nodes 3D element.

SHFSH5(I, J) Is the value of the I-th shape function calculated at the J-th integration point of the edge of the thin shell element.

SHF2D4(I, J) Is the value of the I-th shape function calculated at the J-th integration point of the edge of the 4-nodes 2D element.

SHF2D8(I, J) Is the value of the I-th shape function calculated at the J-th integration point of the edge of the 8-nodes 2D element.

SHF3D8(I, J) Is the value of the I-th shape function calculated at the J-th integration point of the face of the 8-nodes 3D element.

SHSH5(I, J) Is the value of the I-th shape function calculated at the J-th integration point of the thin shell element

SH1D2(I, J) Is the value of the I-th shape function calculated at the J-th integration point of the beam element.

SH2D4(I, J) Is the value of the I-th shape function calculated at the J-th integration point of the 4-nodes 2D element (the case J=5 indicates the element centroid).

SH2D8(I, J) Is the value of the I-th shape function calculated at the J-th integration point of the 8-nodes 2D element.

SH3D20(I, J) Is the value of the I-th shape function calculated at the J-th integration point of the 20-nodes 3D element.

SH3D8(I, J) Is the value of the I-th shape function calculated at the J-th integration point of the 8-nodes 3D element (the case J=9 indicates the element centroid).

WG1D2(I) Is the Gauss integration factor calculated at the I-th integration point of the beam element.

WG1D3(I) Is the Gauss integration factor calculated at the I-th integration point of the edge of the 8-nodes 2D element.

WG2D4(I) Is the Gauss integration factor calculated at the I-th integration point of the 4-nodes 2D element.

WG2D8(I) Is the Gauss integration factor calculated at the I-th integration point of the 8-nodes 2D element.

WG3D20(I) Is the Gauss integration factor calculated at the I-th integration point of the 20-nodes 3D element.

WG3D8(I) Is the Gauss integration factor calculated at the I-th integration point of the 8-nodes 3D element.

VARIABLES OF THE COMMON/SIZE/

This COMMON contains pointers to the start of arrays contained in the work areas.

```
      common /size/ lposgp, lweigp, lcoord, lpresc, lgeom , lprops,
     *              lhard , lhaiso, lhakin, lraph , lrsp0 , lrtie ,
     *              lcsete, lcsetn, lrttem, lbound, lsurf , lversn,
     *              lcpres, lverst, lcfric, lffric, lchan , ltempi,
     *              lfilm , lsink , lchans, lprsca, ldamp , ldispi,
     *              ldisp0, lveloi, laccei, lpmass, ltdisp, lvelo ,
     *              lacce , lasdis, lseldi, lfixed, lfixea, ltreac,
     *              lpoint, lrdist, leload, leloat, leloaf, lrload,
     *              laxis , lcaric, ltload, lrefor, lestif, ltempe,
     *              ldtemp, ldttmp, lsurf0, lcsvlo, learea, leqrhs,
     *              lequat, lvecrv, lgload, lgstif, lisele, lnodth,
     *              lrplan, lilump, liuaxi, llnods, lltype, ljarea,
     *              lmatno, lifpre, lndfro, lickfr, litie , lirtie,
     *              lnsete, llese , lnsetn, llesn , liwset, lnhard,
     *              liraph, lmaso , littem, ldefo , lisur , liboun,
     *              licpre, lkfric, lielpr, linodp, lnchan, lifilm,
     *              liprea, liffix, liffia, lidist, liglue, liplan,
     *              lisur0, ljpcod, lnamev, lndfeq, lnpivo, lndest,
     *              lnacva, lnacid, mrec  , msizi , msizr
```

Variable	Description
LACCE	Pointer to the beginning of array ACCE.
LACCEI	Pointer to the beginning of array ACCEI.
LASDIS	Pointer to the beginning of array ASDIS.
LAXIS	Pointer to the beginning of array AXIS.
LBOUND	Pointer to the beginning of array BOUND.
LCARIC	Pointer to the beginning of array CARIC.
LCFRIC	Pointer to the beginning of array CFRICT.
LCHAN	Pointer to the beginning of array CHAN.
LCHANS	Pointer to the beginning of array CHANS.
LCOORD	Pointer to the beginning of array COORD.

LCPRES	Pointer to the beginning of array CPRES.
LCSETE	Pointer to the beginning of array CSETE.
LCSETN	Pointer to the beginning of array CSETN.
LCSVLO	Pointer to the beginning of array CSVLOA.
LDAMP	Pointer to the beginning of array DAMP.
LDEFO	Pointer to the beginning of array IDEFO.
LDISP0	Pointer to the beginning of array DISP0.
LDISPI	Pointer to the beginning of array DISPI.
LDTEMP	Pointer to the beginning of array DTEMP.
LDTTMP	Pointer to the beginning of array DTTMP.
LEAREA	Pointer to the beginning of array EAREA.
LELOAD	Pointer to the beginning of array ELOAD.
LELOAF	Pointer to the beginning of array ELOAF.
LELOAT	Pointer to the beginning of array ELOAT.
LEQUAT	Pointer to the beginning of array EQUAT.
LEQRHS	Pointer to the beginning of array EQRHS.
LESTIF	Pointer to the beginning of array ESTIF.
LFFRIC	Pointer to the beginning of array FFRICT.
LFILM	Pointer to the beginning of array FILM.
LFIXEA	Pointer to the beginning of array FIXEA.
LFIXED	Pointer to the beginning of array FIXED.
LGEOM	Pointer to the beginning of array GEOM.
LGLOAD	Pointer to the beginning of array GLOAD.
LHAISO	Pointer to the beginning of array HAISO.

LHAKIN	Pointer to the beginning of array HAKIN.
LHARD	Pointer to the beginning of array HARD.
LGSTIF	Pointer to the beginning of array GSTIF.
LIBOUN	Pointer to the beginning of array IBOUND.
LICKFR	Pointer to the beginning of array ICKFR.
LICPRE	Pointer to the beginning of array ICPRES.
LIDIST	Pointer to the beginning of array IDIST.
LIELPR	Pointer to the beginning of array IELPR.
LIFFIA	Pointer to the beginning of array IFFIA.
LIFFIX	Pointer to the beginning of array IFFIX.
LIFILM	Pointer to the beginning of array IFILM.
LIFPRE	Pointer to the beginning of array IFPRE.
LIGLUE	Pointer to the beginning of array IGLUED.
LILUMP	Pointer to the beginning of array ILUMP.
LINODP	Pointer to the beginning of array INODPR.
LIPLAN	Pointer to the beginning of array IPLANE.
LIPREA	Pointer to the beginning of array IPREA.
LIRAPH	Pointer to the beginning of array IRAPH.
LIRTIE	Pointer to the beginning of array IRTIE.
LISELE	Pointer to the beginning of array ISELE.
LISUR	Pointer to the beginning of array ISURF.
LISUR0	Pointer to the beginning of array ISURF0.
LITIE	Pointer to the beginning of array ITIE.
LITTEM	Pointer to the beginning of array ITTEMP.

LIUAXI	Pointer to the beginning of array IUAXI.
LIWSET	Pointer to the beginning of array IWSET.
LJAREA	Pointer to the beginning of array JEAREA.
LJPCOD	Pointer to the beginning of array JPCOD.
LKFRIC	Pointer to the beginning of array KFRICT.
LLESE	Pointer to the beginning of array LESE.
LLESN	Pointer to the beginning of array LESN.
LLNODS	Pointer to the beginning of array LNODS.
LLTYPE	Pointer to the beginning of array LTYPE.
LMASO	Pointer to the beginning of array MASO.
LMATNO	Pointer to the beginning of array MATNO.
LNACID	Pointer to the beginning of array NACID.
LNACVA	Pointer to the beginning of array NACVA.
LNAMEV	Pointer to the beginning of array NAMEV.
LNCHAN	Pointer to the beginning of array NCHAN.
LNDEST	Pointer to the beginning of array NDEST.
LNDFEQ	Pointer to the beginning of array NDFEQ.
LNDFRO	Pointer to the beginning of array NDFRO.
LNHARD	Pointer to the beginning of array NHARD.
LNODTH	Pointer to the beginning of array NODTH.
LNPIVO	Pointer to the beginning of array NPIVO.
LNSETE	Pointer to the beginning of array NSETE.
LNSETN	Pointer to the beginning of array NSETN.
LPMASS	Pointer to the beginning of array PMASS.

LPOINT	Pointer to the beginning of array POINT.
LPOSGP	Pointer to the beginning of array POSGP.
LPRESC	Pointer to the beginning of array PRESC.
LPROPS	Pointer to the beginning of array PROPS.
LPRSCA	Pointer to the beginning of array PRSCA.
LRAPH	Pointer to the beginning of array RAPH.
LRDIST	Pointer to the beginning of array RDIST.
LREFOR	Pointer to the beginning of array REFOR.
LRLOAD	Pointer to the beginning of array RLOAD.
LRPLAN	Pointer to the beginning of array RPLANE.
LRSP0	Pointer to the beginning of array RSP0.
LRTIE	Pointer to the beginning of array RTIE.
LRTTEM	Pointer to the beginning of array RTTEMP.
LSELDI	Pointer to the beginning of array SELDIS.
LSINK	Pointer to the beginning of array SINK.
LSURF	Pointer to the beginning of array SURF.
LSURF0	Pointer to the beginning of array SURF0.
LTDISP	Pointer to the beginning of array TDISP.
LTEMPE	Pointer to the beginning of array TEMPE.
LTEMPI	Pointer to the beginning of array TEMPI.
LTLOAD	Pointer to the beginning of array TLOAD.
LTREAC	Pointer to the beginning of array TREAC.
LVECRV	Pointer to the beginning of array VECRV.
LVELO	Pointer to the beginning of array VELO.

LVELOI	Pointer to the beginning of array VELOI.
LVERSN	Pointer to the beginning of array VERSN.
LVERST	Pointer to the beginning of array VERST.
LWEIGP	Pointer to the beginning of array WEIGP.
MREC	Number of records necessary for the file JFIL3.
MSIZI	Dimensions of the work area IVE containing the integer variables.
MSIZR	Dimensions of the work area VEC containing the real variables.

VARIABLES OF THE COMMON/TIMCPU/

This COMMON contains the CPU time spend by the analysis (the routines for the CPU time calculation are available only for LINUX system).

```
common /timcpu/ icput0, icput
```

Variable	Description
ICPUT	Current number of clock tics.
ICPUT0	Number of clock tics at the beginning of the analysis

VARIABLES OF THE COMMON/TITL/

This COMMON contains the title of the analysis.

```
common /titl/ title(70)
character*1 title
```

Variable	Description
TITLE	Alphanumeric string (maximum 70 characters) which identifies the analysis.

VARIABLES OF THE COMMON/TMPPST/

This COMMON contains the parameters which control the reading of a thermal post-processing file needed for calculating thermal loads..

```
 common/tmppst/ irdtmp, inctmp, lastin, ntvar, ntnod, ntele,
*               ntdof , ntgp  , ntnodv
```

Variable	Description
INCTMP	Number of the increment to be read from post-processing file.
IRDTMP	switch indicating the reading of nodal temperatures = 0, no reading (Default), = 1, reading from input cards,. = 2, reading from binary post-processing file, = 3, reading from formatted post-processing file, = 4, nodal temperatures calculated in the user routine URDTEM.
LASTIN	Number of the last increment read from the post-processing file.
NTDOF	Maximum number of degrees of freedom for each node in the thermal post-processing file.
NTELE	Total number of elements present in the thermal post-processing file.
NTGP	Maximum number of integration points for each element in the thermal post-processing file.
NTNOD	Total number of nodes present in the post-processing file.
NTNODN	Number of nodal variables present in the thermal post-processing file.
NTVAR	Number of elemental variables present in the thermal post-processing file.

G. DESCRIPTION OF THE ARRAYS OF THE WORK AREAS

DESCRIPTION OF THE INTEGER VARIABLES OF ARRAY IVE

The arrays contained in the vector IVE are described following the alphabetical order regardless of their position in IVE.

KFRICT(NPOIN)
KFRICT(I) is the code which indicates the condition of node I with respect to frictional forces; more precisely:
KFRICT(I) = 1 node I is not subjected to frictional forces;
KFRICT(I) = 2 the motion of node I is stopped by the frictional force;
KFRICT(I) = 3 a slip condition occurs at node I.
The array is allocated only if IFRIC=.TRUE..

IBOUND(2, NPOIN)
If IBOUND(1, I) is different from zero, it indicates that node I belongs to the boundary of the deformable body IBOUND(1, I),
IBOUND(2, I) has meaning only if IBOUND(1, I) is different from zero; more precisely:
= 0 node I is free,
>0 node I is in contact with the K-th part of the J-th rigid surface, where J=IBOUND(2, I)/1000 and K = MOD(IBOUND(2, I),1000),
< 0 node I belongs to the interior of the J-th rigid surface, facing the K-th part, where J and K are determined as the previous point by using the absolute value of IBOUND(2, I).
The array is allocated only if MSURF≠0.

ICKFR(2, MDOFN, NPOIN)
ICKFR(1, I, J) is the element number in which the I-th degree of freedom of the J-th node appears for the first time. The I-th degree of freedom of J-th node can be eliminated from the solution front when the element number ICKFR(2, I, J) is processed.

ICPRES(4, NELEM)
If ICPRES(J, I) is non-zero, then face J of element I is subjected to contact forces. The array is allocated only if MSURF≠0.
The array is allocated only if MSURF≠0.

IDEFO(NELEM, MDEFO)
IDEFO(J, I) is the number of the J-th element of the I-th deformable body. The array is allocated only if MSURF≠0.

IDIST(2, MDIST, NELEM)
IDIST(1, J, I) is the type identifier of the J-th distributed load applied to element I.
IDIST(2, J, I) is the ordinal number, as defined in cards DIST LOADS, of the J-th distributed load applied to element I.

IELPR(NELEM)
If IELPR(I) is non-zero, then the data of element I will appear in the printout of the output file.
The array is allocated only if NELPR≠0.

IFFIA(MDOFN, NPOIN)
IFFIA(I, J)≠0 means that value of the acceleration of the degree of freedom I of node J is prescribed. The array is allocated only if IDYNA=.TRUE..

IFFIX(MDOFN, NPOIN)
IFFIX(I, J) is the constraint code for the global degree of freedom I of node J; more precisely:

IFFIX(I, J) = 0, degree of freedom I is not constrained;
IFFIX(I, J) = 1, degree of freedom I is constrained;
IFFIX(I, J) = 2, in correspondence to a first degree of freedom, degree of freedom I has a contact constraint;
IFFIX(I, J) = 2, in correspondence to a second degree of freedom, degree of freedom I is stopped by the friction;
IFFIX(I, J) = 3, in correspondence to a second degree of freedom, degree of freedom I is slipping;
IFFIX(I, J) = 4, degree of freedom I is tied;
IFFIX(I, J) = 5, degree of freedom I is retained and free;
IFFIX(I, J) = 6, degree of freedom I is retained and constrained.

IFILM(2, MFAC, NELEM)
IFILM(I, J, K)<0 means that the value of the film coefficient (I=1) or the sink temperature (I=2) will be calculated by the user routine UFILM for the face J of element K. The array is allocated only if IIFILM=.TRUE..

IFPRE(MDOFN, NPOIN)
If IFPRE(I, J) is non-zero, then the degree of freedom I of node J is constrained; if IFPRE(I, J) is less than zero, then the constraint is managed by the user routine UBOUND.

IGLUE(NELEM)
IGLUE(I) indicates the collapsed edge of the shell element I. The array is allocated only if shell elements are present in the mesh.

ILUMP(NELEM)
The setting of ILUMP(I) to a non zero value means that a lumped mass matrix will be used for the element number I. The array will be allocated only if ILUMP=.TRUE..

INODPR(NPOIN)
If INODPR(I) is non-zero, then the data of node I will appear in the printout of the output file. The array is allocated only if NODPR≠0.

IPLANE(NELEM)
IPLANE(I)≠0 means that the shell element I is planar. The array is allocated only if shell elements are present in the mesh.

IPREA(MDOFN, NPOIN)
IPREA(I, J)≠0 means that the value of the acceleration of the I-th degree of freedom of the J-th node is prescribed. If IPREA(I, J)<0, the acceleration calculation is done by the user routine UBOUND. The array is allocated only if IDYNA=.TRUE..

IRAPH(NMATS)
If IRAPH(I) is different from zero, then the I-th material defined has an anisotropic yield stress. The array is allocated only if NRAPH≠0.

IRTIE(MDOFN, MRET, MDOFN, MTIE)
If different from zero, IRTIE(L, K, J, I), indicates a non-zero value in the I-th tying relation for the coefficient which links the J-th degree of freedom of the tied node to the L-th degree of freedom of the K-th retained node.

ISELE(NELEM)
ISELE(I)≠0 means that only one integration point will be used to integrate the volumetric part of the strain-displacement relation in the setting up of the stiffness matrix of the element I. The parameter has meaning only for 2D and 3D elements with linear shape functions. For shell elements ISELE(I)≠0 means that the shear in plane component of the deformation will be considered as a constant in the setting up of the stiffness matrix.

ISURF(MPART, MSURF)
ISURF(J, I) is the type (segment or circumference arc) of the J-th part of the I-th rigid surface.

ITIE(2+MRET, MTIE)
ITIE(1, I) identifier of the I-th tying relation.
ITIE(2, I) number of the tied node in the I-th tying relation.
ITIE(J+2, I) number of the J-th retained node in the I-th tying relation.

ISURF0(MPART0, MSURF)
ISURF0(J, I) is the type (9 for segments, 64 for circumference arcs) of the J-th post-processing part of the rigid surface I. One or more post-processing parts correspond to each part of a rigid surface, as defined in the cards SURFACE; in other words, only one post-processing part corresponds to a segment, whereas for a circumference arc the corresponding number of post-processing parts (minimum 10) is equal to the value obtained by dividing the angle subtended at center by p/20 and adding 1.

ITTEMP(3, MTTEMP)
ITTEMP(1, J) is the identifier of the material to which the J-th table of temperature variable properties refers;
ITTEMP(2, J) indicates the property in the J-th table:
=1, Young modulus,
=2, Poisson ratio,
=3, coefficient of linear thermal dilatation;
=4, heat conductivity;
=5, specific heat per unit mass.
ITTEMP(3, J) indicates the number of data pairs (temperature-property) in the J-th table.

IUAXI(NELEM)
IUAXI(I) set to a non zero value means that the element number I is a beam element requiring the use of the user routine ULAXIS for defining the local reference system. The array is allocated only if beam element are present.

IWSET(MSETEN)
IWSET(I) is the number of the node or element which is the I-th component of the set resulting from decoding a list given as a group of sets linked by logic operators.

JEAREA(NELEM)
JEAREA(I) is the length, in double words, of the work area of the element number I.

JPCOD(NPOIN)
JPCOD(I) is the post-processing nodal code for node I.

LESE(MSETE)
LESE(I) is the number of items in the I-th set of elements.

LESN(MSETN)
LESE(I) is the number of items in the I-th set of nodes.

LNODS(MNODE, NELEM)
The value of LNODS(J, I) is the identifier number of the J-th node of the I-th element.

LTYPE(NELEM)
LTYPE(I) is the ordinal number of element I in the list of element types defined in cards ELEMENTS or SIZING. If the mesh is composed of a single element type, LTYPE(I) is 1.

MASO(MSHEL, NELEM)
MASO(J, I) indicates that layer J of element I may be made up of a masonry-like material; more precisely:

MASO(J, I) = 0, elastic-plastic material;
MASO(J, I) = 1, masonry-like material not resistant to tension and infinitely resistant to compression;
MASO(J, I) = 2, masonry-like material with bounded tensile strength and bounded compressive strength.

In this version of NOSA the shell-element layers must have the same value of MASO. The array is allocated only if IMASO≠0.

MATNO(MSHEL, NELEM)
MATNO (I, J) is the ordinal number of materials definition constituting layer I of element J. Obviously, the indication of the layer holds only for shell elements, whereas for the other element types I is always 1.

NACID(MFRON)
Position I in the resolution front is occupied by the degree of freedom NACID(I) of node NACVA(I). The array is allocated if IHEAT=.FALSE. or heat transfer shell elements are used.

NACVA(MFRON)
Position I in the resolution front is occupied by a degree of freedom of node NACVA(I).

NAMEV(MBUF2)
The global degree of freedom I (during resolution) occupies position NAMEV(I) in the resolution front at the moment of its elimination.

NCHAN(NELEM)
Reserved for future developments. The array is allocated only if MCHAN≠0.

NDEST(MEVAB)
NDEST(I) is the position of the current element in the resolution front of the I-th degree of freedom of the currently processed element.

NDFEQ(MBUF2)
The global degree of freedom I (during resolution) is the degree of freedom NDFEQ(I) of the node NPIVO(I). The array is allocated if IHEAT=.FALSE. or heat transfer shell elements are used.

NDFRO(NELEM)
NDFRO(I) is the change in length of the resolution front when element I is processed.

NHARD(NMATS)
NHARD(I) is the number of data pairs which define the hardening curve of the I-th material. The array is allocated only if MHARD≠0.

NODTH(NELEM)
NODTH(I) ≠0 means that that the thickness of the shell element I is calculated by the user routine UGEOM.

NPIVO(MBUF2)
The global degree of freedom I (during resolution) is a degree of freedom of node NPIVO(I).

NSETE(MSET, MSETE)
NSETE(J, I) is the number of the J-th element of the I-th set of elements.

NSETN(MSET, MSETN)
NSETN(J, I) is the number of the J-th node of the I-th set of nodes.

DESCRIPTION OF THE REAL VARIABLES OF ARRAY VEC

The arrays contained in VEC are described following the alphabetical order regardless of their position in VEC.

ACCE(MDOFN, NPOIN)
ACCE(I, J)) is the current value of the total generalized acceleration relative to the I-th degree of freedom of the J-th node. The array is allocated only if IDYNA=.TRUE.

ACCEI(MDOFN, NPOIN)
ACCEI(I, J) is the value of the acceleration at the beginning of an increment for the I-th degree of freedom of J-th node. The array is allocated only if IDYNA=.TRUE..

ASDIS(MDOFN, NPOIN)
ASDIS(I, J) is the value of the variation, calculated at the current iteration, of the incremental generalized displacement or temperature relative to the I-th degree of freedom of the J-th node.

AXIS(MCORD, NELEM)
AXIS(I,J), with I=1,2,3, are the components of the unit vector which identifies the first coordinate direction of the local reference system for the beam element number J. The array is allocated only if beam elements are present.

BOUND(MPART, MSURF, NPOIN)
BOUND(K, J, I) is the distance between the node and the K-th part of the J-th rigid surface. This distance is set equal to $1*10^{20}$ if the node does not 'face' that part of the surface; that is, it does not belong to the plane region which would be swept by the part if it were to move perpendicular to itself.

CARIC(MEVAB, NELEM)
CARIC(I, J) is the value of the right hand side in the calculation of the initial acceleration for the degree of freedom I of element J. The array is allocated only if IDYNA=.TRUE.

CFRICT(MPART, MSURF, MDEFO)
CFRICT(K, J, I) is the friction coefficient at the interface between the I-th deformable body and the K-th part of the J-th rigid surface. If the coefficient is less than zero, the value of the limit frictional force at the corresponding interface is calculated by the user routine UFRI. . The array is allocated only if IFRIC=.TRUE..

CHAN(5, NELEM)
Array reserved for future developments. It is allocated only if MCHAN≠0.

CHANS(2, MCHAN1)
Reserved for future developments. The array is allocated only if MCHAN≠0.

COORD(MCORD, NPOIN)
COORD(J, I) is the initial value of the J-th coordinate of node I.

CPRES(MDOFN, MNODE, NELEM)
CPRES(1, J, I) is the contact pressure at the J-th node of element I, if this node is in contact. CPRES(2...MDOFN, J, I) reserved for future developments. The array is allocated only if MSURF≠0.

CSETE(MSETE)
CSETE(I) is the name (maximum eight characters) which identifies the I-th set of elements.

CSETN(MSETN)
CSETN(I) is the name (maximum eight characters) which identifies the I-th set of nodes.

CSVLOA(MEVAB)
Load term due to the anisotropic yield. The array is allocated only if NRAPH≠0.

DAMP(3, NELEM)
DAMP(1, I) is the value of the viscous damping multiplier of the mass matrix of the I-th element; DAMP(2, I) is the valur of the viscous damping multiplier of the stiffness matrix of the I-th element; DAMP(3, I) is the value of the numerical damping multiplier of the stiffness matrix of the I-th element. The array is allocated only if IDAMP=.TRUE..

DISPI(MDOFN, NPOIN)
DISPI(I, J) is the value of the total displacement at the beginning of an increment for the I-th degree of freedom of J-th node. The array is allocated only if IDYNA=.TRUE..

DISP0(MDOFN, NPOIN)
DISP0(I, J) is the value of the total displacement at the beginning of the analysis for the I-th degree of freedom of J-th node. The array is allocated only if IDYNA=.TRUE..

DTEMP(ITYRD, NPOIN)
DTEMP(I, J) is the value of the increment of temperature at layers of node J, as set in the cards THERMAL LOADS. The array is allocated only if IRDTMP≠0.

DTTMP(ITYRD, NPOIN)
DTTMP(I, J) is the total change in temperature at layers of node J, as read in the thermal post-processing file. The array is allocated only if IRDTMP≠0.

EAREA(KEAREA)
Work area containing the data (stresses, strains *etc.*) of the elements. A description of EAREA is given in the next section.

ELOAD(MEVAB, NELEM)
ELOAD(J, I) is the value of the generalized nodal force or flux equivalent to the distributed and point loads or fluxes for the J-th degree of freedom of element I. Such a load is the result of settings in the cards DISTRIBUTED LOADS, POINT LOADS and THERMAL LOADS or DISTRIBUTED FLUXES and POINT FLUXES. The load has not yet been scaled by the proportionality factor, FACTO declared in cards PROPORTIONAL INCREMENT.

ELOAF(MEVAB, NELEM)
ELOAF(J, I) is the value of the correction, for the J-th degree of freedom of element I, calculated in a such way that, if option FOLLOWER FORCES is required, the distributed loads defined in the local reference system maintain the correct orientation in the current deformed configuration. The array is allocated only if IFOLL=.TRUE..

ELOAT(MEVAB, NELEM)
If IRDTMP≠0, ELOAT(J, I) is the value, for the J-th degree of freedom of element I, of the nodal force equivalent to the load due to the temperature changes in the corresponding node.
If IHEAT=.TRUE., ELOAT(1,J) is the value of the convective term of the heat flux.

EQRHS(MBUF2)
EQRHS(I) is the value of the reduced right hand side relative to the I-th global degree of freedom.

EQUAT(MFRON, MBUF2)
EQUAT(J, I) is the value of the J-th coefficient (J-th position in the resolution front) of the reduced equation relative to the I-th global degree of freedom.

ESTIF(MEVAB, MEVAB)
Stiffness matrix of the current element.

FFRICT(2, NPOIN)
FFRICT(J, I) is the value of the J-th component of the force opposing motion under slippage conditions at node I. . The array is allocated only if IFRIC=.TRUE..

FILM(MSIDE, MFAC, NELEM)
FILM(I, J, K) is the value of the film coefficient of the I-th node of the J-th face or edge of the K-th element. The array is allocated only if IIFILM=.TRUE..

FIXEA(MDOFN, NPOIN)
FIXEA(I, J) is the value of the incremental generalized acceleration of the I-th degree of freedom of the J-th node if constrained as specified in IFFIA(I). the array is allocated only if IDYNA=.TRUE..

FIXED(MDOFN, NPOIN)
FIXED(I, J) is the value of the incremental generalized displacement or temperature of the I-th degree of freedom of the J-th node if constrained as specified in IFFIX(I). After solution of the evolution system, at the end of the routine FRONT, FIXED(I, J) is the value of the generalized reaction force or flux variation calculated for the same degree of freedom.

GEOM(MDT, MGEOM, NELEM)
For all types of planar elements, GEOM(1, 1, K) is the thickness value of element K. In the case of shells or beams, GEOM(J, I, K) is the thickness value of the J-th layer or fiber at the I-th corner node of element K.

GLOAD(MFRON)
GLOAD(I) is the value of the relative known term relative to the I-th equation currently active in the resolution front.

GSTIF(MSTIF)
It is the matrix (vectorized and, if symmetric, triangularized) containing the coefficients of the currently active equations in the resolution front.

HAISO(NMATS)
HAISO(I) is the fraction of hardening which must be considered isotropic for the I-th material. The array is allocated only if MHARD≠0.

HAKIN(NMATS)
HAKIN(I) is the fraction of hardening which must be considered kinematic for the I-th material. Obviously HAKIN(I)= 1. - HAISO(I). The array is allocated only if MHARD≠0.

HARD(2, MHARD, NMATS)
HARD(1, J, I) is the J-th value of the accumulated equivalent plastic strain (Odqvist parameter) in the hardening curve relative to the I-th material. Note that HARD(1, 1, I) must be 0.
HARD(2, J, I) is the value of the von Mises equivalent tensile stress corresponding to HARD(1, J, I).

PMASS(MDOFN, NPOIN)
PMASS(I, J) is the value of the concentrated mass for the I-th degree of freedom of J-th node. The array is allocated only if IDYNA=.TRUE..

POINT(MDOFN, NPOIN)
POINT(I, J) is the value of the generalized concentrated load or flux applied at the I-th degree of freedom of node J.

POSGP(MGAUS, MTYPE)
POSGP(J, I) is the local coordinate of the J-th Gauss integration point (on segment [-1, 1]) of the I-th element type.

PRESC(MDOFN, NPOIN)
PRESC(J, I) is the prescribed value of the incremental displacement or temperature of the J-th constrained degree of freedom of the node. This value has meaning only if IFPRE(J, I) is non-zero.

PROPS(MPROP, NMATS)
PROPS(J, I) is the value of the J-th mechanical or thermal property, as shown in cards PROPERTY, of the I-th material.

PRSCA(MDOFN, NPOIN)
PRSCA(I, J) is the value of the prescribed acceleration of the I-th degree of freedom of the J-th node. The array is allocated only if IDYNA=.TRUE..

RAPH(2,NRAPH, NMATS)
RAPH(I, J) is the yield value of the I-th component of the stress of the material number J. The array is allocated only if NRAPH≠0.

RDIST(2*MDOFN, MSIDE, MDIST, NELEM)
RDIST(L, K, J, I) and RDIST(L + MDOFN, K, J, I) are the distributed load or flux values for incremental and total loads, respectively, as defined in the cards DISTRIBUTED LOADS or DISTRIBUTED FLUXES, for the L-th degree of freedom of the K-th face node of the I-th element, for the J-th load condition.

REFOR(MDOFN, NPOIN)
It is used in the routines CONVRS, CONVRD and CONVRT. At first REFOR(I, J) is the value of the residual force or flux for the I-th degree of freedom of node J; subsequently it is the value of the total current force or flux for the same degree of freedom.

RLOAD(MEVAB, NELEM)
RLOAD(J, I) is the value of the generalized nodal residual force or flux for the J-th degree of freedom of element I. Obviously, at the first iteration of an increment, RLOAD(J, I) is the value of the nodal force equivalent to the incremental external load relative to the same degree of freedom.

RPLANE(3, NELEM)
RPLANE(*, I) are the components of the unit vector normal to planar thick shell elements.

RSP0(3, 3, NMATS)
RSP0(*, *, I) is the initial rotation matrix between the global and principal reference systems for the material number I, in case of anisotropic yield stress. The array is allocated only if NRAPH≠0.

RTIE(MDOFN, MRET, MDOFN, MTIE)
RTIE(L, K, J, I) is the coefficient which links the J-the degree of freedom of the tied node to the K-th degree of freedom of the J-th retained node in the I-th tying relation. This coefficient is considered only if IRTIE(L, K, J, I) is non-zero.

RTTEMP(2, MTAB, MTTEMP)
Tables of temperature varying properties.
RTTEMP(1, I, J) and RTTEMP(2, I, J) are, respectively, the temperature value and the corresponding property value at the I-th entry of the J-th table.

SELDIS(MDOFN, MNODE, NELEM)
SELDIS(K, J, I) is the value of the incremental generalized displacement or temperature relative to the K-th degree of freedom of the J-th node of element I.

SINK(MSIDE, MFAC, NELEM)
SINK(I, J, K) is the value of the sink temperature of the I-th node of the J-th face or edge of the K-th element. The array is allocated only if IIFILM=.TRUE..

SURF(8, MPART+1, MSURF)

SURF(1, 1, I) current x coordinate of the guide point of the I-th rigid surface.
SURF(2, 1, I) current y coordinate of the guide point of the I-th rigid surface.
SURF(3, 1, I) current x component of the velocity of the guide point of the I-th rigid surface.
SURF(4, 1, I) current y component of the velocity of the guide point of the I-th rigid surface.
SURF(5, 1, I) angular velocity of the guide point of the I-th rigid surface.
SURF(6, 1, I) initial x coordinate of the guide point of the I-th rigid surface.
SURF(7, 1, I) initial y coordinate of the guide point of the I-th rigid surface.
SURF(8, 1, I) not used.

SURF(K, J+1, I) has different significance depending upon whether the J-th part of the I-th rigid surface is a segment or a circumference arc.

a) Segment
SURF(1, J+1, I) current coordinate x of the segment's starting point.
SURF(2, J+1, I) current coordinate y of the segment's starting point.
SURF(3, J+1, I) current coordinate x of the segment's end point.
SURF(4, J+1, I) current coordinate y of the segment's end point.
SURF(5...8, J+1, I) not used.

b) Circumference arc
SURF(1, J+1, I) current coordinate x of the arc's starting point.
SURF(2, J+1, I) current coordinate y of the arc's starting point.
SURF(3, J+1, I) current coordinate x of the arc's end point.
SURF(4, J+1, I) current coordinate y of the arc's end point.
SURF(5, J+1, I) current coordinate x of the center.
SURF(6, J+1, I) current coordinate y of the center.
SURF(7, J+1, I) angle at center (rad) subtended by the arc.
SURF(8, J+1, I) radius of the circumference.

SURF0(12, MPART0, MSURF)

SURF0(K, J, I) defines the initial position and the current position of the J-th post-processing part of the I-th rigid surface; its meaning differs for segments and circumference arcs.

a) Segment
SURF0(1, J, I) initial value of the x coordinate of the segment's starting point.
SURF0(2, J, I) initial value of the y coordinate of the segment's starting point.
SURF0(3, J, I) initial value of the x coordinate of the segment's end point.
SURF0(4, J, I) initial value of the y coordinate of the segment's end point.
SURF0(5, J, I) not used.
SURF0(6, J, I) not used.
SURF0(7, J, I) current value of the x coordinate of the segment's starting point.
SURF0(8, J, I) current value of the y coordinate of the segment's starting point.
SURF0(9, J, I) current value of the x coordinate of the segment's end point.
SURF0(10, J, I) current value of the y coordinate of the segment's starting point.
SURF0(11, J, I) not used.
SURF0(12, J, I) not used.

b) Circumference arc

SURF0(1, J, I) initial value of the x coordinate of the arc's starting point.
SURF0(2, J, I) initial value of the y coordinate of the arc's starting point.
SURF0(3, J, I) initial value of the x coordinate of the arc's midpoint.
SURF0(4, J, I) initial value of the y coordinate of the arc's midpoint.
SURF0(5, J, I) initial value of the x coordinate of the arc's end point.
SURF0(6, J, I) initial value of the y coordinate of the arc's end point.
SURF0(7, J, I) current value of the x coordinate of the arc's starting point.
SURF0(8, J, I) current value of the y coordinate of the arc's starting point.
SURF0(9, J, I) current value of the x coordinate of the arc's midpoint.
SURF0(10, J, I) current value of the y coordinate of the arc's midpoint.
SURF0(11, J, I) current value of the x coordinate of the arc's end point.
SURF0(12, J, I) current value of the y coordinate of the arc's end point.

TDISP(MDOFN, NPOIN)
TDISP(I, J) is the current value of the total generalized displacement or temperature relative to the I-th degree of freedom of the J-th node.

TEMPE(ITYRD, NPOIN)
TEMPE(I, J) is the current value of the total temperature at layers of node J, used in the thermal loads calculations. The array is allocated only if IRDTMP≠0.

TEMPI(MDOFN, NPOIN)
TEMPI(I, J), is the value of the initial temperature of the I-th degree of freedom (I=1 except for shell elements) of the J-th node. The array is allocated only if IHEAT=.TRUE. or IRDTMP≠0.

TLOAD(MEVAB, NELEM)
TLOAD(J, I) is the value of the total generalized nodal force or flux, including reactions and correctional forces for follower forces, applied to the J-th degree of freedom of element I.

TREAC(MDOFN, NPOIN)
TREAC(J, I) is the value of the generalized reaction force or flux for the J-th degree of freedom of node I, due to kinematic constraints, contact and/or friction constraints and tying relations.

VECRV(MFRON)
VECRV(I) is the resolved value (generalized displacement or reaction) of the I-th variable active in the resolution front at the moment of back-substitution.

VELO(MDOFN, NPOIN)
VELO(I, J)) is the current value of the total generalized velocity relative to the I-th degree of freedom of the J-th node. The array is allocated only if IDYNA=.TRUE.

VELOI(MDOFN, NPOIN)
VELOI(I, J) is the value of the velocity at the beginning of an increment for the I-th degree of freedom of J-th node. The array is allocated only if IDYNA=.TRUE..

VERSN(MDIME, NPOIN)
VERSN(J, I) has meaning only if the node is internal to or in contact with a rigid surface, in this case it is the J-th component of the outward unit normal to the surface at node. The array is allocated only if MSURF≠0.

VERST(MDIME, NPOIN)
VERST(J, I) has meaning only if the node is interior to or in contact with a rigid surface, in this case it is the J-th component of the unit vector tangential to the surface at node I. The array is allocated only if IFRIC=.TRUE..

WEIGP(MGAUS, MTYPE)
WEIGP(J, I) is the weight of the J-th Gauss integration point (on the segment [-1, 1]) of the I-th element type.

DESCRIPTION OF THE REAL VARIABLES OF ARRAY EAREA

This array, which is a part of array VEC, contains both the total and incremental elemental data. The descriptions of the arrays contained in EAREA follow the alphabetical order regardless of their position and refer to the current element.

ACCA(LSTR1, LGASP)
ACCA(J, I) is the value, at the beginning of the load increment, of the J-th component of the tensor which manages the exponential kinematic hardening for infinitesimal strains, calculated at the I-th Gauss point of the element. The array is allocated only for elastic-plastic materials with infinitesimal deformations.

BMAT5(6, 24, LGASP)
BMAT5(I, J, K) is the term of the strain-displacement matrix connecting the I-th strain characteristic to the J-th degree of freedom, calculated at the K-th Gauss point of a thin shell element. The array is allocated only for thin shell elements.

BMAT9(6, 12, LGASP)
BMAT9(I, J, K) is the term of the strain-displacement matrix connecting the I-th strain characteristic to the J-th degree of freedom, calculated at the K-th Gauss point of a beam element. The array is allocated only for beam elements.

BMA10(8, 24, LGASP)
BMA10(I, J, K) is the term of the strain-displacement matrix connecting the I-th strain characteristic to the J-th degree of freedom, calculated at the K-th Gauss point of a thick shell element. The array is allocated only for thick shell elements.

CBASE(3, 3)
For shell elements, CBASE(I, J) is the I-th component of the J-th unit vector of the ortonormal base at the centroid of the element. For beam elements, CBASE(I, J) is the I-th component of the J-th unit vector of the ortonormal local base of the element. The array is allocated only for shell or beam elements.

CDERV(LDIME, LNODE, LGASP)
CDERV(K, J, I) is the current value of the derivative of the J-th shape function with respect to the K-th global coordinate direction, calculated at the I-th Gauss point of the element. The array is not allocated for shell or beam elements.

CDER0(LDIME, LNODE)
CDER0(K, J)is the value of the derivative of the J-th shape function with respect to the K-th global coordinate direction, calculated at the centroid of the element. The array is allocated only for elements with linear shape functions and only if ISELE≠0 for the current element.

CENTR(LSTR1, LGASP)
CENTR(J, I) is the value, at the beginning of the load increment, of the J-th component of the center of the elastic range in the space of deviatoric strains, calculated at the I-th Gauss point

of the element. The array is not allocated if IHEAT=.TRUE., for shell or beam elements, or for masonry-like materials.

CMAT(LEVAB, LEVAB)
CMAT is the linear elastic stiffness matrix, which is a term used to set up the damping matrix used in a dynamic analysis. The array is allocated only if LDAMPK=.TRUE.

COSTRE(LSTRE, 2, LGASP)
Is the incremental and total corrective term to be added to the stiffness matrix in the case of anisotropic yield material. The array is allocated only for elastic-plastic materials with finite deformations and with NRAPH≠0.

DELTAH(MSHEL, LGASP), DELTAH(MBDT, LGASP)
For composite shells/beams, DELTAH(I, J) is the thickness of the I-th layer/fiber at the J-th Gauss point. For homogeneous shells/beams, DELTAH(1, J) is the thickness of any layer/fiber at the J-th Gauss point. The array is allocated only if shell or beam elements are present in the mesh.

DENERG(3, LGASP)
DENERG(1, I) is the density of the kinetic energy at the I-th Gauss point of the element.
DENERG(2, I) is the density of the deformation energy at the I-th Gauss point of the element.
DENERG(3, I) is the increment change of the work done by the externally applied loads, reaction forces included.
The array is allocated only if LENER=.TRUE., *i.e.* if the option ENERGY has been set.

DMATG(LSTRE, LSTRE, LGASP), DMATG(LSTRE, LSTRE, LGASP, MCASI)
DMATG(L, K, J, I) is the current value of coefficient (L, K) of the constitutive tangent matrix calculated at th J-th Gauss point of the I-th layer point of the element (the distinction for layers has meaning only for shell elements). The array is not allocated if IHEAT=.TRUE..

DSTRA(LSTRE, LGASP), DSTRA(LSTRE, LGASP, MCASI)
DSTRA(K, J, I) is the current value of the K-th component of the total strain increment, calculated at the J-th Gauss point for the I-th layer point of the element (the distinction for layers has meaning only for shell elements). The array is not allocated if IHEAT=.TRUE..

DVOLU(LGASP)
DVOLU(I) is the current value of the volume or area differential factor, calculated at the I-th Gauss point of the element, multiplied by the suitable integration weights.

EMASS(LEVAB, LEVAB)
EMASS(I, J) is the mass matrix term connecting degrees of freedom I and J of the currente element. The array is allocated only if IDYNA=.TRUE.

GEI(2, LGASP)
For a beam element, GEI(*, I) are the two moments of inertia along the first two local coordinates directions on the beam cross section at the I-th Gauss point. The array is allocated only for beam elements.

GPBASE(3, 3, LGASP)
GPBASE(I, J, K) is the I-th component of the J-th unit vector forming the ortonormal local basis at the K-th Gauss point on the middle surface of a shell element. The array is allocated only for shell elements.

GPCOD(LCORD, LGASP)
GPCOD(J, I) is the value of the J-th global coordinate of the I-th Gauss point of the element.

GRADT(LCORD, LGASP), GRADT(LCORD, MCASI, LGASP)
GRADT(I, J, K) is the I-th component of the temperature gradient calculated at the J-th layer point (the second index drops for all element types except shells) of the K-th Gauss point. The array is allocated only if IHEAT=.TRUE..

IFBE(MBTHCK)
In case of composite beam elements, IFBE(I)=0 means that the I-th fiber on the beam cross section is dummy. In this way, starting from the default rectangular shape, a differently shaped cross section can be modeled. The array is allocated only for beam elements if COMPOSITE=.TRUE..

RSPTF(3, 3, LGASP)
RSPTF(*, *, I) is the rotation matrix, at the current iteration, between the global and principal reference system at I-th integration point of the element. The array is allocated only for elastic-plastic materials with finite deformations and with NRAPH≠0.

RSPTI(3, 3, LGASP)
RSPTI(*, *, I) is the rotation matrix, at the beginning of the increment, between the global and principal reference system at I-th integration point of the element. The array is allocated only for elastic-plastic materials with finite deformations and with NRAPH≠0.

SDC(LSTR1, LGASP)
SDC(J, I) is the current value of the J-th component of the tensor CENTR, calculated at the I-th Gauss point of the element. The array is not allocated if IHEAT=.TRUE., for shell or beam elements, or for masonry-like materials.

SDEP(LSTR1, LGASP, MCASI), SDEP(LSTR1, LGASP, MCASI)
SDEP(K, J, I) is the current value of the K-th component of the inelastic strain increment, calculated at the J-th Gauss point for the I-th layer point of the element (the distinction for layers has meaning only for shell elements). The array is not allocated if IHEAT=.TRUE..

SDH(LSTR1, LGASP)
SDH(J, I) it is the current value of the J-th component of the increment of tensor ACCA, calculated at the I-th Gauss point of the element. The array is allocated only for elastic-plastic materials with infinitesimal deformations.

SDSTRE(LSTRE, LGASP), SDSTRE(LSTRE, LGASP, MCASI)
SDSTRE(K, J, I) is the current value of the K-th component of the stress increment, calculated at the J-th Gauss point for the I-th layer point of the element (the distinction for layers has meaning only for shell elements). The array is not allocated if IHEAT=.TRUE..

SDUP(LSTR1, LGASP), SDUP(LSTR1, LGASP, MCASI)
SDUP(K, J, I) is the current value of the K-th component of the compressive inelastic strain increment, calculated at the J-th Gauss point for the I-th layer point of the element (the distinction for layers has meaning only for shell elements). SDUP is calculated only if the element is made up of a masonry-like material with bounded compressive strength.

SDZ(LGASP)
SDZ(I) is the current value of the increment of the Odqvist parameter (time integral of the norm of the strain velocity) calculated in the I-th Gauss point of the element. The array is not allocated if IHEAT=.TRUE., for shell or beam elements, or for masonry-like materials.

SECT(LGASP)
For a beam element, SECT(I) is the value of the area of the beam cross section at the I- Gauss point. The array is allocated only for beam elements.

SHAP5(9, LGASP)
SHAP5(I, J) is the value of the I-th shape function calculated at the J-th Gauss point of the thin shell element. The array is allocated only for thin shell elements.

THERM(LGASP), THERM(MCASI, LGASP)
THERM(I, J) is the increment of temperature of the I-th layer point (the first index is dropped for all element types except shells) of the J-th node of the element. The array is allocated only if IHEAT=.TRUE. or IRDTMP≠0.

TSTRA(LSTRE, LGASP), TSTRA(LSTRE, LGASP, MCASI)
TSTRA(K, J, I) is the value, at the beginning of the load increment, of the K-th total strain component in the J-th Gauss point for the I-th layer point of the element (the distinction for layers has meaning only for shell elements). The array is not allocated if IHEAT=.TRUE..

TSTRAP(LSTR1, LGASP), TSTRAP(LSTR1, LGASP, MCASI)
TSTRAP(K, J, I) is the value, at the beginning of the load increment, of the K-th inelastic strain component in the J-th Gauss point for the I-th layer point of the element (the distinction for layers has meaning only for shell elements). The array is not allocated if IHEAT=.TRUE..

TSTRE(LSTRE, LGASP), TSTRE(LSTRE, LGASP, MCASI)
TSTRE(K, J, I) is the value, at the beginning of the load increment, of the K-th stress component in the J-th Gauss point for the I-th layer point of the element (the distinction for layers has meaning only for shell elements). The array is not allocated if IHEAT=.TRUE..

TSTRUP(LSTR1, LGASP), TSTRUP(LSTR1, LGASP, MCASI)
TSTRUP(K, J, I) is the value, at the beginning of the load increment, of the K-th inelastic compressive strain component at the J-th Gauss point for the I-th layer point of the element (the distinction for layers has meaning only for shell elements). TSTRUP is allocated only if the element is made up of a masonry-like material with bounded compressive strength.

TTHERI(LGASP), TTHERI(MCASI, LGASP)

TTHERI(I, J) is the total temperature, at the beginning of the analysis, of the I-th layer point (the first index is dropped for all element types except shells) of the J-th node of the element. The array is allocated only if IHEAT=.TRUE. or IRDTMP≠0.

TTHERM(LGASP), TTHERM(MCASI, LGASP)
TTHERM(I, J) is the total temperature, at the beginning of the increment, of the I-th layer point (the first index is dropped for all element types except shells) of the J-th node of the element. The array is allocated only if IHEAT=.TRUE. or IRDTMP≠0.

ZETA(LGASP)
ZETA(I)I is the value of the Odqvist parameter (time integral of the norm of the plastic strain velocity) at the I-th Gauss point of the element. The array is not allocated if IHEAT=.TRUE., for shell or beam elements, or for masonry-like materials.

ZETSH(MCASI, LGASP), ZETSH(MBCASI, LGASP),
ZETSH(I, J) is the position along the thickness or on the cross section of the I-th Sympson integration point at the J-th Gauss integration point. The array is allocated only if shell or beam elements are present in the mesh.

H. DESCRIPTION OF THE OUTPUT

In this section the information that NOSA provides at the end of a F.E.M. analysis are listed

Information on the work areas

The NOSA output presents much information about the work area, the dimensions of work arrays and their allocations in memory.
More precisely, in the output the following quantities are printed: the dimensions of internal variables, the dimensions of variables relative to the elements, their allocations in memory, the bandwidth of the system stiffness matrix , the number of words necessary for in-core solution of the system.
The technique of in-core solution is used when the dimensions of the work areas specified in the card SIZING are greater than those necessary for in-core memorization of the matrices that have been used.
When the work space required is too large, the techniques of out-of-core solution are used and the length and format of the auxiliary file are printed.

Incremental information

At each load increment and at each iteration of the analysis, the following information can be printed:

1) information on loads. The incremental nodal forces relative to each element and the incremental loads summed over the model are printed.

2) information on CPU time. The CPU time at the beginning and end of assembly, at the end of the solution of the system and after the stress recovering is printed.

3) information on convergence. The values of total external loads, including reactions, and of the residual loads are printed. Finally the flag NCHEK, which indicates if the convergence criterion is satisfied or not, is also printed.

I. ERROR CODES

code	meaning
0	the list of errors encountered is required;
1	the total number, NPOIN, of nodes of the structure is less than or equal to zero;
2	the total number, IELEM, of elements of the structure is less than or equal to zero;
3	the number of constrained loads is less than or equal to zero;
4	the total number NMATS of different types of materials making up the structure is less than or equal to zero;
6	the element type considered does not fall within the range 1 to 10;
7	the maximum number of different element types making up the structure is not within range (1 to 10), or else the first component of the vector NTYPE is equal to zero;
8	the element type considered has not been yet implemented;
9	the SIZING card has not been defined;
10	the TITLE card has not been assigned;
15	a node number equal to zero has been found;
16	the numbering of nodes given is erroneous, the matrix LNODS has an element which is negative or greater than the maximum number of points contained in the mesh;
18	the number corresponding to a constrained node is less than zero or greater than the maximum number of nodes of the mesh;
19	the number relative to the constrained node currently under consideration is the same as the number of a previous node;
21	the interpretation of a card is impossible;
95	the number of the current element is not contained in the list of elements;
110	the coordinates of the current node are the same as those of a previous node;
111	a degree of freedom of the current node is not contained in the list;

112 a node has not been found in the matrix of the mesh connectivity;

113 the sum of the absolute values of the coordinates of an unused node is different from zero;

114 there is a unused node in the list of constrained nodes;

115 the number of a constrained nodes is zero.