SESAM INTERFACE FORMAT DESCRIPTION

Input Interface Format

Finite Element Model and Loads Data Types

Valid from SIF version 10

Sesam Interface Format Description

SIF

Date: 2022-05-09

Valid from SIF version 10

Prepared by: Digital Solutions at DNV

E-mail support: software.support@dnv.com

E-mail sales: digital@dnv.com

Table of contents

1 INTRODUCTION

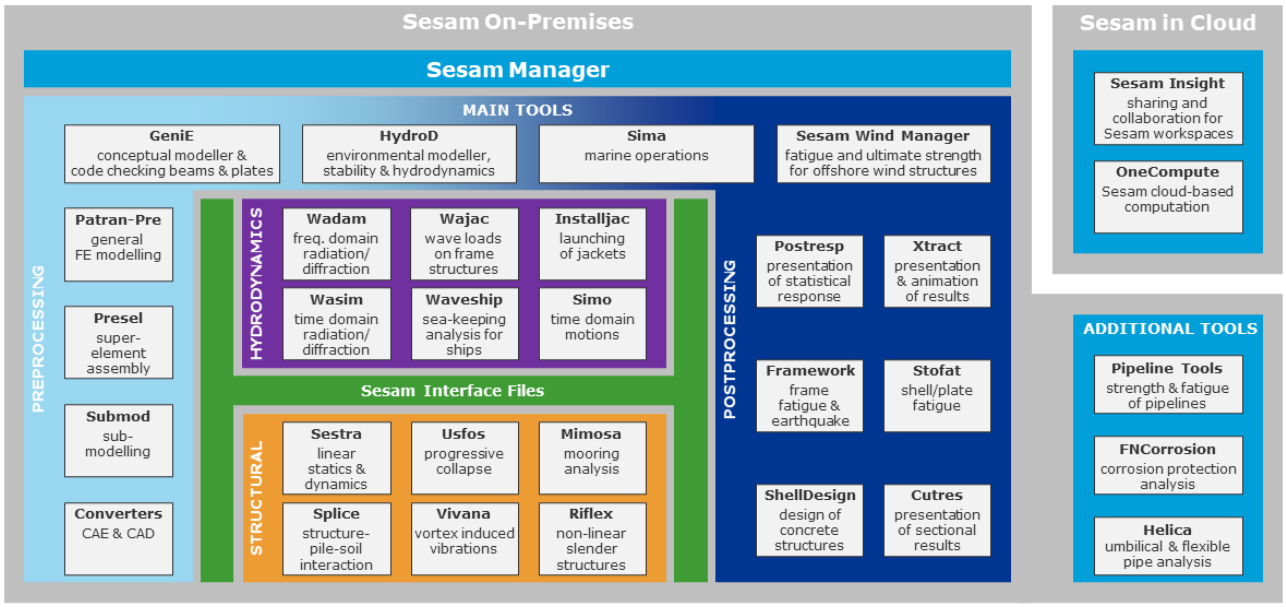


Figure 1.1: The Sesam system

This manual contains the Finite Element Model and Loads Data Types.

For the Finite Element Results Data Types consult [1].

Sesam Interface Format (SIF) provides a standardised basis for data communication in the Sesam system. The data definitions are organised as

- Input Interface data types.
- Load Interface data types.
- Result Interface data types which are further organised into
  - Structural Result Interface data types,
  - Hydrodynamic Result Interface data types.

1.1 Purpose

The purpose of SIF.API is to provide a clear, standardised and versatile data communication in the Sesam system. SIF.API is also intended to be open towards other software systems, as such it is also an interface between Sesam and other program systems.

1.2 How to Read the Manual

SIF.API consists of data types where a data type describes data for a node, an element, a result case etc.

Each data type has its unique name and input reference(s). An input reference may be a node number, an element number, a result case number etc.

2 IDENTIFICATION DATA FOR SUPER ELEMENTS AND TEXT DATA

The term identification data is used on the type of data that identify each super element, and the build up of the super element hierarchy.

2.1 Identification Data

DATE	Date and Program Information	see Section 2.1.1
IDENT	Identification of a Super Element	see Section 2.1.2
IEND	End of a Super Element	see Section 2.1.3
UNITS	Definition of the consistent units used in this Sesam Interface file	see Section 2.1.4

2.1.1 **DATE**: Date and Program Information

DATE	TYPE	SUBTYPE	NRECS	NBYTE

	Text Data

	Text Data

The identifier is used to transfer date and program information on the Interface File.

TYPE	Value giving information on how to use this text. = 1 Text concerning current super element. = 2 Text concerning children of current super element (not implemented).
SUBTYPE	= 0 If current super element (TYPE = 1). > 0 Sub element no. referring to the current super element (only if TYPE = 2).
NRECS	The following NRECS records must be read in A format, 72 characters per record. NRECS Number of records to be read in A format, NRECS $\geq$ 1.
NBYTE	NBYTE Number of significant bytes on the text data types, 1 $\leq$ NBYTE $\leq$ 72. The eight first bytes on the text data types shall be filled with blanks.

Example of the format of a **DATE** data type as used in Sesam:

```
DATE      1.00000000E+00  0.00000000E+00  4.00000000E+00  7.20000000E+01
        DATE:     25-Feb-2014         TIME:          08:37:26
        PROGRAM:  Sesam GeniE         VERSION:       D6.7-07 20-Dec-2013
        COMPUTER: X86 Windows         INSTALLATION:
        USER:     jeft                ACCOUNT:

------------------------------------------------------------------------
123456789.123456789.123456789.123456789.123456789.123456789.123456789.12
         1         2         3         4         5         6         7
------------------------------------------------------------------------
```

Back to Section 2.1 **Identification Data**

2.1.2 **IDENT**: Identification of a Super Element

IDENT	SLEVEL	SELTYP	SELMOD	

SLEVEL — Super element level.

The level of a super element is defined as the highest level number among its sub elements plus 1. (Basic elements, i.e. beams, shells, springs, etc. have level zero.)

SELTYP — Super element type number.

SELMOD — Super element model dimension

= 2, 2-dimensional model.

= 0 or 3, 3-dimensional model.

Back to Section 2.1 **Identification Data**

2.1.3 **IEND**: End of a Super Element

IEND	CONT			

CONT = 0 (Default). This is also end of the file.
= 1 The super elements are concatenated on one file.
More super elements follow.
= 2 Last super element in a structure for a concatenated file.

Back to Section 2.1 **Identification Data**

2.1.4 **UNITS**: Definition of the consistent units used in this Sesam Interface file

UNITS	NFIELD	→ ID	LENFAC	FORFAC
	TEMPFAC			

NFIELD	Number of fields in this record (this is always 5).
ID	Unit set ID (this should always be set to 1. for FEM file units).
LENFAC	Length unit converted to SI base unit [m].
FORFAC	Force unit converted to SI base unit [N].
TEMPFAC	Temperature difference unit converted to SI base unit [delC].

The Time unit is assumed to be seconds, the Angle unit is assumed to be radians, unless other units are specified in the Sesam Interface File card description.

The Sesam Interface File units are always consistent, i.e. the consistent Mass unit is derived from Force. Length and Time units.

When the SI base units (m,N,delC) are used, the 3 conversion factors will be 1.0.

Units and conversion factors

Length

Unit	Factor	
m	1.0	meter
mm	0.001	millimeter
inc	0.0254	inches
ft	0.3048	feet

Force

Unit	Factor	
N	1.0	Newton
kN	1.0e+3	kilo Newton
MN	1.0e+6	Mega Newton
lbf	4.4482216	pound force
kipf	4.4482216+3	kilopound force

Temperature difference

Unit	Factor	
delC	1.0	Celsius

delF	0.5555555	Fahrenheit

Back to Section 2.1 **Identification Data**

2.2 Text Data

TDELEM	Name of an Element and/or comment	see Section 2.2.1
TDLOAD	Name of a Local Load Case and/or comment	see Section 2.2.2
TDMATER	Name of a Material Type and/or comment	see Section 2.2.3
TDNODE	Name of a Node and/or comment	see Section 2.2.4
TDSECT	Name of a General Eccentric Sandwich Section and/or comment	see Section 2.2.5
TDSETNAM	Name of a Set and/or comment	see Section 2.2.6
TDSUPNAM	Name of a Super Element and/or comment	see Section 2.2.7
TEXT	User supplied Text	see Section 2.2.8
TSLAYER	Name of a General Eccentric Sandwich Type and/or comment	see Section 2.2.9

2.2.1 **TDELEM**: Name of an Element and/or comment

TDELEM	NFIELD	→ ELNO	CODNAM	CODTXT
	Name			
	Comment line			
	. . .			
	Comment line			

This data type will associate a name and/or a comment to the element with identification ELNO.

NFIELD — Number of numeric data fields at this data type before text data (MAX = 1024).

ELNO — Internal element number (unique).

CODNAM — Coded dimension of the Name:
CODNAM = NLNAM*100 + NCNAM. The inverse relation will then be:
NLNAM = integer part of (CODNAM/100)
NCNAM = remaindering of (CODNAM/100)
NLNAM number lines used to store the name. Legal range = [0,1]
= 0, no name defined
= 1, name is defined
NCNAM - number of characters in the name. Legal range = [0,64]

CODTXT — Coded dimension of the Comment:
CODTXT = NLTXT*100 + NCTXT. The inverse relation will then be:
NLTXT = integer part of (CODTXT/100)
NCTXT = remaindering of (CODTXT/100)
NLTXT - number of lines used to store the comment. Legal range = [0,5]
= 0, no comments defined
$\geq$ 1, number of physical records with comments
NCTXT - number of characters in the comment – each comment line must be of the same length. Legal range = [0,64]

Name — A user set name.

Comment line — User set comment lines.

Back to Section 2.2 **Text Data**

2.2.2 **TDLOAD**: Name of a Local Load Case and/or comment

TDLOAD	NFIELD	→ LLC	CODNAM	CODTXT
	Name			
	Comment line			
	. . .			
	Comment line			

This data type will associate a name and/or a comment to the local load case with identification LLC.

NFIELD — Number of numeric data fields at this data type before text data (MAX = 1024).

LLC — Local load case number.

CODNAM — Coded dimension of the Name:
CODNAM = NLNAM*100 + NCNAM. The inverse relation will then be:
NLNAM = integer part of (CODNAM/100)
NCNAM = remaindering of (CODNAM/100)
NLNAM number lines used to store the name. Legal range = [0,1]
= 0, no name defined
= 1, name is defined
NCNAM - number of characters in the name. Legal range = [0,64]

CODTXT — Coded dimension of the Comment:
CODTXT = NLTXT*100 + NCTXT. The inverse relation will then be:
NLTXT = integer part of (CODTXT/100)
NCTXT = remaindering of (CODTXT/100)
NLTXT - number of lines used to store the comment. Legal range = [0,5]
= 0, no comments defined
$\geq$ 1, number of physical records with comments
NCTXT - number of characters in the comment – each comment line must be of the same length. Legal range = [0,64]

Name — A user set name.

Comment line — User set comment lines.

Back to Section 2.2 **Text Data**

2.2.3 **TDMATER**: Name of a Material Type and/or comment

TDMATER	NFIELD	→ MATNO	CODNAM	CODTXT
	Name			
	Comment line			
	. . .			
	Comment line			

This data type will associate a name and/or a comment to the material with identification MATNO.

NFIELD — Number of numeric data fields at this data type before text data (MAX = 1024).

MATNO — Material number.

CODNAM — Coded dimension of the Name:
CODNAM = NLNAM*100 + NCNAM. The inverse relation will then be:
NLNAM = integer part of (CODNAM/100)
NCNAM = remaindering of (CODNAM/100)
NLNAM number lines used to store the name. Legal range = [0,1]
= 0, no name defined
= 1, name is defined
NCNAM - number of characters in the name. Legal range = [0,64]

CODTXT — Coded dimension of the Comment:
CODTXT = NLTXT*100 + NCTXT. The inverse relation will then be:
NLTXT = integer part of (CODTXT/100)
NCTXT = remaindering of (CODTXT/100)
NLTXT - number of lines used to store the comment. Legal range = [0,5]
= 0, no comments defined
$\geq$ 1, number of physical records with comments
NCTXT - number of characters in the comment – each comment line must be of the same length. Legal range = [0,64]

Name — A user set name.

Comment line — User set comment lines.

Back to Section 2.2 **Text Data**

2.2.4 **TDNODE**: Name of a Node and/or comment

TDNODE	NFIELD	→ NODENO	CODNAM	CODTXT
	Name			
	Comment line			
	. . .			
	Comment line			

This data type will associate a name and/or a comment to the node with identification NODENO.

NFIELD — Number of numeric data fields at this data type before text data (MAX = 1024).

NODENO — Node number.

CODNAM — Coded dimension of the Name:
CODNAM = NLNAM*100 + NCNAM. The inverse relation will then be:
NLNAM = integer part of (CODNAM/100)
NCNAM = remaindering of (CODNAM/100)
NLNAM number lines used to store the name. Legal range = [0,1]
= 0, no name defined
= 1, name is defined
NCNAM - number of characters in the name. Legal range = [0,64]

CODTXT — Coded dimension of the Comment:
CODTXT = NLTXT*100 + NCTXT. The inverse relation will then be:
NLTXT = integer part of (CODTXT/100)
NCTXT = remaindering of (CODTXT/100)
NLTXT - number of lines used to store the comment. Legal range = [0,5]
= 0, no comments defined
$\geq$ 1, number of physical records with comments
NCTXT - number of characters in the comment – each comment line must be of the same length. Legal range = [0,64]

Name — A user set name.

Comment line — User set comment lines.

Back to Section 2.2 **Text Data**

2.2.5 **TDSECT**: Name of a General Eccentric Sandwich Section and/or comment

TDSECT	NFIELD	→ GEONO	CODNAM	CODTXT
	Name			
	Comment line			
	. . .			
	Comment line			

This data type will associate a name and/or a comment to the general eccentric sandwich section with identification GEONO.

NFIELD — Number of numeric data fields at this data type before text data (MAX = 1024).

GEONO — General eccentric sandwich section number.

CODNAM — Coded dimension of the Name:
CODNAM = NLNAM*100 + NCNAM. The inverse relation will then be:
NLNAM = integer part of (CODNAM/100)
NCNAM = remaindering of (CODNAM/100)
NLNAM number lines used to store the name. Legal range = [0,1]
= 0, no name defined
= 1, name is defined
NCNAM - number of characters in the name. Legal range = [0,64]

CODTXT — Coded dimension of the Comment:
CODTXT = NLTXT*100 + NCTXT. The inverse relation will then be:
NLTXT = integer part of (CODTXT/100)
NCTXT = remaindering of (CODTXT/100)
NLTXT - number of lines used to store the comment. Legal range = [0,5]
= 0, no comments defined
≥ 1, number of physical records with comments
NCTXT - number of characters in the comment – each comment line must be of the same length. Legal range = [0,64]

Name — A user set name.

Comment line — User set comment lines.

Back to Section 2.2 **Text Data**

2.2.6 **TDSETNAM**: Name of a Set and/or comment

TDSETNAM	NFIELD	→ ISREF	CODNAM	CODTXT
	Set - name			
	Text line			
	. . .			
	Text line			

This data type together with the set of nodes or elements data type(s) (**GSETMEMB**) constitute the set (group) datatype.

NFIELD — Number of numeric data fields at this data type before text data (MAX = 1024).

ISREF — Internal set identification number. Legal range [1,NSET], where NSET is number of sets which is equeal to number of "Name and Description of a Set" data types (**TDSETNAM**). Two **TDSETNAM** data types may not have identical set identification numbers (ISREF).

CODNAM — Coded dimension of the Name:
CODNAM = NLNAM*100 + NCNAM. The inverse relation will then be:
NLNAM = integer part of (CODNAM/100)
NCNAM = remaindering of (CODNAM/100)
NLNAM number lines used to store the name. Legal range = [0,1]
= 0, no name defined
= 1, name is defined
NCNAM - number of characters in the name. Legal range = [0,64]

CODTXT — Coded dimension of the Comment:
CODTXT = NLTXT*100 + NCTXT. The inverse relation will then be:
NLTXT = integer part of (CODTXT/100)
NCTXT = remaindering of (CODTXT/100)
NLTXT - number of lines used to store the comment. Legal range = [0,5]
= 0, no comments defined
$\geq$ 1, number of physical records with comments
NCTXT - number of characters in the comment – each comment line must be of the same length. Legal range = [0,64]

Back to Section 2.2 **Text Data**

2.2.7 **TDSUPNAM**: Name of a Super Element and/or comment

TDSUPNAM	NFIELD	→ IHREF	CODNAM	CODTXT
	Super element - name			
	Text line			
	. . .			
	Text line			

This data type will associate a name and/or a comment to a super element in the super element hierarchy.

NFIELD — Number of numeric data fields at this data type before text data (MAX = 1024).

IHREF — Hierarchy reference number. Number 1 is reserved for the top level super element. In Sesam, PRESEL (super element pre-processor) is writing the HIERARCH data types and defining a unique number (IHREF) for each appearance of the different super elements. See also Figure 2.1 below.

CODNAM — Coded dimension of the Name:
CODNAM = NLNAM*100 + NCNAM. The inverse relation will then be:
NLNAM = integer part of (CODNAM/100)
NCNAM = remaindering of (CODNAM/100)
NLNAM number lines used to store the name. Legal range = [0,1]
= 0, no name defined
= 1, name is defined
NCNAM - number of characters in the name. Legal range = [0,64]

CODTXT — Coded dimension of the Comment:
CODTXT = NLTXT*100 + NCTXT. The inverse relation will then be:
NLTXT = integer part of (CODTXT/100)
NCTXT = remaindering of (CODTXT/100)
NLTXT - number of lines used to store the comment. Legal range = [0,5]
= 0, no comments defined
$\geq$ 1, number of physical records with comments
NCTXT - number of characters in the comment – each comment line must be of the same length. Legal range = [0,64]

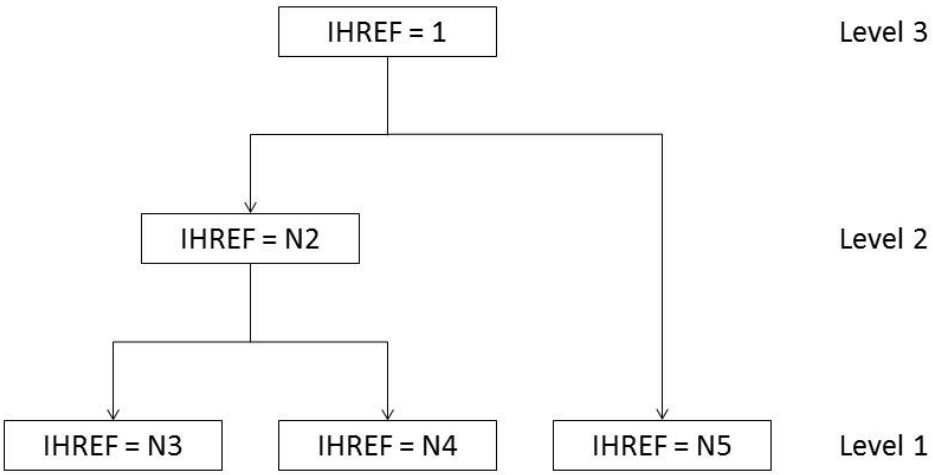


Figure 2.1: Super element hierarchy with 3 levels.

In Figure 2.1 above, please notice that N2, N3, N4, and N5 may take any values as long as they are unique in the hierarchy. Further notice that the top level super element has IHREF = 1 as required for the top level super element.

Back to Section 2.2 **Text Data**

2.2.8 **TEXT**: User supplied Text

TEXT	TYPE	SUBTYPE	NRECS	NBYTE
	...			

The identifier is used to transfer text strings on the interface file. The following NRECS records must be read in A-format, 72 characters per record.

TYPE	Value giving information of how to use this text
	= 1 Texts describing this analysis/global text
	= 2 Texts concerning current super element
	= 3 Text concerning specific load cases
	$\geq$ 4 The meaning of text to be mutually agreed on by pre-processor and analysis program
SUBTYPE	Value giving additional information to TYPE
	Example: For TYPE = 3, SUBTYPE gives load case number.
NRECS	Number of records following to be read in A-format. NRECS $\geq$ 1
NBYTE	Number of significant bytes (characters) on the following NRECS records.
	$1 \leq$ NBYTE ≤ 72
	The eight first bytes on the text records shall be filled with blanks.

Back to Section 2.2 **Text Data**

2.2.9 **TSLAYER**: Name of a General Eccentric Sandwich Type and/or comment

TSLAYER	NFIELD	→ GEONO	CODNAM	CODTXT
	Name			
	Comment line			
	. . .			
	Comment line			

This data type will associate a name and/or a comment to the general eccentric sandwich section layer with identification GEONO.

NFIELD — Number of numeric data fields at this data type before text data (MAX = 1024).

GEONO — General eccentric sandwich section layer number.

CODNAM — Coded dimension of the Name:
CODNAM = NLNAM*100 + NCNAM. The inverse relation will then be:
NLNAM = integer part of (CODNAM/100)
NCNAM = remaindering of (CODNAM/100)
NLNAM number lines used to store the name. Legal range = [0,1]
= 0, no name defined
= 1, name is defined
NCNAM - number of characters in the name. Legal range = [0,64]

CODTXT — Coded dimension of the Comment:
CODTXT = NLTXT*100 + NCTXT. The inverse relation will then be:
NLTXT = integer part of (CODTXT/100)
NCTXT = remaindering of (CODTXT/100)
NLTXT - number of lines used to store the comment. Legal range = [0,5]
= 0, no comments defined
$\geq$ 1, number of physical records with comments
NCTXT - number of characters in the comment – each comment line must be of the same length. Legal range = [0,64]

Name — A user set name.

Comment line — User set comment lines.

Back to Section 2.2 **Text Data**

3 ELEMENT TYPES

Conventions for use of the interface file for the elements in Sesam are defined here. Other element types may be introduced for use in other programs.

The table below contains element type numbers already reserved. (Not all of them are included in Sesam).

Id	Description	Element Type	Section
BEPS	Beam Element for Plane Systems	2	see Section 3.1
CSTA	Plane Constant Strain Triangle	3	see Section 3.2
ILST	Plane Linear Strain Triangle	6	see Section 3.3
IQQE	Plane Isoparametric Quadrilateral Membrane Element	8	see Section 3.4
LQUA	Plane Incompatible Linear Quadrilateral Membrane Element	9	see Section 3.5
TESS	Truss Element	10	see Section 3.6
GMAS	1-Noded Mass Element	11	see Section 3.7
GLMA	General 2-Noded Mass Element	12	see Section 3.8
GLDA	General 2-Noded Damping Element	13	see Section 3.9
BEAS	Beam Element	15	see Section 3.10
AXIS	Axial Spring	16	see Section 3.11
AXDA	Axial Damper	17	see Section 3.12
GSPR	Ground Spring	18	see Section 3.13
GDAM	Damper to Ground	19	see Section 3.14
IHEX	Iso-parametric Hexahedron	20	see Section 3.15
LHEX	Linear Hexahedron	21	see Section 3.16
SECB	Sub-parametric Curved Beam	22	see Section 3.17
BTSS	Sub-parametric General Curved Beam	23	see Section 3.18
FQUS	Flat Quadrilateral Thin Shell	24	see Section 3.19
FTRS	Flat Triangular Thin Shell	25	see Section 3.20
SCTS	Sub-parametric Curved Triangular Shell	26	see Section 3.21
MCTS	Sub-parametric Multi-layered Curved Triangular Shell	27	see Section 3.22
SCQS	Sub-parametric Curved Quadrilateral Shell	28	see Section 3.23
MCQS	Sub-parametric Multilayered Curved Quadrilateral Shell	29	see Section 3.24
IPRI	Iso-parametric Prism	30	see Section 3.25
ITET	Iso-parametric Tetrahedron	31	see Section 3.26
TPRI	Triangular Prism	32	see Section 3.27
TETR	Tetrahedron	33	see Section 3.28
LCTS	Sub-parametric Layered Curved Triangular Shell	34	see Section 3.29

continued . . .

Id	Description	Element Type	Section
LCQS	Sub-parametric Layered Curved Quadrilateral Shell	35	see Section 3.30
TRSI	Transition Elements between Solids and Shells	36, 37, or 38	see Section 3.31
GLSH	General 2-Noded Spring / Shim Element	40	see Section 3.32
AXCS	Axi-symmetric Constant Strain Triangle	41	see Section 3.33
AXLQ	Axi-symmetric Constant Strain Quadrilateral	42	see Section 3.34
AXLS	Axi-symmetric Linear Strain Triangle	43	see Section 3.35
AXQQ	Axi-symmetric Linear Strain Quadrilateral	44	see Section 3.36
CTCP	2-Noded (1+1) Contact Element	51	see Section 3.37
CTCL	4-Noded (2+2) Contact Element	52	see Section 3.38
CTAL	4-Noded (2+2) Axi-symmetric Contact Element	53	see Section 3.39
CTCC	6-Noded (3+3) Contact Element	54	see Section 3.40
CTAQ	6-Noded (3+3) Axi-symmetric Contact Element	55	see Section 3.41
CTLQ	8-Noded (4+4) Contact Element	56	see Section 3.42
CTCQ	16-Noded (8+8) Contact Element	57	see Section 3.43
CTMQ	18-Noded (9+9) Contact Element	58	see Section 3.44
FTAS	Flat Triangular Thin Shell – Drilling degree of freedom	59	see Section 3.45
FQAS	Flat Quadrilateral Thin Shell – Drilling degree of freedom	60	see Section 3.46
HCQS	Heterosis Curved Quadrilateral Shell	61	see Section 3.47
THTS	Flat 3-noded Thick Triangular Shell	63	see Section 3.48
THQS	Flat 4-noded Thick Quadrilateral Shell	64	see Section 3.49
MATR	General Matrix Element	70	see Section 3.50
GHEX	General Hexahedron	100, . . . , 163	see Section 3.51

3.1 BEPS: Beam Element for Plane Systems

Element Type 2.

- 2 nodes
- 6 degrees of freedom, 3 (u,v and q) at each of the two nodes
- Bending, shear and axial deformations are considered
- The element is straight and has a constant cross section
- offset nodes (i.e. the nodes may be located eccentric in space)
- element loads:
  - load linearly distributed over all, or a part, of the element (Figure 5-1 b)
  - gravitational load
  - general inertia load
  - initial strain (temperature load)

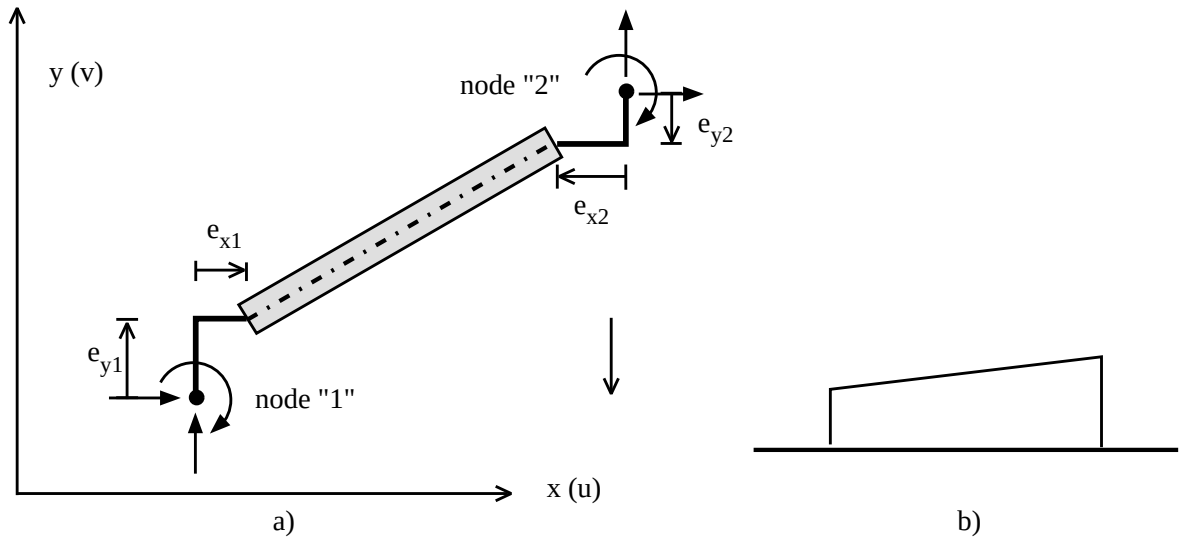


Figure 5-1 a)2Ð Beam Element b)Linearly Distributed Load

Element coordinate system (reference axes):

The local x-axis is directed along the beam, coinciding with the center of gravity and pointing from the beam node "1" to node "2". The local z-axis is defined on GUNIVEC-record.

Data types used for this element:

GELMNT1 *
GELREF1 *
GBEAMG *
GIORH, GUSYI, GCHAN, GBOX, GPIPE, GBARM, GTONP or GDOBO;
for SESTRA these reecords are transferred to postprocessor, and only referred when storing on result file.

MISOSEL *
GUNIVEC *
GECCEN
BELOAD1
BGRAV
BNACCLO
BEISTE
BELFIX

*) Mandatory

Back to Section 3 **ELEMENT TYPES**

3.2 CSTA: Plane Constant Strain Triangle

Element Type 3, see reference [7]

- 3 nodes
- 3 x 2 degrees of freedom
- straight (two dimensional)
- linearly varying thickness
- deformation considered: translational strain
- element loads:
  - line loads
  - initial strain (temperature load)
  - gravitational load (only in the mebrane plane)
  - general inertia load (only in the membrane plane)
- isotropic or anisotropic material data

Local node numbering:

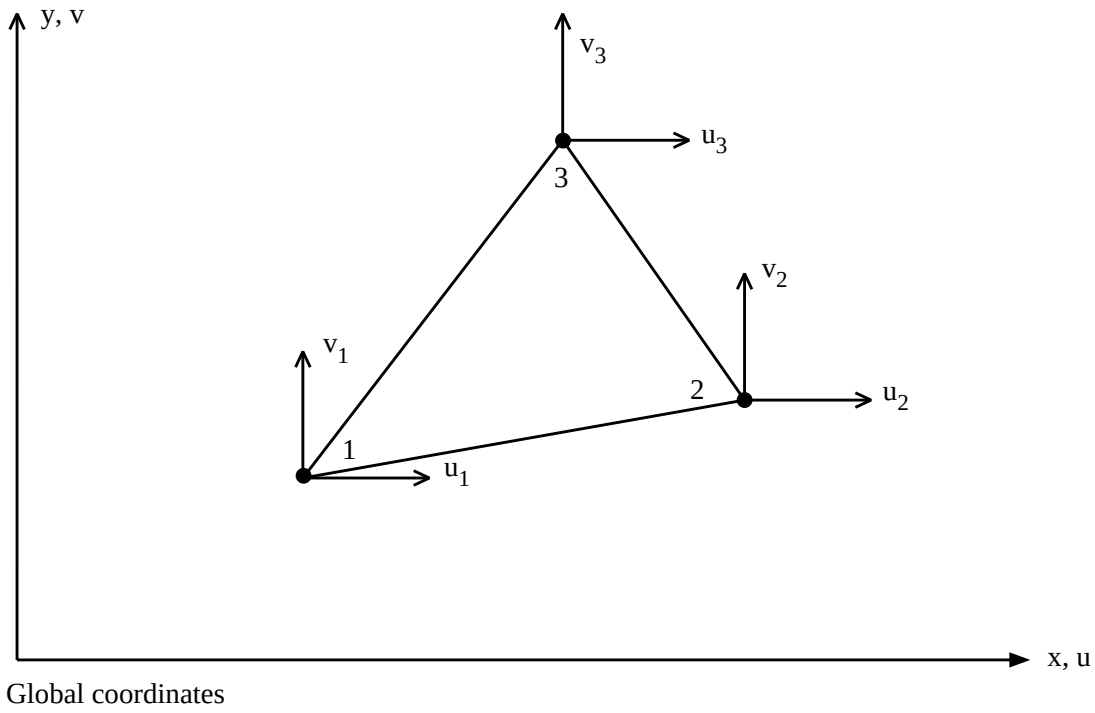


Figure 5-2 Plane constant strain triangle.

When line load is specified, the relation between local node numbers and loaded line will be:

Line 1 means load along the line defined by the nodes 2 and 3.
Line 2 means load along the line defined by the nodes 1 and 3.
Line 3 means load along the line defined by the nodes 1 and 2.

The direction of node numbering can be as well clockwise as counterclockwise.

Data types used for this element:

GELMNT1 *
GELREF1 *
GNODE *
GCOORD *
GELTH *

MISOSEL * or
MORSMEL *
MTRMEL

BELLO2
BEISTE
BGRAV
BNACCLO

*) Mandatory

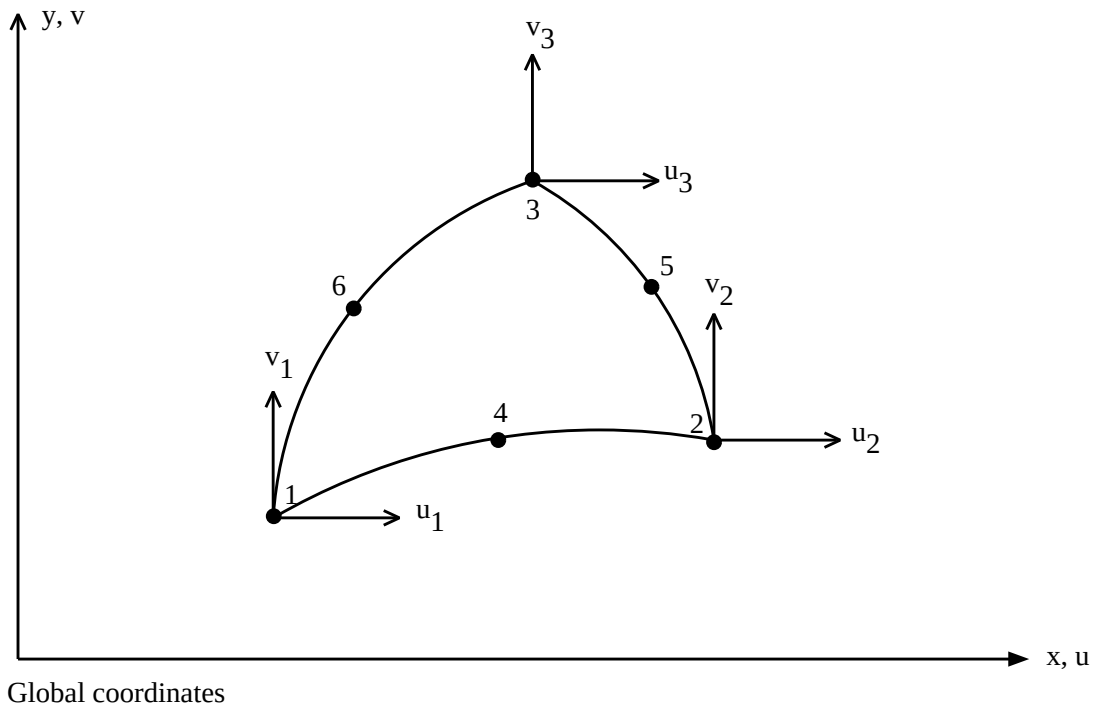
Back to Section 3 **ELEMENT TYPES**

3.3 ILST: Plane Linear Strain Triangle

Element Type 6, see reference [7]

- 6 nodes
- 6 x 2 degrees of freedom
- curved (two-dimensional)
- linearly varying thickness
- deformations considered: translational strain
- element loads:
  - line loads
  - initial strain (temperature loads)
  - gravitational load
  - general inertia load
- isotropic or anisotropic material data

Local node numbering:



Global coordinates

Figure 5-3 Plane linear strain triangle.

When line load is specified, the relation between local node numbers and loaded line will be:

Line 1 means load along the line defined by the nodes 2, 5 and 3.
Line 2 means load along the line defined by the nodes 1, 6 and 3.
Line 3 means load along the line defined by the nodes 1, 4 and 2.

The direction of node numbering can be as well clockwise as counterclockwise.

The following restrictions are put on the data types

GELINT — The integration stations must be distributed according to the Gaussian integration scheme, i.e.
INTYPE =1. For this element type the GELINT specifications consist of the first filerecord (line) only.

Stiffness matrix: For stiffness matrix calculations the number of integration stations will be the same in both coordinate directions and equal to the value specified for N1.
Legal values are 3 and 4.
Default value is 3.

Mass matrix: For mass matrix calculations the number of integration stations will be the same in both coordinate directions and equal to the value specified for N1.
Legal values are 1,3,4 and 7.
Default value is 7.

Load vector: For load vector calculations due to initial strains the number of integration stations will be the same in both coordinate directions and equal to the value specified for N1.
Legal values are 1,3 and 4.
Default value is 3.

Data types used for this element:

GELMNT1 *
GELREF1 *
GNODE *
GCOORD *
GELTH *
GELINT
MISOSEL or MORSMEL *

MTRMEL
BELLO2
BEISTE
BGRAV
BNACCLO

*) Mandatory

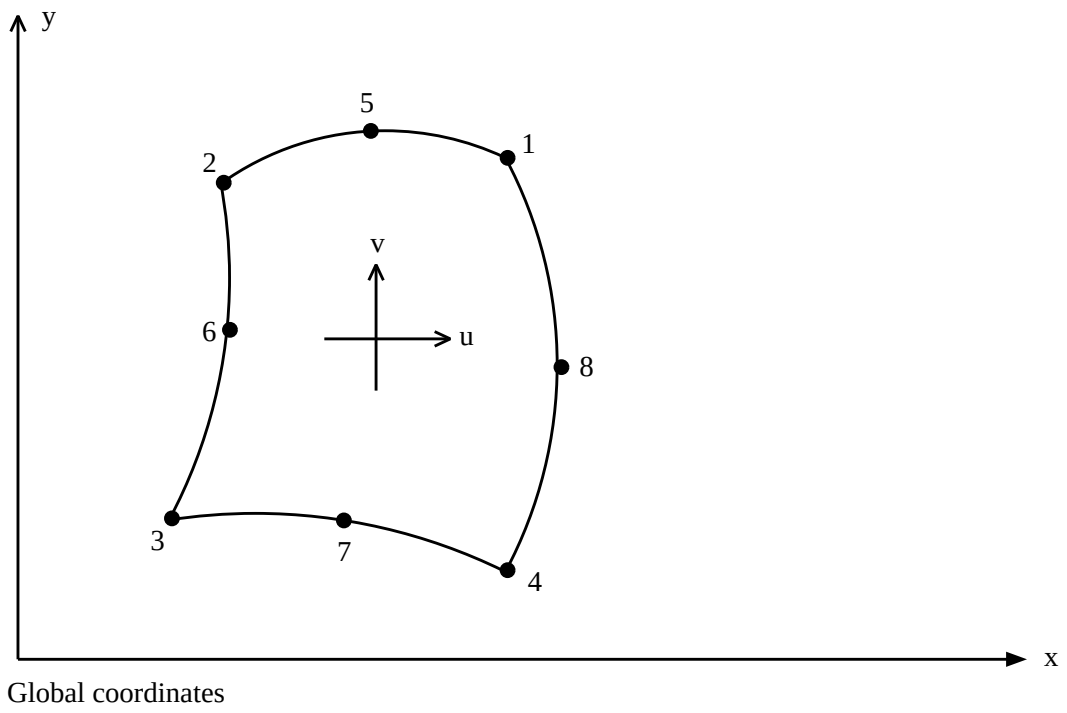
Back to Section 3 **ELEMENT TYPES**

3.4 IQQE: Plane Isoparametric Quadrilateral Membrane Element

Element Type 8, see reference [7]

- 8 nodes
- 8 x 2 degrees of freedom
- curved (two-dimensional)
- linearly varying thickness
- deformations considered: translational strain
- element loads:
  - line loads
  - initial strain (temperature loads)
  - gravitational load
  - general inertia load
- isotropic or anisotropic material data

Local node numbering:



Global coordinates

Figure 5-4 Plane quadrilateral membrane element.

When line load is specified, the relation between local node numbers and loaded line will go:

LINE 1 means load along the line defined by the nodes 2, 6 and 3
LINE 2 means load along the line defined by the nodes 5 and 7
LINE 3 means load along the line defined by the nodes 1, 8 and 4
LINE 4 means load along the line defined by the nodes 1, 5 and 2
LINE 5 means load along the line defined by the nodes 8 and 6
LINE 6 means load along the line defined by the nodes 4, 7 and 3

The direction of node numbering can be as well clockwise as counterclockwise.

The following restrictions are put on the data types

GELINT — The integration stations must be distributed according to the Gaussian integration scheme, i.e.
INTYPE =1. For this element type the GELINT specifications consist of the first filerecord only.

Stiffness matrix: For stiffness matrix calculations the number of integration stations will be the same in both coordinate directions and equal to the value specified for N1.
Legal values are 2, 3 and 4. Default value is 2.

Mass matrix: For mass matrix calculations the number of integration stations will be the same in both coordinate directions and equal to the value specified for N1.
Legal values are 2, 3 and 4. Default value is 4.

Load vector: For load vector calculations due to initial strains the number of integration stations will be the same in both coordinate directions and equal to the value specified for N1.
Legal values are 2, 3 and 4.
Default value is 2.

BELLO2 — LINE = 2, and LINE = 5 are not operative.

Data types used for this element:

GELMNT1 *
GELREF1 *
GNODE *
GCOORD *
GELTH *
GELINT
MISOSEL or MORSMEL *
MTRMEL
BELLO2
BEISTE
BGRAV
BNACCLO

*) Mandatory

Back to Section 3 **ELEMENT TYPES**

3.5 LQUA: Plane Incompatible Linear Quadrilateral Membrane Element

Element Type 9, see reference [7]

- 4 nodes
- 4 x 2 degrees of freedom
- straight (two-dimensional)
- linearly varying thickness
- isotropic or anisotropic material data
- deformations considered: translational strain
- element loads
  - line loads
  - initial strain (temperature load)
  - gavitational load
  - general inertia load

Local node numbering:

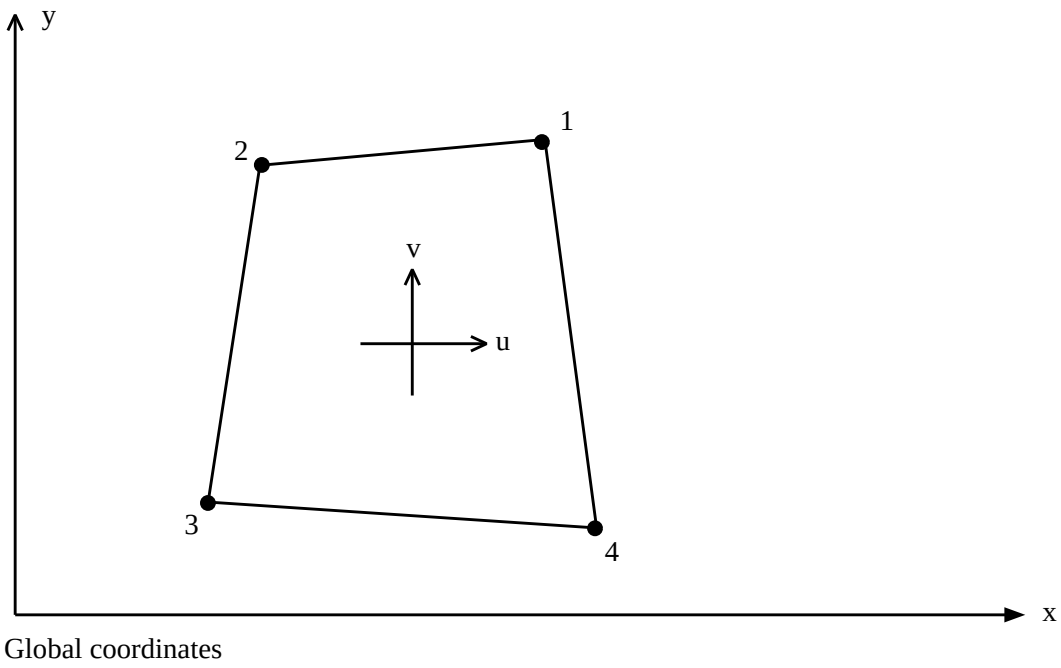


Figure 5-5 Plane Quadrilateral Membrane Element.

When line load is specified, the relation between local node numbers and loaded line will go:

LINE 1 means load along the line defined by the nodes 1 and 2
LINE 2 means load along the line defined by the nodes 2 and 3
LINE 3 means load along the line defined by the nodes 3 and 4
LINE 4 means load along the line defined by the nodes 4 and 1

The direction of node numbering can be as well clockwise as counterclockwise.

Data types used for this element:

GELMNT1 *
GELREF1 *
GNODE *
GCOORD *
GELTH *
GELINT
MISOSEL or *
MORSMEL
BELLO2
MTRMEL
BEISTE
BGRAV
BNACCLO

*) Mandatory

The following restrictions are put on data types:

GELINT The integration stations must be distributed according to the Gaussian integration scheme, i.e.
INTYPE =1. For this element type the GELINT specifications consist of the first line.

Stiffness matrix:
Legal values for N1 and N2 are 1, 2, 3 and 4.
Default value is N1=N2=2.

Mass matrix: For mass matrix calculations the number of integration stations will be the same
in both coordinate directions and equal to the value specified for N1.
Legal values are 2 and 3.
Default value is N1=N2=3.

Back to Section 3 **ELEMENT TYPES**

3.6 TESS: Truss Element

Element Type 10, see reference [7]

- 2 nodes
- 2 x 3 = 6 degrees of freedom
- straight
- constant cross section
- axial stiffness only
- element loads:
  - initial strain (temperature)

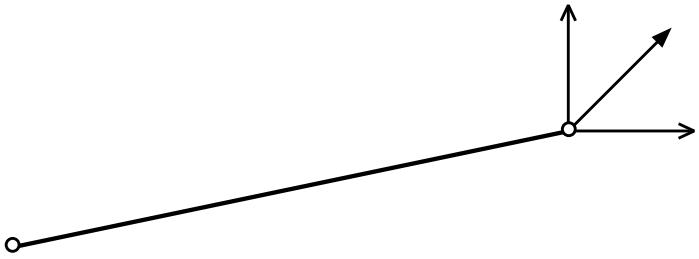


Figure 5-6 Truss element

Data types used for this element:

BEISTE

GELMNT1 *
GBEAMG (only AREA) *
GELREF1 *

MISOSEL

*) Mandatory

Back to Section 3 **ELEMENT TYPES**

3.7 GMAS: 1-Noded Mass Element

Element Type 11

- 1 node
- degrees of freedom, arbitrary
- mass matrix

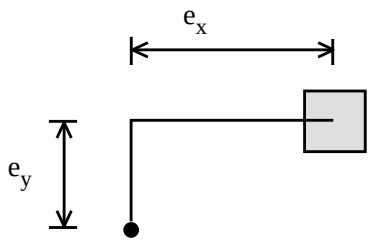


Figure 5-7 1-noded mass element

The mass point may be specified with eccentricities (e_x, e_y and e_z) in all three global directions and the mass matrix may be specified in a transformed local coordinate system.

The mass matrix is a full symmetric matrix where all values on and below the diagonal are stored.

Data types used for this element:

GELMNT1 *
MGMASS * (NDOF must be equal to NDOF on data type GNODE)
GELREF1 *
GECCEN
BNTRCOS (NDOF = 3 or 6 is required for transformations)

*) Mandatory

Back to Section 3 **ELEMENT TYPES**

3.8 GLMA: General 2-Noded Mass Element

Element Type 12

- 2 nodes
- degrees of freedom, arbitrary
- general mass matrix

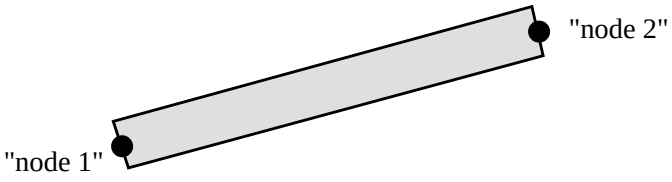


Figure 5-8 General 2-noded mass element.

The resulting mass matrix is a full symmetric matrix where all values on and below the main diagonal are stored.

Data types used for this element:

GELMNT1 *
MGLMASS * (NDOF1 and NDOF2 must be equal to NDOF on data type GNODE for "node 1" and "node 2")
GELREF1 *
BNTRCOS (Transformation in the two nodes may be different. But NDOF1 = NDOF2 = 3 or 6 is required for transformation).

*) Mandatory

Back to Section 3 **ELEMENT TYPES**

3.9 GLDA: General 2-Noded Damping Element

Element Type 13

- 2 nodes
- degrees of freedom, arbitrary
- general damping matrix

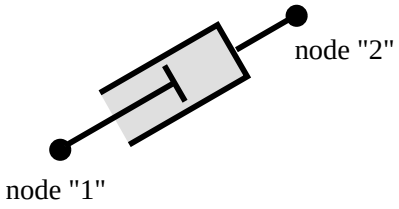


Figure 5-9 General 2-noded damping element

The resulting damping matrix is a full symmetric matrix where all values on and below the main diagonal are stored.

Data types used for this element:

GELMNT1 *
MGLDAMP * (NDOF1 and NDOF2 must be equal to NDOF on data type GNODE for node "1" and node "2")
GELREF1 *
BNTRCOS (Transformation in the two nodes may be different. But NDOF1 = NDOF2 = 3 or 6 is required for transformation).

*) Mandatory

Back to Section 3 **ELEMENT TYPES**

3.10 BEAS: Beam Element

Element Type 15, see reference [7]

- 2 nodes
- $2 \times 6 = 12$ degrees of freedom
- straight
- constant cross section
- offset nodes (i.e. the nodes may be located eccentrically in space)
- deformations considered: bending and shear about the two principal axes, axial deformations and St. Venant torsion
- the transverse load must be located in the shear centre of the beam
- eccentric shear centre
- element loads:
  - load linearly distributed over all, or a part, of the element (see Figure 3.1 b))
  - gravitational load
  - general inertia load
  - initial strain (temperature load)

Element co-ordinate system (reference axes):

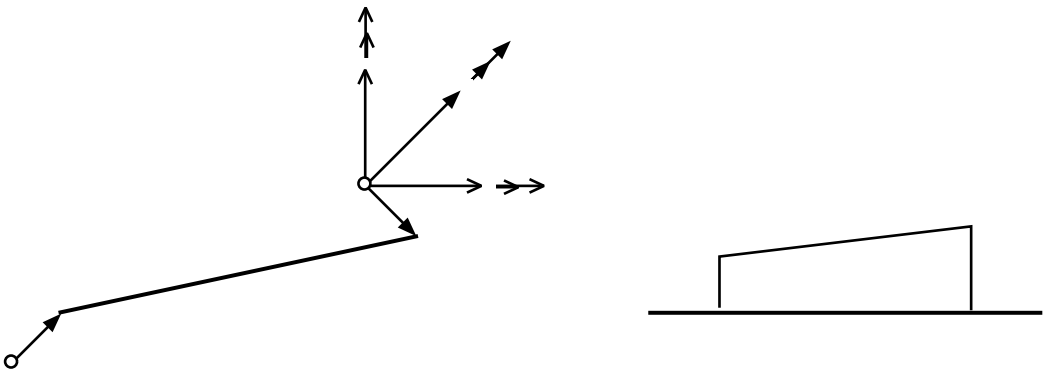


Figure 3.1: a) A beam element b) Linearly distributed load

The local x-axis is directed along the beam, coinciding with the centre of gravity and pointing from node "1" to node "2". The local z-axis is defined on the **GUNIVEC** data type.

Data types used for this element:

Data type	Mandatory	Note
GELMNT1	yes	
GBEAMG	yes	The following data types can be used to specify the cross-section of a BEAS element: **GIORH**, **GUSYI**, **GCHAN**, **GBOX**, **GPIPE**, **GBARM**, **GTONP** or, **GDOBO** for Sestra these data types are transferred to post-processor, and only referred when storing on result file
GELREF1	yes	
MISOSEL	yes	
GUNIVEC	yes	
GECCEN		
BEDRAG1		
BEMASS1		
BELOAD1		
BGRAV		
BNACCLO		
BEISTE		
BELFIX		

Back to Section 3 **ELEMENT TYPES**

3.11 AXIS: Axial Spring

Element Type 16

- 2 nodes
- degrees of freedom at each node: 2, 3 or 6
- axial stiffness

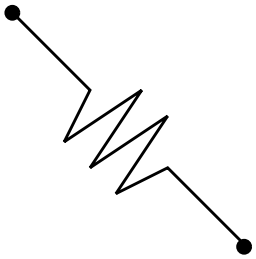


Figure 5-11 Axial spring

Data types used for this element:

GELMNT1 *
GELREF1 *
MAXSPR *

*) Mandatory

Back to Section 3 **ELEMENT TYPES**

3.12 AXDA: Axial Damper

Element Type 17

- 2 nodes
- degrees of freedom at each node: 2, 3 or 6.
- axial damping

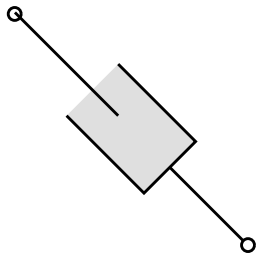


Figure 5-12 Axial damper.

Data types used for this element:

GELMNT1 *
GELREF1 *
MAXDMP *

*) Mandatory

Back to Section 3 **ELEMENT TYPES**

3.13 GSPR: Ground Spring

Element Type 18

- 1 node
- degrees of freedom, arbitrary
- stiffness matrix

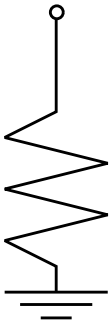


Figure 5-13 Ground spring

Data types used for this element:

GELMNT1 *
GELREF1 *
MGSPRNG * (NDOF must be equal to NDOF on data type GNODE)
BNTRCOS (NDOF = 3 or 6 is required for transformations)

*) Mandatory

Back to Section 3 **ELEMENT TYPES**

3.14 GDAM: Damper to Ground

Element Type 19

- 1 node
- degrees of freedom arbitrary
- damping matrix

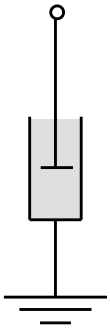


Figure 5-14 Damper to ground

Data types used for this element:

GELMNT1 *
GELREF1 *
MGDAMP * (NDOF must be equal to NDOF on data type GNODE)
BNTRCOS (NDOF = 3 or 6 is required for transformations)

*) Mandatory

Back to Section 3 **ELEMENT TYPES**

3.15 IHEX: Iso-parametric Hexahedron

Element Type 20, see reference [7]

- 20 nodes
- 20 x 3 degrees of freedom
- curved element sides
- isotropic or anisotropic material data
- deformations considered: translational strain
- element loads:
  - initial strain (temperature load)
  - surface forces
  - line loads
  - gravitational load
  - general inertia load

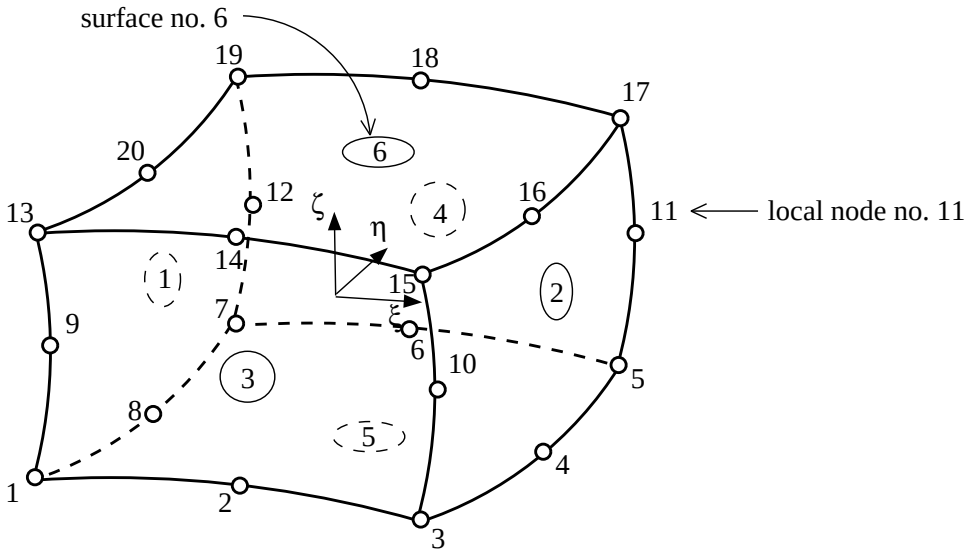


Figure 5-15 Isoparametric hexahedron with local nodenumbering and corresponding surface numbering

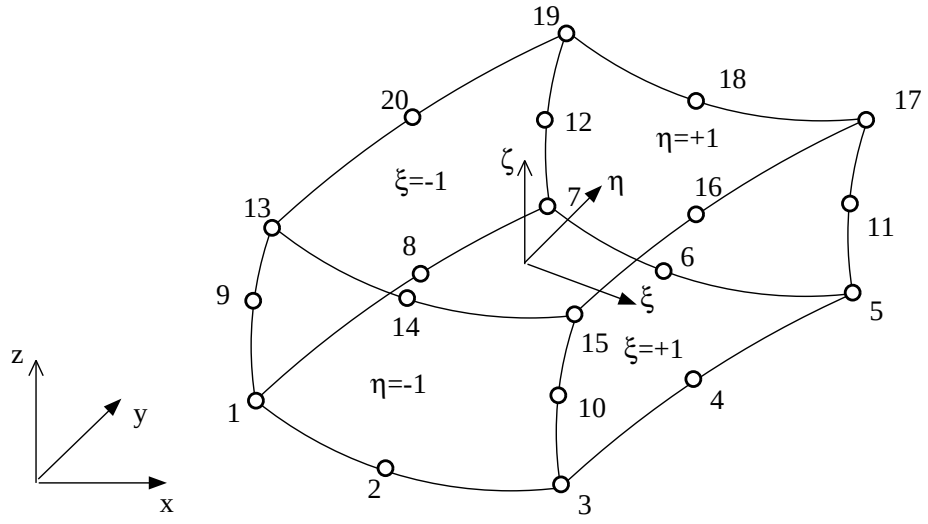


Figure 5-16 Typical isoparametric hexahedron

When surface loads are specified for the element side, the surface numbers shown are used for identification of the side in question.

The local node numbering for each side is defined as follows:

	Node number							
	1	2	3	4	5	6	7	8
Side no.								
1	7	8	1	9	13	20	19	12
2	5	11	17	16	15	10	3	4
3	1	2	3	10	15	14	13	9
4	7	12	19	18	17	11	5	6
5	7	6	5	4	3	2	1	8
6	19	20	13	14	15	16	17	18

When line load is specified, the relation between the local node numbers and the loaded line will be as follows:

	Node number		
	1	2	3
Line no.			
1	1	2	3
2	3	4	5
3	5	6	7
4	7	8	1
5	1	9	13
6	3	10	15
7	5	11	17
8	7	12	19
9	13	14	15
10	15	16	17
11	17	18	19
12	19	20	13
13	8	4	
14	6	2	
15	9	10	
16	2	14	
17	10	11	
18	4	16	
19	11	12	
20	6	18	
21	12	9	
22	8	20	
23	20	16	
24	14	18	

Data types used for this element:

GELMNT1 *
GNODE *
GCOORD *
GELREF1 *
GELSTRP
MISOSEL * or
MISOPL * or
MORSSOL *
MTRSOL
BEUSLO
BELLO2
BEISTE
BGRAV
BNACCLO

*) Mandatory

The following restrictions are put on the data types

GELINT — The integration stations must be distributed according to the Gaussian integration scheme, i.e. INTYPE=1. For this element type the GELINT specifications consist of the first two lines.

Stiffness matrix: For stiffness matrix calculations the number of integration stations will be the same in all coordinate directions and equal to the value specified for N1.
Legal values are 2, 3 and 4.
For a regular element N1 = 2 may cause a singular stiffness matrix.
Default value is 3.

Load calculations: Here, the number of integration stations in each coordinate direction must be specified individually. If volume forces are calculated, legal values for N1, N2 and N3 are 2, 3 and 4.
For surface forces, the legal value for N1, N2 and N3 is 2.
Default value is 2.

Initial strain: The number of integration stations in each coordinate direction must be specified individually.
Legal values for N1, N2 and N3 are 2, 3 and 4.
Default value is 2.

Mass matrix: Again the number of integration stations in each direction must be specified individually.
Legal values for N1, N2 and N3 are 3 and 4.
Default value is 3.

GELSTRP — The stress points must be distributed according to the Gaussian integration scheme, i.e. STRPTYP=1. Only the first two records of the GELSTRP specification are therefore used in this element type.
Legal values for N1, N2 and N3 are 1, 2, 3 and 4.
Default value is 2.

BELLO2 — The SIDE definition is not used. The load components are given nodewise in global coordinates.

BEUSLO — Only one side can be loaded for each BEUSLO record. For the same reason only one side identification may be given in SIDE on the BEUSLO record.

Back to Section 3 **ELEMENT TYPES**

3.16 LHEX: Linear Hexahedron

Element Type 21, see reference [7]

- 8 nodes
- 8 x 3 = 24 degrees of freedom
- linear element sides
- isotropic or anisotropic material data
- deformation considered: translational displacement
- element load
  - gravitational load
  - general inertia load
  - initial strain (temperature load)
  - surface forces
  - line loads

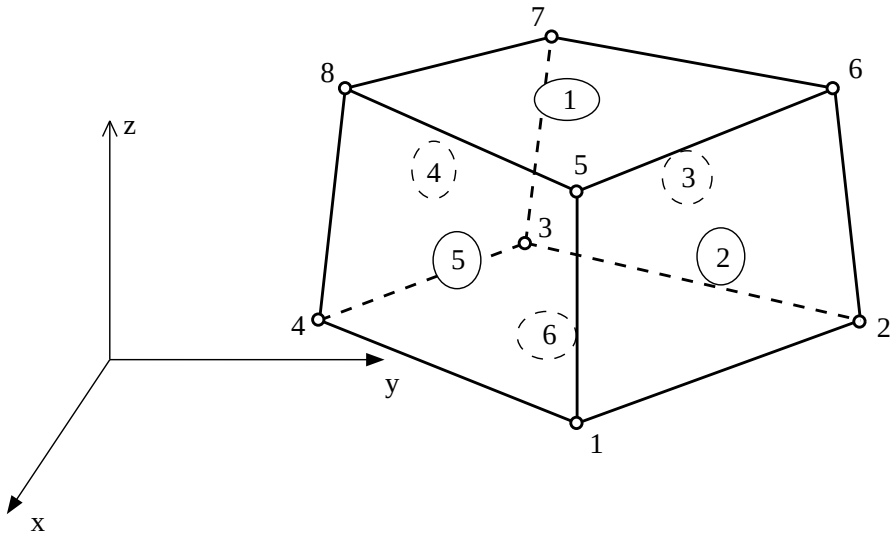


Figure 5-17 Linear hexahedron solid element with local nodenumbering and corresponding surface numbering.

When surface loads are specified for an element side, the surface numbers shown are used for identification of the surface in question.
The local nodenumbering for each side is defined as follows:

	Node number			
	1	2	3	4
Side no.				
1	5	6	7	8
2	1	2	6	5
3	2	3	7	6
4	3	4	8	7
5	4	1	5	8
6	1	4	3	2

When line load is specified, the relation between the local node numbers and the loaded line will be as follows:

	Node number	
	1	2
Line no.		
1	1	2
2	2	3
3	3	4
4	4	1
5	1	5
6	2	6
7	3	7
8	4	8
9	5	6
10	6	7
11	7	8
12	8	5

Data types used for this element:

GELMNT1 *
GNODE *
GCOORD *
GELREF1 *
MISOSEL * or
MORSSOL *
BELLO2
BEUSLO
BEISTE
BGRAV
BNACCLO

*) Mandatory

The following restrictions are put on data type:

GELINT	The integration stations must be distributed according to the Gaussian integration scheme, i.e. INTYPE = 1. For this element type the GELINT specifications consist of only the first line. Stiffness matrix calculation: Number of integration stations will be the same in all coordinate directions and equal to the value specified for N1. Legal values are 2 and 3. Default value is 2. Load calculations: Number of integration stations is the same in all coordinate directions. The only legal value is N1=2. Initial strain: Number of integration stations is the same in all coordinate directions and is given by N1. Legal values are 2 and 3. Default value for N1 is 2. Mass matrix calculation: The same number of integration points in each coordinate direction as in stiffness matrix calculation is also used in mass matrix calculation.
GELSTRP	The stress points must be distributed according to the Gaussian integration scheme, i.e. STRPTYP=1. Number of stress points in each of the coordinate directions are the same and equal to the number specified by N1. Only the first record of the GELSTRP specification is therefore used for this element type. Legal values of N1 are 2 and 3. The default value is 2.
BELLO2	The SIDE definition is not used. The load components are given nodewise in global coordinates.
BEUSLO	An element side may only be loaded once for each BEUSLO-record.

Back to Section 3 **ELEMENT TYPES**

3.17 SECB: Sub-parametric Curved Beam

Element Type 22, see reference [7]

- 3 nodes
- 3 x 6 degrees of freedom
- curved element
- isotropic material data
- deformations considered:
  bending, shear and axial strain
- element loads:
  - line loads
  - gravitational load
  - general inertia load

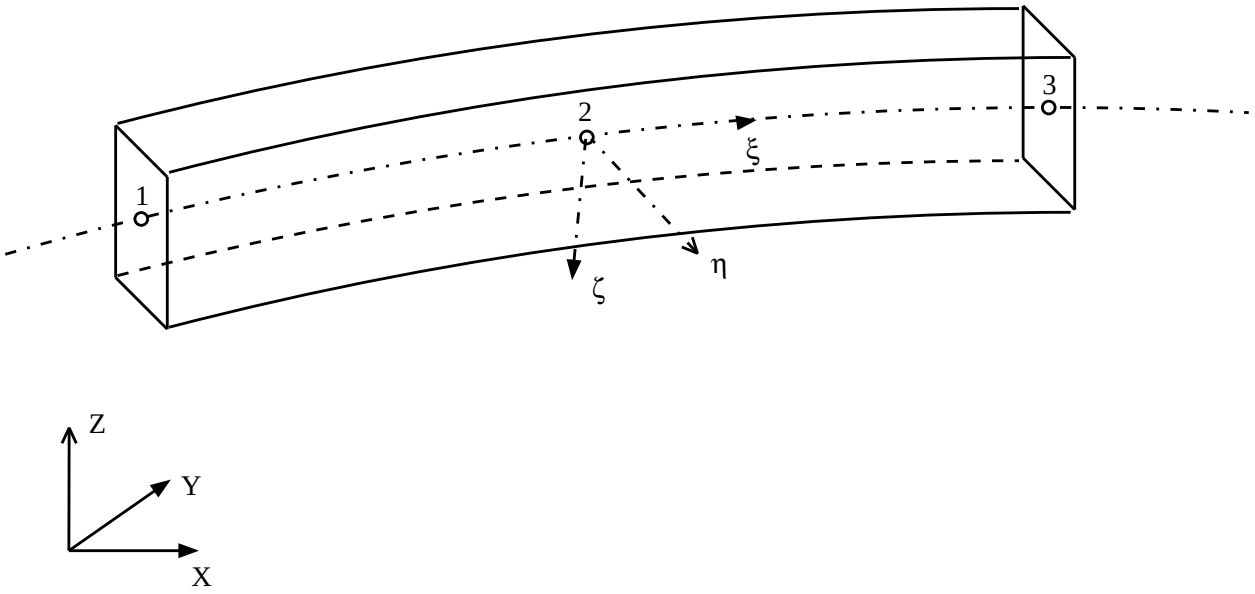


Figure 5-18 Typical beam element
Global cartesian - local curvilinear coordinate system

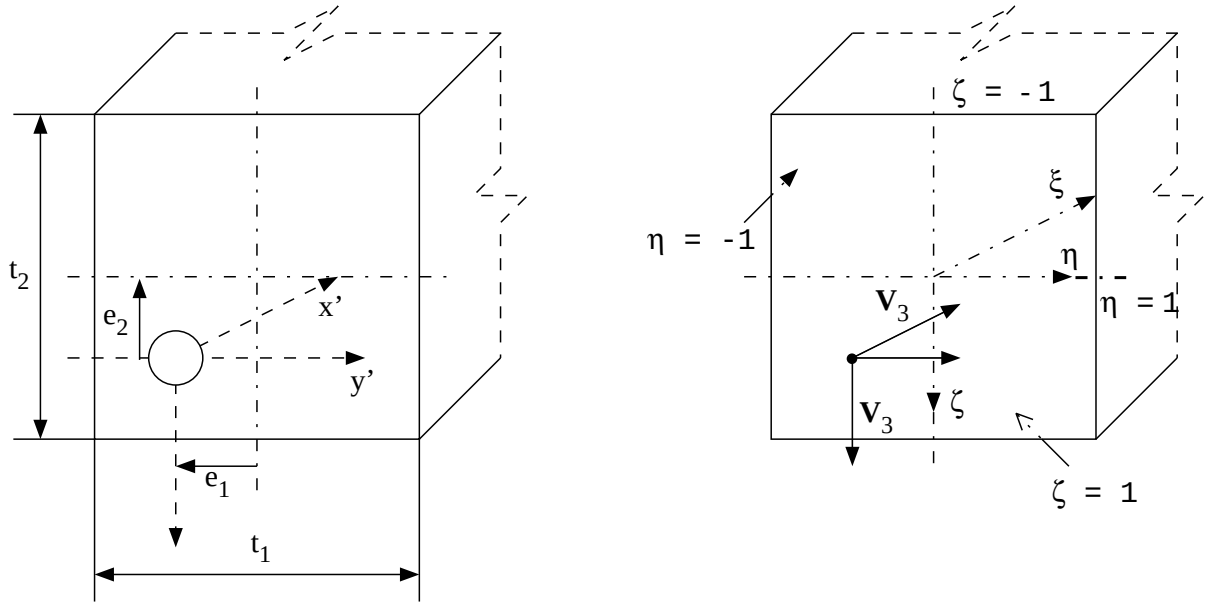


Figure 5-19 Cross-section data, for the beam element.

Data types used for this element:

GELMNT1 *
GELREF1 *
MISOSEL *
GUNIVEC *
GECCEN
GELINT
GBARM *
BELLO2
BGRAV
BNACCLO

*) Mandatory

The following restrictions are put on the data-types.

GECCEN	For this element only eccentricities in the local (h, z)-plane is allowed.
GELINT	The integration stations must be distributed according to the Gaussian integration scheme, i.e. INTYPE=1. For this element type the GELINT specifications consist of the first two lines. Stiffness matrix: Here, the number of integration stations in each coordinate direction must be specified individually. N1 must always be equal to 2. Legal values for N2 and N3 are 1 and 2. In the directions where one integration station is specified, analytical integration is used, else numerical integration is used. Default value for N1, N2 and N3 is 2. Load calculations: Only the number of integration stations in the first coordinate direction is used (line load along beam axis). Legal values are 2, 3 or 4. Default value for N1 is 2. Mass matrix: As for stiffness matrix calculations.
GBARM	Since the element cross-section must be rectangular, only H2I and BT are needed to specify the cross-section geometry at a node.
BELLO2	LINE and SIDE will not be employed for this element.

Back to Section 3 **ELEMENT TYPES**

3.18 BTSS: Sub-parametric General Curved Beam

Element Type 23, see reference [7]

- 3 nodes
- 3 x 6 degrees of freedom
- curved element
- isotropic material data
- constant cross section along the beam
- general cross section
- offset nodes (i.e. the nodes may be located eccentrically in space)
- deformations considered: bending and shear, axial deformations and St.Venant torsion
- the transverse load must be located in the shear centre of the beam
- eccentric shear center
- element loads:
  - line load
  - line moment load
  - gravitational load
  - general inertia load
  - temperature load

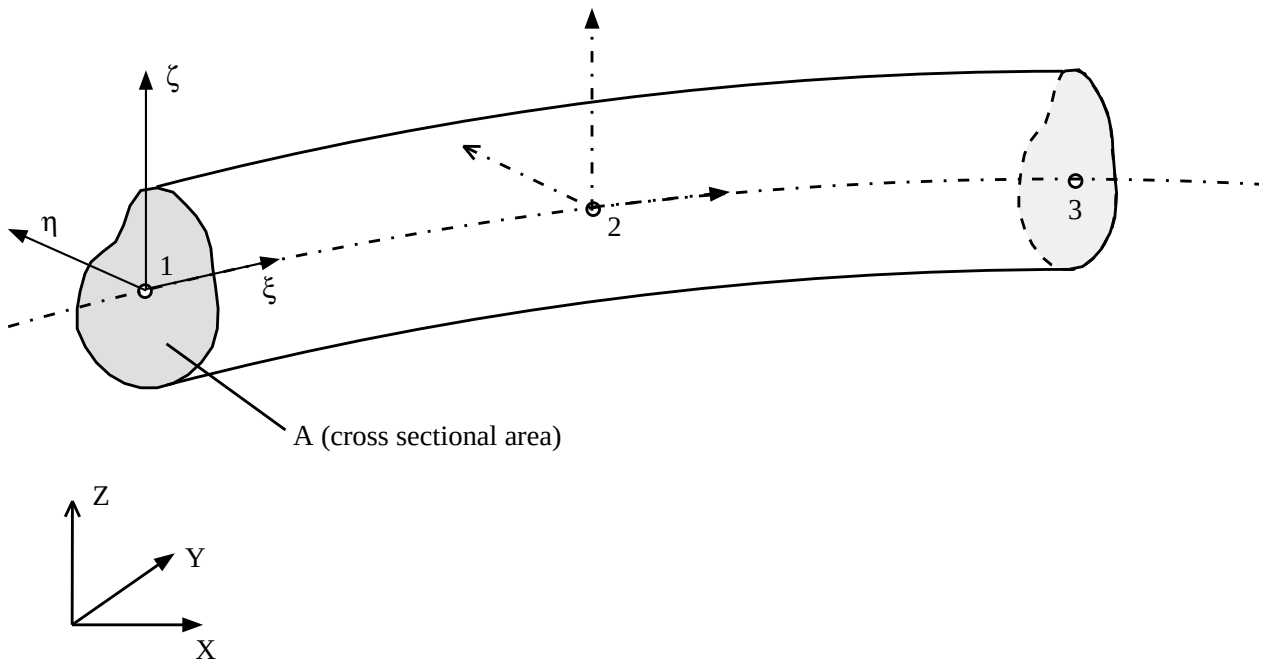


Figure 5-20 Typical general beam element
Global cartesian - local curvilinear coordinate system

Data types used for this element:

GELMNT1 *
GELREF1 *
MISOSEL *
GUNIVEC *
GECCEN
GBEAMG *
GELREF1 *
GIORH, GUSYI, GCHAN, GBOX, GPIPE, GBARM, GTONP or,GDOBO;
for SESTRA these records are transferred to postprocessor, and only referred when storing on result file.

BELLO2
BGRAV
BNACCLO
BEISTE

*) Mandatory

The following restrictions are put on the data-types.

GECCEN	General eccentricities in the local (ξ, η, ζ)-directions are allowed.
GBEAMG	Cross section properties are fetched from this record in SESTRA. Geometry of cross sections specified on other records (GBARM, GIORH etc.) are only transferred to the result file.
BELLO2	LINE and SIDE will not be employed for this element.

Back to Section 3 **ELEMENT TYPES**

3.19 FQUS: Flat Quadrilateral Thin Shell

Element Type 24, see reference [7]

- 4 nodes
- 4 x 5 degrees of freedom
- linear element sides
- isotropic or anisotropic material data
- deformations considered:
  bending, shear and translational strain
- constant element thickness
- element loads:
  - initial strain (temperature loads)
  - surface forces
  - line loads
  - line moment load
  - gravitational load
  - general inertia loade

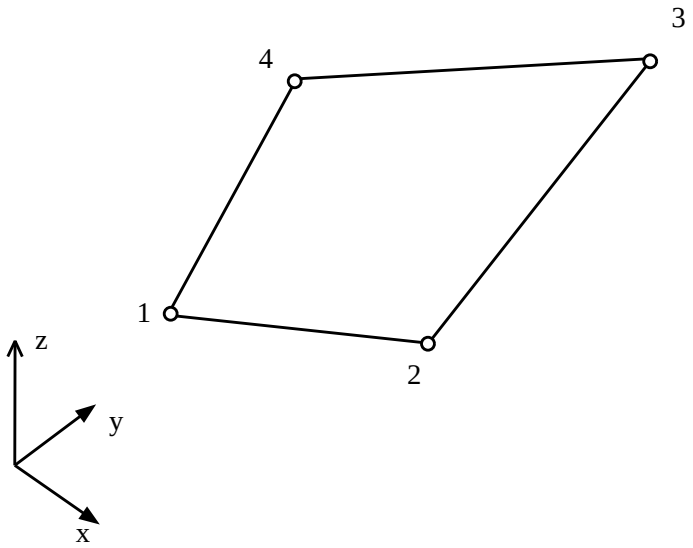


Figure 5-21 Element node numbering on flat quadrilateral shell element.

When line load is specified, the relation between local node numbers and loaded line will be:

LINE =1	means line load between node 1 and 2
LINE =2	means line load between node 2 and 3
LINE =3	means line load between node 3 and 4
LINE =4	means line load between node 4 and 1

Data types for this element:

GELMNT1	*
GNODE	*
GCOORD	*
GELREF1	*
GELTH	*
MISOSEL	* or
MORSMEL	*
MTRMEL	
BEUSLO	
BELLO2	
BEISTE	
BGRAV	
BNACCLO	

*) Mandatory

The following restrictions apply to the data types:

GELREF1 - INTNO and ISPONO are not read because number of integration and stress points are constant in the program and can not be set by user.

- No thickness variation is allowed for this element. If thickness variation is specified by the GEONO / OPT option, the finite element program executing this element must use a mean thickness calculated from the nodal thicknesses.

BELL02 - The SIDE definition is not used.

BEUSL0 - This element type is only able to calculate surface loads which are acting perpendicular to the element surface. For LOTYP=2 (loads given in component form), the in-plane components are ignored by the program.

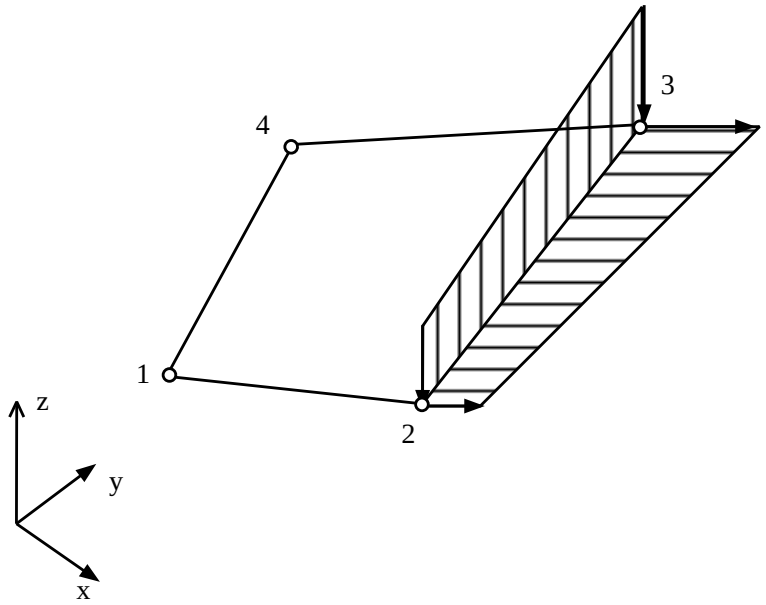


Figure 5-22 Linear variation of line load component normal to the element plane and in the plane.

Back to Section 3 **ELEMENT TYPES**

3.20 FTRS: Flat Triangular Thin Shell

Element Type 25, see reference [7]

- 3 nodes
- 3 x 5 degrees of freedom
- linear element sides
- isotropic or anisotropic material data
- deformations considered:
  bending, shear and translational strain
- constant element thickness
- element loads:
  - initial strain (temperature loads)
  - surface forces
  - line loads
  - line moment load
  - gravitational load
  - general inertia load

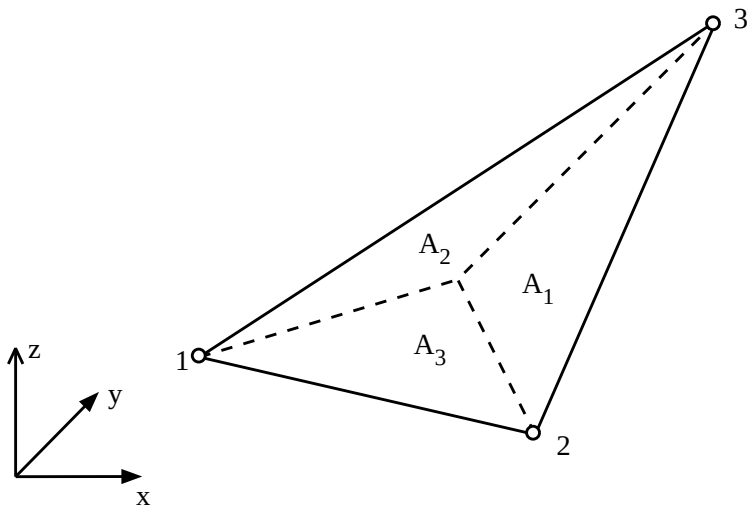


Figure 5-23 Element node numbering and local area coordinates (A_1, A_2 and A_3) on flat triangular shell element.

When line load is specified for one of the element sides, the relation between local node numbers and loaded line will be:

LINE=1	means line load between node 2 and 3
LINE=2	means line load between node 1 and 3
LINE=3	means line load between node 1 and 2

Data types used for this element:

GELMNT1	*
GNODE	*
GCOORD	*
GELREF1	*
GELTH	*
MISOSEL	*
MORSMEL	*
MTRMEL	
BEUSLO	
BELLO2	
BEISTE	
BGRAV	
BNACCLO	

*) Mandatory

The following restrictions apply to the data types:

GELREF1 - INTNO and ISPONO are not read because number of integration and stress points are constant in the program and can not be set by user.

- GEONO / OPT should be set>0 because no thickness variation is allowed for this element type.

BELLO2 - The SIDE definition is not used.

BEUSLO - This element type is only able to calculate surface loads which are acting perpendicular to the element surface. For LOTYP=2 (loads given in component form), the in-plane components are ignored by the program.

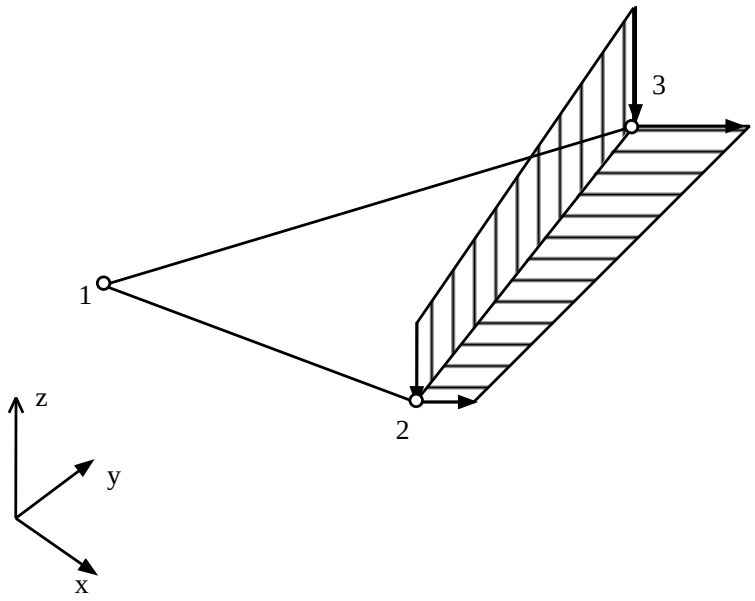


Figure 5-24 Constant line load in element plane and linear variation of component normal to the plane

Back to Section 3 **ELEMENT TYPES**

3.21 SCTS: Sub-parametric Curved Triangular Shell

Element Type 26, see reference [7]

- 6 nodes
- 6 x 6 degrees of freedom
- curved element shape
- isotropic or anisotropic material data
- deformations considered:
  bending, shear and translational strain
- parabolically varying element thickness
- element loads:
  - initial strain
  - surface forces
  - line loads
  - line moment load
  - gravitational load
  - general inertia load

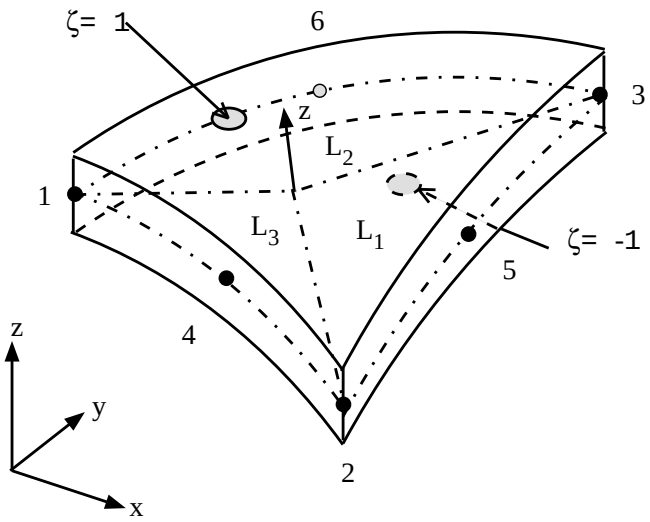


Figure 5-25 Element node numbering. Global Cartesian and local area coordinate system.

When line load is specified for one of the element sides, the relation between local node numbers and loaded line will be:

LINE=1 means line load on the element side defined by nodes 2, 5 and 3
LINE=2 means line load on the element side defined by nodes 1, 6 and 3
LINE=3 means line load on the element side defined by nodes 1, 4 and 2

The orientation of the local nodal coordinate system is related to the nodenumbering sequence (see Figure 5-25). The positive z-direction, normal to the element middle surface, is chosen according to the normal convention of the right hand rule, i.e. the positive ζ-direction is found when the node numbers are followed in ascending order, counterclockwise.

The SIDE definition on the BEUSLO records is as follows:

SIDE=1 means that load is given on the element surface where ζ=-1,
SIDE=2 that ζ=0 is loaded, and
SIDE=3 that ζ=1 is loaded.

Data types for this element:

GELMNT1 *
GNODE *
GCOORD *
GELREF1 *
GELINT
GELTH *
GELSTRP
MISOSEL * or
MORSSEL *
MTRSEL
BEUSLO
BELLO2
BEISTE
BGRAV
BNACCLO

*) Mandatory

The following restrictions are put on the data types.

GELINT	The integration stations must be distributed according to a scheme similar to the Gaussian integration scheme, i.e. INTYPE=1. For this element type the GELINT specifications consist of the first line only. The integration stations will be specified in the triangle plane, and not along local coordinate axes. The value specified for N1 is **not** the number of integration stations but a reference number to specified distributions of integration stations, which may have the same number of integration stations, but different positions in the triangle plane. Therefore, N3 has no meaning for this element type. Legal values for N1 are 1, 2, 3, 4, 5, 6, 7, 8, 9, see description of subroutine HAMC30 in the finite element library, /2/. N2 ("ζ-direction") is not possible to specify and the value 2 is used for each layer. The integration in ζ-direction will be performed analytically if max deviation from mean thickness is 5% and the element is nearly flat. Stiffness matrix: Default value for N1 is 7. Load calculations: Default value for N1 is 6 for surface loads. Initial strain: Default value for N1 is 6. Mass matrix: N1 should be specified ³ 7 to ensure positive definite mass matrix. Default value is N1=7.
GELSTRP	The stress points must be distributed according to the same scheme as the integration points in the GELINT specification, for stress point coordinates in the triangular plane. The stress point coordinates in the ζ-direction are distributed according to the usual Gaussian integration scheme STRPTYP=1. Legal values for N1 are 1, 3, 4, 7, 9, and for N2 2, 3, 4. Default values for N1 is 3 and for N2 the default value is 2. Only the first record of the GELSTRP specification is used for this element type.
BELLO2	The SIDE-definition is not used.

Back to Section 3 **ELEMENT TYPES**

3.22 MCTS: Sub-parametric Multi-layered Curved Triangular Shell

Element Type 27

- 6 nodes
- 6 x 6 degrees of freedom
- curved element shape
- isotropic or anisotropic material data
- two or more material layers (sandwich)
- deformations considered:
  bending, shear and translational strain
- parabolically varying element thickness
- element loads:
  - initial strain
  - surface forces
  - line loads
  - line moment load
  - gravitational load
  - general inertia load

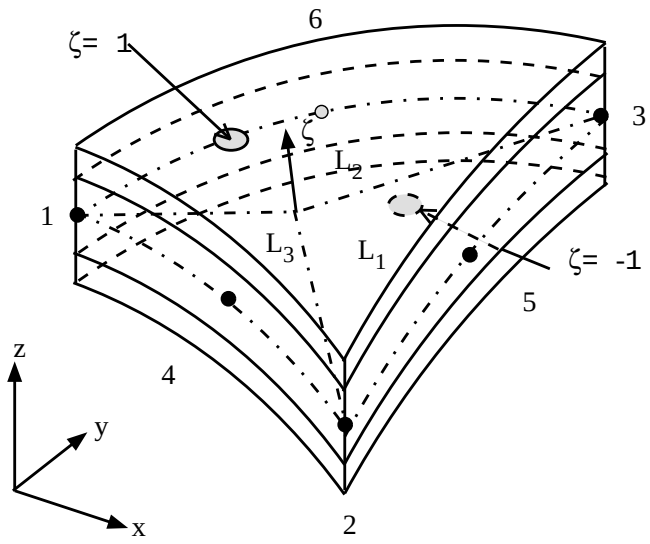


Figure 5-26 Element node numbering. Global Cartesian and local area coordinate system. Three layers.

When line load is specified for one of the element sides, the relation between local node numbers and loaded line will be:

LINE=1 means line load on the element side defined by nodes 2, 5 and 3
LINE=2 means line load on the element side defined by nodes 1, 6 and 3
LINE=3 means line load on the element side defined by nodes 1, 4 and 2

The orientation of the local nodal coordinate system is related to the nodenumbering sequence (see Figure 5-26). The positive ζ-direction, normal to the element middle surface, is chosen according to the normal convention of the right hand rule, i.e. the positive ζ-direction is found when the node numbers are followed in ascending order.

The SIDE definition on the BEUSLO records is as follows:

SIDE=1 means that load is given on the element surface where ζ=-1,
SIDE=2 that ζ=0 is loaded, and
SIDE=3 that ζ=1 is loaded.

Data types for this element:

GELMNT1 *
GNODE *
GCOORD *
GELREF1 *
GELINT
GELTH *
GELSTRP
MORSSEL *
MTRSEL
BEUSLO
BELLO2
BEISTE
BGRAV
BNACCLO

*) Mandatory

The following restrictions are put on the data types.

GELINT — The integration stations must be distributed according to a scheme similar to the Gaussian integration scheme, i.e. INTYPE=1. For this element type the GELINT specifications consist of the first line only. The integration stations will be specified in the triangle planes, and not along local coordinate axes. The value specified for N1 is **not** the number of integration stations but a reference number to specified distributions of integration stations, which may have the same number of integration stations, but different positions in the triangle planes. Therefore, N3 has no meaning for this element type. Legal values for N1 are 1, 2, 3, 4, 5, 6, 7, 8, 9, see description of subroutine HAMC30 in the finite element library, /2/. N2 ("ζ-direction") is not possible to specify and the value 2 is used for each layer. The integration in ζ-direction will be performed analytically if max deviation from mean thickness is 5% and the element is nearly flat.

Stiffness matrix: Default value for N1 is 7.

Load calculations: Default value for N1 is 6 for surface loads.

Initial strain: Default value for N1 is 6.

Mass matrix: N1 should be specified ³ 7 to ensure positive definite mass matrix. Default value is N1=7.

GELSTRP — The stress points must be distributed according to the same scheme as the integration points in the GELINT specification, for stress point coordinates in the triangular plane. The stress point coordinates in the ζ-direction are distributed according to the usual Gaussian integration scheme STRPTYP=1. Legal values for N1 are 1, 3, 4, 7, 9, and for N2 2, 3, 4.
Default values for N1 is 3 and for N2 ("z-direction") the default value is 2.
Only the first record of the GELSTRP specification is used for this element type.

BELLO2 — The SIDE-definition is not used.

Back to Section 3 **ELEMENT TYPES**

3.23 SCQS: Sub-parametric Curved Quadrilateral Shell

Element Type 28, see reference [7]

- 8 nodes
- 8 x 6 degrees of freedom
- curved element shape
- isotropic or anisotropic material data
- deformations considered:
  bending, shear and translational strain
- parabolically varying element thickness
- element loads:
  - initial strain
  - surface forces
  - line loads
  - line moment load
  - gravitational load
  - general inertia load

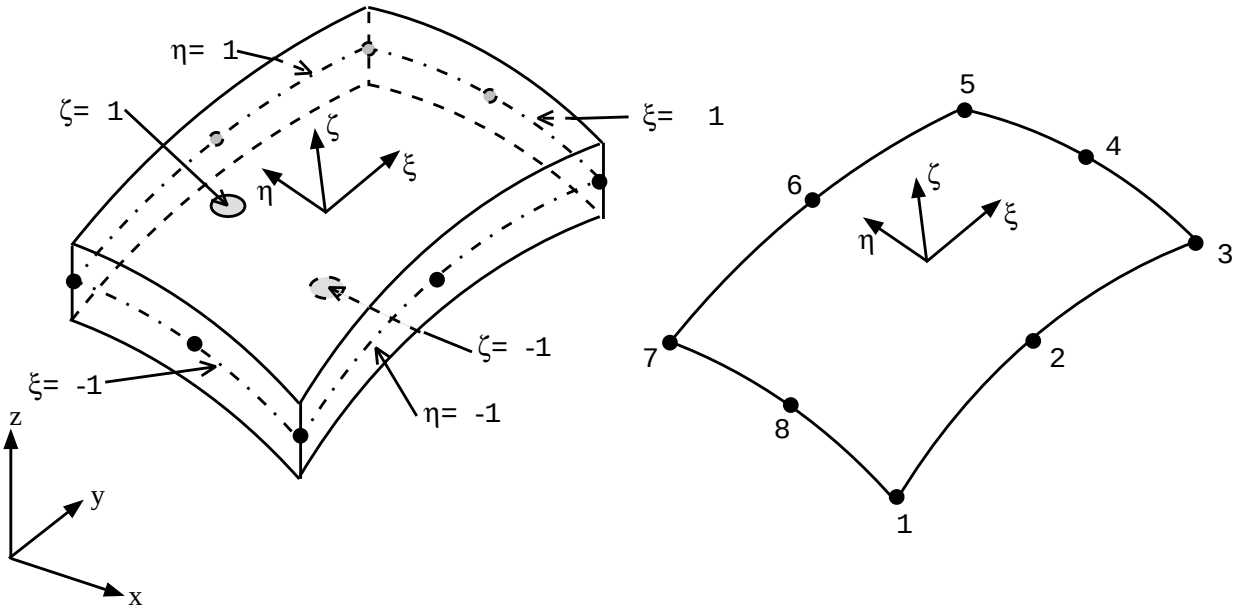


Figure 5-27 Element node numbering. Global Cartesian and local normalized coordinate system.

When line load is specified, the relation between local node numbers and loaded line will go:

LINE=1 means line load along the line defined by the nodes 3, 4 and 5
LINE=2 means line load along the line defined by the nodes 2 and 6
LINE=3 means line load along the line defined by the nodes 1, 8 and 7
LINE=4 means line load along the line defined by the nodes 7, 6 and 5
LINE=5 means line load along the line defined by the nodes 8 and 4
LINE=6 means line load along the line defined by the nodes 1, 2 and 3

The orientation of the local nodal coordinate system is related to the nodenumbering sequence (see Figure 5-27). The positive ζ-direction, normal to the element middlesurface, is chosen according to the normal convention of the right hand rule, i.e. the positive ζ-direction is found when the node numbers are followed in ascending order.

The SIDE definition on the BEUSLO records is as follows:

SIDE=1 means that load is given on the element surface where ζ= -1,
SIDE=2 that ζ= 0 is loaded, and
SIDE=3 that ζ= 1 is loaded.

Data types for this element:

GELMNT1 *
GNODE *
GCOORD *
GELREF1 *
GELINT
GELTH *
GELSTRP
MISOSEL * or
MISOPL * or
MORSSEL *
MTRSEL
BEUSLO
BELLO2
BEISTE
BGRAV
BNACCLO

*) Mandatory

The following restrictions are put on the data types.

GELINT	The integration stations must be distributed according to the Gaussian integration scheme, i.e. INTYPE=1. For this element type only the first line of the GELINT specifications is used. N3 ("ζ-direction") is not possible to specify and the value 2 is used. The integration in ζ-direction will be performed analytically if max deviation from mean thickness is 5% and the element is nearly flat. Stiffness matrix: Legal values for N1 and N2 are 2 and 3. Default value is 2 for both. Load calculations: Legal values for N1 and N2 are 2, 3 and 4. Default value is 2 for both. Initial strain: Legal values for N1 and N2 are 2 and 3. Default value is 2 for both. Mass matrix: Legal values for N1 and N2 are 1, 2, 3 and 4. To ensure positive definite mass matrix N1=N2=4 is recommended. Default value is N1=N2=4.
GELSTRP	The stress points must be distributed according to the Gaussian integration scheme, i.e. STRPTYP=1. The only legal value for N1 and N2 is 2. N3 may be 2, 3 or 4. This gives 4*N3 stress points within each element.
BELLO2	The SIDE-definition is not used.

Back to Section 3 **ELEMENT TYPES**

3.24 MCQS: Sub-parametric Multilayered Curved Quadrilateral Shell

Element Type 29

- 8 nodes
- 8 x 6 degrees of freedom
- curved element shape
- isotropic or anisotropic material data
- two or more material layers (sandwich)
- deformations considered:
  bending, shear and translational strain
- parabolically varying element thickness
- element loads:
  - initial strain
  - surface forces
  - line loads
  - line moment load
  - gravitational load
  - general inertia load

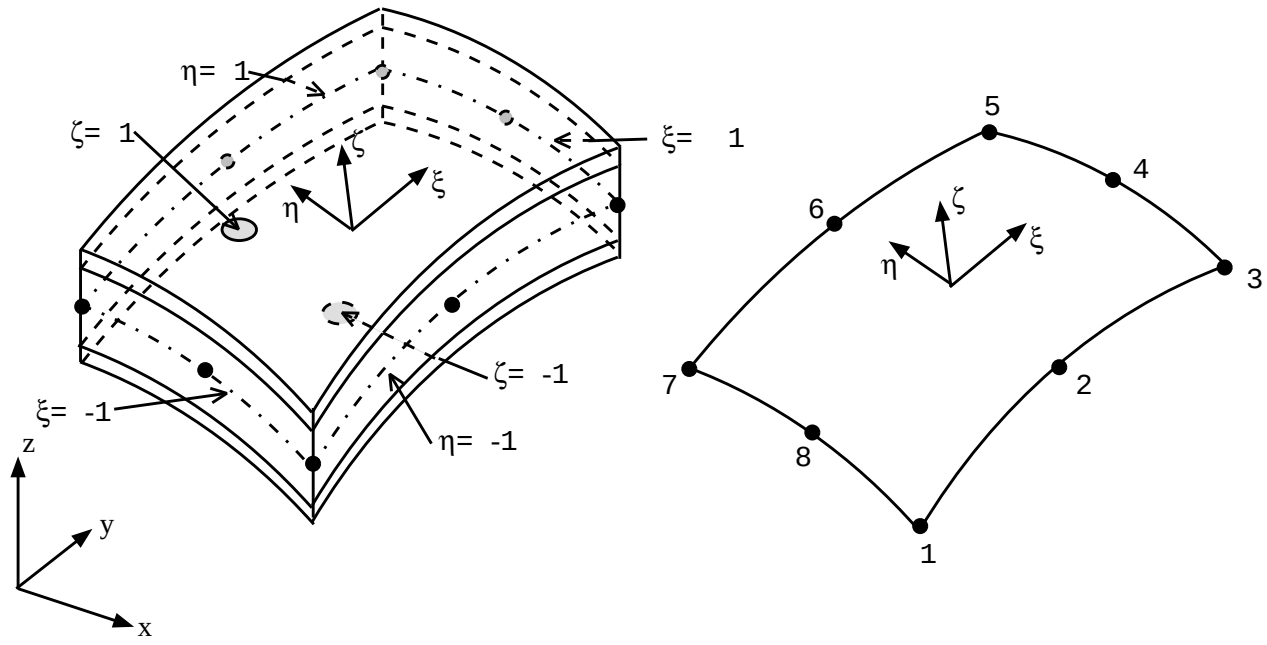


Figure 5-28 Element node numbering. Global Cartesian and local normalized coordinate system. Three layers.

When line load is specified, the relation between local node numbers and loaded line will go:

LINE=1 means line load along the line defined by the nodes 3, 4 and 5
LINE=2 means line load along the line defined by the nodes 2 and 6
LINE=3 means line load along the line defined by the nodes 1, 8 and 7
LINE=4 means line load along the line defined by the nodes 7, 6 and 5
LINE=5 means line load along the line defined by the nodes 8 and 4
LINE=6 means line load along the line defined by the nodes 1, 2 and 3

The orientation of the local nodal coordinate system is related to the nodenumbering sequence (see Figure 5-28). The positive ζ-direction, normal to the element middlesurface, is chosen according to the normal convention of the right hand rule, i.e. the positive ζ-direction is found when the node numbers are followed in ascending order, counterclockwise.

The SIDE definition on the BEUSLO records is as follows:

SIDE=1 means that load is given on the element surface where ζ = -1,
SIDE=2 that ζ= 0 is loaded, and
SIDE=3 that ζ= 1 is loaded.

Data types for this element:

GELMNT1 *
GNODE *
GCOORD *
GELREF1 *
GELINT
GELTH *
GELSTRP
MORSSEL *
MTRSEL
BEUSLO
BELLO2
BEISTE
BGRAV
BNACCLO

*) Mandatory

The following restrictions are put on the data types.

GELINT — The integration stations must be distributed according to the Gaussian integration scheme, i.e. INTYPE=1. For this element type only the first line of the GELINT specifications is used. N3 ("ζ-direction") is not possible to specify and the value 2 is used for each layer. The integration in ζ-direction will be performed analytically if max deviation from mean thickness is 5% and the element is nearly flat.

Stiffness matrix: Legal values for N1 and N2 are 2 and 3.
Default value is 2 for both.

Load calculations: Legal values for N1 and N2 are 2, 3 and 4.
Default value is 2 for both.

Initial strain: Legal values for N1 and N2 are 2 and 3.
Default value is 2 for both.

Mass matrix: Legal values for N1 and N2 are 1, 2, 3 and 4. To ensure positive definite mass matrix N1=N2=4 is recommended.
Default value is N1=N2=4.

GELSTRP — The stress points must be distributed according to the Gaussian integration scheme, i.e. STRPTYP=1. The only legal value for N1 and N2 is 2. N3 may be 2, 3 or 4. This gives 4*N3 stress points within each layer of the element.

BELLO2 — The SIDE-definition is not used.

Back to Section 3 **ELEMENT TYPES**

3.25 IPRI: Iso-parametric Prism

Element Type 30, see reference [7]

- 15 nodes
- 15 x 3 degrees of freedom
- curved element sides
- isotropic or anisotropic material data
- deformations considered: translational strain
- element loads:
  - initial strain (temperature load)
  - surface forces
  - line forces
  - gravitational load
  - general inertia load

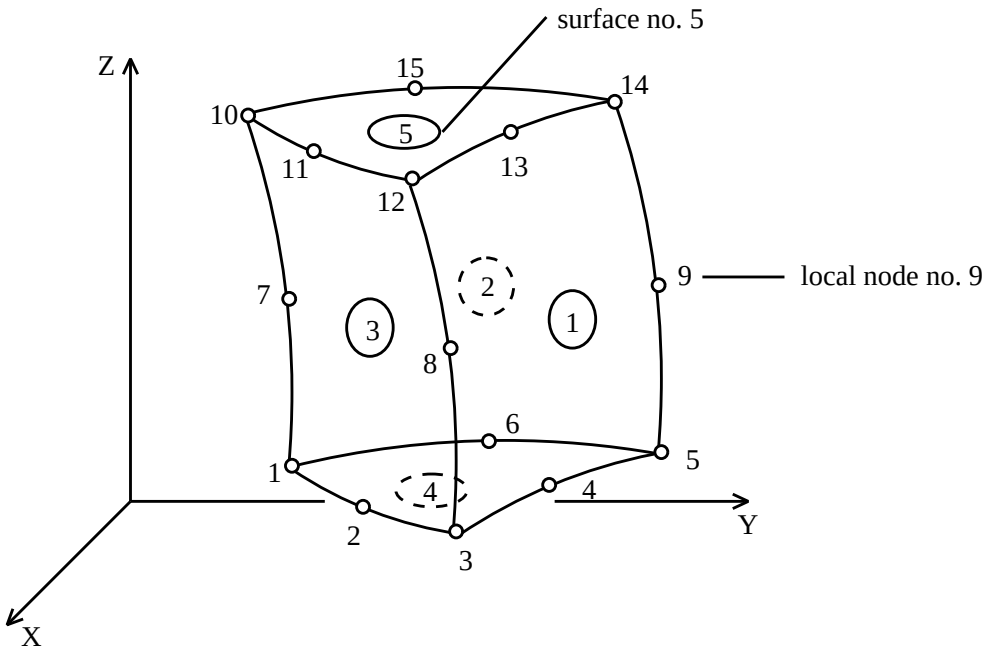


Figure 5-29 Isoparametric triangular prism with local nodenumbering and corresponding surface numbering.

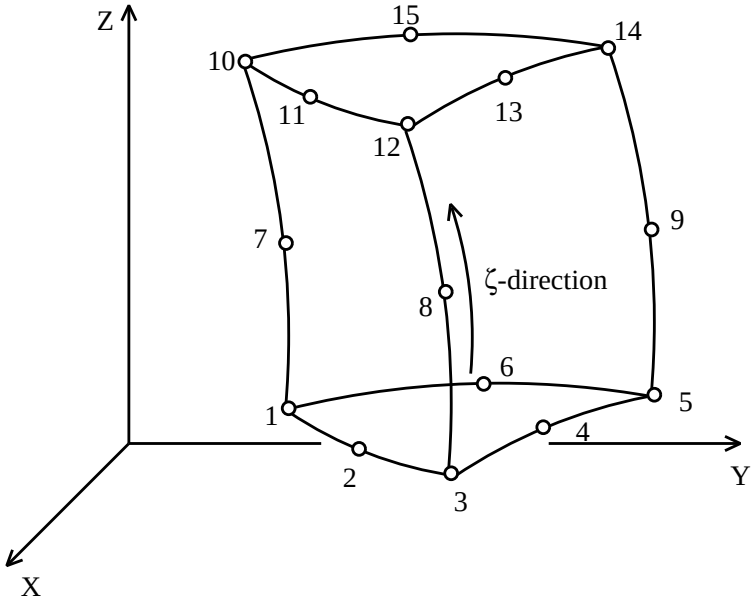


Figure 5-30 Typical isoparametric triangular prism with local node numbering

When surface loads are specified for an element side, the surface numbers shown are used for identification of the surface in question.

The local nodenumbering for each side is defined as follows:

	Node number							
	1	2	3	4	5	6	7	8
Side no.								
1	3	4	5	9	14	13	12	8
2	5	6	1	7	10	15	14	9
3	1	2	3	8	12	11	10	7
4	1	2	3	4	5	6		
5	14	13	12	11	10	15		

When line load is specified, the relation between the local node numbers and the loaded line will be as follows:

	Node number		
	1	2	3
Line no.			
1	1	2	3
2	3	4	5
3	5	6	1
4	1	7	10
5	3	8	12
6	5	9	14
7	10	11	12
8	12	13	14
9	14	15	10
10	7	8	
11	2	11	
12	8	9	
13	4	13	
14	9	7	
15	6	15	

Data types used for this element:

GELMNT1	*	
GNODE	*	
GCOORD	*	
GELREF1	*	
GELINT		
GELSTRP		
MISOSEL	*	or
MORSSOL	*	
MTRSOL		
BEUSLO		
BELLO2		
BEISTE		
BGRAV		
BNACCLO		

*) Mandatory

The following restrictions are put on the data type.

GELINT — The integration stations must be distributed according to the Gaussian integration scheme, i.e. INTYPE=1. For this element type the GELINT specifications consist of the two first lines. For the triangle plane the integration stations will be distributed according to the usual Gaussian integration scheme. N1 will be employed for the specification of number of integration stations in the triangular plane; N2 for the number of integration stations in the ζ-direction. N3 will be employed for the specification of number of integration stations in the direction perpendicular to the ζ-direction in the four edged surfaces, used in the calculation of surfaces, used in the calculation of surface loads.

Stiffness matrix: Legal values for N1 are 3, 4 and 7, and for N2 legal values are 2, 3 and 4. Default values are N1=7 and N2=3. N3 is not used in this case.

Load calculations: For surface loads N3 will be employed as described above. Legal values for N3 are 1, 2, 3 and 4. Default value is N3=3. For surface loads legal values for N1 are 3, 4 and 7, and for N2 2, 3 and 4. Default values are N1=7 and N2=3.

Initial strain: Only N2 is used in this case. Legal values for N2 are 1, 2, 3 and 4. Default value is N2=3.

Mass matrix: Only N2 is used in this case. Legal values for N2 are 3 and 4. Default value is N2=3.

GELSTRP — The stress points must be distributed according to the same scheme as the integration points in the GELINT specification, i.e. STRPTYP=1. Legal values for N1 are 1, 4 and 7 and for N2 1, 2 and 3. Default values are N1=4 and N2=2. Only the first record of the GELSTRP specification is used for this element type.

BEUSLO — A element side may only be loaded once for each BEUSLO-record.

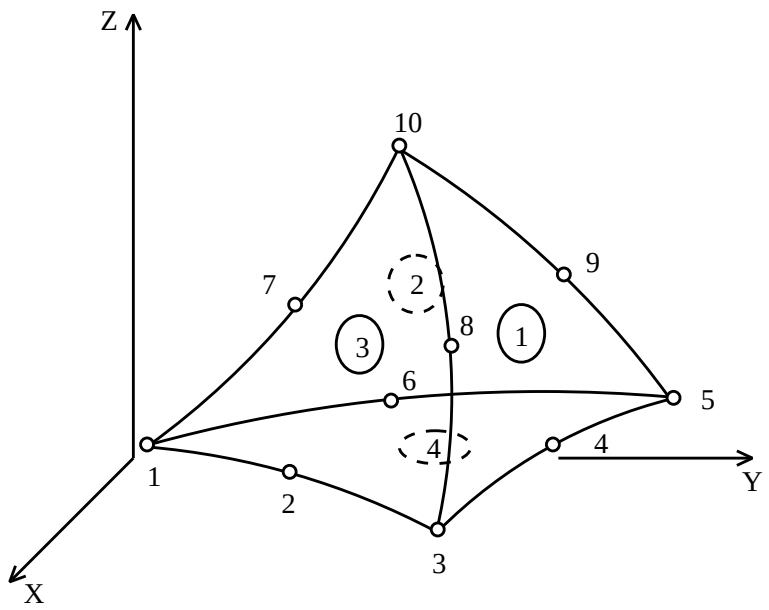
BELLO2 — The SIDE definition is not used. The load components are given nodewise in global coordinates.

Back to Section 3 **ELEMENT TYPES**

3.26 ITET: Iso-parametric Tetrahedron

Element Type 31, see reference [7]

- 10 nodes
- 10 x 3 degrees of freedom
- curved element sides
- isotropic or anisotropic material data
- deformations considered: translational strain
- element loads:
  - initial strain (temperature load)
  - surface forces
  - line forces
  - gravitational load
  - general inertia load



In surface No. 1 we have L_1=0 and so on for surface No. 2, 3 and 4. See "volume coordinates" in /3/

Figure 5-31 Isoparametric tetrahedron with local nodenumbering and corresponding surface numbering.

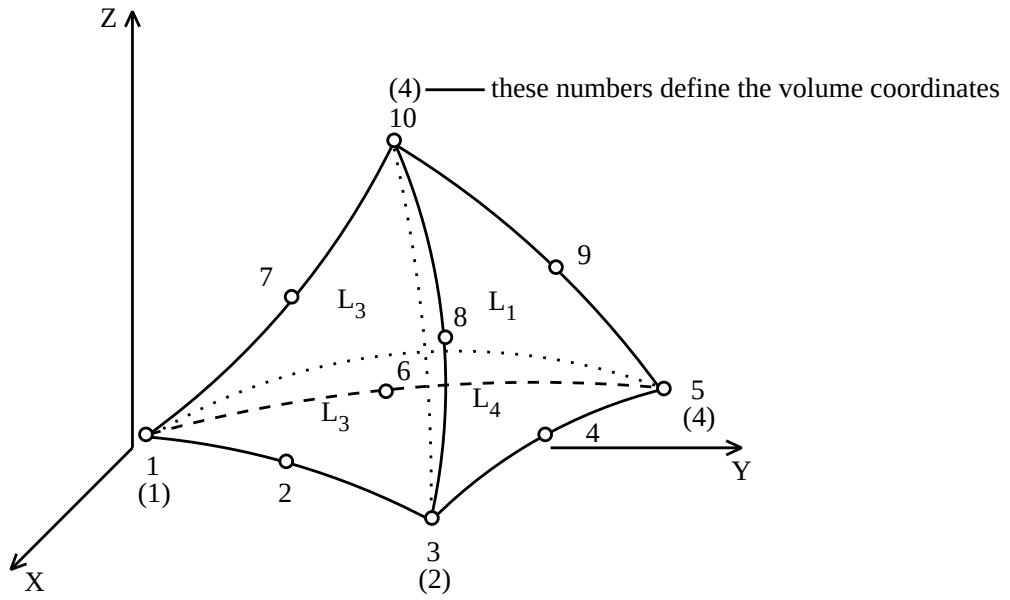


Figure 5-32 Typical isoparametric tetrahedron with node numbering and numbers defining the volume coordinates L_1, L_2, L_3, L_4.

When surface loads are specified for an element side, the surface numbers shown are used for identification of the side in question.

The local nodenumbering for each side is defined as follows:

	Node number					
	1	2	3	4	5	6
Side no.						
1	3	4	5	9	10	8
2	5	6	1	7	10	9
3	1	2	3	8	10	7
4	1	6	5	4	3	2

When line load is specified, the relation between local node numbers and loaded line will be as follows:

	Node number		
	1	2	3
Line no.			
1	1	2	3
2	3	4	5
3	5	6	1
4	1	7	10
5	3	8	10
6	5	9	10

Data types for this element:

GELMNT1 *
GNODE *
GCOORD *
GELREF1 *
GELINT
GELSTRP
MISOSEL * or
MORSSOL *
MTRSOL
BEUSLO
BEISTE
BGRAV
BNACCLO

*) Mandatory

The following restrictions are put on the data types:

GELINT	The integration stations must be distributed according to a scheme similar to the Gaussian integration scheme, i.e. INTYPE=1. Only the first record of the GELINT specification is used for these element types. The number of integration stations specified for N1 covers the whole volume of the tetrahedron. N2 will be used to specify the number of integration points in the triangular surfaces. The specification of integration points in the triangular surfaces is used in surface load calculations. N1 must always be specified equal to 5. Load calculations: Legal values for N2 are 1, 2, 3, 4 and 7. Default value is 4.
GELSTRP	The stress points must be distributed according to the same scheme as the integration points in the GELINT specification, i.e. STRPTYP=1. Legal values for N1 are 1, 4 and 5. Default value is 4.
BELLO2	The SIDE-definition is not used. The load components are given nodewise in global coordinates.

Back to Section 3 **ELEMENT TYPES**

3.27 TPRI: Triangular Prism

Element Type 32, see reference [7]

- 6 nodes
- 6 x 3 degrees of freedom
- linear element sides
- isotropic or anisotropic material data
- deformation considered: translational displacement
- element loads
  - initial strain (temperature loads)
  - volume forces
  - surface forces
  - line forces
  - gravitational load
  - general acceleration load

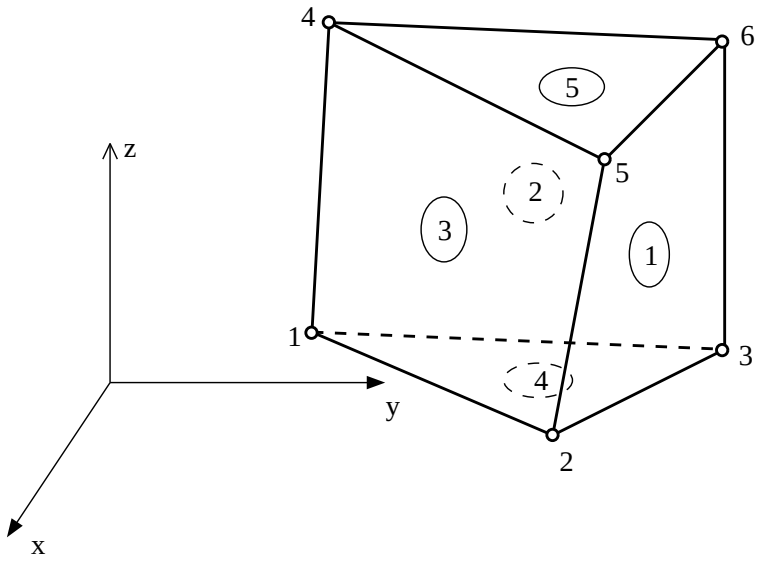


Figure 5-33 Triangular prism solid element with local nodenumbering and corresponding surface numbering.

When surface loads are specified for an element side, the surface numbers shown are used for identification of the surface in question.
The local nodenumbering for each side is defined as follows:

	Node number			
	1	2	3	4
Side no.				
1	2	3	6	5
2	3	1	4	6
3	1	2	5	4
4	1	2	3	0
5	6	5	4	0

When line load is specified, the relation between the local node numbers and the loaded line will be as follows:

	Node number	
	1	2
Line no.		
1	1	2
2	2	3
3	3	1
4	1	4
5	2	5
6	3	6
7	4	5
8	5	6
9	6	4

Data types used for this element:

GELMNT1 *
GNODE *
GCOORD *
GELREF1 *
MISOSEL * or
MORSSOL *
MTRSOL
BELLO2
BEUSLO
BEISTE
BGRAV
BNACCLO

*) Mandatory

The following restrictions are put on data type:

GELINT	The integration stations must be distributed according to the Gaussian integration sceme, i.e. INTYPE=1. For this element type the GELINT specifications consist of only the first line. For the triangular plane the integration stations will be specified in the plane and not along local coordinate axis. In the z-direction, integration stations will be distributed according to the usual Gaussian integration shceme. N1 will be used for specification of number of integration points in the triangular plane, N2 for the number of integration stations in the z-direction. Stiffness matrix calculation: Legal values for N1 are 3, 4 and 7, for N2, 2, 3 and 4. Default values are N1=4 and N2=3. Load calculation: For surface loads N1 will not be used. Number of integration stations in z-direction is given by N2 as before. Legal values of N2 are 2, 3 and 4, with 3 as default. Number of integration stations normal to the z-direction is set equal to one less than N2. This gives a default value equal to 2, and with legal values 1, 2 and 3. Initial strain: Legal values for N1 are 3, 4 and 7, and for N2, 2, 3 and 4. Default values are N1=4 and N2=3. Mass matrix calculations: The only legal values for N1 and N2 are N1=4 and N2=3.
GELSTRP	The stress points must be distributed according to the Gaussian integration scheme, i.e. STRPTYP=1. Number of stress points in the triangular planes is given by N1. Legal values are 1, 3 and 4. The default value of N1 is 1. Number of stress points in the z-direction is specified by N2. The only legal value of N2 is 2.
BEUSLO	An element side may only be loaded once for each BEUSLO-record.
BELLO2	The SIDE definition is not used. The load components are given nodewise in global coordinates.

Back to Section 3 **ELEMENT TYPES**

3.28 TETR: Tetrahedron

Element Type 33, see reference [7]

- 4 nodes
- 4 x 3 degrees of freedom
- linear element sides
- isotropic or anisotropic material data
- deformation considered: translational displacement
- element loads
  - initial strain (temperature loads)
  - volume forces
  - surface forces
  - line forces
  - gravitational load
  - general inertia load

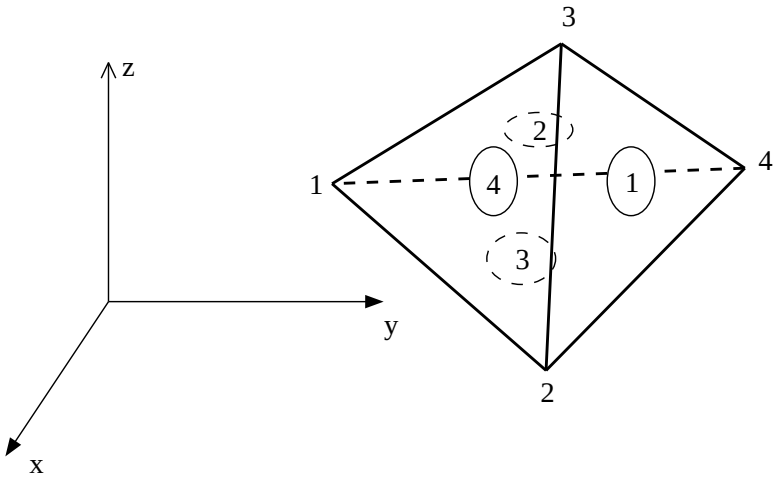


Figure 5-34 Tetrahedron solid element with local node numberieng and corresponding surface numbering.

When surface loads are specified for an element side, the surface numbers shown are used for identification of the surface in question. The local nodenumbering for each side is defined as follows:

	Node number		
	1	2	3
Side no			
1	3	2	4
2	1	3	4
3	2	1	4
4	1	2	3

When line load is specified, the relation between the local node numbers and the loaded line will be as follows:

	Node number	
	1	2
Line no.		
1	1	2
2	2	3
3	3	1
4	1	4
5	2	4
6	3	4

Data types used for this element:

GELMNT1	*	
GNODE	*	
GCOORD	*	
GELREF1	*	
MISOSEL	*	or
MORSSOL	*	
BEUSLO		
BEEO2		
BEISTE		
BGRAV		
BNACCLO		

*) Mandatory

The following restrictions are put on data type:

GELINT	Only the default values for number of integration points are used. This means that if GELINT is given, N1 must be specified equal to the default value for the calculation type in question: Stiffness matrix: The centroid of the element is used as the only calculation point. Load calculation: Constant volume force is only calculated in the centroid of the element. Surface forces in arbitrary direction are calculated in three points on the loaded side, and constant normal pressure is calculated in the midpoint on the loaded side. Initial strain: The centroid of the element is used as the only calculation point. Mass matrix: Four integration points are used. These are distributed according to the Gaussian integration scheme.
GELSTRIP	Stresses are only calculated in the centroid of the element. If the GELSTRP specification is given, only the first record is used. This must be given with STRPTYP=1 and N1=1. N2 should not be specified.
BEUSLO	Only one side can be loaded in each load case. This means that only one side identification can be given in SIDE. The element can only reproduce a constant strain situation. Variation in loads will cause a stress situation which the element is not able to reproduce. Therefore this element should only be used for constant loads, i.e. the load intensity should be the same for all the nodes on the loaded side.
BELLO2	The SIDE definition is not used. The load components are given nodewise in global coordinates.

Back to Section 3 **ELEMENT TYPES**

3.29 LCTS: Sub-parametric Layered Curved Triangular Shell

Element Type 34

- 6 nodes
- 6 x 6 degrees of freedom
- curved element shape
- isotropic or anisotropic material data (anisotropic only, in ’plate layers’)
- one or more material layers
- the layers may be eccentric plate layers and / or eccentric stiffener layers with bar stiffeners in one arbitrary direction. Each layer may have different stiffener direction
- deformations considered:
  bending, shear and translational strain
- constant element thickness
- element loads:
  - initial strain
  - surface forces
  - line loads
  - line moment load
  - gravitational load
  - general inertia load

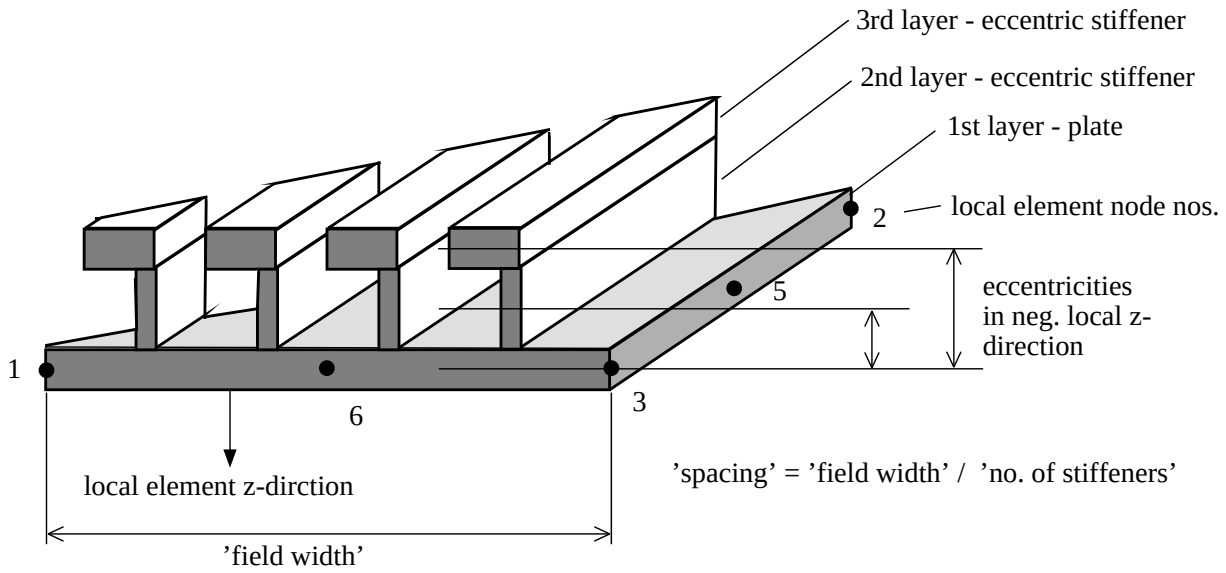


Figure 5-35 Element local node numbering. Three layers - one plate layer and two stiffener layers.

The stiffeners are transformed to layers with no stiffness in the direction lateral to the stiffener direction, and smeared stiffness in the stiffener direction. The direction defining the local element coordinate axes (local x-,

y- and z-axes) are specified on a BNTRCOS record for this element. Only the x-direction (first line) in the direction cosine matrix defined on the BNTRCOS record is used.

The projection of this vector onto the middle of the element surface defines the local x-axis in different points on the element. The positive local z-direction, normal to the element middlesurface, is chosen according to the normal convention of the right hand rule, i.e. the positive z-direction is found when the local element node numbers are followed in ascending order. The BNTRCOS record is referred on the GELREF1 record.

The material may be general anisotropic in the plate layer(s) of the element, but if stiffener layers are specified, they must have isotropic material. For anisotropic materials, each layer must have a separate anisotropic material specification (MORSSEL record).

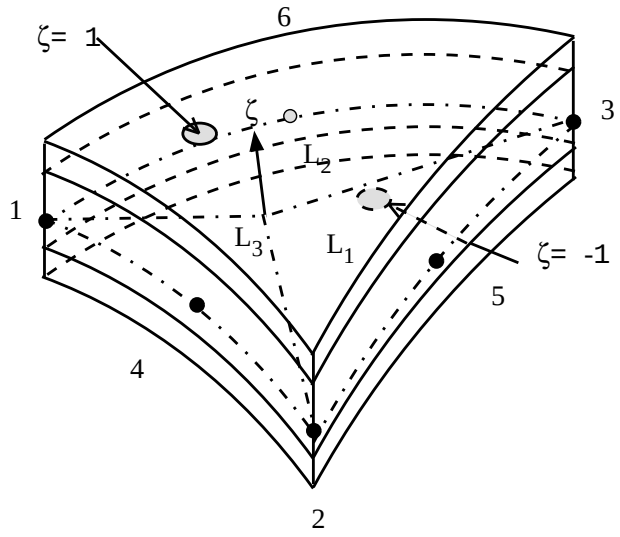


Figure 5-36 Element local node numbering and local normalised coordinate system.

When line load is specified, the relation between local node numbers and loaded line will go:

LINE=1	means line load on the element side defined by nodes 2, 5 and 3
LINE=2	means line load on the element side defined by nodes 1, 6 and 3
LINE=3	means line load on the element side defined by nodes 1, 4 and 2

The orientation of the local normalised coordinate system is related to the local node numbering sequence (see Figure 5-36). The positive ζ-direction, normal to the element middlesurface, is chosen according to the normal convention of the right hand rule, i.e. the positive ζ-direction is found when the node numbers are followed in ascending order.

The SIDE definition on the BEUSLO records is as follows:

SIDE=1	means that load is given on the element surface where ζ= -1,
SIDE=2	that ζ= 0 is loaded, and
SIDE=3	that ζ= 1 is loaded.

Data types for this element:

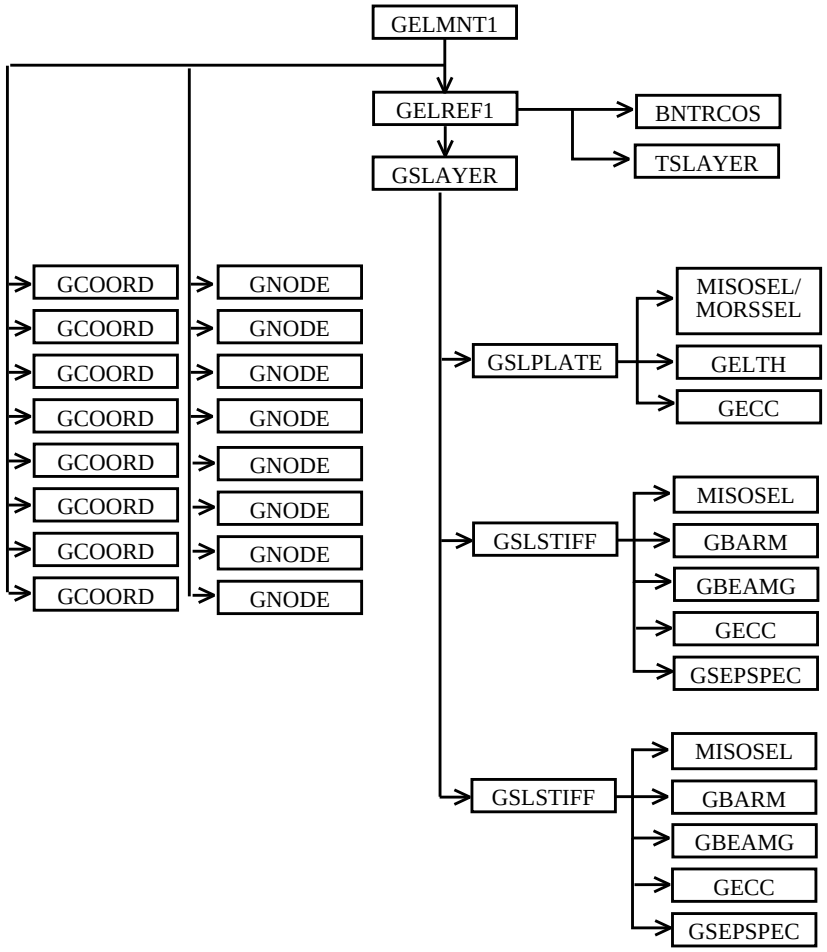


Figure 5-37 Normal data reference for a Subparametric Layered Curved Triangular Thin / Thick Shell . The layered element in the figure is having three layers, one plate layer and two eccentric stiffener layers.

GELMNT1	GELSTRP	BEUSLO
GNODE	GELTH	BELLO2
GCOORD	GSLAYER	BEISTE
GBARM	GSLPLATE	BGRAV
GBEAMG	GSLSTIFF	BNACCLO
GECC	GSEPSPEC	TSLAYER
GELREF1	MISOSEL	
GELINT	MORSSEL	

The following restrictions are put on the data types.

GELINT — The integration stations must be distributed according to a scheme similar to the Gaussian integration scheme, i.e. INTYPE=1. For this element type the GELINT specifications consist of the first line only. The integration stations will be specified in the triangle planes, and not along local coordinate axes. The value specified for N1 is **not** the number of integration stations but a reference number to specified distributions of integration stations, which may have the same number of integration stations, but different positions in the triangle planes. Therefore, N3 has no meaning for this element type. Legal values for N1 are 1, 2, 3, 4, 5, 6, 7, 8, 9, see description of subroutine HAMC30 in the finite element library, /2/. N2 ("ζ-direction") is not possible to specify and the value 2 is used for each layer. The integration in ζ-direction will be performed analytically if max deviation from mean thickness is 5% and the element is nearly flat.

Stiffness matrix: Default value for N1 is 7.

Load calculations: Default value for N1 is 6 for surface loads.

Initial strain: Default value for N1 is 6.

Mass matrix: N1 should be specified ³ 7 to ensure positive definite mass matrix. Default value is N1=7.

GELSTRP — The stress points must be distributed according to the same scheme as the integration points in the GELINT specification, for stress point coordinates in the triangular plane. The stress point coordinates in the ζ-direction are distributed according to the usual Gaussian integration scheme STRPTYP=1. Legal values for N1 are 1, 3, 4, 7, 9, and for N2 2, 3, 4.
Default values for N1 is 3 and for N2 ("z-direction") the default value is 2.
Only the first record of the GELSTRP specification is used for this element type.

BELLO2 — The SIDE-definition is not used.

Back to Section 3 **ELEMENT TYPES**

3.30 LCQS: Sub-parametric Layered Curved Quadrilateral Shell

Element Type 35

- 8 nodes
- 8 x 6 degrees of freedom
- curved element shape
- isotropic or anisotropic material data (anisotropic only, in ’plate layers’)
- one or more material layers
- the layers may be eccentric plate layers and / or eccentric stiffener layers with bar stiffeners in one arbitrary direction. Each layer may have different stiffener direction
- deformations considered:
  bending, shear and translational strain
- constant element thickness
- element loads:
  - initial strain
  - surface forces
  - line loads
  - line moment load
  - gravitational load
  - general inertia load

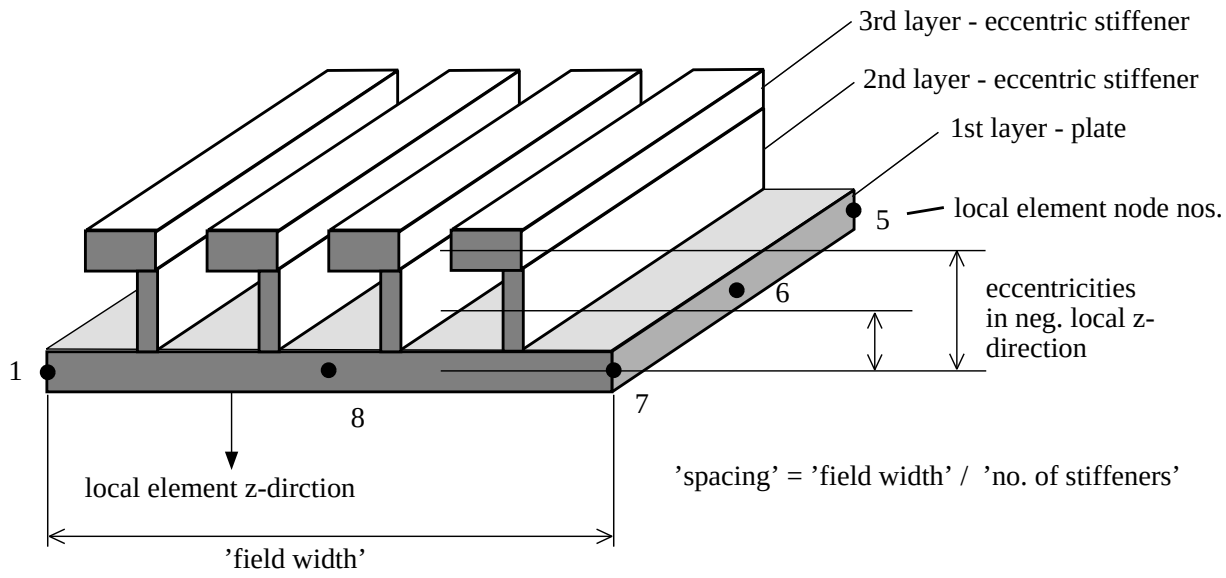


Figure 5-38 Element local node numbering. Three layers - one plate layer and two stiffener layers.

The stiffeners are transformed to layers with no stiffness in the direction lateral to the stiffener direction, and smeared stiffness in the stiffener direction. The direction defining the local element coordinate axes (local x-, y- and z-axes) are specified on a BNTRCOS record for this element. Only the x-direction (first line) in the direction cosine matrix defined on the BNTRCOS record is used.

The projection of this vector onto the middle of the element surface defines the local x-axis in different points on the element. The positive local z-direction, normal to the element middlesurface, is chosen according to the normal convention of the right hand rule, i.e. the positive z-direction is found when the local element node numbers are followed in ascending order. The BNTRCOS record is referred on the GELREF1 record.

The material may be general anisotropic in the plate layer(s) of the element, but if stiffener layers are specified, they must have isotropic material. For anisotropic materials, each layer must have a separate anisotropic material specification (MORSSEL record).

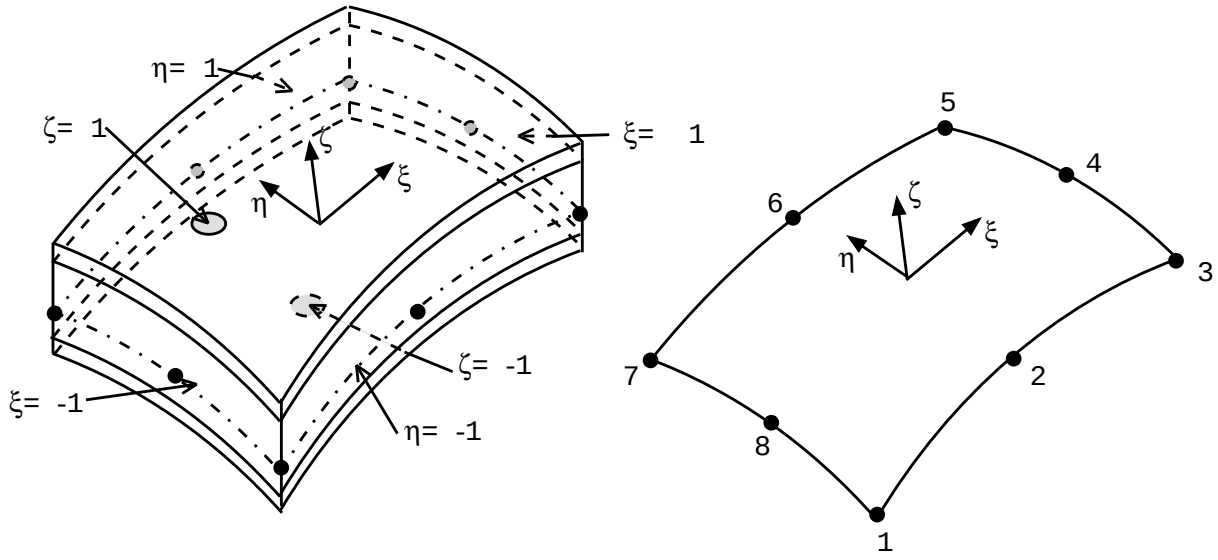


Figure 5-39 Element local node numbering and local normalised coordinate system.

When line load is specified, the relation between local node numbers and loaded line will go:

LINE=1	means line load along the line defined by the nodes 3, 4 and 5
LINE=2	means line load along the line defined by the nodes 2 and 6
LINE=3	means line load along the line defined by the nodes 1, 8 and 7
LINE=4	means line load along the line defined by the nodes 7, 6 and 5
LINE=5	means line load along the line defined by the nodes 8 and 4
LINE=6	means line load along the line defined by the nodes 1, 2 and 3

The orientation of the local normalised coordinate system is related to the local node numbering sequence (see Figure 5-39). The positive ζ-direction, normal to the element middlesurface, is chosen according to the normal convention of the right hand rule, i.e. the positive ζ-direction is found when the node numbers are followed in ascending order.

The SIDE definition on the BEUSLO records is as follows:

SIDE=1	means that load is given on the element surface where ζ= -1,
SIDE=2	that ζ= 0 is loaded, and
SIDE=3	that ζ= 1 is loaded.

Data types for this element:

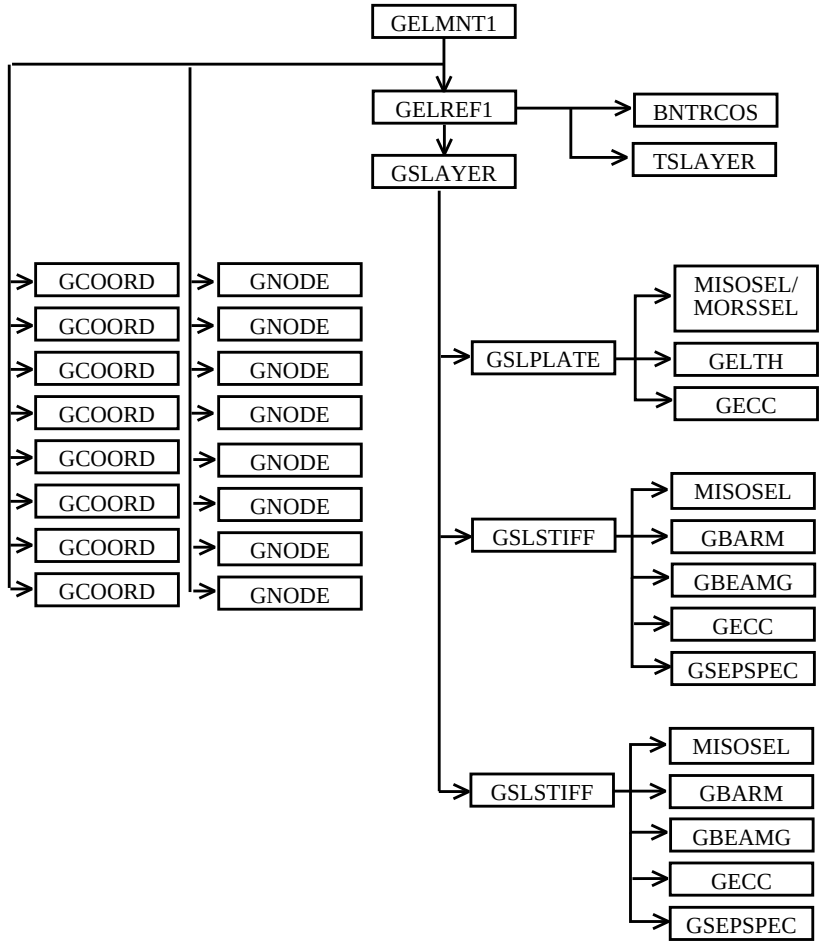


Figure 5-40 Normal data reference for a Subparametric Layered Curved Quadrilateral Thin / Thick Shell . The layered element in the figure is having three layers, one plate layer and two eccentric stiffener layers.

GELMNT1	GELSTRP	BEUSLO
GNODE	GELTH	BELLO2
GCOORD	GSLAYER	BEISTE
GBARM	GSLPLATE	BGRAV
GBEAMG	GSLSTIFF	BNACCLO
GECC	GSEPSPEC	TSLAYER
GELREF1	MISOSEL	
GELINT	MORSSEL	

The following restrictions are put on the data types.

GELINT — The integration stations must be distributed according to the Gaussian integration scheme, i.e. INTYPE=1. For this element type only the first line of the GELINT specifications is used. N3 ("ζ-direction") is not possible to specify and the value 2 is used for each layer. The integration in ζ-direction will be performed analytically if max deviation from mean thickness is 5% and the element is nearly flat.

Stiffness matrix: Legal values for N1 and N2 are 2 and 3.
Default value is 2 for both.

Load calculations: Legal values for N1 and N2 are 2, 3 and 4.
Default value is 2 for both.

Initial strain: Legal values for N1 and N2 are 2 and 3.
Default value is 2 for both.

Mass matrix: Legal values for N1 and N2 are 1, 2, 3 and 4. To ensure positive definite mass matrix N1=N2=4 is recommended.
Default value is N1=N2=4.

GELSTRP — The stress points must be distributed according to the Gaussian integration scheme, i.e. STRPTYP=1. The only legal value for N1 and N2 is 2. N3 may be 2, 3 or 4. This gives 4*N3 stress points within each layer of the element.

BELLO2 — The SIDE-definition is not used.

Back to Section 3 **ELEMENT TYPES**

3.31 TRSI: Transition Elements between Solids and Shells

Element Type 36, 37, or 38, see reference [3]

- 18, 15 or 12 nodes
- 57, 54 or 51 degrees of freedom
- curved or linear element sides
- parabolically varying element thickness (in relevant nodes of ELTYP=37 and 38 only)
- isotropic or anisotropic material data
- deformations considered: translational strain
- element loads:
  - initial strain (temperature loads)
  - surface forces
  - line forces
  - gravitational load
  - general inertia load

The TRSI elements are described in detail in /6/

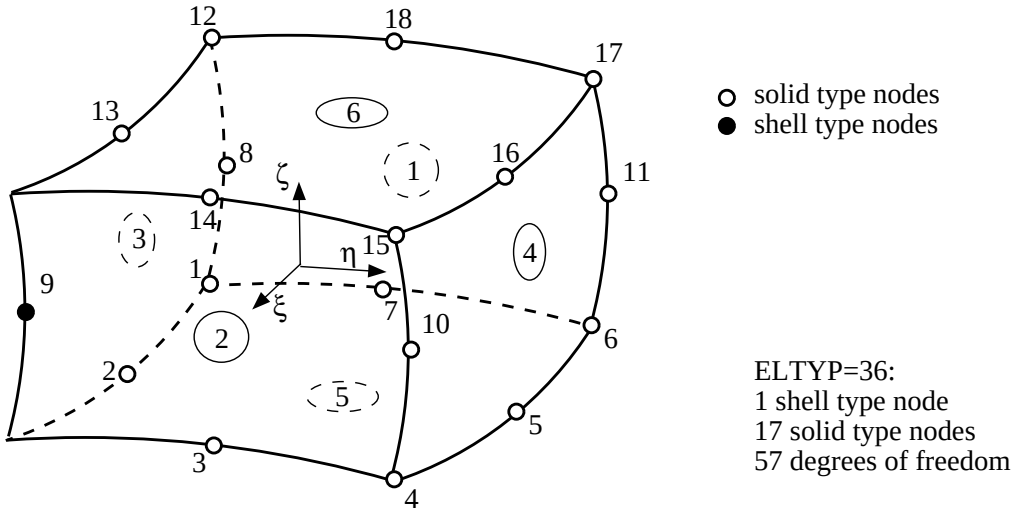


Figure 5-41 Local node numbering and corresponding surface numbering for transition element with one "shell type node", the "shell type node" is node number 9.

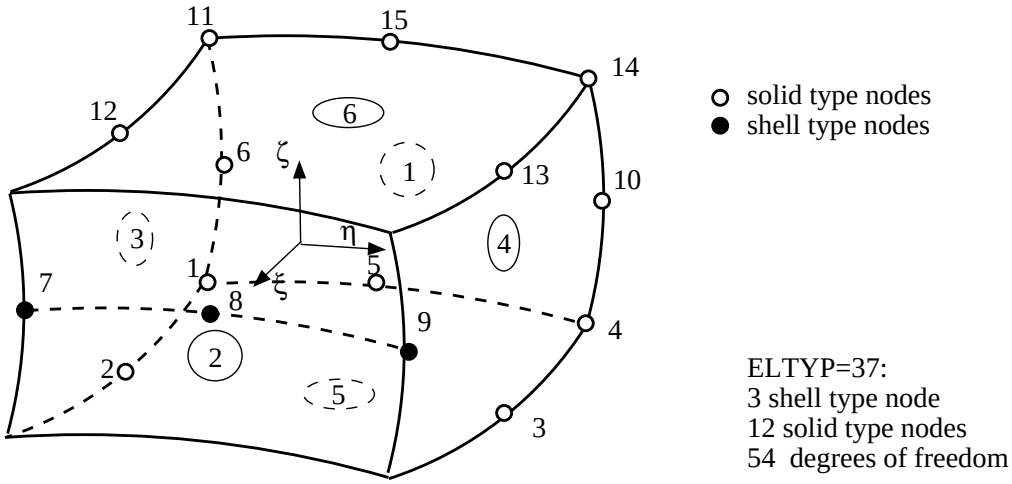


Figure 5-42 Local node numbering and corresponding surface numbering for transition element with three "shell type nodes", the "shell type nodes" is node number 7, 8 and 9.

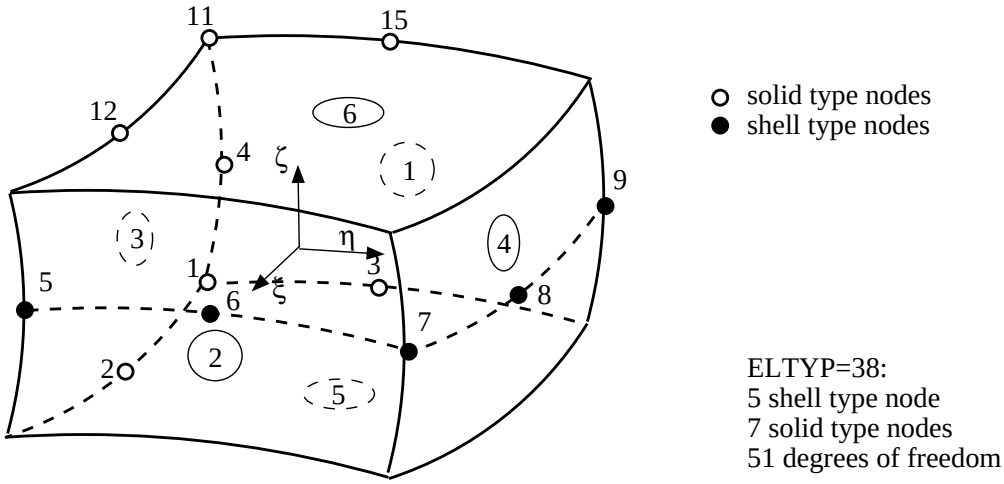


Figure 5-43 Local node numbering and corresponding surface numbering for transition element with five "shell type nodes", the "shell type nodes" are nodes number 5 through 9.

Loads on the element are specified as if given for an IHEX element (ELTYP=20) (the so-called originating IHEX element) with that element's number of nodes, type of nodes (3 translational degrees of freedom) and sequence of nodes specifying the originating element. (see the tables and figures below for TRSI(36), TRSI(37) and TRSI(38)).
When surface load is specified for one element side, the load intensities are always given in eigth points on each side, regardless of this beeing a "solid side" or a "shell side". These points are the nodes on the "solid sides", and the points on the upper and lower element side corresponding to the nodes on the "shell sides". The points on the upper and lower shell element side is marked with the same number as the node in the 'shell middle plane' with a ' (prime) and " (double prime) to separate them from the 'real nodes'.

With correspondence to Figure 5-41, 5-42 and 5-43 the local node ("point") numbering of each side is defined as follows:

It should be noted that the points in the three tables and Figs. 5-41, 5-42 and 5-43 below are in the same topological positions on the sides, but the node (point) numbers are different, since number of 'real nodes' are different. The point numbers are sort of dummy numbers and only used to indicate the sequence of the load intensities to be specified.

TRSI(36)

	Local node ("point") number on side							
	1	**2**	**3**	**4**	**5**	**6**	**7**	**8**
Side no.								
1	6	7	1	8	12	18	17	11
2	4	10	15	14	9"	9	9'	3
3	1	2	9'	9	9"	13	12	8
4	6	11	17	16	15	10	4	5
5	6	5	4	3	9'	2	1	7
6	17	18	12	13	9"	14	15	16

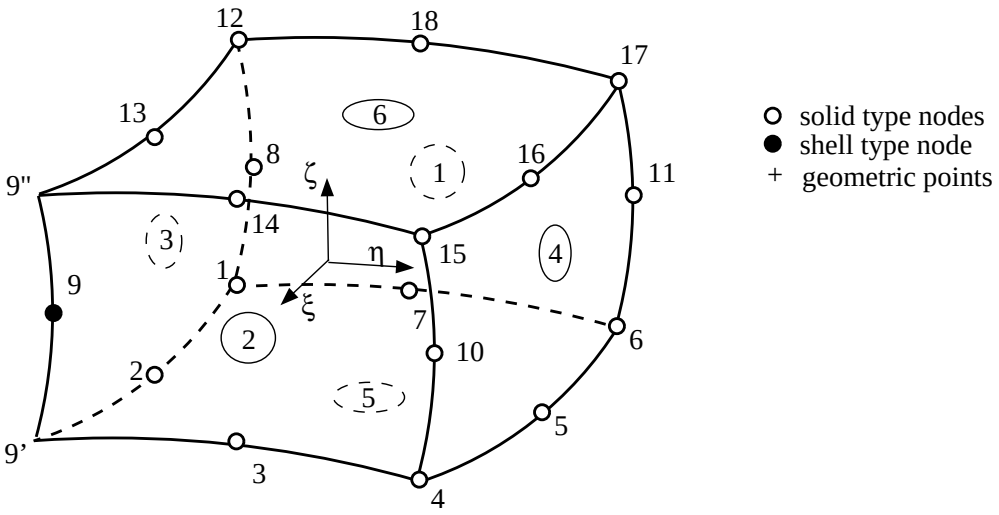


Figure 5-44 Numbering of nodes (points) where surface and line load values are specified for TRSI(36)-elements

	Local node ("point") number on side							
	1	**2**	**3**	**4**	**5**	**6**	**7**	**8**
Side no.								
1	4	5	1	6	11	15	14	10
2	9’	9	9"	8"	7"	7	7’	8’
3	1	2	7’	7	7"	12	11	6
4	4	10	14	13	9"	9	9’	3
5	4	3	9’	8’	7’	2	1	5
6	14	15	11	12	7"	8"	9"	13

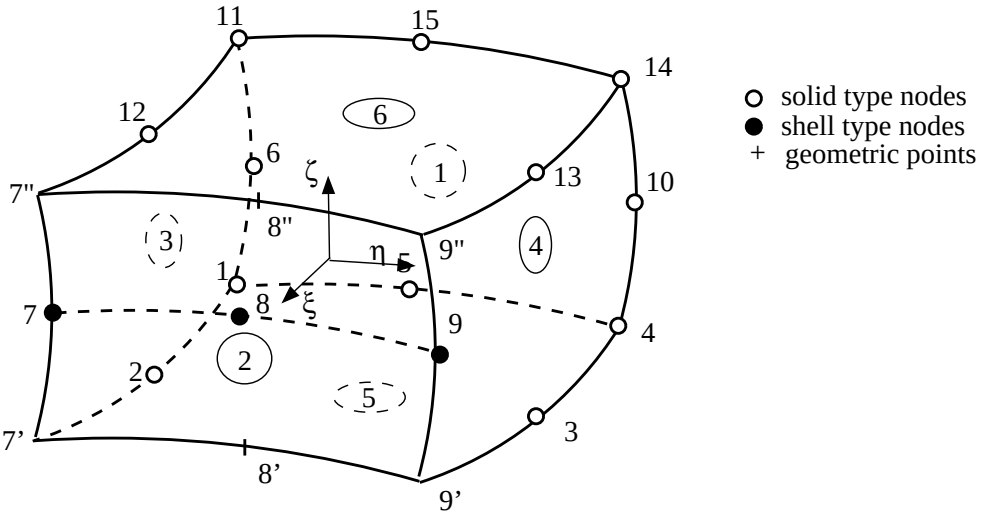


Figure 5-45 Numbering of nodes (points) where surface and line load values are specified for TRSI(37)-elements

	Local node ("point") number on side							
	1	**2**	**3**	**4**	**5**	**6**	**7**	**8**
Side no.								
1	9’	3	1	4	10	12	9"	9
2	7’	7	7"	6"	5"	5	5’	6’
3	1	2	5’	5	5"	11	10	4
4	9’	9	9"	8"	7"	7	7’	8’
5	9’	8’	7’	6’	5’	2	1	3
6	9"	12	10	11	5"	6"	7"	8"

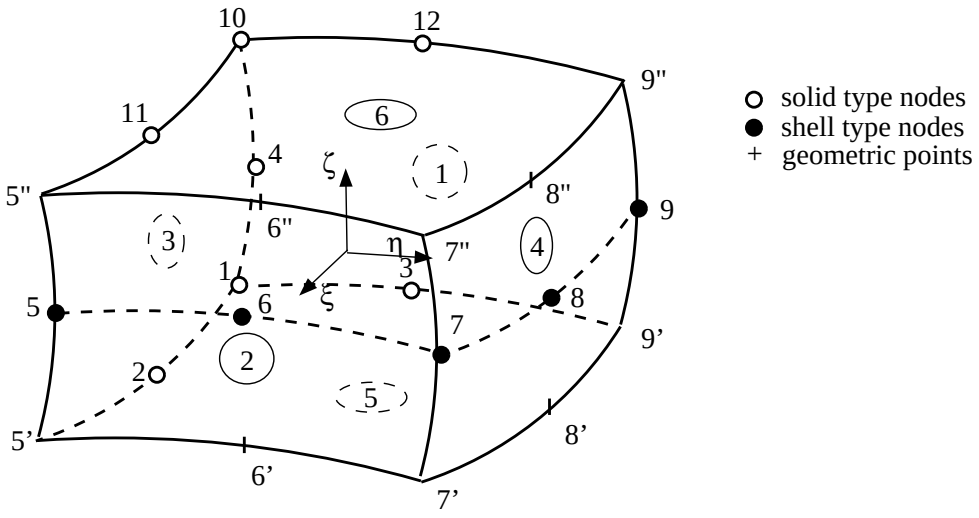


Figure 5-46 Numbering of nodes (points) where surface and line load values are specified for TRSI(37)-elements

Line loads are specified in the same manner as surface loads. The load intenisties are always given in points along lines as defined for the IHEX-element (the so-called originating IHEX element). In the table below the numbers without prime and double prime refer to the node numbers of the actual TRSI element. The the nmbers with prime or double prime refers to the geometrical point below or above the TRSI node. With referrence to Figs. 5-41, 5-42 and 5-43 the local node (point) numbering for each line is defined as follows:

	TRSI(36)			**TRSI(37)**			**TRSI(38)**		
	Local node ("point") number on line:								
	1	**2**	**3**	**1**	**2**	**3**	**1**	**2**	**3**
Line no.:									
1	1	2	9’	1	2	7’	1	2	5’
2	9’	3	4	7’	8’	9’	5’	6’	7’
3	4	5	6	9’	3	4	7’	8’	9’
4	6	7	1	4	5	1	9’	3	1
5	1	8	12	1	6	11	1	4	10
6	9’	9	9"	7’	7	7"	5’	5	5"
7	4	10	15	9’	9	9"	7’	7	7"
8	6	11	17	4	10	14	9’	9	9"
9	12	13	9"	11	12	7"	10	11	5"
10	9"	14	15	7"	8"	9"	5"	6"	7"
11	15	16	17	9"	13	14	7"	8"	9"
12	17	18	12	14	15	11	9"	12	10
13	7	3		5	8’		3	6’	
14	5	2		3	2		8’	2	
15	8	9		6	7		4	5	
16	2	13		2	12		2	11	
17	9	7		7	9		5	7	
18	3	14		8’	8"		6’	6"	
19	10	11		9	10		7	9	
20	5	16		3	13		8’	8"	
21	11	8		10	6		9	4	
22	7	18		5	15		3	12	
23	18	14		15	8"		12	6"	
24	13	16		12	13		11	8"	

Data types used for this element:

GELMNT1 *
GNODE *
GCOORD *
GELREF1 *
GELTH *
GELINT
GELSTRP
MISOSEL * or
MISOPL * or
MORSSOL *
MTRSOL
BEUSLO
BELLO2
BEISTE
BGRAV
BNACCLO

*) Mandatory

Restrictions on modelling with transition elements:

The "shell type nodes" have no stiffness connected to the rotational degree of freedom perpendicular to the middle plane. In shell elements this is taken care of by inserting a small stiffness value to avoid a rectangular system. Hence the "shell type nodes" of the transition elements must always be coupled to a shell element in order to avoid a singular stiffness matrix.

Restrictions on data types:

GELREF1	Parameters INTNO, MINTNO, STRANO, STRENO and STREPONO all refer to data groups were the data are specified as if given for an IHEX element (ELTYP=20). The element thickness of the shell type node(s) must be referred to, either directly by GEONO/OPT or indirectly by GEONO(i) (with GEONO/OPT=-1). In the latter case the GEONO(i) for solid type numbers are of no consequence (set equal to zero).
GELINT GELSTRP BEUSLO BEISTE	The data is as if given for an IHEX element (ELTYP=20). When giving nodal values (intensities) they shall refer to a so-called originating IHEX element. See Figure 5-41 above.
GNODE	The values of NDOF and ODOF must be consistent with the type of node: solid type: NDOF=3, ODOF=123 shell type: NDOF=6, ODOF=123456

- 2 nodes
- degrees of freedom, arbitrary (max 12)
- stiffness matrix

Data types used for this element:

GELMNT1	*
GELREF1	*
MSHGLSP	*
BNTRCOS	

*) Mandatory

Back to Section 3 **ELEMENT TYPES**

3.32 GLSH: General 2-Noded Spring / Shim Element

Element Type 40

- 2 nodes
- degrees of freedom, arbitrary
- general spring matrix

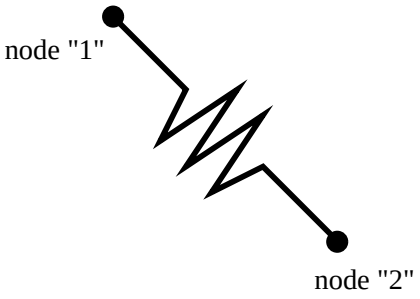


Figure 5-47 General 2-noded spring element

As General Spring it is just a 2-noded spring (12x12 matrix) which may be in a local coordinate system. As a shim element the preprocessor(s) will only insert stiffness in the local x- and y-direction. In the analysis program(s), shim members and general springs are treated exactly in the same manner.

The resulting spring matrix is a full symmetric matrix where all values on and below the main diagonal are stored.

Data types used for this element:

GELMNT1 *
MSHGLSP * (NDOF1 and NDOF2 must be equal to NDOF on data type GNODE for node "1" and node "2")
GELREF1 *
BNTRCOS (Transformation in the two nodes may be different. But NDOF1 = NDOF2 = 3 or 6 is required for transformation).

*) Mandatory

Back to Section 3 **ELEMENT TYPES**

3.33 AXCS: Axi-symmetric Constant Strain Triangle

Element Type 41

- 3 nodes
- 3 x 3 degrees of freedom
- straight
  - line loads
  - initial strain (temperature load)
  - gravitational load (only in the membrane plane)
  - general inertia load (only in the membrane plane)
- isotropic or anisotropic material data

Local node numbering:

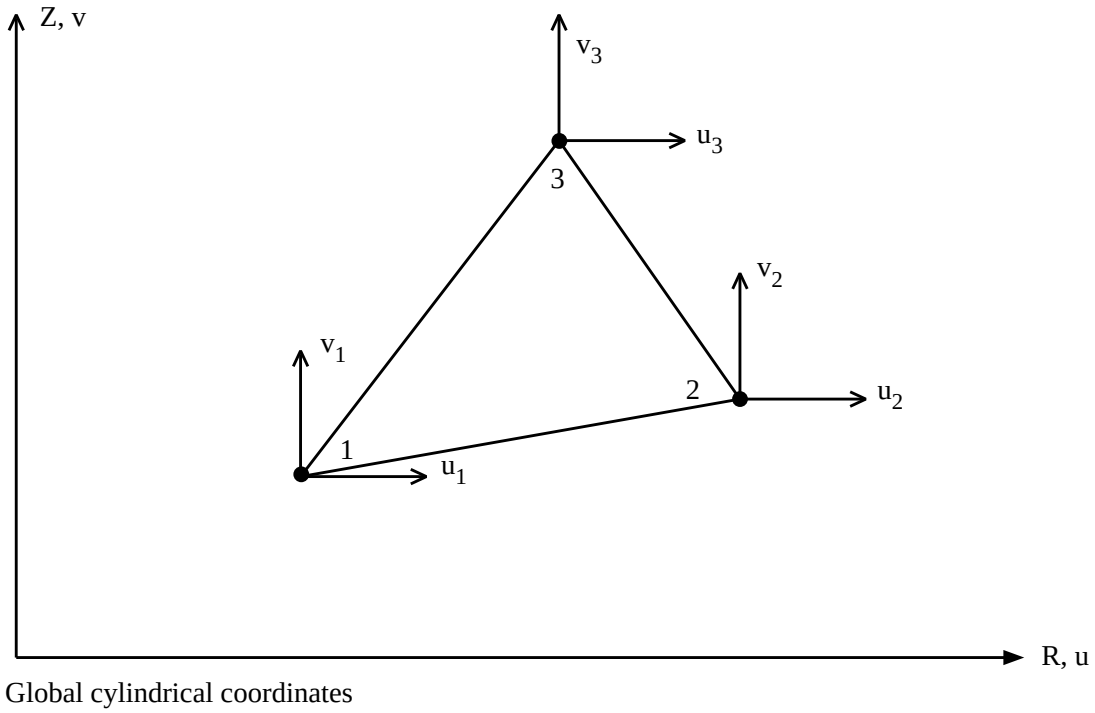


Figure 5-48 Element node numbering. Global cylindrical coordinate system.

When line load is specified, the relation between local node numbers and loaded line will be:

Line 1 means load along the line defined by the nodes 2 and 3.
Line 2 means load along the line defined by the nodes 1 and 3
Line 3 means load along the line defined by the nodes 1 and 2.

The direction of node numbering must be counterclockwise.

Data types used for this element:

GELMNT1 *
GELREF1 *
GNODE *
GCOORD *

MISOSEL * or
MORSMEL *
MTRMEL

BELLAX
BEISTE
BGRAV
BNACCLO

*) Mandatory

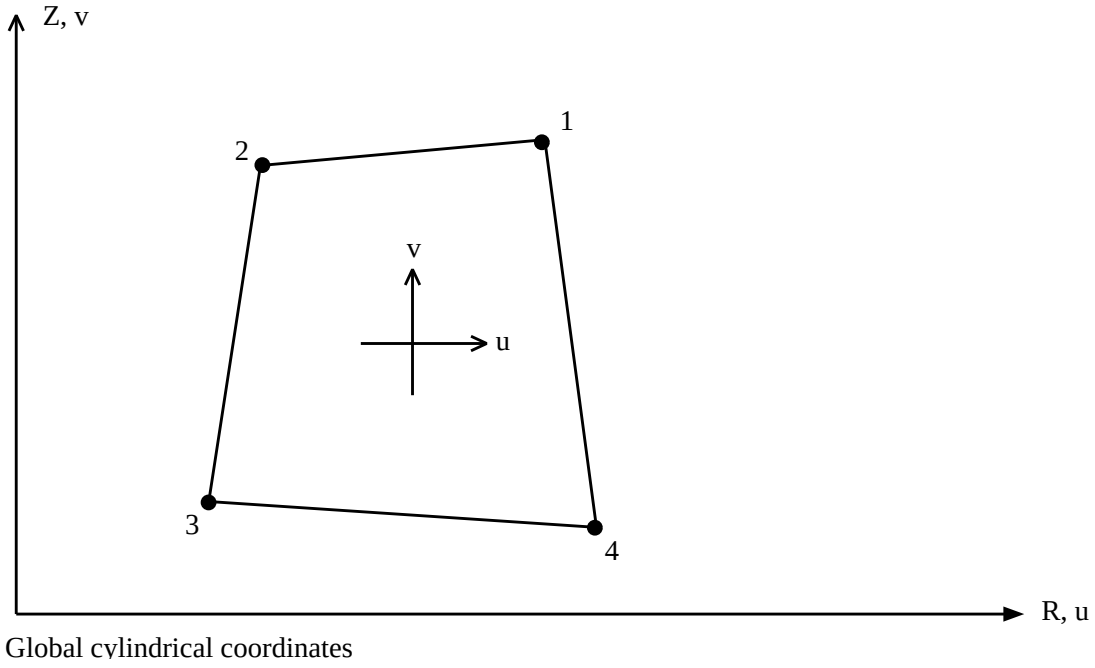
Back to Section 3 **ELEMENT TYPES**

3.34 AXLQ: Axi-symmetric Constant Strain Quadrilateral

Element Type 42

- 4 nodes
- 4 x 3 degrees of freedom
- straight
- element loads
  - line loads
  - initial strain (temperature load)
  - gavitational load (only in the membrane plane)
  - general inertia load (only in the membrane plane)
- isotropic or anisotropic material data
- deformations considered: translational strain

Local node numbering:



Global cylindrical coordinates

Figure 5-49 Element node numbering. Global cylindrical coordinate system.

When line load is specified, the relation between local node numbers and loaded line will be:

LINE 1 means load along the line defined by the nodes 1 and 2
LINE 2 means load along the line defined by the nodes 2 and 3
LINE 3 means load along the line defined by the nodes 3 and 4
LINE 4 means load along the line defined by the nodes 4 and 1

The direction of node numbering must be counterclockwise.

Data types used for this element:

GELMNT1 *
GELREF1 *
GNODE *
GCOORD *
GELINT

MISOSEL or *
MORSMEL
MTRMEL

BELLAX
BEISTE
BGRAV
BNACCLO

*) Mandatory

The following restrictions are put on data types:

GELINT — The integration stations must be distributed according to the Gaussian integration scheme, i.e.
INTYPE =1. For this element type the GELINT specifications consist of the first line.

Stiffness matrix:
Legal values for N1 and N2 are 2 and 3. N1 and N2 must have the same value.
Default value is N1=N2=2.

Mass matrix: For mass matrix calculations the number of integration stations will be the same
in both coordinate directions and equal to the value specified for N1 for the stiffness matrix.

Back to Section 3 **ELEMENT TYPES**

3.35 AXLS: Axi-symmetric Linear Strain Triangle

Element Type 43

- 6 nodes
- 6 x 3 degrees of freedom
- curved
- deformations considered: translational strain
- element loads:
  - line loads
  - initial strain (temperature loads)
  - gravitational load (only in the membrane plane)
  - general inertia load (only in the membrane plane)
- isotropic or anisotropic material data

Local node numbering:

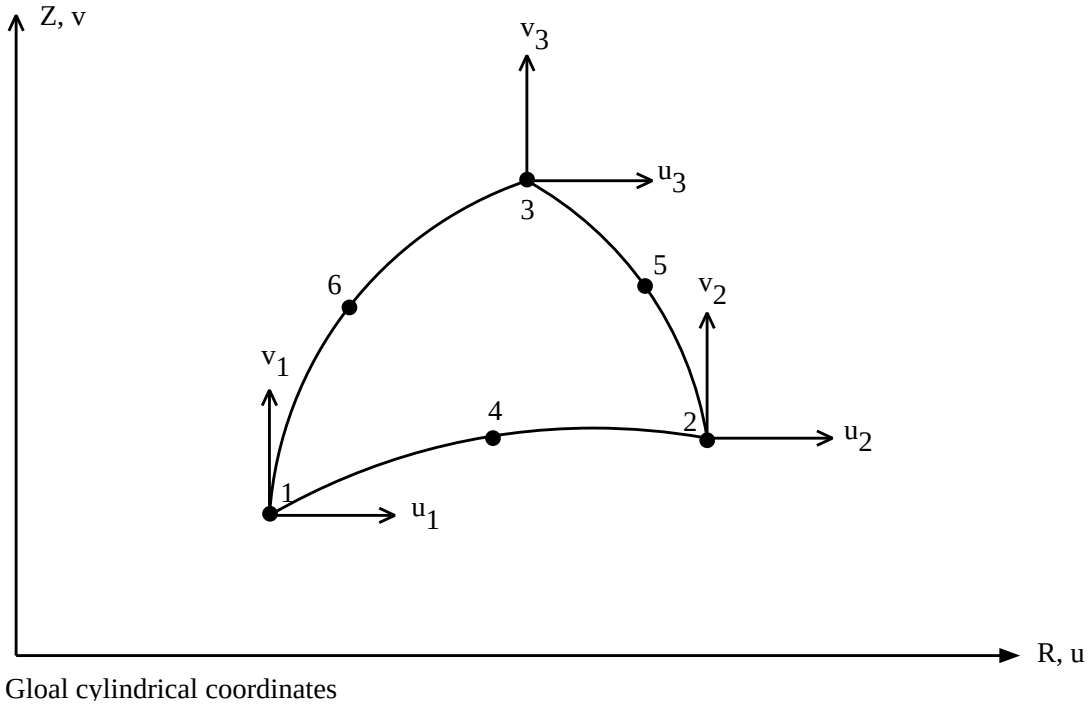


Figure 5-50 Element node numbering. Global cylindrical coordinate system.

When line load is specified, the relation between local node numbers and loaded line will be:

Line 1 means load along the line defined by the nodes 2, 5 and 3.
Line 2 means load along the line defined by the nodes 1, 6 and 3
Line 3 means load along the line defined by the nodes 1, 4 and 2.

The direction of node numbering must be counterclockwise.

The following restrictions are put on the data types

GELINT — The integration stations must be distributed according to the Gaussian integration scheme, i.e.
INTYPE =1. For this element type the GELINT specifications consist of the first filerecord (line) only.

Stiffness matrix: For stiffness matrix calculations the number of integration stations will be equal to the value specified for N1.
Legal values are 1, 3, 4 and 7.
Default value is 4.

Mass matrix: Only possible value and default value is 7.

Load vector: Only possible value and default value is 3.

Data types used for this element:

GELMNT1 *
GELREF1 *
GNODE *
GCOORD *
GELINT

MISOSEL or *
MORSMEL
MTRMEL

BELLAX
BEISTE
BGRAV
BNACCLO

*) Mandatory

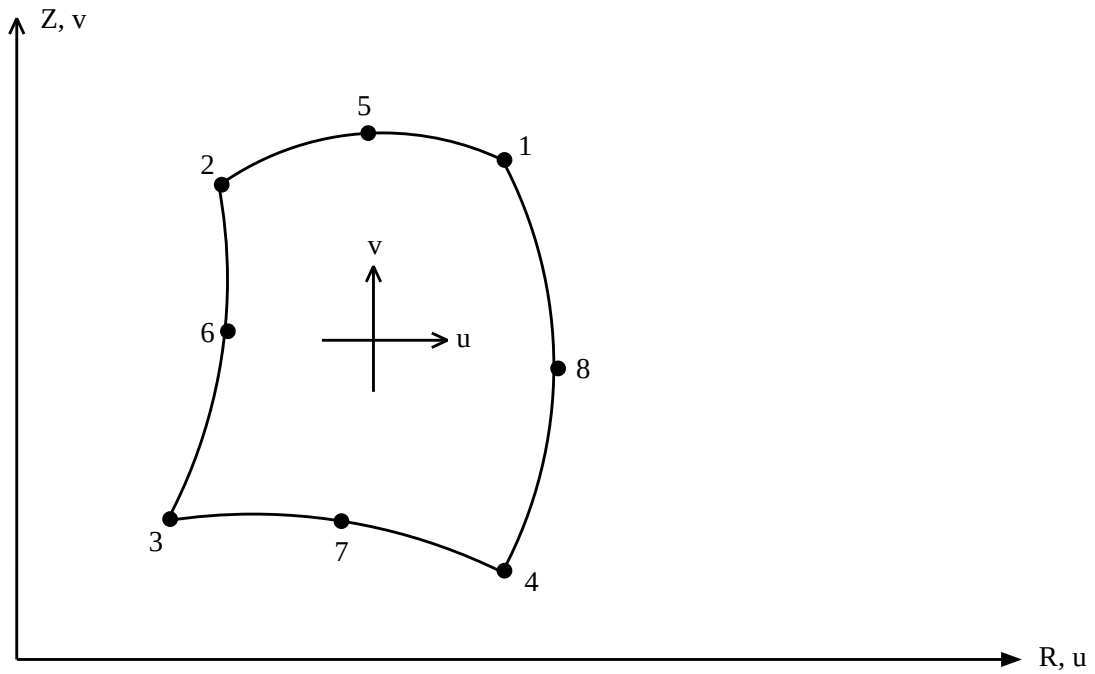
Back to Section 3 **ELEMENT TYPES**

3.36 AXQQ: Axi-symmetric Linear Strain Quadrilateral

Element Type 44

- 8 nodes
- 8 x 3 degrees of freedom
- curved
- deformations considered: translational strain
- element loads:
  - line loads
  - initial strain (temperature loads)
  - gravitational load (only in the membrane plane)
  - general inertia load (only in the membrane plane)
- isotropic or anisotropic material data

Local node numbering:



Gloal cylindrical coordinates

Figure 5-51 Element node numbering. Global cylindrical coordinate system.

When line load is specified, the relation between local node numbers and loaded line will be:

LINE 1 means load along the line defined by the nodes 2, 6 and 3
LINE 2 means load along the line defined by the nodes 5 and 7
LINE 3 means load along the line defined by the nodes 1, 8 and 4
LINE 4 means load along the line defined by the nodes 1, 5 and 2
LINE 5 means load along the line defined by the nodes 8 and 6
LINE 6 means load along the line defined by the nodes 4, 7 and 3

The direction of node numbering must be counterclockwise.

The following restrictions are put on the data types

GELINT	The integration stations must be distributed according to the Gaussian integration scheme, i.e. INTYPE =1. For this element type the GELINT specifications consist of the first filerecord only. Stiffness matrix: For stiffness matrix calculations the number of integration stations will be the same in both coordinate directions and equal to the value specified for N1. Legal values are 2 and 3. Default value is 3. Mass matrix: For mass matrix calculations the number of integration stations will be the same in both coordinate directions and equal to the value specified for N1 for the stiffness martix.. Load vector: 3 is used, the variable is not possible to specify for the user.
BELLO2	LINE = 2 and LINE = 5 is not operative in SESTRA.

Data types used for this element:

GELMNT1 *
GELREF1 *
GNODE *
GCOORD *
GELINT

MISOSEL or *
MORSMEL
MTRMEL

BELLAX
BELLO2
BEISTE
BGRAV
BNACCLO

*) Mandatory

Back to Section 3 **ELEMENT TYPES**

3.37 CTCP: 2-Noded (1+1) Contact Element

Element Type 51

- 2 nodes
- 2 x 6 degrees of freedom
- contact material (see the MCNT record)
- deformations considered:
  separation and relative tangential desplacement between surfaces.
- zero thickness.
- element loads:
  - none

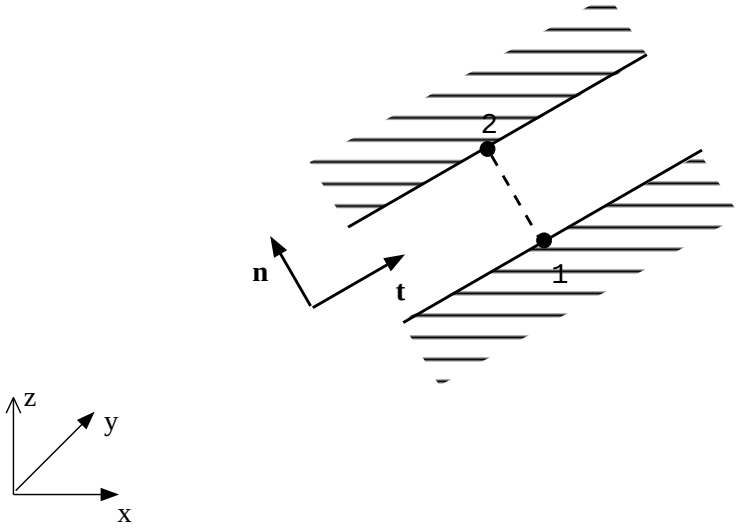


Figure 5-52 Element node numbering.

Data types for this element:

GELMNT1	*
GNODE	*
GCOORD	*
GELREF1	*
GELTH	**
GELINT	
MCNT	*

*) Mandatory
**) If the nodes in the two layers (sides) of the element in the model do not coincide, thickness of the element may be specified by GELTH records.
Any distance between the two layers (sides) which is different from specified thickness is interpreted as initial gap or initial penetration of the contact surface.

Back to Section 3 **ELEMENT TYPES**

3.38 CTCL: 4-Noded (2+2) Contact Element

Element Type 52

- 4 nodes
- 4 x 6 degrees of freedom
- linear element shape
- contact material (see the MCNT record)
- deformations considered:
  separation and relative tangential desplacement between surfaces.
- zero thickness.
- element loads:
  - none

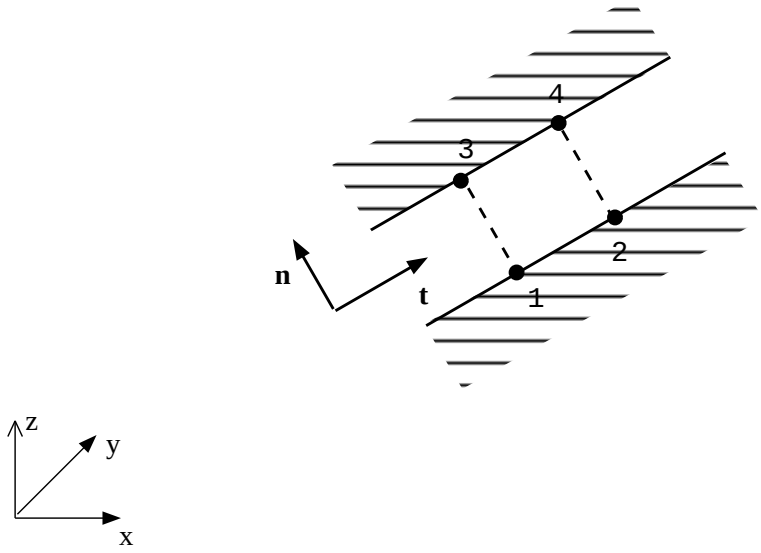


Figure 5-53 Element node numbering.

Data types for this element:

GELMNT1	*
GNODE	*
GCOORD	*
GELREF1	*
GELTH	**
GELINT	
MCNT	*

*) Mandatory
**) If the nodes in the two layers (sides) of the element in the model do not coincide, thickness of the element may be specified by GELTH records.
Any distance between the two layers (sides) which is different from specified thickness is interpreted as initial gap or initial penetration of the contact surface.

Back to Section 3 **ELEMENT TYPES**

3.39 CTAL: 4-Noded (2+2) Axi-symmetric Contact Element

Element Type 53

- 4 nodes
- 4 x 3 degrees of freedom
- linear element shape
- contact material (see the MCNT record)
- deformations considered:
  separation and relative tangential desplacement between surfaces.
- zero thickness.
- element loads:
  - none

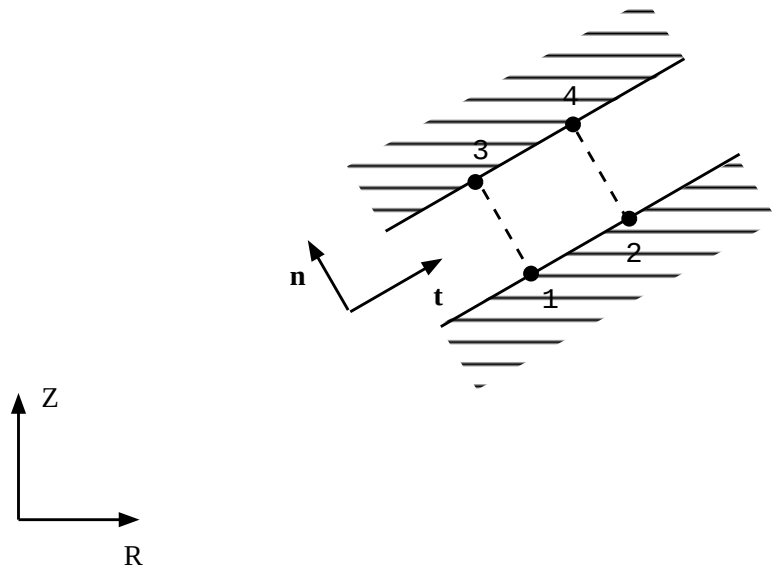


Figure 5-54 Element node numbering.

Data types for this element:

GELMNT1	*
GNODE	*
GCOORD	*
GELREF1	*
GELTH	**
GELINT	
MCNT	*

*) Mandatory
**) If the nodes in the two layers (sides) of the element in the model do not coincide, thickness of the element may be specified by GELTH records.
Any distance between the two layers (sides) which is different from specified thickness is interpreted as initial gap or initial penetration of the contact surface.

Back to Section 3 **ELEMENT TYPES**

3.40 CTCC: 6-Noded (3+3) Contact Element

Element Type 54

- 6 nodes
- 6 x 6 degrees of freedom
- curved element shape
- contact material (see the MCNT record)
- deformations considered:
  separation and relative tangential desplacement between surfaces.
- zero thickness.
- element loads:
  - none

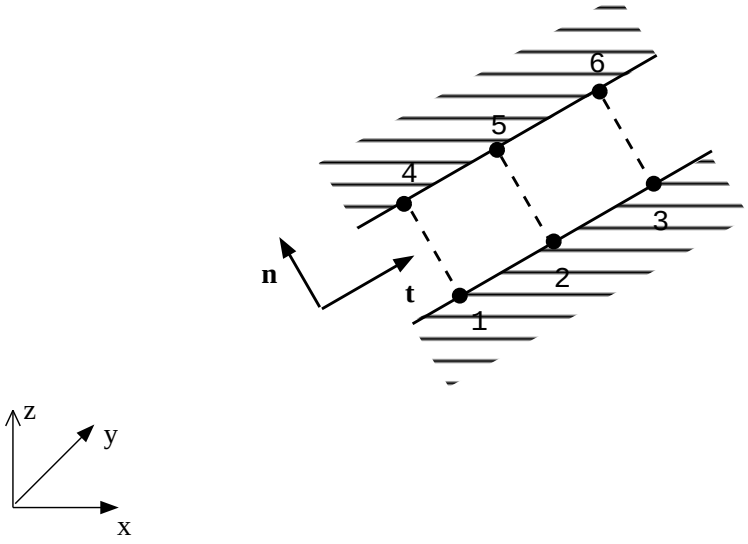


Figure 5-55 Element node numbering.

Data types for this element:

GELMNT1	*
GNODE	*
GCOORD	*
GELREF1	*
GELTH	**
GELINT	
MCNT	*

*) Mandatory
**) If the nodes in the two layers (sides) of the element in the model do not coincide, thickness of the element may be specified by GELTH records.
Any distance between the two layers (sides) which is different from specified thickness is interpreted as initial gap or initial penetration of the contact surface.

Back to Section 3 **ELEMENT TYPES**

3.41 CTAQ: 6-Noded (3+3) Axi-symmetric Contact Element

Element Type 55, see references [6] and [5]

- 6 nodes
- 6 x 3 degrees of freedom
- curved element shape
- contact material (see the MCNT record)
- deformations considered:
  separation and relative tangential desplacement between surfaces.
- zero thickness.
- element loads:
  - none

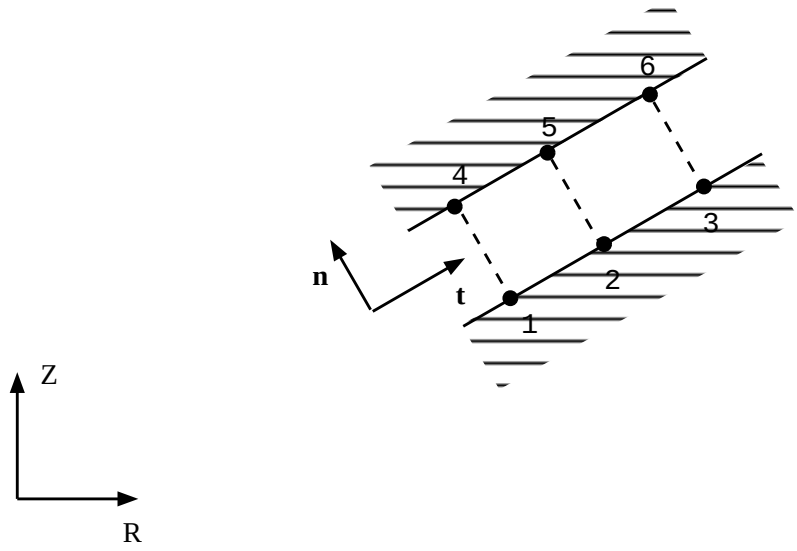


Figure 5-56 Element node numbering.

Data types for this element:

GELMNT1	*
GNODE	*
GCOORD	*
GELREF1	*
GELTH	**
GELINT	
MCNT	*

*) Mandatory
**) If the nodes in the two layers (sides) of the element in the model do not coincide, thickness of the element may be specified by GELTH records.
Any distance between the two layers (sides) which is different from specified thickness is interpreted as initial gap or initial penetration of the contact surface.

Back to Section 3 **ELEMENT TYPES**

3.42 CTLQ: 8-Noded (4+4) Contact Element

Element Type 56, see references [6] and [5]

- 8 nodes
- 8 x 3 degrees of freedom
- flat elemtent shape
- contact material (see the MCNT record)
- deformations considered:
  penetration prevented
- zero thickness when beeing between two solid elements and linearly varying element thickness according to the adjacent shell elements when connecting two shell elements.
- element loads:
  - none

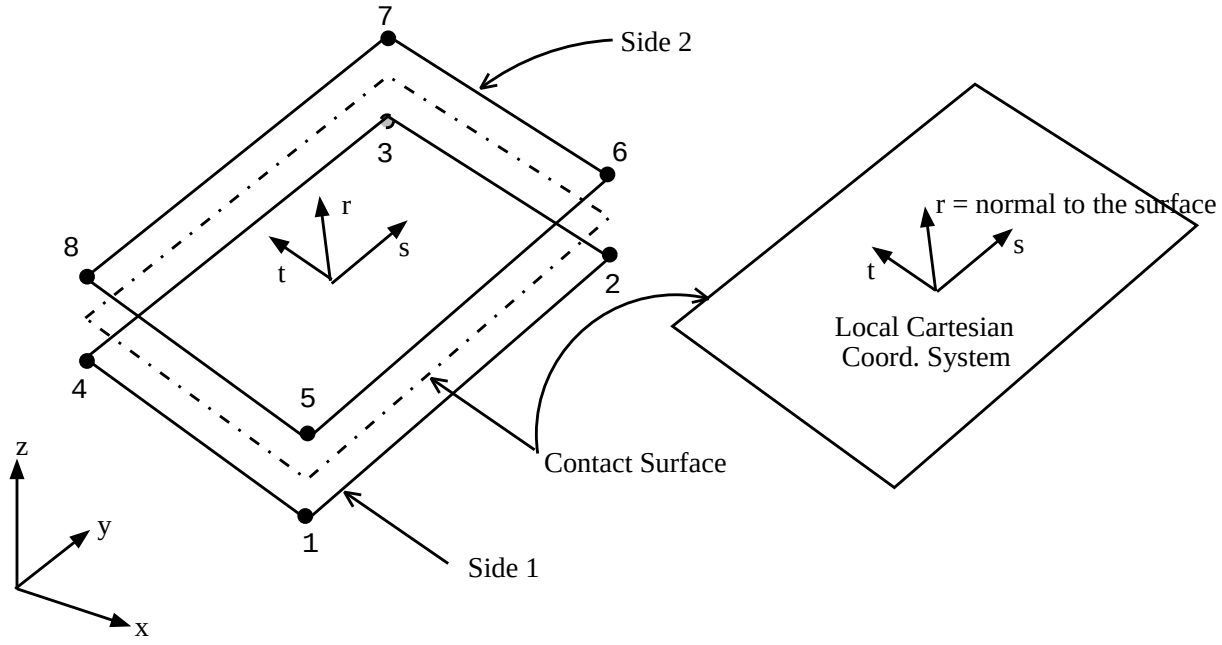


Figure 5-57 Element node numbering. Global Cartesian and local coordinate system.

Line no. definition:

The orientation of the local nodal coordinate system is related to the nodenumbering sequence (see Figure 5-57). The positive r-direction, normal to the element contact surface, is chosen according to the normal convention of the right hand rule, i.e. the positive r-direction is found when the node numbers are followed in ascending order, counterclockwise.

The SIDE definition is as follows:

SIDE=1	means the element surface where r is negative
SIDE=2	means the element surface where r is positive

Data types for this element:

GELMNT1	*
GNODE	*
GCOORD	*
GELREF1	*
GELTH	**
GELINT	
MCNT	*

*) Mandatory
**) If the nodes in the two layers (sides) of the element in the model do not coincide, thickness of the element may be specified by GELTH records.
Any distance between the two layers (sides) which is different from specified thickness is interpreted as initial gap or initial penetration of the contact surface.

Back to Section 3 **ELEMENT TYPES**

3.43 CTCQ: 16-Noded (8+8) Contact Element

Element Type 57, see references [6] and [5]

- 16 nodes
- 16 x 3 degrees of freedom
- curved element shape
- contact material (see the MCNT record)
- deformations considered:
  penetration prevented
- zero thickness when beeing between two solid elements and parabolically varying element thickness according to the adjacent shell elements when connecting two shell elements.
- element loads:
  - none

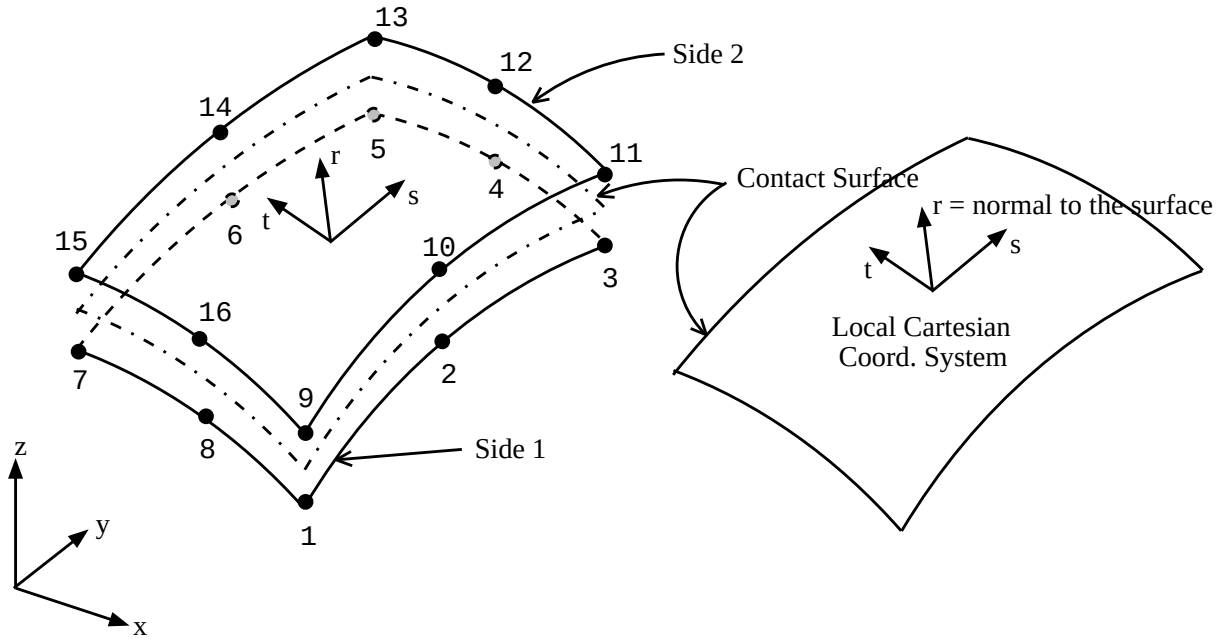


Figure 5-58 Element node numbering. Global Cartesian and local coordinate system.

Line no. definition:

LINE=1	means line defined by the nodes 3, 4 and 5
LINE=2	means line defined by the nodes 2 and 6
LINE=3	means line defined by the nodes 1, 8 and 7
LINE=4	means line defined by the nodes 7, 6 and 5
LINE=5	means line defined by the nodes 8 and 4
LINE=6	means line defined by the nodes 1, 2 and 3
LINE=7	means line defined by the nodes 11, 12 and 13
LINE=8	means line defined by the nodes 10 and 14

LINE=9 means line defined by the nodes 9, 16 and 15
LINE=10 means line defined by the nodes 15, 14 and 13
LINE=11 means line defined by the nodes 16 and 12
LINE=12 means line defined by the nodes 9, 10 and 11

The orientation of the local nodal coordinate system is related to the nodenumbering sequence (see Figure 5-58). The positive r-direction, normal to the element contact surface, is chosen according to the normal convention of the right hand rule, i.e. the positive r-direction is found when the node numbers are followed in ascending order, counterclockwise.

The SIDE definition is as follows:

SIDE=1 means the element surface where r is negative
SIDE=2 means the element surface where r is positive

Data types for this element:

GELMNT1	*
GNODE	*
GCOORD	*
GELREF1	*
GELTH	
GELINT	
MCNT	*

*) Mandatory
**) If the nodes in the two layers (sides) of the element in the model do not coincide, thickness of the element may be specified by GELTH records.
Any distance between the two layers (sides) which is different from specified thickness is interpreted as initial gap or initial penetration of the contact surface.

Back to Section 3 **ELEMENT TYPES**

3.44 CTMQ: 18-Noded (9+9) Contact Element

Element Type 58, see references [6] and [5]

- 18 nodes
- 18 x 3 degrees of freedom
- curved element shape
- contact material (see the MCNT record)
- deformations considered:
  penetration prevented
- zero thickness when beeing between two solid elements and parabolically varying element thickness according to the adjacent shell elements when connecting two shell elements.
- element loads:
  - none

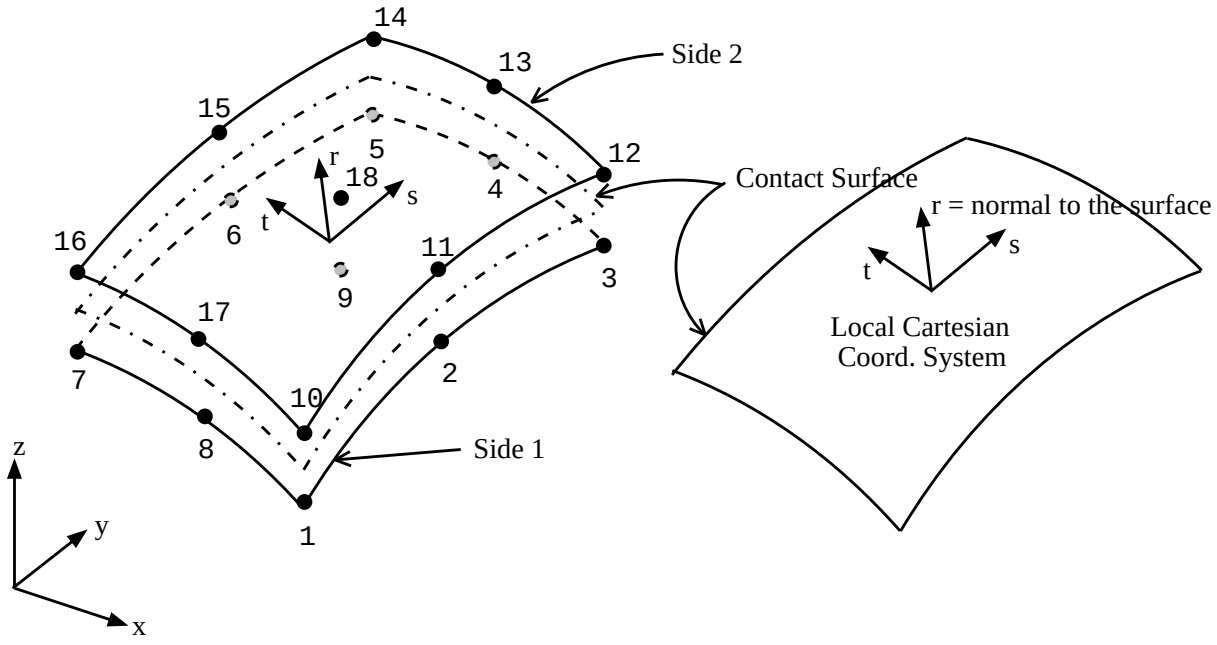


Figure 5-59 Element node numbering. Global Cartesian and local coordinate system.

The orientation of the local nodal coordinate system is related to the nodenumbering sequence (see Figure 5-59). The positive r-direction, normal to the element contact surface, is chosen according to the normal convention of the right hand rule, i.e. the positive r-direction is found when the node numbers are followed in ascending order, counterclockwise.

The SIDE definition is as follows:

SIDE=1	means the element surface where r is negative
SIDE=2	means the element surface where r is positive

Data types for this element:

GELMNT1	*
GNODE	*
GCOORD	*
GELREF1	*
GELTH	
GELINT	
MCNT	*

*) Mandatory
**) If the nodes in the two layers (sides) of the element in the model do not coincide, thickness of the element may be specified by GELTH records.
Any distance between the two layers (sides) which is different from specified thickness is interpreted as initial gap or initial penetration of the contact surface.

Back to Section 3 **ELEMENT TYPES**

3.45 FTAS: Flat Triangular Thin Shell – Drilling degree of freedom

Element Type 59, see reference [7] for the plate bending formulaton (FTRS), and reference [2] for the membrane formulation.

- 3 nodes
- $3 \times 6 = 18$ degrees of freedom
- linear element sides
- isotropic or anisotropic material data
- deformations considered:
  - bending,
  - shear, and
  - translational strain
- constant element thickness
- element loads:
  - initial strain (temperature load)
  - surface forces
  - line loads
  - line moment load
  - gravitational load
  - general inertia load

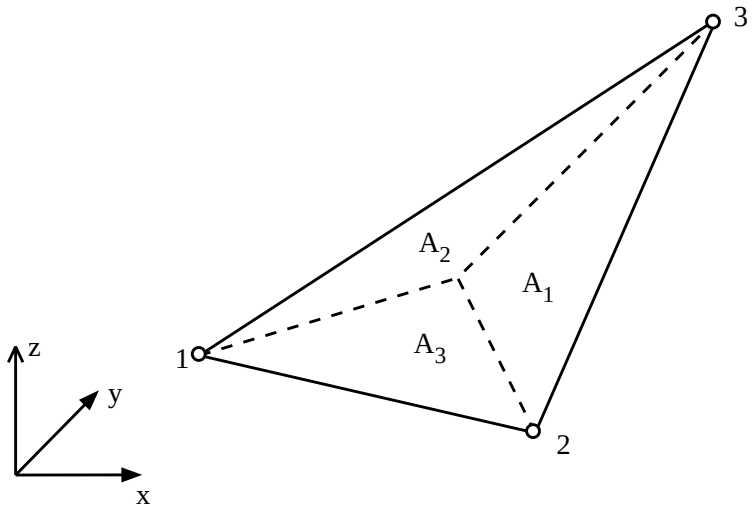


Figure 3.2: Element node numbering and local area coordinates (A_1, A_2 and A_3) on flat triangular shell element.

Data types used for this element:

Data type	Mandatory	Note
GELMNT1	yes	
GELREF1	yes	
GNODE	yes	
GCOORD	yes	
GELTH	yes	
MISOSEL	yes	**MISOSEL** or **MORSMEL** is mandatory.
MORSMEL	yes	
MTRMEL		Sometimes referred to from **MORSMEL**.
BEUSLO		
BELLO2		
BEISTE		
BGRAV		
BNACCLO		

The following restrictions apply to the data types:

`GELREF1` - INTNO and ISPONO are not read because number of integration and stress points are constant in the program and can not be set by user.

- GEONO / OPT should be set>0 because no thickness variation is allowed for this element type.

`BELLO2` - The SIDE definition is not used.

`BEUSLO` - This element type is only able to calculate surface loads which are acting perpendicular to the element surface. For LOTYP=2 (loads given in component form), the in-plane components are ignored by the program.

When line load (**BELLO2**) is specified for one of the element sides, then the relation between local node numbers and loaded line will be:

- LINE = 1 – means line load betweeen node 2 and node 3
- LINE = 2 – means line load betweeen node 1 and node 3
- LINE = 3 – means line load betweeen node 1 and node 2

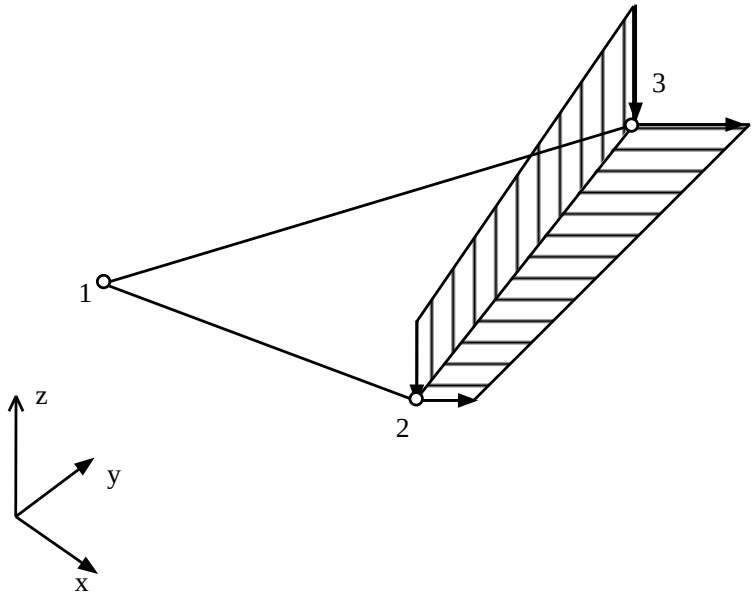


Figure 3.3: Constant line load in the element plane and linear variation of component normal to the plane.

Back to Section 3 **ELEMENT TYPES**

3.46 FQAS: Flat Quadrilateral Thin Shell – Drilling degree of freedom

Element Type 60, see reference [7] for the plate bending formulaton (FQUS), and reference [2] for the membrane formulation.

- 4 nodes
- $4 \times 6 = 24$ degrees of freedom
- linear element sides
- isotropic or anisotropic material data
- deformations considered:
  - bending,
  - shear, and
  - translational strain
- constant element thickness
- element loads:
  - initial strain (temperature load)
  - surface forces
  - line loads
  - line moment load
  - gravitational load
  - general inertia load

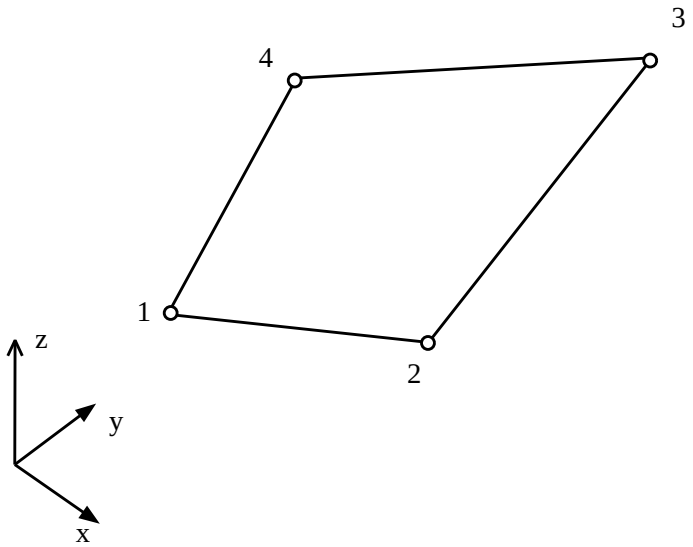


Figure 3.4: Element node numbering on flat quadrilateral shell element.

Data types used for this element:

Data type	Mandatory	Note
GELMNT1	yes	
GELREF1	yes	
GNODE	yes	
GCOORD	yes	
GELTH	yes	
MISOSEL	yes	**MISOSEL** or **MORSMEL** is mandatory.
MORSMEL	yes	
MTRMEL		Sometimes referred to from **MORSMEL**.
BEUSLO		
BELLO2		
BEISTE		
BGRAV		
BNACCLO		

The following restrictions apply to the data types:

`GELREF1` - INTNO and ISPONO are not read because number of integration and stress points are constant in the program and can not be set by user.

- No thickness variation is allowed for this element. If thickness variation is specified by the GEONO / OPT option, the finite element program executing this element must use a mean thickness calculated from the nodal thicknesses.

`BELLO2` - The SIDE definition is not used.

`BEUSLO` - This element type is only able to calculate surface loads which are acting perpendicular to the element surface. For LOTYP=2 (loads given in component form), the in-plane components are ignored by the program.

When line load (**BELLO2**) is specified for one of the element sides, then the relation between local node numbers and loaded line will be:

- LINE = 1 – means line load betweeen node 1 and node 2
- LINE = 2 – means line load betweeen node 2 and node 3
- LINE = 3 – means line load betweeen node 3 and node 4
- LINE = 4 – means line load betweeen node 4 and node 1

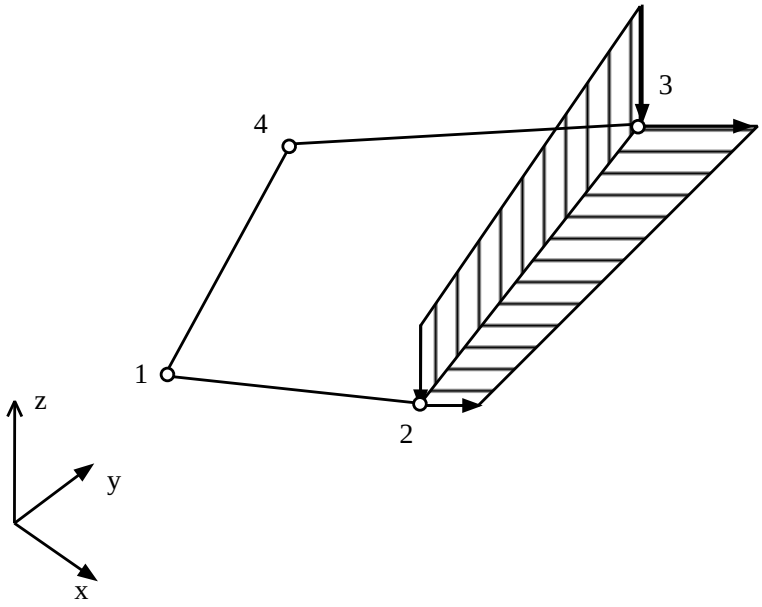


Figure 3.5: Linear variation of line load component normal to the element plane and in the element plane.

Back to Section 3 **ELEMENT TYPES**

3.47 HCQS: Heterosis Curved Quadrilateral Shell

Element Type 61

- 9 nodes
- 9 x 6 degrees of freedom
- curved element shape
- isotropic or anisotropic material data
- deformations considered:
  bending, shear and translational strain
- parabolically varying element thickness
- element loads:
  - initial strain
  - surface forces
  - line loads
  - gravitational load
  - general inertia load

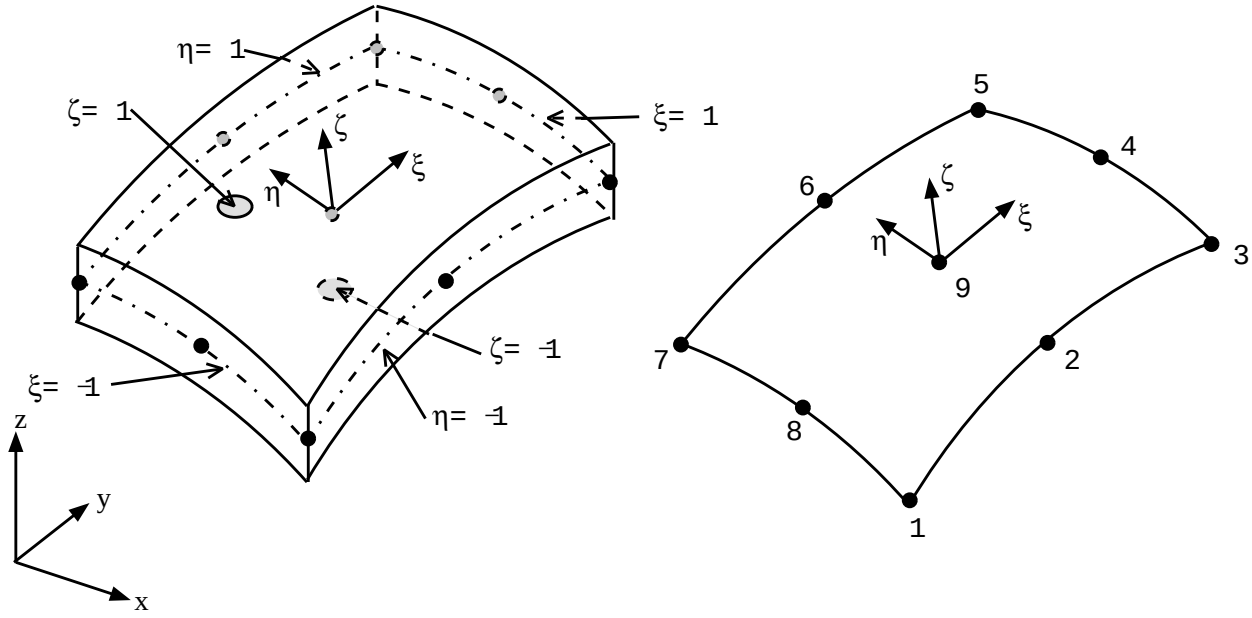


Figure 5-60 Element node numbering. Global Cartesian and local normalized coordinate system.

When line load is specified, the relation between local node numbers and loaded line will go:

LINE=1 means line load along the line defined by the nodes 3, 4 and 5
LINE=2 means line load along the line defined by the nodes 2, 9 and 6
LINE=3 means line load along the line defined by the nodes 1, 8 and 7
LINE=4 means line load along the line defined by the nodes 7, 6 and 5
LINE=5 means line load along the line defined by the nodes 8, 9 and 4
LINE=6 means line load along the line defined by the nodes 1, 2 and 3

The orientation of the local nodal coordinate system is related to the nodenumbering sequence (see Figure 5-60). The positive ζ-direction, normal to the element middlesurface, is chosen according to the normal convention of the right hand rule, i.e. the positive ζ-direction is found when the node numbers are followed in ascending order, counterclockwise.

The SIDE definition on the BEUSLO records is as follows:

SIDE=1 means that load is given on the element surface where ζ= -1,
SIDE=2 that ζ= 0 is loaded, and
SIDE=3 that ζ= 1 is loaded.

Data types for this element:

GELMNT1 *
GNODE *
GCOORD *
GELREF1 *
GELINT
GELTH *
GELSTRP
MISOSEL * or
MISOPL * or
MORSSEL *
MTRSEL
BEUSLO
BELLO2
BEISTE
BGRAV
BNACCLO

*) Mandatory

The following restrictions are put on the data types.

BELLO2	The SIDE-definition is not used.

Back to Section 3 **ELEMENT TYPES**

3.48 THTS: Flat 3-noded Thick Triangular Shell

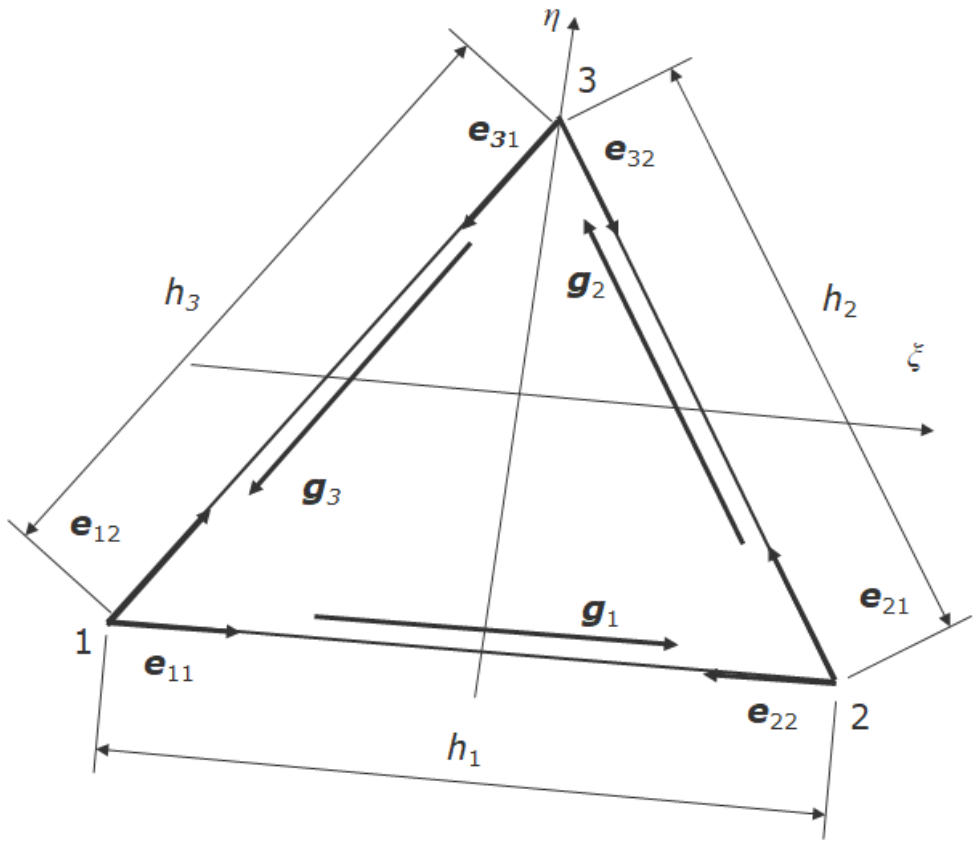


Figure 3.6: Element node numbering, and geometric and kinematic data for the 3-node, linear, triangular element, THTS.

3.48.1 Element Type 63

Reference [4].

- 3 nodes
- 3 × 5 (6) = 15 (18) degrees of freedom
- linear element sides
- isotropic or anisotropic material data
- deformations considered:
  - strain due to bending – linear through the thickness of the element
  - transverse shear strain due to bending – constant through the thickness of the element
  - in-plane strain due to membrane action – constant through the thickness of the element
- linear element thickness
- element loads:
  - initial strain (temperature load)

- surface forces
- line loads
- line moment load
- gravitational load
- general inertia load

3.48.2 Data types used

Data type	Mandatory	Note
GELMNT1	yes	
GELREF1	yes	
GNODE	yes	
GCOORD	yes	
GELTH	yes	
MISOSEL	yes	**MISOSEL**
MORSSEL	yes	or **MORSSEL** is mandatory.
MTRSEL		Sometimes referred to from **MORSSEL**.
BEUSLO		
BELLO2		
BEISTE		
BGRAV		
BNACCLO		

3.48.3 Restrictions

The following interpretations and restrictions apply to the data types:

GELREF1 INTNO and ISPONO are not yet read and thus the number of integration and stress points are constant in the program.

- The element can model linear variation of the thickness. At present the element input data handles one thickness. I.e. the current implementation has constant thickness.

BELLO2 - The SIDE definition is not used.

BEUSLO- Loads are computed as if they act in the middle-plane (for local $z = 0$) of the element.

When line load (**BELLO2**) is specified for one of the element sides, then the relation between local node numbers and loaded line will be:

- LINE = 1 – means line load betweeen node 2 and node 3
- LINE = 2 – means line load betweeen node 1 and node 3

- LINE = 3 – means line load betweeen node 1 and node 2

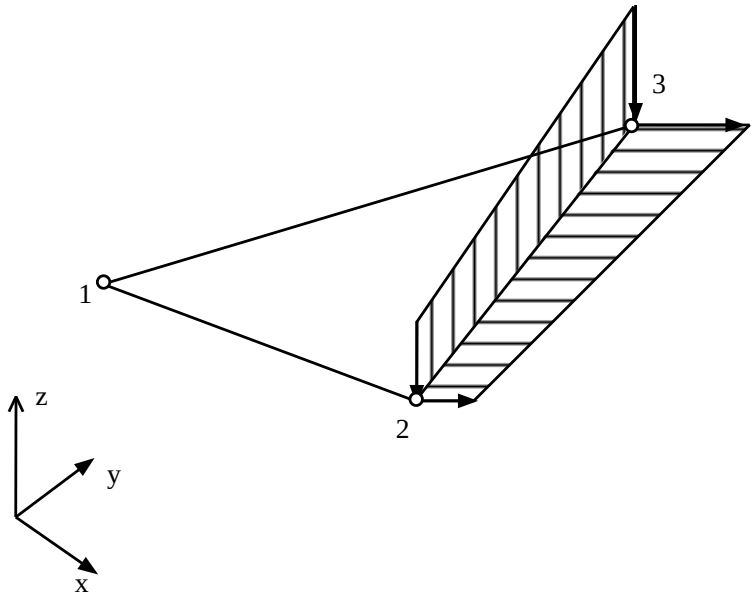


Figure 3.7: Constant line load in the element plane and linear variation of component normal to the plane.

Back to Section 3 **ELEMENT TYPES**

3.49 THQS: Flat 4-noded Thick Quadrilateral Shell

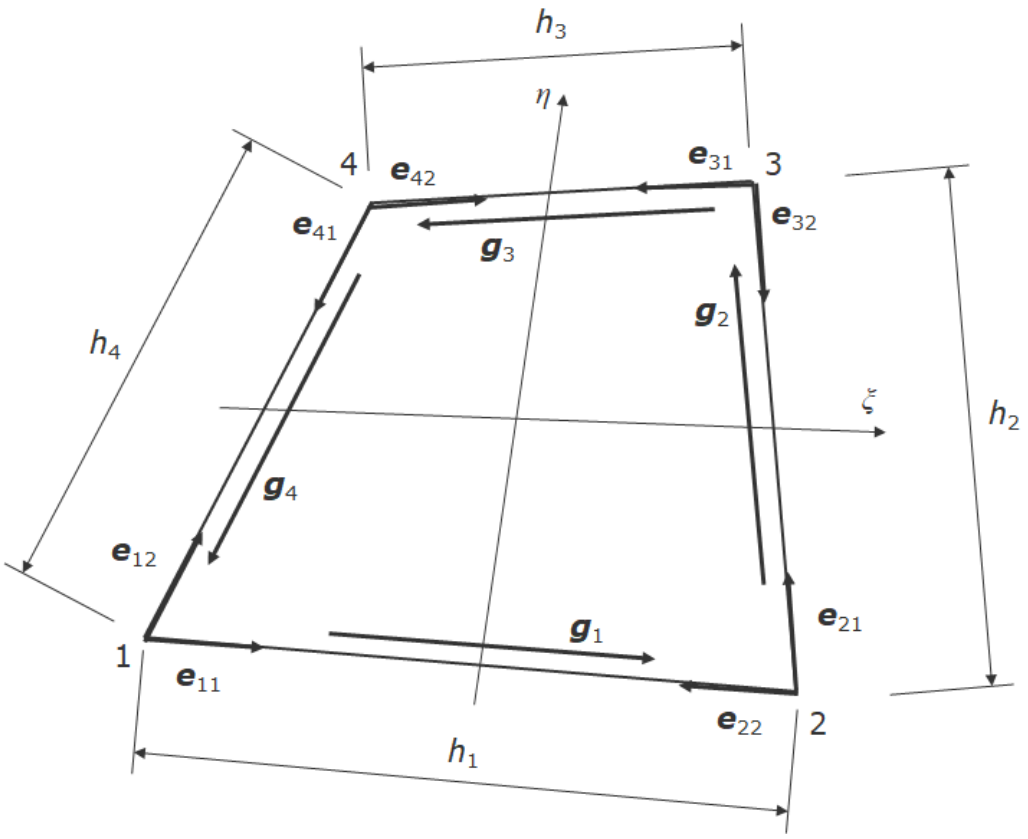


Figure 3.8: Element node numbering, and geometric and kinematic data for the 4-node, bilinear, quadrilateral element, THQS.

3.49.1 **Element Type** 64

Reference [4].

- 4 nodes
- 4 × 5 (6) = 20 (24) degrees of freedom
- linear element sides
- isotropic or anisotropic material data
- deformations considered:
  - strain due to bending – linear through the thickness of the element
  - transverse shear strain due to bending – constant through the thickness of the element
  - in-plane strain due to membrane action – constant through the thickness of the element
- linear element thickness
- element loads:
  - initial strain (temperature load)
  - surface forces

- line loads
- line moment load
- gravitational load
- general inertia load

3.49.2 Data types used

Data type	Mandatory	Note
GELMNT1	yes	
GELREF1	yes	
GNODE	yes	
GCOORD	yes	
GELTH	yes	
MISOSEL	yes	**MISOSEL**
MORSSEL	yes	or **MORSSEL** is mandatory.
MTRSEL		Sometimes referred to from **MORSSEL**.
BEUSLO		
BELLO2		
BEISTE		
BGRAV		
BNACCLO		

3.49.3 Restrictions

The following interpretations and restrictions apply to the data types:

GELREF1 INTNO and ISPONO are not yet read and thus the number of integration and stress points are constant in the program.

- The element can model linear variation of the thickness. At present the element input data handles one thickness. I.e. the current implementation has constant thickness.

BELLO2 - The SIDE definition is not used.

BEUSLO- Loads are computed as if they act in the middle-plane (for local $z = 0$) of the element.

When line load (**BELLO2**) is specified for one of the element sides, then the relation between local node numbers and loaded line will be:

- LINE = 1 – means line load betweeen node 1 and node 2
- LINE = 2 – means line load betweeen node 2 and node 3
- LINE = 3 – means line load betweeen node 3 and node 4

- LINE = 4 – means line load betweeen node 4 and node 1

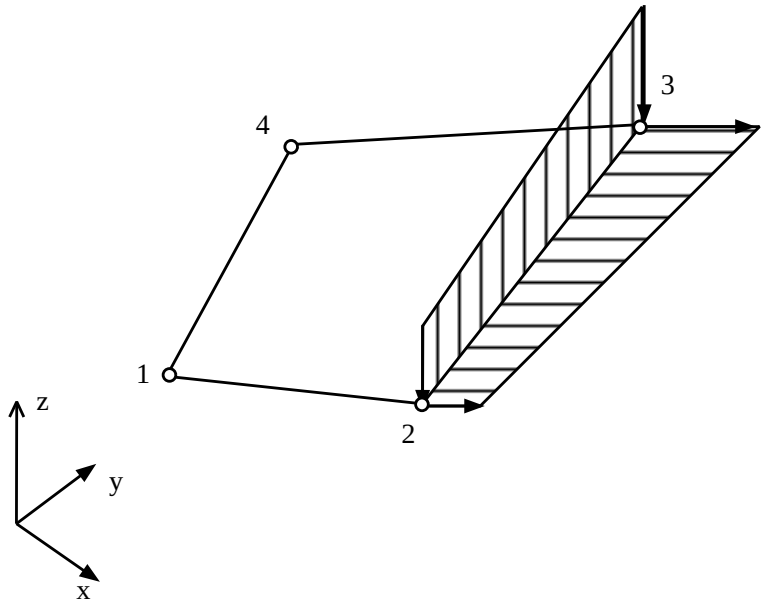


Figure 3.9: Linear variation of line load component normal to the element plane and in the element plane.

Back to Section 3 **ELEMENT TYPES**

3.50 MATR: General Matrix Element

Element Type 70

This element is a way of transferring stiffness / damping / mass matrices and / or load / displacement / velocity / acceleration vectors between different models or analysis programs.

- variable number of nodes - maximum 999 nodes
- possible with different number of degrees of freedom in each node
- real or complex matrices and vectors
- possible with e.g. frequency dependent matrices
- stiffness matrix
- damping matrix
- mass matrix
- nodal load vectors
- no element loads
- nodal displ. / velocity / acceleration vectors

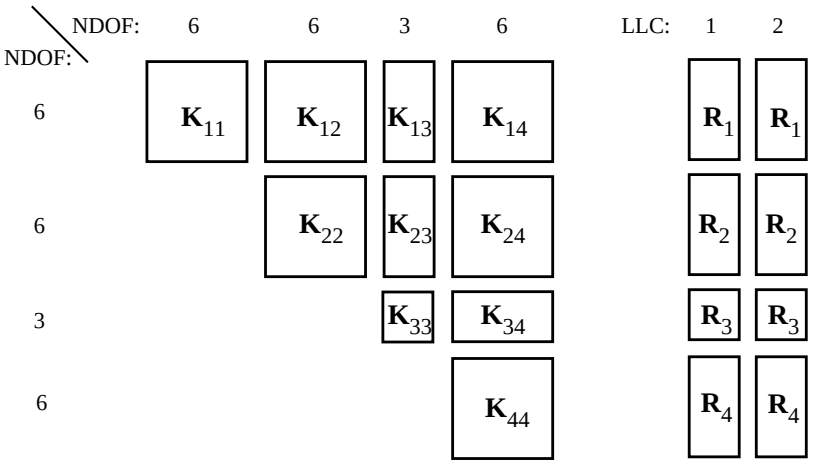


Figure 5-61 Example of 4 noded stiffness and load matrix / vector, with 6, 6, 3 and 6 degrees of freedom in the nodes respectively and 2 loadcases.

Data types for this element:

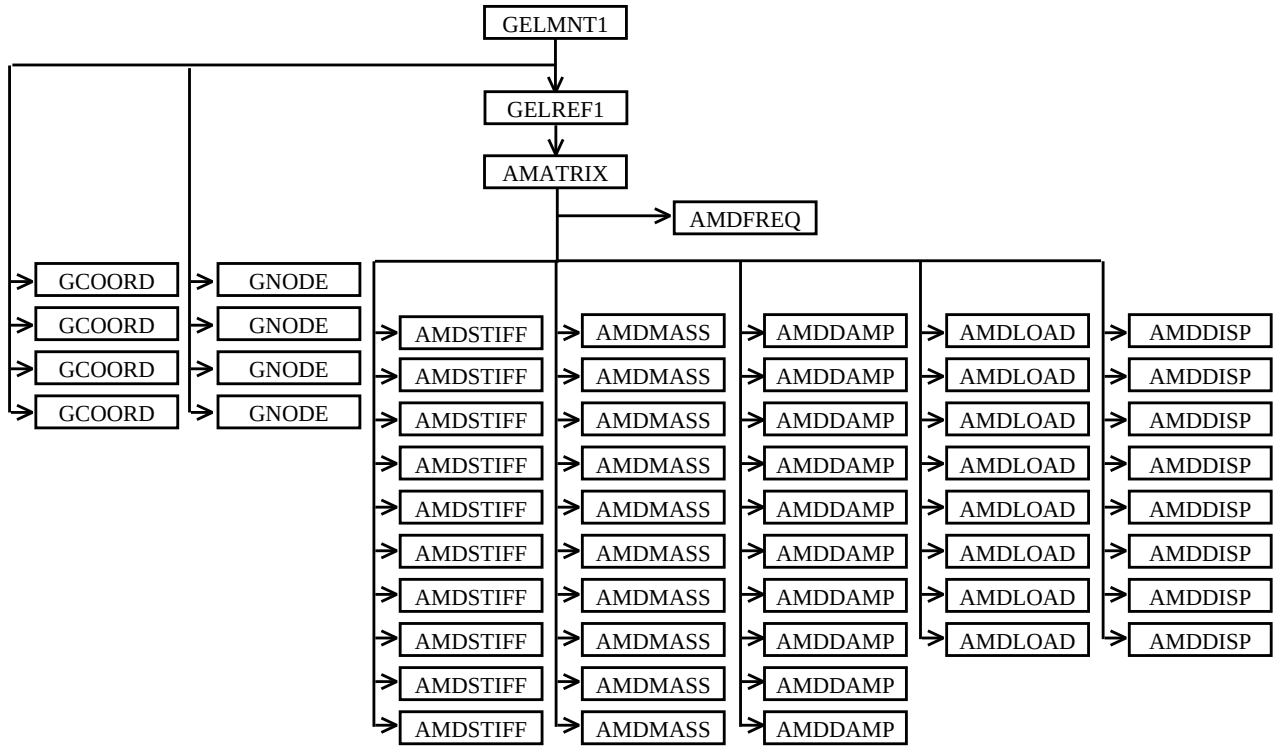


Figure 5-62 Normal data reference for a matrix element with 4 nodes and 2 loadcases. The element is having stiffness, mass, damping, load and resulting displacement matrices / vectors. At least some of the vectors / matrices are also frequency dependent.

AMATRIX [1]	AMDSTIFF [2]
AMDDAMP [2]	AMDMASS [2]
AMDLOAD [2]	AMDDISP [3]
AMDFREQ	AMDVELO [3]
AMDACCL [3]	GCOORD [1]
GELMNT1 [1]	GELMNT2
GELREF1 [1]	GNODE [1]

[1] Mandatory

[2] At least one of these records must be present.

[3] Nodal results may alternatively be defined through records described in 'SIF, Results Interf. File, File Descr.'. See also figure 5-63.

Below is a figure showing the datatypes for nodal displacement, velocity or acceleration results associated with a matrix element, when using the records described in the ’SIF, Results Interf. File, File Descr.’ In the example the results are displacements.

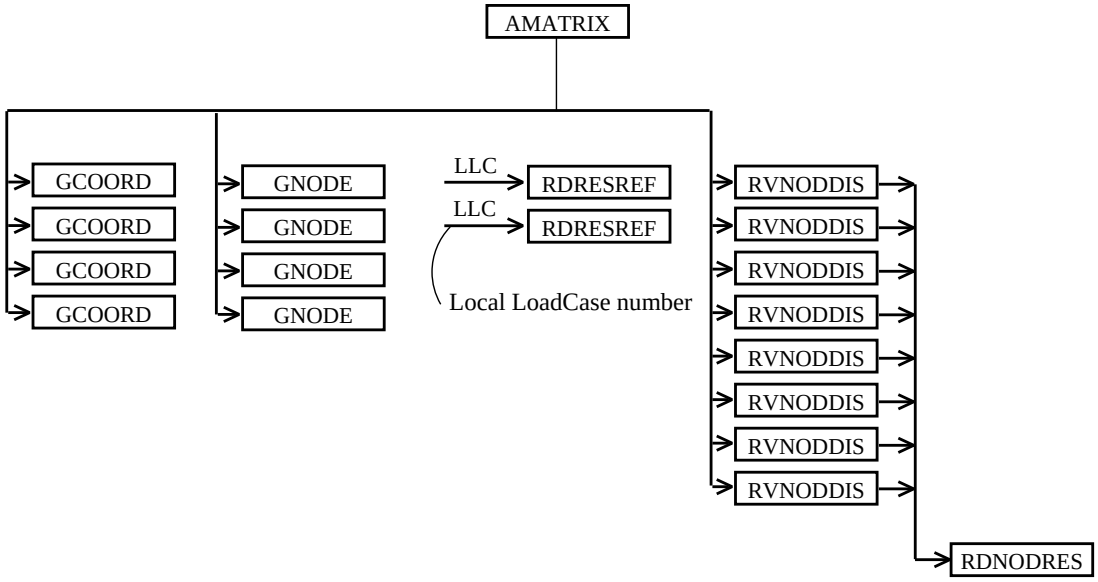


Figure 5-63 Normal data reference for results defined on records described in the ’SIF, Results Interf. File, File Descr.’ for a matrix element with 4 nodes and 2 loadcases.

AMATRIX	[1]	RDRESREF	[1]
GCOORD	[1]	RVNODACC	[2]
GNODE	[1]	RVNODDIS	[2]
RDNODRES	[1]	RVNODVEL	[2]

[1] Mandatory.

[2] At least one of these records must be present.

Back to Section 3 **ELEMENT TYPES**

3.51 GHEX: General Hexahedron

Element Type 100, . . . , 163

- 21 to 27 nodes
- (21 to 27) x 3 degrees of freedom
- curved element sides
- isotropic or anisotropic material data
- deformations considered: translational strain
- element loads:
  - initial strain (temperature load)
  - surface forces
  - line loads
  - gravitational load
  - general inertia load

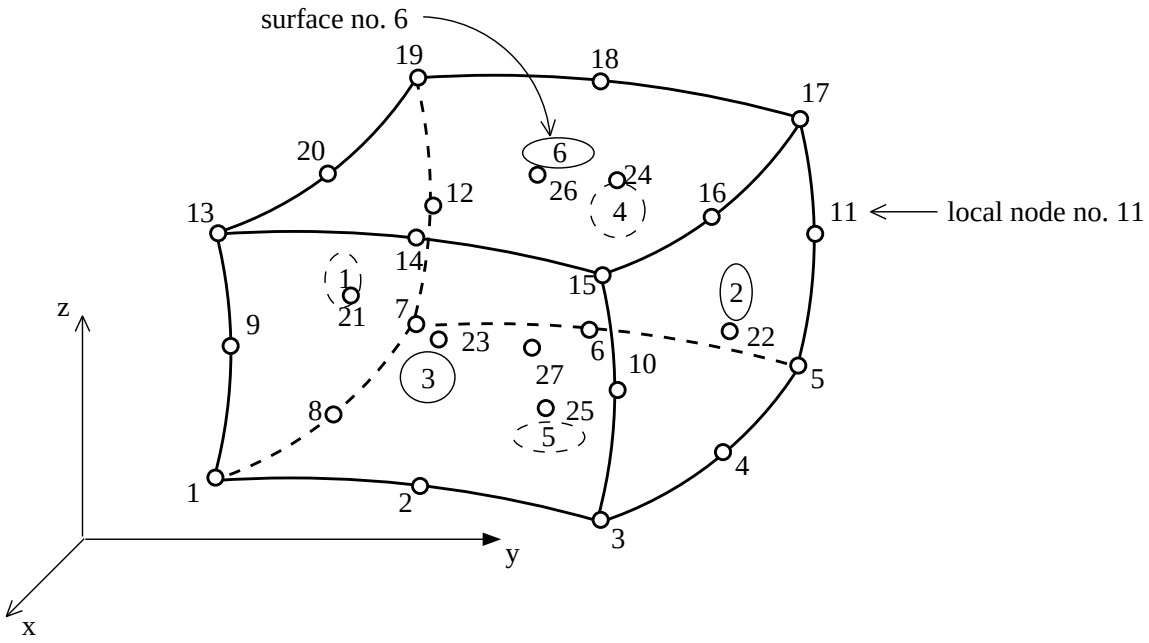


Figure 5-64 General hexahedron with local nodenumbering and corresponding surface numbering

The element may be without some of the nodes. Which nodes are not present is reflected in the element number:

100	The basic element with node 1 to 20 and node 27 present.
101	Nodes 1 to 20, node 21 and node 27 are present.
102	Nodes 1 to 20, node 22 and node 27 are present.
103	Nodes 1 to 20, node 21, node 22 and node 27 are present.

ETC.

If node 21 is present, 1 is added to the basic element type number.
If node 22 is present, 2 is added to the basic element type number.
If node 23 is present, 4 is added to the basic element type number.
If node 24 is present, 8 is added to the basic element type number.
If node 25 is present, 16 is added to the basic element type number.
If node 26 is present, 32 is added to the basic element type number.
If more than one of these nodes are present, the sum of the additions above is added.

element type number	Node position contains node number						
Node positions:	21	22	23	24	25	26	27
100	-	-	-	-	-	-	21
101	21	-	-	-	-	-	22
102		21	-	-	-	-	22
103	21	22	-	-	-	-	23
104	-	-	21	-	-	-	22
105	21	-	22	-	-	-	23
106	-	21	22	-	-	-	23
107	21	22	23	-	-	-	24
108	-	-	-	21	-	-	22
109	21	-	-	22	-	-	23
110	-	21	-	22	-	-	23
111	21	22	-	23	-	-	24
112	-	-	21	22	-	-	23
113	21	-	22	23	-	-	24
114	-	21	22	23	-	-	24
115	21	22	23	24	-	-	25
116	-	-	-	-	21	-	22
117	21	-	-	-	22	-	23
118	-	21	-	-	22	-	23
119	21	22	-	-	23	-	24
120	-	-	21	-	22	-	23
121	21	-	22	-	23	-	24
122	-	21	22	-	23	-	24
123	21	22	23	-	24	-	25
124	-	-	-	21	22	-	23
125	21	-	-	22	23	-	24
126	-	21	-	22	23	-	24
127	21	22	-	23	24	-	25
128	-	-	21	22	23	-	24
129	21	-	22	23	24	-	25
130	-	21	22	23	24	-	25
131	21	22	23	24	25	-	26
132	-	-	-	-	-	21	22
133	21	-	-	-	-	22	23
134	-	21	-	-	-	22	23

135	21	22	-	-	-	23	24
136	-	-	21	-	-	22	23
137	21	-	22	-	-	23	24
138	-	21	22	-	-	23	24
139	21	22	23	-	-	24	25
140	-	-	-	21	-	22	23
141	21	-	-	22	-	23	24
142	-	21	-	22	-	23	24
143	21	22	-	23	-	24	25
144	-	-	21	22	-	23	24
145	21	-	22	23	-	24	25
146	-	21	22	23	-	24	25
147	21	22	23	24	-	25	26
148	-	-	-	-	21	22	23
149	21	-	-	-	22	23	24
150	-	21	-	-	22	23	24
151	21	22	-	-	23	24	25
152	-	-	21	-	22	23	24
153	21	-	22	-	23	24	25
154	-	21	22	-	23	24	25
155	21	22	23	-	24	25	26
156	-	-	-	21	22	23	24
157	21	-	-	22	23	24	25
158	-	21	-	22	23	24	25
159	21	22	-	23	24	25	26
160	-	-	21	22	23	24	25
161	21	-	22	23	24	25	26
162	-	21	22	23	24	25	26
163	21	22	23	24	25	26	27

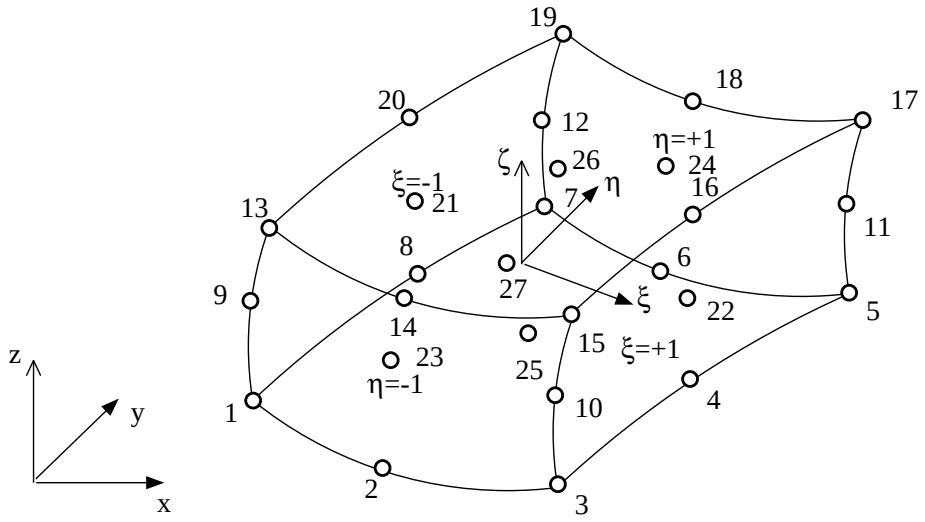


Figure 5-65 Typical general hexahedron

When surface loads are specified for the element side, the surface numbers shown are used for identification of the side in question. The node numbers 21 to 27 signifies the node position number. The actual node number within the actual element is 20 + number of nodes up to node position which are present in the actual element. Which node number shall actually be used in the tables below may be determined by the table of nodes in node position above. If the node position in question is empty the number of nodes on the line or side is one less, and compacted when the line is missing one node.

The local node numbering for each side is defined as follows:

	Node number								
	1	2	3	4	5	6	7	8	(9)
Side no.									
1	7	8	1	9	13	20	19	12	(21)
2	5	11	17	16	15	10	3	4	(22)
3	1	2	3	10	15	14	13	9	(23)
4	7	12	19	18	17	11	5	6	(24)
5	7	6	5	4	3	2	1	8	(25)
6	19	20	13	14	15	16	17	18	(26)

When line load is specified, the relation between the local node numbers and the loaded line will be as follows:

	Node number		
	1	2	3
Line no.			
1	1	2	3
2	3	4	5
3	5	6	7
4	7	8	1
5	1	9	13
6	3	10	15
7	5	11	17
8	7	12	19
9	13	14	15
10	15	16	17
11	17	18	19
12	19	20	13
13	8	(25)	4
14	6	(25)	2
15	9	(23)	10
16	2	(23)	14
17	10	(22)	11
18	4	(22)	16
19	11	(24)	12
20	6	(24)	18
21	12	(21)	9
22	8	(21)	20
23	20	(26)	16
24	14	(26)	18

Data types used for this element:

GELMNT1 *
GNODE *
GCOORD *
GELREF1 *
GELSTRP
MISOSEL * or
MISOPL * or
MORSSOL *
MTRSOL
BEUSLO
BELLO2
BEISTE
BGRAV
BNACCLO

*) Mandatory

The following restrictions are put on the data types

BELLO2	The SIDE definition is not used. The load components are given nodewise in global coordinates.
BEUSLO	Only one side identification can be given in SIDE.

Back to Section 3 **ELEMENT TYPES**

4 STRUCTURAL CONCEPTS AND CONCEPT ATTRIBUTES

4.1 The structure concept model

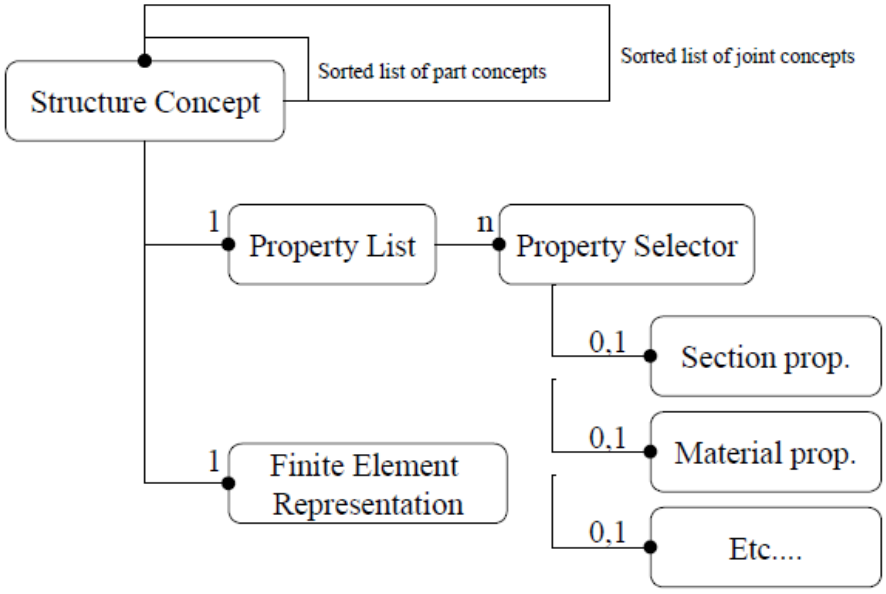


Figure 4.1: The structure concept model

The structure concept may represent concepts on any level (assemblies or basic parts).

The function of a concept is determined by its type and role.

The structure concepts can contain other structure concepts (part concepts).

The structure concept can refer joint concepts.

The structure concept has a Finite Element representation given on lowest concept level.

The structure concept has properties available via a property list and property selectors.

The structure properties are recursively inherited from parent concepts if not specified on current level.

There is only one property list to each concept.

Each property selector may be referred to from several property lists (i.e. concepts may share property selectors).

Each property value may be referred to from several property selectors (i.e. property selectors may share property values).

4.2 Structure concept descriptions

SCONCEPT	Structure concept definition	see Section 4.2.1
SCONMESH	Structure concept finite element representation	see Section 4.2.2
SCONPLIS	Structure concept property list	see Section 4.2.3
SPROCODE	Structure concept code checking attributes	see Section 4.2.4
SPROECCE	Structure concept eccentricity attributes	see Section 4.2.5
SPROHYDR	Structure concept hydrodynamic attributes	see Section 4.2.6
SPROMATR	Structure concept material attributes	see Section 4.2.7
SPROORIE	Structure concept orientation attributes	see Section 4.2.8
SPROPILE	Structure concept pile attributes	see Section 4.2.9
SPROSECT	Structure concept cross section attributes	see Section 4.2.10
SPROSEGM	Structure concept segment attributes	see Section 4.2.11
SPROSELE	Structure concept property selectors	see Section 4.2.12
SPROSOIL	Structure concept soil data	see Section 4.2.13
TDSCONC	Structure concept referable text	see Section 4.2.14

The structure concept data types are described on the following pages.

4.2.1 **SCONCEPT**: Structure concept definition

This data type defines a structural concept, with definition of the function of the concept, and reference to its parent, parts and joints concepts.

Data type definition:

	SCONCEPT	Data type reference.
	NFIELD	Number of data fields on this data type (including this field).
→	IRCON	Concept reference number (unique).
	SCONTYPE	Structure concept type number (see Table 4.1).
	SCONROLE	Structure concept role number (see Table 4.2).
	IRPARENT	Parent concept reference number (or 0.0 if no parent).
	NPART	Number of part concepts.
	NJOINT	Number of joint (connection) concepts.
	Repeat for NPART	
	IRPART	Part concept reference numbers.
	Next PART	
	Repeat for NJOINT	
	IRJOINT	Joint concept reference numbers.
	Next JOINT	

Note:

- Typically, a member will refer to n number of part concepts (segments), and two joint concepts.

Table 4.1: Structure concept type definitions

SCONTYPE	Data type	Comment
0	Undefined	
1	Joint	Is a connection between Members.
2	Segment	Is a part of a Member with uniform or tapered cross section.
3	Member	Runs between two Joints.
4	Pile joint	Is a start or end node for a Pile.
5	Pile segment	Is a part of a Pile with uniform cross section.
6	Pile	Runs between two Pile joints.
7	Segmented beam	One or more segments, where internal segments may be cans and cones. Stubs may only be at beam end segments. May include no reference to joints.

Table 4.2: Structure concept role definitions

SCONROLE	Data type	Comment
0	Undefined	
1	Stub	Is a Segment role.
2	Can	Is a Segment role.
3	Cone	Is a Segment role.
4	MidSection	Is a Segment role.
5	Chord	Is a Member role.
6	Brace	Is a Member role.
7	MainPile	Is a Pile role.
8	InnerPile	Is a Pile role.
9	ConductorPile	Is a Pile role.
10	PileGroup	Is a Pile role.

Back to Section 4.2 **Structure concept descriptions**

4.2.2 **SCONMESH**: Structure concept finite element representation

This data type defines the finite element representation of the lowest level structural concepts, i.e. the connection to internal element or node identities.

Data type definition:

	SCONMESH	Data type reference.
	NFIELD	Number of data fields on this data type (including this field).
→	IRCON	Concept reference number (unique).
	NUMREP	Number of finite element representation types.
	Repeat for NUMREP	
	TYPREP	finite element representation type (see Table 4.3).
	NFEREP	Number of finite element representations.
	Repeat for NFEREP	
	IRFEREP	finite element representation reference numbers (internal id).
	Next FEREP	
	Next REP	

Note:

- **SCONMESH** to be given on lowest concept level, i.e. for segments and joints.

Table 4.3: Definition of finite element representation types

TYPREP	Data type	Comment
1	Node	Refers internal node number (**GNODE**).
2	Element	Refers internal element number (**GELMNT1**).
3	Set	Refers internal set number (**GSETMEMB**) (not to be implemented).

Back to Section 4.2 **Structure concept descriptions**

4.2.3 **SCONPLIS**: Structure concept property list

This data type defines the property list. Through the list the concept indirectly refers to actual properties. The property items do not contain the properties themselves, but refer to them through property selectors.

There is only one property list to each concept.

Data type definition:

	SCONPLIS	Data type reference.
	NFIELD	Number of data fields on this data type (including this field).
→	IRCON	Concept reference number (unique).
	NUMPSEL	Number of property selectors.
	Repeat for NUMPSEL	
	TYPEPROP	Property attribute type number (must be a **SPROSELE** in first implementation) (see Table 4.4).
	IRPSELE	Property selector reference numbers (internal id).
	Next PSEL	

Table 4.4: Definition of property attribute types

TYPEPROP	Data type	Type of property	Comment
1	**SPROSELE**	Structure concept property selectors	
2	**SPROSECT**	Structure concept cross section attributes	
3	**SPROMATR**	Structure concept material attributes	
4	**SPROSEGM**	Structure concept segment attributes	E.g. cone data.
5	**SPROHYDR**	Structure concept hydrodynamic attributes	
6	**SPROCODE**	Structure concept code checking attributes	
7	**SPROORIE**	Structure concept orientation attributes	
8	**SPROECCE**	Structure concept eccentricity attributes	
9	**SPROPILE**	Structure concept pile attributes	

Back to Section 4.2 **Structure concept descriptions**

4.2.4 **SPROCODE**: Structure concept code checking attributes

This data type defines the property attributes connected to code checking.

A property attribute may be shared by many property selectors.

Data type definition:

	SPROCODE	Data type reference.
	NFIELD	Number of data fields on this data type (including this field).
→	IRPATTR	Property attribute reference number (unique).
	NUMATTR	Number of attributes for current property.
	Repeat for NUMATTR	
	TYPEATTR	Attribute number, see Table 4.5.
	VALUE	Attribute value.
	Next PSEL	

Note:

- For rule based alternatives (i.e. 602 and 605, see Table 4.5), the attribute value is of no concern and may be set to any number.

Table 4.5: Definition of property values

TYPEATTR	Attribute value	Comment
600	Buckling length Y	Concept local axis.
601	Buckling length Z	Concept local axis.
602	Buckling length joint-to-joint	
603	Effective length factor Y	Concept local axis.
604	Effective length factor Z	Concept local axis.
605	Calculated acc. to NPD-NS3472	
606	Moment amplification factor Y-direction	Concept local axis.
607	Moment amplification factor Z-direction	Concept local axis.
608	Moment amplification method Y-direction	Concept local axis.
609	Moment amplification method Z-direction	Concept local axis.
610	Unbraced length upper flange	
611	Unbraced length lower flange	
612	Stiffener spacing member	
613	Stiffener spacing cone	
614	Bending coefficient	
615	Bending coefficient rulebased	

Back to Section 4.2 **Structure concept descriptions**

4.2.5 **SPROECCE**: Structure concept eccentricity attributes

This data type defines the property attributes connected to eccentricity / gap of concepts.

A property attribute may be shared by many property selectors.

Data type definition:

	SPROECCE	Data type reference.
	NFIELD	Number of data fields on this data type (including this field).
→	IRPATTR	Property attribute reference number (unique).
	NUMATTR	Number of attributes for current property.
	Repeat for NUMATTR	
	TYPEATTR	Attribute number, see Table 4.6.
	VALUE	Attribute value.
	Next PSEL	

Table 4.6: Definition of property values

TYPEATTR	Attribute value	Comment

Back to Section 4.2 **Structure concept descriptions**

4.2.6 **SPROHYDR**: Structure concept hydrodynamic attributes

This data type defines the hydrodynamic property attributes. A property attributes may be shared by many property selectors.

Data type definition:

	SPROHYDR	Data type reference.
	NFIELD	Number of data fields on this data type (including this field).
→	IRPATTR	Property attribute reference number (unique).
	NATTR	Number of attributes for current property.
	Repeat for NATTR	
	TYPEATTR	Attribute number, see Table 4.7.
	VALUE	Attribute value.
	Next ATTR	

Note:

- Even if **SPROHYDR** is revised to include the additional hydrodynamic attributes as shown in the table above, there will still be a need to document actual values used in Wajac, as Wajac input may override the information stored on **SPROHYDR**. For documenting Wajac values used, we have **WHYCOEL** and **WHYPREL**.

Table 4.7: Definition of property values

TYPEATTR	Attribute value	Comment
500	Flooding coefficient	1.0 = completely flooded (open). 0.0 = not flooded (sealed).
501	Drag coefficient X (CDx)	Concept local co-ordinate system.
502	Drag coefficient Y (CDy)	
503	Drag coefficient Z (CDz)	
504	Inertia coefficient X (CMx)	Concept local co-ordinate system.
505	Inertia coefficient Y (CMy)	
506	Inertia coefficient Z (CMz)	
507	Hydrodynamic diameter	
508	Buoyancy nonflooded area	
509	Buoyancy flooded area	
510	Marine growth	

Back to Section 4.2 **Structure concept descriptions**

4.2.7 **SPROMATR**: Structure concept material attributes

This data type defines the material property attributes. The property value contains the actual values for a particular material definition.

A property attribute may be shared by many property selectors.

Data type definition:

	SPROMATR	Data type reference.
	NFIELD	Number of data fields on this data type (including this field).
→	IRPATTR	Property attribute reference number (unique).
	NUMATTR	Number of attributes for current property.
	Repeat for NUMATTR	
	TYPEATTR	Attribute number, see Table 4.8.
	VALUE	Attribute value.
	Next PSEL	

Table 4.8: Definition of property values

TYPEATTR	Attribute value	Comment
300	Yield strength	Currently in use by pile only.
301	Tensile strength	Currently not in use.

Back to Section 4.2 **Structure concept descriptions**

4.2.8 **SPROORIE**: Structure concept orientation attributes

This data type defines the property attributes connected to orientation of concepts.

A property attribute may be shared by many property selectors.

Data type definition:

	SPROORIE	Data type reference.
	NFIELD	Number of data fields on this data type (including this field).
→	IRPATTR	Property attribute reference number (unique).
	NUMATTR	Number of attributes for current property.
	Repeat for NUMATTR	
	TYPEATTR	Attribute number, see Table 4.9.
	VALUE	Attribute value.
	Next PSEL	

Table 4.9: Definition of property values

TYPEATTR	Attribute value	Comment

Back to Section 4.2 **Structure concept descriptions**

4.2.9 **SPROPILE**: Structure concept pile attributes

This data type defines the specific property attributes connected to pile concepts.

A property attribute may be shared by many property selectors.

Data type definition:

	SPROPILE	Data type reference.
	NFIELD	Number of data fields on this data type (including this field).
→	IRPATTR	Property attribute reference number (unique).
	NUMATTR	Number of attributes for current property.
	Repeat for NUMATTR	
	TYPEATTR	Attribute number, see Table 4.10.
	VALUE	Attribute value.
	Next PSEL	

Table 4.10: Definition of property values

TYPEATTR	Attribute value	Comment
900	Pile tip boundary condition	"Tip code".
901	Pile head fixed to dummy pile, i.e. rigidely connected to	Refers to internal node number in the structure (dummy pile not modelled).

Back to Section 4.2 **Structure concept descriptions**

4.2.10 **SPROSECT**: Structure concept cross section attributes

This data type defines the cross section property attributes. The property value contains the actual values for a particular property, e.g. the necessary data to define a stub section.

A property attribute may be shared by many property selectors.

Data type definition:

	SPROSECT	Data type reference.
	NFIELD	Number of data fields on this data type (including this field).
→	IRPATTR	Property attribute reference number (unique).
	NUMATTR	Number of attributes for current property.
	Repeat for NUMATTR	
	TYPEATTR	Attribute number, see Table 4.11.
	VALUE	Attribute value.
	Next PSEL	

Table 4.11: Definition of property values

TYPEATTR	Attribute value	Comment
200	Outer diameter	Currently not in use.
201	Wall thickness	Currently not in use.
202	Axial stiffness modifier	Pile modelling (EA-NEW).
203	Bending stiffness modifier	Pile modelling (EI-NEW).
204	Shear stiffness modifier	Pile modelling (GA-NEW).
205	Torsional stiffness modifier	Pile modelling (GIP-NEW).
206	Unit density of fluid inside pile	Pile modelling (DENS-FLUID).

Back to Section 4.2 **Structure concept descriptions**

4.2.11 **SPROSEGM**: Structure concept segment attributes

This data type defines the property attributes connected to specific segment information.

A property attribute may be shared by many property selectors.

Data type definition:

	SPROSEGM	Data type reference.
	NFIELD	Number of data fields on this data type (including this field).
→	IRPATTR	Property attribute reference number (unique).
	NUMATTR	Number of attributes for current property.
	Repeat for NUMATTR	
	TYPEATTR	Attribute number, see Table 4.12.
	VALUE	Attribute value.
	Next PSEL	

Table 4.12: Definition of property values

TYPEATTR	Attribute value	Comment
400	Cone diameter at start	Currently not in use.
401	Cone diameter at end	Currently not in use.
402	Cone wall thickness	Currently not in use.
403	Cone inclination	Currently not in use.
404	Segment length	Currently not in use.

Back to Section 4.2 **Structure concept descriptions**

4.2.12 **SPROSELE**: Structure concept property selectors

This data type defines the property selector. The property selector represents the sharing of property values between different concepts.

A property selector selects one or more property attribute types. It may be referenced by many concepts. Changing the content of a property selector will change the connected properties of all the concepts that references it.

Data type definition:

	SPROSELE	Data type reference.
	NFIELD	Number of data fields on this data type (including this field).
→	IRPSELE	Property selector reference number (unique).
	NUMTYPE	Number of property attribute types.
	Repeat for NUMTYPE	
	TYPEPROP	Property attribute type number (ref. Table 4.1).
	IRPATTR	Property attribute reference numbers (internal id).
	Next TYPE	

Note:

- A property selector may only refer to one property attribute of each type, e.g. only one **SPROSECT** may be referred from each **SPROSELE**.

Back to Section 4.2 **Structure concept descriptions**

4.2.13 **SPROSOIL**: Structure concept soil data

This data type defines the soil related data types.

Data type definition:

	SPROSOIL	Data type reference.
	NFIELD	Number of data fields on this data type (including this field).
→	IRPATTR	Property reference number (unique).
	NUMTYP	Type of data.
	ATTRIB	Common for all **SPROSOIL** data types (value = 1000) data field 5 through n varies with NUMTYP, see Table 4.13.

Table 4.13: Definition of data types

NUMTYP	Data type	Data fields 5 through n
10	Soil types	5: soil type (3 = sand, 1 = clay). 6: soil number. 7 - n: ref. data definitions below.
11	Profile / layer definition	5: soil id. 6: number of soil divisions. 7 - n: ref. data definitions below.
12	Soil (Gensod) parameters	5: parameter set number (1 through 6). 6 - n: ref. data definitions below.
13	PY TZ QZ codes	5: z - level. 6 - n: ref. data definitions below.
14	Skin friction data	5: z - level. 6 - n: ref. data definitions below.
15	Tip resitance data	5: z - level. 6 - n: ref. data definitions below.

Soil types definitions:

SPROSOIL	NFIELD	IRPATTR	NUMTYP	ATTRIB
	SOILTYP	SOILNO	GAMTOT	PHI
	SU.Z=0	SU.Z=100	EPSC	OCR
	API-J	OPEN	R/P-RAT	TZZREZ

SOILTYP	Soil type (1 = clay, 3 = sand (2 not in use)).
SOILNO	Soil id (number).
GAMTOT	Total unit weigth.
PHI	Angle of internal friction (degrees).
SU.Z=0	Undrained shear strength at z=0.
SU.Z=100	Undrained shear strength at z=100.
EPSC	Strain at half of max stress.
OCR	Over-consolidation ratio.
API-J	J factor for API code.
OPEN	Code for open gap (0 or 1).
R/P-RAT	Residual / peak skin friction ratio.
TZZES	T-Z curve z displacement from peak to residual skin friction.

Soil profile / layer definitions:

SPROSOIL	NFIELD	IRPATTR	NUMTYP	ATTRIB
	SOILID	NDIV	Z-BOTM(1)	NOFLAY(1)
	SOILNO(1)	. . .	. . .	. . .
	Z-BOTM(NDIV)	NOFLAY(NDIV)	SOILNO(NDIV)	

SOILID	Soil profile id (number). Currently only id 1 allowed.
NDIV	Number of soil divisions (types) in profile.
Z-BOTM	Z level (global co-ordinates) at bottom of division (type) n.
NOFLAY	Number of layers within the soil division (type).
SOILNO	Soil type to be used in soil division.

Gensod control data definitions, (SETNUM = 1):

SPROSOIL	NFIELD	IRPATTR	NUMTYP	ATTRIB
	SETNUM	MUDLINE	ZONINF	RFTZ

MUDLINE	The Z level defining mudline.
ZONINF	The value to be used for zone of influence.
RFTZ	The value to be used for curve fitting.

Gensod control data definitions, CONTROL SECTION (SETNUM = 2):

SPROSOIL	NFIELD	IRPATTR	NUMTYP	ATTRIB
	SETNUM	CONFRC	CONLTH	GAMMAW
	ATMPRS	ZCYCL	SUSTIF	

CONFRC	Old-force-unit = confrc * new-force-unit 1mn = 1000*1kn.
CONLTH	Old-length-unit = conlth * new-length-unit 1m = 3.28*1ft.
GAMMAW	Unit weight of water.
ATMPRS	Atmospheric pressure.
ZCYCL	Z-level down to which cyclic p-y data shall be generated.
SUSTIF	Use stiff clay p-y procedures if su.gt.sustf (api only).

Gensod control data definitions, MATERIAL COEFFICIENT SECTION (SETNUM = 3):

SPROSOIL	NFIELD	IRPATTR	NUMTYP	ATTRIB
	SETNUM	SFTPHI	SFSU	SFSKF
	SFSIGT			

SFTPHI	Material coefficient on tan(phi).
SFSU	Material coefficient on undrained shear strength.
SFSKF	Material coefficient on pile skin friction.
SFSIGT	Material coefficient on pile tip resistance.

Gensod control data definitions, PILE DIAM. AND GROUP EFFECT SECTION (SETNUM = 4):

SPROSOIL	NFIELD	IRPATTR	NUMTYP	ATTRIB
	SETNUM	ESOL0	ESOL1	POSAVR

ESOL0	E-soil for group effect calculation:
ESOL1	ESOIL(Z) = ESOL0 + ESOL1*Z
POSAVR	Soil average poisson ratio for group effects.

Gensod control data definitions, SOIL SURFACE & GROUND WATER SECTION (SETNUM = 5):

SPROSOIL	NFIELD	IRPATTR	NUMTYP	ATTRIB
	SETNUM	ZSURF	SCRGEN	SCRLOC

continued . . .

	SLOPE	ZGRWT	GAMPWP	

ZSURF	Z-level of non-scoured soil surface.
SCRGEN	Depth of general scour below zsurf.
SCRLOC	Depth of local scour below zsurf.
SLOPE	Side slope (degrees) of local scour holes.
ZGRWT	Z-level of ground water table.
GAMPWP	Unit weight of ground water (used to find pore water pressure).

Gensod control data definitions, LOADS AT SOIL SURFACE SECTION (SETNUM = 6):

SPROSOIL	NFIELD	IRPATTR	NUMTYP	ATTRIB
	SETNUM	SIGSRF	DPEMB	AEMB
	BEMB	DPCIRC	RADIUS	

SIGSRF	Vertical stress at surface.
DPEMB	Vertical stress under embankment loading.
AEMB	Width a of embankment sloping part.
BEMB	Pile position with respect to embankment toe (positive outside).
DPCIRC	Vertical stress under circular loaded area.
RADIUS	Radius of circular loaded area (pile is in center).

PY - TZ - QZ code definitions:

SPROSOIL	NFIELD	IRPATTR	NUMTYP	ATTRIB
	Z-LEVEL	PY-CODE	TZ-CODE	QZ-CODE

Z-LEVEL	Z-level where values are given.
PY-CODE	PY code to be used.
TZ-CODE	TZ code to be used.
QZ-CODE	QZ code to be used.

Skin friction & tip resistance data definitions:

SPROSOIL	NFIELD	IRPATTR	NUMTYP	ATTRIB

continued . . .

	Z-LEVEL DS/DIA-RAT	SKIN-CMP SIG-TIP	SKIN-TNS POIS	G0-SOIL DT/DIA-RAT

Z-LEVEL	Z-level where values are given.
SKIN-CMP	Peak skin friction in compression.
SKIN-TNS	Peak skin friction in tension.
G0-SOIL	Initial value of soil shear modulus.
DS/DIA-RAT	Ratio between dipl. to reach peak skin friction and pile diameter.
SIG-TIP	Peak tip stress acting at pile tip.
POIS Soil	Poisson ration.
DT/DIA-RAT	Ratio between dipl. to reach peak tip stress and pile diameter.

Back to Section 4.2 **Structure concept descriptions**

4.2.14 **TDSCONC**: Structure concept referable text

TDSCONC	NFIELD	→ IRCON	CODNAM	CODTXT
	Name			
	Comment line			
	. . .			
	Comment line			

This data type will associate a name to a structure concept.

NFIELD — Number of numeric data fields at this data type before text data (MAX = 1024).

IRCON — Concept reference number (unique).

CODNAM — Coded dimension of the Name:
CODNAM = NLNAM*100 + NCNAM. The inverse relation will then be:
NLNAM = integer part of (CODNAM/100)
NCNAM = remaindering of (CODNAM/100)
NLNAM number lines used to store the name. Legal range = [0,1]
= 0, no name defined
= 1, name is defined
NCNAM - number of characters in the name. Legal range = [0,64]

CODTXT — Coded dimension of the Comment:
CODTXT = NLTXT*100 + NCTXT. The inverse relation will then be:
NLTXT = integer part of (CODTXT/100)
NCTXT = remaindering of (CODTXT/100)
NLTXT - number of lines used to store the comment. Legal range = [0,5]
= 0, no comments defined
≥ 1, number of physical records with comments
NCTXT - number of characters in the comment – each comment line must be of the same length. Legal range = [0,64]

Name — A user set name.

Comment line — User set comment lines.

Note:

- **TDSCONC** is currently in use only for **SCONCEPT** of type Member and Pile.

Back to Section 4.2 **Structure concept descriptions**

4.2.15 Example

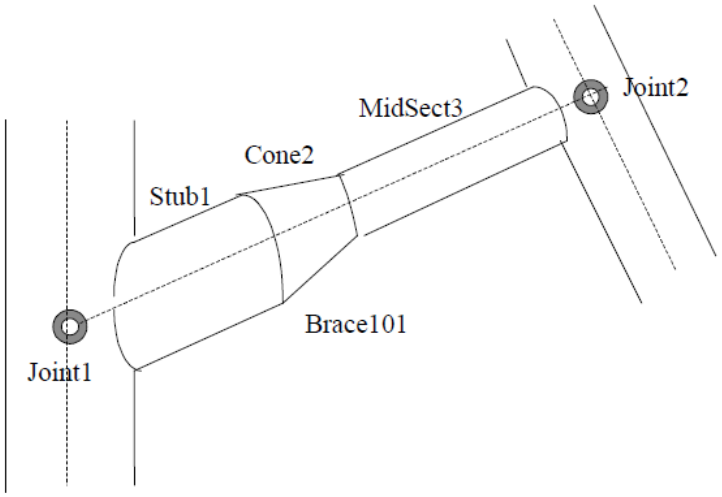


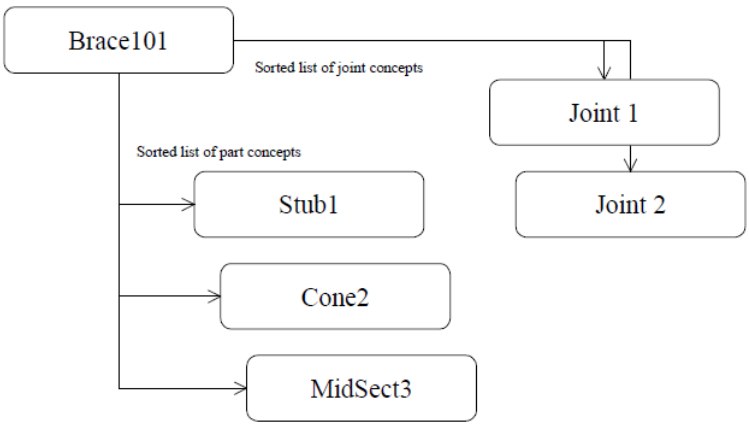
Figure 4.2: The structure concept example model

The simple model has a member with a conical transition and stub at one end, see Figure 4.2 above.

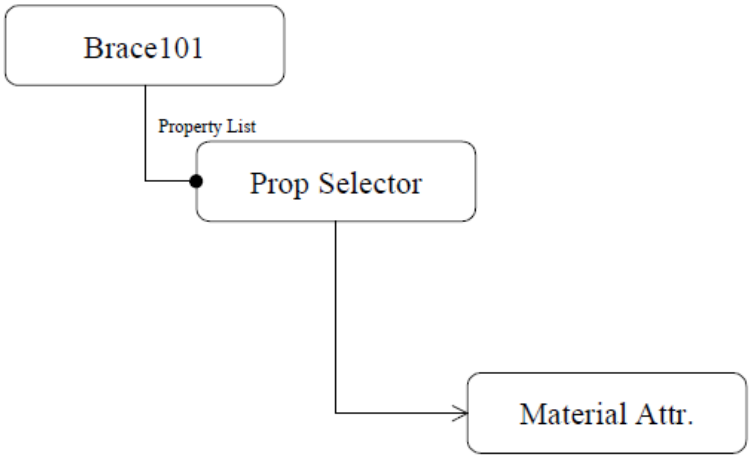
The following example SIF file shows:

- Concepts with types and roles, and the part and joint relations.
- Specification of material yield strength.
- References to FE representation with three beam elements.
- Specification of joint coordinates. (Defined via reference to internal node number.)

The instansiated object model is as follows:



The material attributes are assigned to the Brace101:



SESAM Interface Input file (T1.FEM) representing "Brace 101":

```
IDENT     1.00000000E+00  1.00000000E+00  3.00000000E+00  0.00000000E+00
DATE      1.00000000E+00  0.00000000E+00  4.00000000E+00  7.20000000E+01
        DATE:     10-Sep-2016         TIME:          14:05:46
        PROGRAM:  SESAM GeniE         VERSION:       V7.3-15 07-Mar-2016
        COMPUTER: X86 Windows         INSTALLATION:
        USER:     lzhan               ACCOUNT:
GNODE     1.00000000E+00  1.00000000E+00  6.00000000E+00  1.23456000E+05
GNODE     3.00000000E+00  2.00000000E+00  6.00000000E+00  1.23456000E+05
GNODE     4.00000000E+00  3.00000000E+00  6.00000000E+00  1.23456000E+05
GNODE     2.00000000E+00  4.00000000E+00  6.00000000E+00  1.23456000E+05
GCOORD    1.00000000E+00  0.00000000E+00  0.00000000E+00  0.00000000E+00
GCOORD    2.00000000E+00  2.78543019E+00  0.00000000E+00  1.11417210E+00
GCOORD    3.00000000E+00  4.64238358E+00  0.00000000E+00  1.85695338E+00
GCOORD    4.00000000E+00  1.00000000E+01  0.00000000E+00  4.00000000E+00
GELMNT1   1.00000000E+00  1.00000000E+00  1.50000000E+01  0.00000000E+00
          1.00000000E+00  2.00000000E+00  0.00000000E+00  0.00000000E+00
GELMNT1   2.00000000E+00  2.00000000E+00  1.50000000E+01  0.00000000E+00
          2.00000000E+00  3.00000000E+00  0.00000000E+00  0.00000000E+00
GELMNT1   3.00000000E+00  3.00000000E+00  1.50000000E+01  0.00000000E+00
          3.00000000E+00  4.00000000E+00  0.00000000E+00  0.00000000E+00
GPIPE     1.00000000E+00  7.40000010E-01  8.00000012E-01  2.99999993E-02
          1.00000000E+00  1.00000000E+00  0.00000000E+00  0.00000000E+00
GBEAMG    1.00000000E+00  0.00000000E+00  7.25707933E-02  1.07731372E-02
          5.38656861E-03  5.38656861E-03  0.00000000E+00  2.69328430E-02
          1.34664215E-02  1.34664215E-02  3.63221057E-02  3.63221057E-02
          0.00000000E+00  0.00000000E+00  8.89800023E-03  8.89800023E-03
GPIPE     2.00000000E+00  5.50000012E-01  6.00000024E-01  2.50000004E-02
          1.00000000E+00  1.00000000E+00  0.00000000E+00  0.00000000E+00
GBEAMG    2.00000000E+00  0.00000000E+00  4.51603979E-02  3.73984477E-03
          1.86992239E-03  1.86992239E-03  0.00000000E+00  1.24661485E-02
          6.23307424E-03  6.23307424E-03  2.26086304E-02  2.26086304E-02
          0.00000000E+00  0.00000000E+00  4.13541729E-03  4.13541729E-03
GPIPE     3.00000000E+00  6.39999986E-01  6.99999988E-01  2.99999993E-02
          1.00000000E+00  1.00000000E+00  0.00000000E+00  0.00000000E+00
GBEAMG    3.00000000E+00  0.00000000E+00  6.31460175E-02  7.10076885E-03
          3.55038443E-03  3.55038443E-03  0.00000000E+00  2.02879105E-02
          1.01439552E-02  1.01439552E-02  3.16151790E-02  3.16151790E-02
```

```
          0.00000000E+00  0.00000000E+00  6.73799962E-03  6.73799962E-03
MISOSEL   1.00000000E+00  2.10000003E+11  3.00000012E-01  7.85000000E+03
          0.00000000E+00  1.20000004E-05  0.00000000E+00  0.00000000E+00
GUNIVEC   1.00000000E+00 -3.71390700E-01  0.00000000E+00  9.28476691E-01
GUNIVEC   2.00000000E+00 -3.71390671E-01  0.00000000E+00  9.28476751E-01
GUNIVEC   3.00000000E+00 -3.71390671E-01  0.00000000E+00  9.28476691E-01
GELREF1   1.00000000E+00  1.00000000E+00  0.00000000E+00  0.00000000E+00
          0.00000000E+00  0.00000000E+00  0.00000000E+00  0.00000000E+00
          1.00000000E+00  0.00000000E+00  0.00000000E+00  1.00000000E+00
GELREF1   2.00000000E+00  1.00000000E+00  0.00000000E+00  0.00000000E+00
          0.00000000E+00  0.00000000E+00  0.00000000E+00  0.00000000E+00
          3.00000000E+00  0.00000000E+00  0.00000000E+00  2.00000000E+00
GELREF1   3.00000000E+00  1.00000000E+00  0.00000000E+00  0.00000000E+00
          0.00000000E+00  0.00000000E+00  0.00000000E+00  0.00000000E+00
          2.00000000E+00  0.00000000E+00  0.00000000E+00  3.00000000E+00
IEND                0.00            0.00            0.00            0.00
```

Example of the new data types:

TDSCONC and **SCONCEPT** for Brace 101:

```
TDSCONC   4.00000000E+00  1.01000000E+02  1.08000000E+02  0.00000000E+00
        BRACE101
SCONCEPT  1.20000000E+01  1.01000000E+02  3.00000000E+00  6.00000000E+00
          0.00000000E+00  3.00000000E+00  2.00000000E+00  1.04000000E+02
          1.05000000E+02  1.06000000E+02  1.02000000E+02  1.03000000E+02
```

SCONCEPT and **SCONMESH** for Joint 2:

```
SCONCEPT  3.00000000E+00  1.03000000E+02  1.00000000E+00
SCONMESH  6.00000000E+00  1.03000000E+02  1.00000000E+00  1.00000000E+00
          1.00000000E+00  4.00000000E+00
```

SCONCEPT and **SCONMESH** for Stub 1:

```
SCONCEPT  5.00000000E+00  1.04000000E+02  2.00000000E+00  1.00000000E+00
          1.01000000E+02
SCONMESH  6.00000000E+00  1.04000000E+02  1.00000000E+00  2.00000000E+00
          1.00000000E+00  1.00000000E+00
```

SCONCEPT and **SCONMESH** for Cone 1:

```
SCONCEPT  5.00000000E+00  1.05000000E+02  2.00000000E+00  3.00000000E+00
          1.01000000E+02
SCONMESH  6.00000000E+00  1.05000000E+02  1.00000000E+00  2.00000000E+00
          1.00000000E+00  2.00000000E+00
```

SCONPLIS, **SPROSELE** and **SPROMATR** supporting concept 101 (referes to one property value; Yield strength with value = 3.45e+8, given in **SPROMATR** no. 28, via **SPROSELE** no. 10):

```
SCONPLIS  5.00000000E+00  1.01000000E+02  1.00000000E+00  1.00000000E+00
          1.00000000E+01
SPROSELE  5.00000000E+00  1.00000000E+01  1.00000000E+00  3.00000000E+00
          2.80000000E+01
SPROMATR  5.00000000E+00  2.80000000E+01  1.00000000E+00  3.00000000E+02
          3.45000000E+08
```

Back to Section 4.2 **Structure concept descriptions**

5 FIRST LEVEL DATA

5.1 Additional Element Data

First level data

ACFD	General Crack Data	see Section 5.1.1
ADDATA	Additional User defined Basic Element Data	see Section 5.1.2

5.1.1 **ACFD**: General Crack Data

ACFD	→ IGLB	→ ILOK	NEP	IMOD
	MKP	DX	DY	DZ
	$I_{(1)}$	$I_{(2)}$	...	$I_{(NEP)}$
	NSIF	$ISIF_{(1)}$	$ISIF_{(2)}$	...
	$ISIF_{(NSIF)}$			

A crack is defined by a set of **ACFD** data types on the Interface file. There is one **ACFD**-data type for each crack front node.

IGLB — Sequence numbering of crack front nodes.

ILOK — External node number of the crack front node.

NEP — Perturbation option for nodes to be perturbed.

- < 0 — Nodes not supplied; use all side nodes closest to the crack tip and nodes coinciding with ILOK.
- = 0 — Crack tip node only is perturbed.
- > 0 — Number of nodes supplied. The nodes are perturbed in addition to the crack tip node.

IMOD — Conversion option.

Governs the conversion from crack driving force to stress intensity factor according to:

- = 1 — $K = \sqrt{E/(1-\nu^2)}\sqrt{G}$
- = 2 — $K = \sqrt{EG}$
- = 3 — $K = \sqrt{E/(1+\nu^2)}\sqrt{G}$

where K is stress intensity factor, G is crack driving force, E is YoungâĂŹs modulus and ν is PoissonâĂŹs ratio.

MKP — Perturbation direction option:

- = 0 — Perturbation direction given in (DX,DY,DZ)
- > 0 — MKP=external node number of node which defines direction together with ILOK
- = -1 — Automatic computation by analysis program (only relevant for double crack surface).
- = -2 — Point (DX,DY,DZ) in crack surface is given.

The perturbation direction must be normal to the crack front and tangential to the crack surface.

DY, DY, DZ — Coordinates (in super element global co-ordinate system) as MKP specifies.

$I_{(1)}$

$I_{(2)}$ — If NEP > 0: External node numbers of nodes to be perturbed together with crack front node ILOK. Nodes with the same position as ILOK may be left out of specification.

⋮

$I_{(NEP)}$

NSIF — Number of associated nodes used in calculation of Stress Intensity Factors (SIF) in node ILOK.

- = 2 — Node ILOK is a mid-side node.
- = 4 — Node ILOK is a corner node.
- = 0 — Node ILOK is a corner node and the last node along crack front. NSIF = 0 is treated as NSIF = 4 (four associated nodes).

Note that two crack fronts may be present in a super element. Separation between the two crack fronts is identified with NSIF = 0. IGLB goes from 1 to the total number of crack front nodes along the two crack fronts.

$ISIF_{(1)}$

$ISIF_{(2)}$ — External node numbers for nodes in crack surface.

⋮

$ISIF_{(NSIF)}$

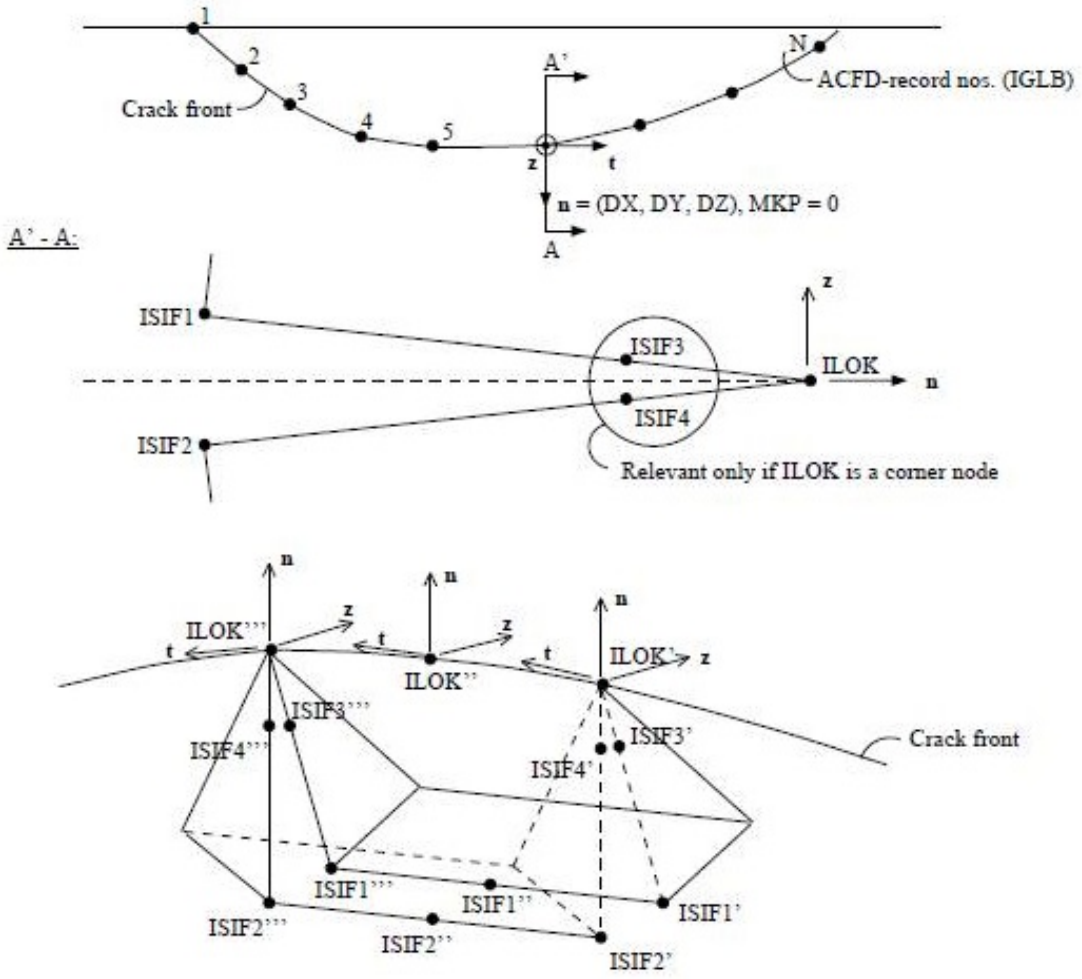


Figure 5.1: Definition of local axis system

Back to Section 5.1 **Additional Element Data**

5.1.2 **ADDATA**: Additional User defined Basic Element Data

ADDATA	→ ADDNO	NPAR	$PAR_{(1)}$	$PAR_{(2)}$
	...	...	$PAR_{(NPAR)}$	

ADDNO	Additional data type number, i.e. reference number referring to additional data specifications.
NPAR	Number of parameters specified by the user.
$PAR_{(1)}$	
$PAR_{(2)}$	Values for the different terms of the matrix input. The sequence is according to the convention of the analysis program. Relevant only if UNIT=0.
⋮	
$PAR_{(NPAR)}$	

Whenever the analysis program requires data that are particular to a basic element of the super element type in question (and which are not defined elsewhere) this data type may be employed to assign the data.

The data assigned above are data intended for one or several basic elements of a referenced super element type. In the data type labelled **GELREF1** of that particular super element type an ADDNO must be included which corresponds with the ADDNO on the **ADDATA** data type.

The definition of the various parameters is depending on the analysis program to be used and must be revised when switching to another analysis program.

Back to Section 5.1 **Additional Element Data**

5.2 Boundary Conditions, Loads and Point Masses

First level data

BAHAMAS	Element with Added Mass	see Section 5.2.1
BEDRAG1	Hydrodynamic Drag and Damping from Wave Load Program	see Section 5.2.2
BEISTE	Element with Initial Strain Due to Thermal Expansion	see Section 5.2.3
BELFIX	Flexible Joint/Hinge	see Section 5.2.4
BELLAX	Surface Load on Axi-symmetric Solid	see Section 5.2.5
BELLO2	Elements with Line Load, Solid, 3D-Shell, 2D-Shell, Membrane and Curved Beam Elements	see Section 5.2.6
BELOAD1	Beam with Line Load	see Section 5.2.7
BEMASS1	Hydrodynamic added Mass from Wave Load Program	see Section 5.2.8
BEUSLO	Element with Surface Load	see Section 5.2.9
BEUVLO	Elements with Volume Force, 3-D Solid, 2-D Shell and Membrane Elements	see Section 5.2.10
BEWAKIN	Wave Kinematics	see Section 5.2.11
BEWALO1	Element Loads from Wave Load Program	see Section 5.2.12
BGRAV	Gravitational Load (Constant of Gravity)	see Section 5.2.13
BLDEP	Node with Linear Dependency	see Section 5.2.14
BNACCLO	Node with Acceleration Load	see Section 5.2.15
BNBCD	Node with Boundary Condition	see Section 5.2.16
BNDISPL	Node with Displacement, Velocity and/or Acceleration	see Section 5.2.17
BNDOF	Node with Transformation	see Section 5.2.18
BNIEQ	Nodes with Non-Linear Contact Dependence	see Section 5.2.19
BNINCO	Node with Initial Condition If Arbitrary Time Dependent Loading	see Section 5.2.20
BNLOAD	Node with Load	see Section 5.2.21
BNLOAX	Node with Load (Line Load) for Axi-symmetric Solid (proposal)	see Section 5.2.22
BNMASS	Node with Point Mass	see Section 5.2.23
BNTEMP	Node with Temperature and Derivative for Temperature	see Section 5.2.24
BNTRCOS	Transformation from Global to Local Co-ordinate System, Direction Cosines	see Section 5.2.25
BNWALO	Node Load from Wave Load Program	see Section 5.2.26
BQDP	Node with Simple Quadratic Dependence	see Section 5.2.27

BRIGAC	Rigid Body Acceleration	see Section 5.2.28
BRIGDI	Rigid Body Displacement	see Section 5.2.29
BRIGVE	Rigid Body Velocity	see Section 5.2.30

5.2.1 **BAHAMAS**: Element with Added Mass

BAHAMAS	→ ELNO	NORMAL		

ELNO — Program defined internal number for the element for which added mass contribution should be computed.

NORMAL — Element side normal vector. NORMAL= ±1.0 indicates the direction of the hydrodynamic pressure.

= 1 hydrodynamic pressure direction is out from the element.

= -1 hydrodynamic pressure direction is in to the element.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.2 **BEDRAG1**: Hydrodynamic Drag and Damping from Wave Load Program

BEDRAG1	Not Used	$\rightarrow$ ELNO	Not Used	NP
	XI(1)	XI(2)	...	XI(NP)
	DRG$(1,1)$	DRG$(2,1)$	DRG$(3,1)$	DRG$(1,2)$
	...	DRG$(3,$NP$)$		

ELNO	Program defined internal number for the element with load condition.	
NP	Number of points.	
XI(i)	(Distance from end 1 to load point "i") / LTOT	
DRG(j,i)	Drag or Damping intensity for the j'th degree of freedom in member local co-ordinate system as defined by relevant **GUNIVEC** data type at point "i".	
Not Used	= 0.	Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

The data type **BEDRAG1** is generated by Wajac. This data type may only be used for element type BEAS(15), with no user defined local co-ordinate system.

LTOT is the total length of the beam.

BEDRAG1 should not be used for new applications.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.3 **BEISTE**: Element with Initial Strain Due to Thermal Expansion

BEISTE	→ LLC	LOTYP	Not Used	TOP
	→ ELNO	NNOD	INTNO	$T(1_1)$
	$T(2_1)$	...	$T(NNOD_1)$	$T(1_2)$
	$T(2_2)$	...	$T(NNOD_2)$	

LLC	Local load case number (positive integer number).	
LOTYP	Load type.	
	= 1	For all element types including shell elements with constant temperature across element thickness.
	= 2	For shell elements with temperature difference across element thickness.
TOP	Option to temperature difference specification.	
	= 1	Only one temperature difference is specified. It will be the same for all nodes.
	= 2	Temperature differences will be specified in all nodes.
ELNO	Program defined internal number for the element	
NNOD	Number of element nodes.	
INTNO	Integration station reference number to data type **GELINT**	
	This reference is usually not used (=0). It means that the program performing the load calculation chooses integration points.	
$T(1_1)$	Temperature difference at node 1, referred to element surface where z=-1.	
$T(2_1)$	Temperature difference at node 2, referred to element surface where z=-1.	
⋮		
$T(NNOD_1)$	Temperature difference at node NNOD, referred to element surface where z=-1.	
$T(1_2)$	Temperature difference at node 1, referred to element surface where z=1.	
$T(2_2)$	Temperature difference at node 2, referred to element surface where z=1.	
⋮		
$T(NNOD_2)$	Temperature difference at node NNOD, referred to element surface where z=1.	
Not Used	= 0.	Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

Reference to element side is used for shell elements with temperature difference across thickness. For all other situations only one temperature difference has to be specified in each node. By "temperature difference at node i" is meant the difference between the present temperature and a reference temperature where no initial strain is acting at the node.

Elements for which **BEISTE** can be applied:

3.10 **BEAS**

3.19 **FQUS**

etc ...

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.4 **BELFIX**: Flexible Joint/Hinge

BELFIX	→ FIXNO	OPT	TRANO	Not Used
	A(1)	A(2)	A(3)	A(4)
	A(5)	A(6)		

FIXNO — Fixation number to a node.

FIXNO is referenced from the **GELREF1** data type.

OPT

= 1 — A$(i) = a_i$ is a value between 0 and 1 that gives the degree of fixation (connectivity) to degree of freedom number i in the node. The extreme values of a is described by:

$a = 0$, fully released

$a = 1$, fully connected

= 2 — A$(i) = C_i$ is the inter element elastic spring stiffness to degree of freedom number i in the node. The degrees of freedom which are neither flexible nor free will be given $C_i = -1$ (instead of $C_i = \infty$). The relation between C_i and a_i is

$$a_i = C_i/(k_{ii} + C_i) \geq 0.0$$

where k_{ii} is the diagonal term of the element stiffness matrix corresponding to degree of freedom number i of the current node.

= 3 — As OPT = 1, where the hinge is located at the end of the beam – and not in the node as for OPT = 1. This option is thus relevant for eccentric beams.

= 4 — As OPT = 2, where the hinge is located at the end of the beam – and not in the node as for OPT = 2. This option is thus relevant for eccentric beams.

TRANO

= -1 — The fixation/flexibility (=A(i)) is given in the super element co-ordinate system.

= 0 — A(i) is given in the local element co-ordinate system.

> 0 — A(i) is given in a local co-ordinate system defined by TRANO, which refers to a transformation matrix given on data type **BNTRCOS**. The transformation matrix is defined by transformation from global to local system.

A(i) — See above (under the explanation of OPT).

Not Used = 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.5 **BELLAX**: Surface Load on Axi-symmetric Solid

BELLAX	→ LLC	LOTYP	COMPLEX	Not Used
	→ ELNO	NDOF	INTNO	LINE
	LD	IZERO	NFF	NFL
	NFS	NF	$FC_{(1,1)}$	$FC_{(1,2)}$
	...	$FC_{(1,NDOF)}$	$FC_{(2,1)}$	$FC_{(2,2)}$
	...	$FC_{(2,NDOF)}$	...	$FC_{(NF,1)}$
	...	$FC_{(NF,NDOF)}$		

LLC	Local load case number (positive integer number).	
LOTYP	Type of load at the element ELNO.	
	= 0	not decided whether conservative or non-conservative load.
	= 1	conservative load.
	= -1	non-conservative load.
COMPLEX	Phase shift definition.	
	= 0	no phase shift.
	= 1	phase shift.
	Currently not used.	
ELNO	Program defined internal number for the element with load condition.	
NDOF	Number of degrees of freedom along the given load line = 3 * the number of nodes.	
INTNO	Integration station reference number to data type **GELINT**	
	This reference is usually not used (=0). It means that the program performing the load calculation chooses integration points.	
	Currently not used.	
LINE	Line specification.	
	See the element description in section 3.	
LD	= 1	if loading is symmetric with respect to the $q = 0$ axis.
	= 2	if loading is antisymmetric with respect to the $q = 0$ axis.
	= 3	if loading is not symmetric with respect to the $q = 0$ axis.
	q is the angle co-ordinate of the cylindrical co-ordinate system (r, z, q).	
IZERO	= 0	if the zero harmonic is NOT included in the Fourier expansion.
	= 1	if the zero harmonic is included in the Fourier expansion.
NFF	The first non-zero harmonic in the Fourier expansion of the load.	
NFL	The last non-zero harmonic in the Fourier expansion of the load.	
NFS	The harmonic increment in the Fourier expansion of the load.	
NF	The total number of coefficients of the Fourier expansion.	
	NF = 1	Axi-symmetric load

	NF =	(NFL NFF)/ NFS + 1 + IZERO for LD = 1, 2 (non axi-symmetric load)
	NF =	2 * ((NFL NFF)/ NFS + 1) + IZERO for LD = 3 (non axi-symmetric load)
$FC_{(1,1)}$		The Fourier coefficient for the radial (R) degree of freedom of the first node for the first harmonic.
$FC_{(1,2)}$		The Fourier coefficient for the tangential (q) degree of freedom of the first node for the first harmonic.
$FC_{(1,3)}$		The Fourier coefficient for the axial (Z) degree of freedom of the first node for the first harmonic.
$FC_{(1,4)}$		The Fourier coefficient for the radial (R) degree of freedom of the second node for the first harmonic.
⋮		
$FC_{(1,NDOF)}$		The Fourier coefficient for the axial (Z) degree of freedom of the last node for the first harmonic.
$FC_{(2,1)}$		
$FC_{(2,2)}$		
⋮		The Fourier coefficients for the degrees of freedom of the load line for the second harmonic.
$FC_{(2,NDOF)}$		
⋮		
⋮		
$FC_{(NF,1)}$		
$FC_{(NF,2)}$		
⋮		The Fourier coefficients for the degrees of freedom of the load line for the last harmonic.
$FC_{(NF,NDOF)}$		
Not Used	= 0.	Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

Note:

E.g. SYMMETRIC LOAD

Consider a load line with 3 nodes each with RADIAL (R), TANGENTIAL (q) and AXIAL (Z) degrees of freedom. The loads are described using the zero, first, second and third harmonics.

NDOF = 9
LD = 1
IZERO = 1
NFF = 1
NFL = 3
NFS = 1
NF = 4

$FC_{(1,1)} \ldots FC_{(1,9)}$: Fourier coefficients for the zero harmonic
$FC_{(2,1)} \ldots FC_{(2,9)}$: Fourier coefficients for the first harmonic
$FC_{(3,1)} \ldots FC_{(3,9)}$: Fourier coefficients for the second harmonic
$FC_{(4,1)} \ldots FC_{(4,9)}$: Fourier coefficients for the third harmonic

$P_{RADIAL} = FC_{(1,1)} + FC_{(2,1)}\cos q + FC_{(3,1)}\cos 2q + FC_{(4,1)}\cos 3q$
$P_{TANGENTIAL} = FC_{(1,2)} + FC_{(2,2)}\sin q + FC_{(3,2)}\sin 2q + FC_{(4,2)}\sin 3q$
$P_{AXIAL} = FC_{(1,3)} + FC_{(2,3)}\cos q + FC_{(3,3)}\cos 2q + FC_{(4,3)}\cos 3q$

Similarly the loads for the second and third node on the load line are described using the values 4, 5, 6 and 7, 8, 9 respectively for the last index of FC.

SYMMETRIC loads (LD=1) of the RADIAL and AXIAL degrees of freedom are described using coefficients of COSINE, while the TANGENTIAL d.o.f.s are described using coefficients of SINE.

E.g. ANTISYMMETRIC LOAD

Consider a load line with 2 nodes, each with 3 degrees of freedom.

The first node has load in RADIAL(R)and AXIAL(Z)direction. The second node has load in RADIAL(R)and TANGENTIAL(q)direction. The loads are described using the second, fifth and eighth harmonics.

NDOF = 6
LD = 2
IZERO = 0
NFF = 2
NFL = 8
NFS = 3
NF = 3

$FC_{(1,1)} \ldots FC_{(1,6)}$: second harmonics
$FC_{(2,1)} \ldots FC_{(2,6)}$: fifth harmonics
$FC_{(3,1)} \ldots FC_{(3,6)}$: eighth harmonics

The loads for the first node on the load line are thus given by:

P_{RADIAL} = $FC_{(1,1)}$ sin 2q + $FC_{(2,1)}$ sin 5q + $FC_{(3,1)}$ sin 8q
P_{AXIAL} = $FC_{(1,3)}$ sin 2q + $FC_{(2,3)}$ sin 5q + $FC_{(3,3)}$ sin 8q

$FC_{(1,2)}$ = $FC_{(2,2)}$ = $FC_{(3,2)}$ = 0.0 (no tangential load)

The loads for the second node on the load line is thus given by:

P_{RADIAL} = $FC_{(1,4)}$ sin 2q + $FC_{(2,4)}$ sin 5q + $FC_{(3,4)}$ sin 8q
$P_{TANGENTIAL}$ = $FC_{(1,5)}$ cos 2q + $FC_{(2,5)}$ cos 5q + $FC_{(3,5)}$ cos 8q

$FC_{(1,6)}$ = $FC_{(2,6)}$ = $FC_{(3,6)}$ = 0.0 (no axial load)

Totally NDOF $\times$ NF = 6 $\times$ 3 = 18 coefficients will be written.

ANTISYMMETRIC loads of the RADIAL and AXIAL d.o.f.s are described using coefficients of SINE, while the TANGENTIAL degrees of freedom are described using coefficients of COSINE.

E.g. NONSYMMETRIC LOAD

Not implemented in Sestra.

Consider 3 nodes on a load line each with RADIAL(R), AXIAL(Z) and TANGENTIAL(q) degrees of freedom. The loads are described using the zero, first and second harmonics

NDOF = 9
LD = 3
IZERO = 1
NFF = 1
NFL = 2
NFS = 1
NF = 3

$FC_{(1,1)} \ldots FC_{(1,9)}$: Fourier coefficients for the zero harmonic
$FC_{(2,1)} \ldots FC_{(2,9)}$: Fourier coefficients for the first COSINE harmonic
$FC_{(3,1)} \ldots FC_{(3,9)}$: Fourier coefficients for the first SINE harmonic
$FC_{(4,1)} \ldots FC_{(4,9)}$: Fourier coefficients for the second COSINE harmonic
$FC_{(5,1)} \ldots FC_{(5,9)}$: Fourier coefficients for the second SINE harmonic

The loads for the 3 degrees of freedom of the first node on the load line is thus given by.

$P_{RADIAL} = FC_{(1,1)} + FC_{(2,1)}\cos q + FC_{(3,1)}\sin q + FC_{(4,1)}\cos 2q + FC_{(5,1)}\sin 2q$
$P_{TANGENTIAL} = FC_{(1,2)} + FC_{(2,2)}\cos q + FC_{(3,2)}\sin q + FC_{(4,2)}\cos 2q + FC_{(5,2)}\sin 2q$
$P_{AXIAL} = FC_{(1,3)} + FC_{(2,3)}\cos q + FC_{(3,3)}\sin q + FC_{(4,3)}\cos 2q + FC_{(5,3)}\sin 2q$

Similarly the load for the second and third node on the load line is described using the values 4, 5, 6 and 7, 8, 9, respectively for the last index of FC.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.6 **BELLO2**: Elements with Line Load, Solid, 3D-Shell, 2D-Shell, Membrane and Curved Beam Elements

BELLO2	→ LLC	LOTYP	COMPLEX	LAYER
	→ ELNO	NDOF	INTNO	LINE
	SIDE	$\text{RLOAD}_{(1)}$	$\text{RLOAD}_{(2)}$	...
	...	...	$\text{RLOAD}_{(\text{NDOF})}$	$\text{ILOAD}_{(1)}$
	$\text{ILOAD}_{(2)}$	...	...	...
	$\text{ILOAD}_{(\text{NDOF})}$			

LLC — Local load case number (positive integer number).

LOTYP — Type of load at the element ELNO. Decision whether load being force or moment and if it is conservative or non-conservative. Conservative or non-conservative is usually not of interest to linear programs.

- = 0 not decided whether conservative or non-conservative force per length.
- = 1 conservative force per length.
- = -1 non-conservative force per length.
- = 3 conservative moment per length.
- = -3 non-conservative moment per length.

COMPLEX — Phase shift definition.

- = 0 no phase shift.
- = 1 phase shift.

LAYER — Layer number for elements with more than one layer.

If LAYER equals 0, the line load will be positioned in the shell layer (in opposition to a stiffener layer).

If more than one shell layer is defined for an element type and LAYER equals 0, the programs shall stop and give an error message.

For elements which are treated as one layer in the load calculations (e.g. the sandwich element in Sestra), LAYER does not have any meaning.

ELNO — Program defined internal number for the element with load condition.

NDOF — Number of translational degrees of freedom along the given load line for line force. For line moment the variable is specifying number of rotational degrees of freedom along the given load line.

INTNO — Integration station reference number to data type **GELINT**

This reference is usually not used (=0). It means that the program performing the load calculation chooses integration points.

LINE — Line specification.

See the element description in section 3.

SIDE — Element side definition.

See the element description in section 3.

$RLOAD_{(1)}$	The real part of the load condition with respect to the first degree of freedom at current LINE of element ELNO.
$RLOAD_{(2)}$	The real part of the load condition with respect to the second degree of freedom at current LINE of element ELNO.
⋮	
$RLOAD_{(NDOF)}$	The real part of the load condition with respect to the last degree of freedom at current LINE of element ELNO.
$ILOAD_{(1)}$	The imaginary part of the load condition with respect to the first degree of freedom at current LINE of element ELNO.
$ILOAD_{(2)}$	The imaginary part of the load condition with respect to the second degree of freedom at current LINE of element ELNO.
⋮	
$ILOAD_{(NDOF)}$	The imaginary part of the load condition with respect to the last degree of freedom at current LINE of element ELNO.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $RLOAD_{(NDOF)}$ – and the first imaginary number – $ILOAD_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions $ILOAD_{(1)}$, $ILOAD_{(2)}$, etc. are left out.

For quadrilateral membrane elements line loads can only be specified along element sides.

It will make no sense to specify line moment for membrane and solid type of elements.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.7 **BELOAD1**: Beam with Line Load

BELOAD1	→ LLC	LOTYP	COMPLEX	OPT
	→ ELNO	L1	L2	NDOF
	INTNO	$RLOAD_{(1)}$	$RLOAD_{(2)}$	. . .
	. . .	. . .	$RLOAD_{(NDOF)}$	$ILOAD_{(1)}$
	$ILOAD_{(2)}$	. . .	. . .	. . .
	$ILOAD_{(NDOF)}$			

LLC — Local load case number (positive integer number).

LOTYP — Type of load at the element ELNO. Decision whether load being force or moment and if it is conservative or non-conservative. Conservative or non-conservative is usually not of interest to linear programs.

- = 0 not decided whether conservative or non-conservative load.
- = 1 True distributed force, conservative load.
- = 2 Simulated concentrated force, conservative load.
- = 3 True distributed moment, conservative load.
- = 4 Simulated concentrated moment, conservative load.
- = -1 True distributed force, non-conservative load.
- = -2 Simulated concentrated force, non-conservative load.
- = -3 True distributed moment, non-conservative load.
- = -4 Simulated concentrated moment, non-conservative load.

COMPLEX — Phase shift definition.

- = 0 no phase shift.
- = 1 phase shift.

OPT — Option for reference of L1 and L2 (see below).

- = 0 L1 and L2 are distances measured from the ends of the flexible part of the beam.
- = 1 L1 and L2 are distances measured along the beam axis from the projection of the end nodes on the beam axis.

ELNO — Program defined internal number for the element with load condition.

L1 — Distance along the beam from end node 1 to the position on the beam where the line load starts acting. The distance L1 is in the interval [0, Length–of–the–Beam].

L2 — Distance along the beam from end node 2 to the position on the beam where the line load ends acting. The distance L2 is in the interval [L1, Length–of–the–Beam].

NDOF — Product of last degree of freedom with specified load and the number of nodes of the beam in question (= 3*NNOD for 3-dimensional beams).

INTNO — Integration station reference number to data type **GELINT**

	This reference is usually not used (=0). It means that the program performing the load calculation chooses integration points.
$RLOAD_{(1)}$	The real part of the intensity with respect to the first degree of freedom of the first load point.
$RLOAD_{(2)}$	The real part of the global intensity with respect to the second degree of freedom of the first load point.
⋮	
$RLOAD_{(NDOF)}$	The real part of the global intensity with respect to the last degree of freedom with load of the last load point.
$ILOAD_{(1)}$	The imaginary part of the global intensity with respect to the first degree of freedom of the first load point.
$ILOAD_{(2)}$	The imaginary part of the global intensity with respect to the second degree of freedom of the first load point.
⋮	
$ILOAD_{(NDOF)}$	The imaginary part of the global intensity with respect to the last degree of freedom with load of the last load point.

The legal degrees of freedom are the translational components, given in the global co-ordinate system.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $RLOAD_{(NDOF)}$ – and the first imaginary number – $ILOAD_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions $ILOAD_{(1)}$, $ILOAD_{(2)}$, etc. are left out.

For the beam element BEAS with two nodes, the line of action of the load will be the axes through the shear centre.

In order to clarify the meaning of NDOF, imagine a beam element with three nodal points. If translatory loads are applied in the global y- and z-direction, the last degree of freedom to have load specified is the third degree of freedom. NDOF will therefore in this case be 3 x 3 = 9. The load intensity for the x-direction will be included, but with the value zero of course.

If the line of application of the line load does not match the node positioning along the beam, the following example will hopefully explain how this problem is solved on the interface file. Node1 (see Figure 5.2) will have the intensity zero, whereas node2 will have the intensity corresponding to the correct intensity at the starting point of the line load. The real point of application (used in the integration process) of course is governed by L1.

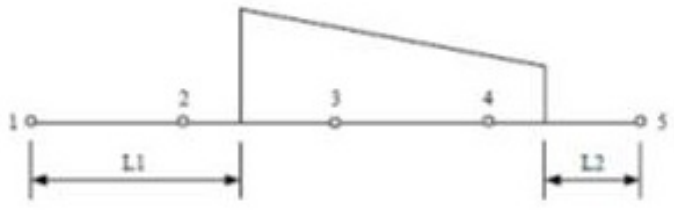


Figure 5.2: Interpretation of BELOAD1 for a beam with 5 nodes

Elements for which **BELOAD1** can be applied:

3.10 **BEAS**

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.8 **BEMASS1**: Hydrodynamic added Mass from Wave Load Program

BEMASS1	Not Used	$\rightarrow$ ELNO	Not Used	NP
	XI(1)	XI(2)	...	XI(NP)
	AM$(1,1)$	AM$(2,1)$	AM$(3,1)$	AM$(1,2)$
	...	AM$(3,$ NP$)$		

ELNO	Program defined internal number for the element with load condition.	
NP	Number of points.	
XI(i)	(Distance from end 1 to load point "i") / LTOT	
AM(j,i)	Added mass intensity for the j'th degree of freedom in member local co-ordinate system as defined by relevant **GUNIVEC** data type at point "i".	
Not Used	= 0.	Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

The data type **BEMASS1** is generated by Wajac and Wadam and may only be read by Sestra. This data type may only be used for element type BEAS(15), with no user defined local co-ordinate system.

LTOT is the total length of the beam.

BEMASS1 should not be used for new applications.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.9 **BEUSLO**: Element with Surface Load

BEUSLO	→ LLC	LOTYP	COMPLEX	LAYER
	→ ELNO	NDOF	INTNO	SIDE
	$RLOAD_{(1)}$	$RLOAD_{(2)}$	. . .	. . .
	. . .	. . .	$RLOAD_{(NDOF)}$	$ILOAD_{(1)}$
	$ILOAD_{(2)}$	. . .	. . .	. . .
	$ILOAD_{(NDOF)}$			

LLC — Local load case number (positive integer number).

LOTYP — Type of load at the element ELNO. Decision whether pressure is conservative or non-conservative. Conservative or non-conservative load is usually not of interest to linear programs. However, see LOTYP = 3.

- = 1 — normal pressure, conservative load.
- = 2 — load given in component form, conservative load.
- = 3 — dummy hydro pressure, used to indicate the direction of hydrostatic / hydrodynamic pressure to be computed by a hydrodynamic load program.

  Hydro pressure is non-conservative.
- = -1 — normal pressure, non-conservative load.
- = -2 — load given in component form, non-conservative load.

COMPLEX — Phase shift definition.

- = 0 — no phase shift.
- = 1 — phase shift.

LAYER — Layer number for elements with more than one layer.

If LAYER equals 0, the line load will be positioned in the shell layer (in opposition to a stiffener layer).

If more than one shell layer is defined for an element type and LAYER equals 0, the programs shall stop and give an error message.

For elements which are treated as one layer in the load calculations (e.g. the sandwich element in Sestra), LAYER does not have any meaning.

ELNO — Program defined internal number for the element with load condition.

NDOF

- LOTYP = 1 — number of nodes of the specified element side.
- LOTYP = 2 — number of translational degrees of freedom of the specified element side.
- LOTYP = 3 — Then NDOF = 1.

  No transformation needed when the super elements are assembled. The reason is that this load condition is a message to the hydrodynamic load programs whether the element is wet.

  If the element is wet, then this data type does also specify for which side of the element to apply the hydrostatic pressure.

INTNO — Integration station reference number to data type **GELINT**

	This reference is usually not used (=0). It means that the program performing the load calculation chooses integration points.
SIDE	Element side definition. See the element description in section 3. For shell elements: = 1 loads referred to element side where z=-1 = 2 loads referred to element side where z=0 = 3 loads referred to element side where z=1 For solid elements: SIDE will be a six figures number for a hexahedron, a five figures number for a prism and a four figures number for a tetrahedron, consisting of zeroes with 1 for the side with load. For instance 000100 for a hexahedron means load on side no. 4.
$\text{RLOAD}_{(1)}$	The real part of the load with respect to the first degree of freedom or first node of the element side if normal pressure.
$\text{RLOAD}_{(2)}$	The real part of the load with respect to the second degree of freedom or second node of the element side.
⋮	
$\text{RLOAD}_{(\text{NDOF})}$	The real part of the load with respect to the last degree of freedom or last node of the element side.
$\text{ILOAD}_{(1)}$	The imaginary part of the load with respect to the first degree of freedom or first node of the element side.
$\text{ILOAD}_{(2)}$	The imaginary part of the load with respect to the second degree of freedom or second node of the element side.
⋮	
$\text{ILOAD}_{(\text{NDOF})}$	The imaginary part of the load with respect to the last degree of freedom or last node of the element side.

If LOTYP= 3, RLOAD= $\pm$1.0 indicating which side the element pressure comes from. (+1 indicates shell element SIDE=1 (z=1), -1 indicates shell element SIDE=3 (z=-1)).

For solids when LOTYP = 3, RLOAD must be +1.0.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $\text{RLOAD}_{(\text{NDOF})}$ – and the first imaginary number – $\text{ILOAD}_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions $\text{ILOAD}_{(1)}$, $\text{ILOAD}_{(2)}$, etc. are left out.

Normal pressure means that only one pressure component is specified for each node, and this pressure component is acting normal to the surface.

For volume elements a positive value means normal pressure directed into the element. For shell elements, a positive value means normal pressure in the local z-direction.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.10 **BEUVLO**: Elements with Volume Force, 3-D Solid, 2-D Shell and Membrane Elements

BEUVLO	→ LLC	LOTYP	COMPLEX	Not Used
	→ ELNO	NDOF	INTNO	$\text{RLOAD}_{(1)}$
	$\text{RLOAD}_{(2)}$	...	$\text{RLOAD}_{(\text{NDOF})}$	$\text{ILOAD}_{(1)}$
	$\text{ILOAD}_{(2)}$	...	$\text{ILOAD}_{(\text{NDOF})}$	

LLC	Local load case number (positive integer number).	
LOTYP	Type of load at the element ELNO. Decision whether pressure is conservative or non-conservative. Conservative or non-conservative is usually not of interest to linear programs.	
	= 0	Not decided whether conservative or non-conservative load.
	= 1	Loads are given as nodal accelerations, conservative load.
	= 2	Loads are given as nodal force intensities, conservative load.
	= -1	Loads are given as nodal accelerations, non-conservative load.
	= -2	Loads are given as nodal force intensities, non-conservative load.
COMPLEX	Phase shift definition.	
	= 0	no phase shift.
	= 1	phase shift.
ELNO	Program defined internal number for the element with load condition.	
NDOF	Number of translational degrees of freedom.	
INTNO	Integration station reference number to data type **GELINT**	
	This reference is usually not used (=0). It means that the program performing the load calculation chooses integration points.	
$\text{RLOAD}_{(1)}$	The real part of the load with respect to the first degree of freedom.	
$\text{RLOAD}_{(2)}$	The real part of the load with respect to the second degree of freedom.	
⋮		
$\text{RLOAD}_{(\text{NDOF})}$	The real part of the load with respect to the last degree of freedom.	
$\text{ILOAD}_{(1)}$	The imaginary part of the load with respect to the first degree of freedom.	
$\text{ILOAD}_{(2)}$	The imaginary part of the load with respect to the second degree of freedom.	
⋮		
$\text{ILOAD}_{(\text{NDOF})}$	The imaginary part of the load with respect to the last degree of freedom.	
Not Used	= 0.	Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $\text{RLOAD}_{(\text{NDOF})}$ – and the first imaginary number – $\text{ILOAD}_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions $\text{ILOAD}_{(1)}$, $\text{ILOAD}_{(2)}$, etc. are left out.

Note: This data type is *not* available in Sesam.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.11 **BEWAKIN**: Wave Kinematics

BEWAKIN	→ LLC	→ ELNO	COMPLEX	NP
	LTOT	$L_{(1)}$	$L_{(2)}$	...
	$L_{(NP)}$	$\eta_{(1)}$	$VX_{(1)}$	$VY_{(1)}$
	$VZ_{(1)}$	$AX_{(1)}$	$AY_{(1)}$	$AZ_{(1)}$
	$\eta_{(2)}$	$VX_{(2)}$	$VY_{(2)}$	...
	$AY_{(NP)}$	$AZ_{(NP)}$		

LLC	Local load case number (positive integer number).
ELNO	Program defined internal number for the element with load condition.
COMPLEX	Phase shift definition.
	= 0 no phase shift.
	= 1 phase shift.
NP	Number of points.
LTOT	Total element length.
$L_{(i)}$	Distance from end 1 to point "i".
$\eta_{(i)}$	Sea surface position above point "i".
$VX_{(i)}$	Wave particle velocity in X-direction.
$VY_{(i)}$	Wave particle velocity in Y-direction.
$VZ_{(i)}$	Wave particle velocity in Z-direction.
$AX_{(i)}$	Wave particle acceleration in X-direction.
$AY_{(i)}$	Wave particle acceleration in Y-direction.
$AZ_{(i)}$	Wave particle acceleration in Z-direction.

Elevation, particle velocity and particle acceleration are given in the co-ordinate system of the super element in question.

The data type **BEWAKIN** may optionally be generated by Wajac, and should not be used for new applications.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.12 **BEWALO1**: Element Loads from Wave Load Program

BEWALO1	→ LLC	→ ELNO	COMPLEX	NP
	LTOT	$L_{(1)}$	$L_{(2)}$	...
	$L_{(NP)}$	$FR_{(1,1)}$	$FR_{(2,1)}$	$FR_{(3,1)}$
	$FR_{(1,2)}$	...	$FR_{(3,NP)}$	$FI_{(1,1)}$
	$FI_{(2,1)}$	...	$FI_{(3,NP)}$	

LLC	Local load case number (positive integer number).
ELNO	Program defined internal number for the element with load condition.
COMPLEX	Phase shift definition.
	= 0 no phase shift.
	= 1 phase shift.
NP	Number of load points.
LTOT	Total element length.
$L_{(i)}$	Distance from end 1 to load point "i".
$FR_{(j,i)}$	Real component of force for the j'th degree of freedom at load point i. Here, $j \in [1,3]$ for the Y and Z directions, respectively.
$FI_{(j,i)}$	Imaginary component of force for the j'th degree of freedom at load point i. Here, $j \in [1,3]$ for the Y and Z directions, respectively.

Forces are given in the co-ordinate system of the super element in question.

When unformatted the records are packed in the following manner:

The first record contains 6 words (as for all other data types).

The next record contains the rest of the information, and the variable record length must be computed as:

NW = 1 + (4 + COMPLEX*3)*NP,

or NW = 1 + 4*NP if real loads,

and NW = 1 + 7*NP if complex loads.

The data type **BEWALO1** is only generated by Wajac and may only be read from Sestra. This data type should not be used for new applications.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.13 **BGRAV**: Gravitational Load (Constant of Gravity)

BGRAV	→ LLC	ModelNode	Not Used	OPT
	GX	GY	GZ	

LLC	Local load case number (positive integer number).	
ModelNode	If ModelNode > 0, then it represents the super element assembly origin, or part of the original super element model. That is, ModelNode is equal to the unique tree node identifier created when super element models are assembled to global models for direct analysis in Sestra. If ModelNode ≤ 0 it means that the data applies for the whole assembled model.	
OPT	Option for weight contribution of stiff ends for beam element.	
	= 0	only flexible part of the beam contributes to the gravitational load.
	= 1	stiff ends also contribute to the gravitational load.
GX	Component of constant of gravity in the global x-direction.	
GY	Component of constant of gravity in the global y-direction.	
GZ	Component of constant of gravity in the global z-direction.	
Not Used	= 0.	Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

Only the constant of gravity is stored on the Interface File. The actual gravitational load is computed in the analysis program where the element routines are available.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.14 **BLDEP**: Node with Linear Dependency

BLDEP	→ SLAVE	MASTER	NDDOF	NDEP
	s_1	m_1	β_1	Not Used
	s_2	m_2	β_2	Not Used
	...	...	...	...
	s_{NDEP}	m_{NDEP}	β_{NDEP}	Not Used

SLAVE	Program defined internal number for the node which is dependent.
MASTER	Program defined internal number for the node which is independent.
NDDOF	Number of dependent degrees of freedom at node SLAVE. When not specified, NDDOF is equal to NDEP.
NDEP	Number of triplets with s_i, m_i and β_i.
s_i	Dependent node's degree of freedom ("slave" degree of freedom).
m_i	Independent node's degree of freedom ("master" degree of freedom).
β_i	The contribution of the degree of freedom m_i of the master (independent) node to degree of freedom s_i on the slave (dependent) node.
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

Each line specifies one dependent degree of freedom which is dependent on the independent node's specified degree of freedom with the factor β_i. The degrees of freedom must also be specified on **BNBCD** data types as linear dependent, i.e. $\text{FIX}_{(i)}$ = 3 for the dependent node; and as retained, i.e. $\text{FIX}_{(i)}$ = 4 for the independent node if the current analysis is a super element analysis where only the Sestra super-matrix solver is used.

If linear dependencies are restricted to the first-level super elements and the Sestra multi-frontal solver is used, then the independent node does not need to be set as retained. It can, however, be a super node but is not required to be a super node only to resolve linear dependency relations. This does also apply for direct analyses — i.e. only one super element— when the Sestra multi-front solver is used. An implication is that the Sestra super-matrix solver cannot be used when linear dependencies are specified unless there are at least two super elements involved.

A node may be dependent on many nodes. For each combination of SLAVE and MASTER a new data type, starting with the identifier **BLDEP**, is given.

The **same** combination of SLAVE and MASTER may occur only once.

When node transformations have been specified for any of the nodes implicated in the linear dependence, the degrees of freedom refer to the transformed local co-ordinate system.

Multipoint constraints (2nd and higher order dependence) may be specified through more **BLDEP** data types with the same linear dependent node and different independent nodes. The factors β_i may be found as Lagrange multipliers or coefficients (Lagrange interpolation polynomial). For 2nd-order dependence this may as well be specified on one **BQDP** data type.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.15 **BNACCLO**: Node with Acceleration Load

BNACCLO	→ LLC	LOTYP	COMPLEX	ModelNode
	→ NODENO	NDOF	$RACC_{(1)}$	$RACC_{(2)}$
	...	...	...	$RACC_{(NDOF)}$
	$IACC_{(1)}$	$IACC_{(2)}$	...	...
	...	$IACC_{(NDOF)}$		

LLC	Local load case number (positive integer number).
LOTYP	Load type. Not used.
COMPLEX	Phase shift definition. = 0 no phase shift. = 1 phase shift.
ModelNode	If ModelNode > 0, then it represents the super element assembly origin, or part of the original super element model. That is, ModelNode is equal to the unique tree node identifier created when super element models are assembled to global models for direct analysis in Sestra. If ModelNode $\leq$ 0 it means that the data applies for the whole assembled model.
NODENO	Program defined internal number for the node with load condition.
NDOF	Number of degrees of freedom at the node NODENO.
$RACC_{(1)}$	The real part of the acceleration with respect to the first degree of freedom at NODENO.
$RACC_{(2)}$	The real part of the acceleration with respect to the second degree of freedom at NODENO.
⋮	
$RACC_{(NDOF)}$	The real part of the acceleration with respect to the last degree of freedom at NODENO.
$IACC_{(1)}$	The imaginary part of the acceleration with respect to the first degree of freedom at NODENO.
$IACC_{(2)}$	The imaginary part of the acceleration with respect to the second degree of freedom at NODENO.
⋮	
$IACC_{(NDOF)}$	The imaginary part of the acceleration with respect to the last degree of freedom at NODENO.
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

Note that the **BNACCLO** data type expects acceleration values and not inertia load values. The nodal acceleration describes an acceleration field at the node, and not prescribed acceleration.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $RACC_{(NDOF)}$ – and the first imaginary number – $IACC_{(1)}$. If no phase shift is specified

(that is COMPLEX= 0), then the fields or positions $IACC_{(1)}$, $IACC_{(2)}$, etc. are left out.

Elements for which **BNACCLO** can be applied:

3.10 **BEAS**

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.16 **BNBCD**: Node with Boundary Condition

BNBCD	→ NODENO	NDOF	$FIX_{(1)}$	$FIX_{(2)}$
	...	...	...	$FIX_{(NDOF)}$

NODENO — Program defined internal number for the node with specified boundary condition.

NDOF — Number of degrees of freedom at the node NODENO.

$FIX_{(1)}$

$FIX_{(2)}$

⋮ — Specification of boundary condition codes for the relevant degrees of freedom.

$FIX_{(NDOF)}$

The following boundary condition codes, i.e. the values for $FIX_{(1)}$, $FIX_{(2)}$, ..., $FIX_{(NDOF)}$ - are defined for the status of the degrees of freedom:

$FIX_{(i)}$	= 0	no boundary condition is applied to the degree of freedom.
$FIX_{(i)}$	= 1	fixed at zero displacement, temperature, etc.
$FIX_{(i)}$	= 2	prescribed displacement, temperature, velocity, acceleration, etc. different from zero.
$FIX_{(i)}$	= 3	linearly dependent.
$FIX_{(i)}$	= 4	retained degree of freedom, i.e. super node.

The code $FIX_{(i)}$ = 2 just indicates specified condition for the relevant degree of freedom. Whether it is displacement, first time derivative of the displacement etc. is defined on the **BNDISPL** data type.

Degrees of freedom with $FIX_{(i)}$ = 2 which are not defined on the **BNDISPL** data type will be fixed (have zero displacement, velocity and acceleration).

The node numbers (degrees of freedom) with $FIX_{(i)}$ = 4 are called super nodes. The super node numbering is according to the increasing order of their internal node number.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.17 **BNDISPL**: Node with Displacement, Velocity and/or Acceleration

BNDISPL	→ LLC	DTYPE	COMPLEX	Not Used
	→ NODENO	NDOF	$RDISP_{(1)}$	$RDISP_{(2)}$
	...	...	...	$RDISP_{(NDOF)}$
	$IDISP_{(1)}$	$IDISP_{(2)}$	...	...
	...	$IDISP_{(NDOF)}$		

LLC — Local load case number (positive integer number).

DTYPE — Type of boundary condition.

= 1 specified displacement, temperature, etc.

= 2 specified velocity, first time derivative of the temperature, etc.

= 3 specified acceleration, etc.

COMPLEX — Phase shift definition.

= 0 no phase shift.

= 1 phase shift.

NODENO — Program defined internal number for the node with specified boundary condition.

NDOF — Number of degrees of freedom at the node NODENO.

$RDISP_{(1)}$ — The real part of the specified boundary condition with respect to the first degree of freedom at NODENO.

$RDISP_{(2)}$ — The real part of the specified boundary condition with respect to the second degree of freedom at NODENO.

⋮

$RDISP_{(NDOF)}$ — The real part of the specified boundary condition with respect to the last degree of freedom at NODENO.

$IDISP_{(1)}$ — The imaginary part of the specified boundary condition with respect to the first degree of freedom at NODENO.

$IDISP_{(2)}$ — The imaginary part of the specified boundary condition with respect to the second degree of freedom at NODENO.

⋮

$IDISP_{(NDOF)}$ — The imaginary part of the specified boundary condition with respect to the last degree of freedom at NODENO.

Not Used — = 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

RDISP and IDISP refer to the transformed co-ordinate system if the node NODENO is transformed, else to the global co-ordinate system of the super element.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $RDISP_{(NDOF)}$ – and the first imaginary number – $IDISP_{(1)}$. If no phase shift is specified

(that is COMPLEX= 0), then the fields or positions $IDISP_{(1)}$, $IDISP_{(2)}$, etc. are left out.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.18 **BNDOF**: Node with Transformation

BNDOF	→ NODENO	TRANSD	TRANSR	

NODENO	Program defined internal number for the node with transformation.
TRANSD	Reference number to the transformed co-ordinate system of the displacements, given on **BNTRCOS** for NODENO.
TRANSR	Reference number to the transformed co-ordinate system of the rotations, given on **BNTRCOS** for NODENO.

If no reference number is given, no transformation is relevant to the relevant type of degree of freedom, i.e. translations or rotations for NODENO.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.19 **BNIEQ**: Nodes with Non-Linear Contact Dependence

BNIEQ	→ NODEA	NODEB	Not Used	NDEP
	DOFA_1	DOFB_1	SCALE_1	OPT_1
	DOFA_2	DOFB_2	SCALE_2	OPT_2
	...	...	...	...
	$\text{DOFA}_{\text{NDEP}}$	$\text{DOFB}_{\text{NDEP}}$	$\text{SCALE}_{\text{NDEP}}$	OPT_{NDEP}

NODEA — Program defined internal number for a node with contact dependency.

NODEB — Program defined internal number for another node with contact dependency.

NDEP — Number of specification groups following.

DOFA_i — Degree of freedom of NODEA which defines the left-hand side of a contact condition.

DOFB_i — Degree of freedom of NODEB which contributes to the contact condition associated with degree of freedom DOFA_i on node NODEA.

SCALE_i — Contribution factor.

$|\text{OPT}_i| \neq 1$: SCALE_i is the contribution of degree of freedom DOFB_i on node NODEB to the contact condition associated with degree of freedom DOFA_i on node NODEA.

$|\text{OPT}_i| = 1$: SCALE_i is a constant contribution to the contact relation associated with degree of freedom DOFA_i on node NODEA.

OPT_i — Contact condition.

$+2$: $\text{DOFA}_i \geq \cdots + \text{DOFB}_j \cdot \text{SCALE}_i$
-2: $\text{DOFA}_i \leq \cdots + \text{DOFB}_j \cdot \text{SCALE}_i$
0: $\text{DOFA}_i = \cdots + \text{DOFB}_j \cdot \text{SCALE}_i$
$+1$: $\text{DOFA}_i \geq \cdots + \text{SCALE}_i$
-1: $\text{DOFA}_i \leq \cdots + \text{SCALE}_i$

Each line specifies one degree of freedom which is related to another degree of freedom with a contact condition. In the definition of OPT_i above, the "..." indicate that each new line is an addition to the right-hand side of any already defined inequality with degree of freedom DOFA_i on the left-hand side. For example, the BNIEQ card

```
BNIEQ   1       2               2
        1       1       0.5     2
        1       2       0.3     2
```

defines the inequality $x_{1,1} \geq 0.5x_{2,1} + 0.3x_{2,2}$, where $x_{m,n}$ denotes degree of freedom n on node m.

When several contributions to the same DOFA_i are present the OPT_i field for each contribution must have the same sign.

A node may have contact condition specification involving several nodes. For each combination of NODEA and NODEB, a new data typewith the identifier **BNIEQ** is given.

Analysis of non-linear contact conditions needs identification of nodes and degrees of freedom involved, and definition of the conditions to be applied. Nodes to be analysed by contact conditions must be specified

as boundary condition nodes by **BNBCD** data types, where degrees of freedom to be involved in contact conditions must be specified as retained (FIX$_{(i)}$ = 4) degrees of freedom.

As long as the condition involves 2 degrees of freedom in a point to point contact, the factor SCALE$_i$ is typically 1.0 and the OPT$_i$ field is either $+2$ or -2.

Initial gap (or penetration) may be specified by OPT$_i = 1$ or OPT$_i = -1$ giving a constant contribution to the inequality associated with degree of freedom DOFA$_i$ on node NODEA. When the constant contribution is specified, the DOFB$_i$ field is not used.

When node transformations have been specified for any of the nodes involved in the contact condition, the degrees of freedom refer to the transformed local co-ordinate system. Local transformation of nodes is available in the **BNDOF** specification.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.20 **BNINCO**: Node with Initial Condition If Arbitrary Time Dependent Loading

BNINCO	→ INCONO	DTYPE	Not Used	Not Used
	→ NODENO	NDOF	RVALUE$_{(1)}$	RVALUE$_{(2)}$
	...	...	...	RVALUE$_{(NDOF)}$

INCONO	Initial condition number.
DTYPE	Type of initial condition. = 1 displacement. = 2 velocity. Both initial displacements and velocities may be specified for a node, but then on separate **BNINCO** data types.
NODENO	Program defined internal number for the node with initial condition.
NDOF	Number of degrees of freedom at the node NODENO.
RVALUE$_{(1)}$	The initial condition with respect to the first degree of freedom at NODENO.
RVALUE$_{(2)}$	The initial condition with respect to the second degree of freedom at NODENO.
⋮	
RVALUE$_{(NDOF)}$	The initial condition with respect to the last degree of freedom at NODENO.
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.21 **BNLOAD**: Node with Load

BNLOAD	→ LLC	LOTYP	COMPLEX	Not Used
	→ NODENO	NDOF	$\text{RLOAD}_{(1)}$	$\text{RLOAD}_{(2)}$
	...	...	...	$\text{RLOAD}_{(\text{NDOF})}$
	$\text{ILOAD}_{(1)}$	$\text{ILOAD}_{(2)}$	...	...
	...	$\text{ILOAD}_{(\text{NDOF})}$		

LLC	Local load case number (positive integer number).	
LOTYP	Type of load at the node NODENO.	
	= 0	not decided whether conservative or non-conservative load.
	= 1	conservative load.
	= -1	non-conservative load.
COMPLEX	Phase shift definition.	
	= 0	no phase shift.
	= 1	phase shift.
NODENO	Program defined internal number for the node with load condition.	
NDOF	Number of degrees of freedom at the node NODENO.	
$\text{RLOAD}_{(1)}$	The real part of the load condition with respect to the first degree of freedom at NODENO.	
$\text{RLOAD}_{(2)}$	The real part of the load condition with respect to the second degree of freedom at NODENO.	
⋮		
$\text{RLOAD}_{(\text{NDOF})}$	The real part of the load condition with respect to the last degree of freedom at NODENO.	
$\text{ILOAD}_{(1)}$	The imaginary part of the load condition with respect to the first degree of freedom at NODENO.	
$\text{ILOAD}_{(2)}$	The imaginary part of the load condition with respect to the second degree of freedom at NODENO.	
⋮		
$\text{ILOAD}_{(\text{NDOF})}$	The imaginary part of the load condition with respect to the last degree of freedom at NODENO.	
Not Used	= 0.	Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $\text{RLOAD}_{(\text{NDOF})}$ – and the first imaginary number – $\text{ILOAD}_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions $\text{ILOAD}_{(1)}$, $\text{ILOAD}_{(2)}$, etc. are left out.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.22 **BNLOAX**: Node with Load (Line Load) for Axi-symmetric Solid (proposal)

BNLOAX	→ LLC	LOTYP	COMPLEX	Not Used
	→ NODENO	NDOF	Not Used	Not Used
	LD	IZERO	NFF	NFL
	NFS	NF	$FC_{(1,1)}$	$FC_{(1,2)}$
	...	$FC_{(1,NDOF)}$	$FC_{(2,1)}$	$FC_{(1,2)}$
	...	$FC_{(2,NDOF)}$	...	$FC_{(NF,1)}$
	...	$FC_{(NF,NDOF)}$		

LLC — Local load case number (positive integer number).

LOTYP — Type of load at the node NODENO. Usually not of interest to linear programs.

- = 0 not decided whether conservative or non-conservative load.
- = 1 conservative load.
- = -1 non-conservative load.

COMPLEX — Phase shift definition.

- = 0 no phase shift.
- = 1 phase shift.

Currently not used.

NODENO — Program defined internal number for the node with load condition.

NDOF — Number of degrees of freedom at the node NODENO.

LD

- = 1 if loading is symmetric with respect to the $q = 0$ axis.
- = 2 if loading is antisymmetric with respect to the $q = 0$ axis.
- = 3 if loading is not symmetric with respect to the $q = 0$ axis.

q is the angle co-ordinate of the cylindrical coordinate system (r, z, q).

IZERO

- = 0 if the zero harmonic is **NOT** included in the Fourier expansion of the load.
- = 1 if the zero harmonic is included in the Fourier expansion of the load.

NFF — The first nonzero harmonic in the Fourier expansion.

NFL — The last nonzero harmonic in the Fourier expansion.

NFS — The harmonic increment in the Fourier expansion.

NF — The total number of harmonics of the Fourier expansion.

Given by

NF = (NFL - NFF) / NFS + 1 + IZERO.

$FC_{(1,1)}$

$FC_{(1,2)}$

⋮	The Fourier coefficients for the degrees of freedom of the node, for the first harmonic.	
$FC_{(1,NDOF)}$		
$FC_{(2,1)}$		
$FC_{(2,2)}$		
⋮	The Fourier coefficients for the degrees of freedom of the node, for the second harmonic.	
$FC_{(2,NDOF)}$		
⋮		
⋮		
$FC_{(NF,1)}$		
$FC_{(NF,2)}$		
⋮	The Fourier coefficients for the degrees of freedom of the node, for the last harmonic.	
$FC_{(NF,NDOF)}$		
Not Used	= 0.	Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

Please, see the **BELLAX** data type for examples.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.23 **BNMASS**: Node with Point Mass

BNMASS	→ NODENO	NDOF	$\text{MASS}_{(1)}$	$\text{MASS}_{(2)}$
	...	...	...	$\text{MASS}_{(\text{NDOF})}$
	ModelNode			

NODENO	Program defined internal number for the node with point mass.
NDOF	Number of degrees of freedom at the node NODENO.
$\text{MASS}_{(1)}$	The point mass with respect to the first degree of freedom at NODENO.
$\text{MASS}_{(2)}$	The point mass with respect to the second degree of freedom at NODENO.
⋮	
$\text{MASS}_{(\text{NDOF})}$	The point mass with respect to the last degree of freedom at NODENO.
ModelNode	If ModelNode > 0, then it represents the super element assembly origin, or part of the original super element model. That is, ModelNode is equal to the unique tree node identifier created when super element models are assembled to global models for direct analysis in Sestra. If ModelNode $\leq$ 0 it means that the data applies for the whole assembled model. For the BNMASS datatype ModelNode is optional. If ModelNode is not given the data applies for the whole model.

The point mass contributions in $\text{MASS}_{(i)}$ refer to the transformed co-ordinate system if the node NODENO is transformed, else to the global co-ordinate system of the super element.

A more general method for specifying mass is the mass element **GMAS**(Sesam element type number 11), specified with the **GELMNT1** and the **MGMASS** data type.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.24 **BNTEMP**: Node with Temperature and Derivative for Temperature

BNTEMP	→ LLC	DTYPE	COMPLEX	Not Used
	→ NODENO	NDOF	$\text{RTEMP}_{(1)}$	$\text{RTEMP}_{(2)}$
	...	...	...	$\text{RTEMP}_{(\text{NDOF})}$
	$\text{ITEMP}_{(1)}$	$\text{ITEMP}_{(2)}$	...	...
	...	$\text{ITEMP}_{(\text{NDOF})}$		

LLC	Local load case number (positive integer number).	
DTYPE	Type of load at the node NODENO.	
	= 1	specified temperature.
	= 2	specified first time derivative of the temperature.
	= 3	specified second time derivative of the temperature.
COMPLEX	Phase shift definition.	
	= 0	no phase shift.
	= 1	phase shift.
NODENO	Program defined internal number for the node with load condition.	
NDOF	Number of degrees of freedom at the node NODENO.	
$\text{RTEMP}_{(1)}$	The real part of the specified boundary condition with respect to the first degree of freedom at NODENO.	
$\text{RTEMP}_{(2)}$	The real part of the specified boundary condition with respect to the second degree of freedom at NODENO.	
⋮		
$\text{RTEMP}_{(\text{NDOF})}$	The real part of the specified boundary condition with respect to the last degree of freedom at NODENO.	
$\text{ITEMP}_{(1)}$	The imaginary part of the specified boundary condition with respect to the first degree of freedom at NODENO.	
$\text{ITEMP}_{(2)}$	The imaginary part of the specified boundary condition with respect to the second degree of freedom at NODENO.	
⋮		
$\text{ITEMP}_{(\text{NDOF})}$	The imaginary part of the specified boundary condition with respect to the last degree of freedom at NODENO.	
Not Used	= 0.	Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

RTEMP and ITEMP refer to the transformed co-ordinate system if the node NODENO is transformed, else to the global co-ordinate system of the super element.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $\text{RTEMP}_{(\text{NDOF})}$ – and the first imaginary number – $\text{ITEMP}_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions $\text{ITEMP}_{(1)}$, $\text{ITEMP}_{(2)}$, etc. are left out.

Note: This data type is *not* available in Sesam.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.25 **BNTRCOS**: Transformation from Global to Local Co-ordinate System, Direction Cosines

BNTRCOS	→ TRANSNO	$c_{1,1}$	$c_{2,1}$	$c_{3,1}$
	$c_{1,2}$	$c_{2,2}$	$c_{3,2}$	$c_{1,3}$
	$c_{2,3}$	$c_{3,3}$		

TRANSNO	Reference number to the transformed co-ordinate system.
$c_{1,1}$	
$c_{2,1}$	
$c_{3,1}$	
$c_{1,2}$	
$c_{2,2}$	Terms (9 direction cosines) of the transformation matrix C.
$c_{3,2}$	
$c_{1,3}$	
$c_{2,3}$	
$c_{3,3}$	

The transformation matrix C describes the transformation defined by

$$r' = Cr \tag{5.1}$$

where r' refers to the local co-ordinate system and r to the global (super element) co-ordinate system.

C is applied to both translations and rotations.

For 2D models all 9 values should be given and the program reading the data type must extract the values $c_{1,1}$, $c_{2,1}$, $c_{1,2}$ and $c_{2,2}$ from this matrix.

The **GUNIVEC** data types are used for beam elements only, i.e. basic element types 2, 15 and 23. Other basic element types may refer to **BNTRCOS** data types. No ambiguity thus exists if both a **GUNIVEC** and **BNTRCOS** data type have same TRANSNO, but they should preferably have separate numbering (TRANSNO) to avoid possible program problems.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.26 **BNWALO**: Node Load from Wave Load Program

BNWALO	→ LLC	→ NODENO	COMPLEX	NDOF
	$FR_{(1)}$	...	$FR_{(NDOF)}$	$FI_{(1)}$
	...	$FI_{(NDOF)}$		

LLC	Local load case number (positive integer number).
NODENO	Program defined internal number for the node with load condition.
COMPLEX	Phase shift definition.
	= 0 no phase shift.
	= 1 phase shift.
NDOF	Number of degrees of freedom at the node NODENO for which loads are given
$FR_{(j)}$	Real component of force for the j'th degree of freedom.
$FI_{(j)}$	Imaginary component of force for the j'th degree of freedom.

Forces are given in the co-ordinate system of the super element in question.

When unformatted the records are packed in the following manner:

The first record contains 6 words (as for all other record types).

The next records contain the rest of the information, and the variable records length must be computed as:

NW = 1 + (1 + COMPLEX)*NDOF,

or NW = NDOF if real loads,

and NW = 2*NDOF if complex loads.

All other data types on the interface file are written with 6 words on each record, where the first two words are reserved for the character string identifier. These positions are blank on the second and following images, until the beginning of a new data type is reached.

The data type **BNWALO** is only generated by Wajac and may only be read from Sestra. This data type should not be used for new applications.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.27 **BQDP**: Node with Simple Quadratic Dependence

BQDP	→ NODENO	DDOF	Not Used	Not Used
	$\text{CNOD}_{(1)}$	$\delta_{(1,1)}$	$\beta_{(1,1)}$	$\delta_{(1,2)}$
	$\beta_{(1,2)}$	...	...	$\delta_{(1,\text{DDOF})}$
	$\beta_{(1,\text{DDOF})}$	Not Used	Not Used	Not Used
	$\text{CNOD}_{(2)}$	$\delta_{(2,1)}$	$\beta_{(2,1)}$	$\delta_{(2,2)}$
	$\beta_{(2,2)}$	...	...	$\delta_{(2,\text{DDOF})}$
	$\beta_{(2,\text{DDOF})}$	Not Used	Not Used	Not Used

⋮

	$\text{CNOD}_{(\text{N})}$	$\delta_{(\text{N},1)}$	$\beta_{(\text{N},1)}$	$\delta_{(\text{N},2)}$
	$\beta_{(\text{N},2)}$	...	...	$\delta_{(\text{N},\text{DDOF})}$
	$\beta_{(\text{N},\text{DDOF})}$			

NODENO — Program defined internal number for the node with dependency.

DDOF — Number of dependent degrees of freedom at the node NODENO. (Must correspond with the number of degrees of freedom with the code FIX = 3 referenced in **BNBCD** for NODENO.)

$\text{CNOD}_{(1)}$ — Program defined internal node number for the first independent node.

$\delta_{(1,1)}$ — The first relevant independent degree of freedom at $\text{CNOD}_{(1)}$ which is coupled to the corresponding dependent degree of freedom at NODENO.

$\beta_{(1,1)}$ — The corresponding contribution of the first relevant degree of freedom at $\text{CNOD}_{(1)}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.

$\delta_{(1,2)}$ — The second relevant independent degree of freedom at $\text{CNOD}_{(1)}$ which is coupled to the corresponding dependent degree of freedom at NODENO.

$\beta_{(1,2)}$ — The corresponding contribution of the second relevant degree of freedom at $\text{CNOD}_{(1)}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.

⋮

$\delta_{(1,\text{DDOF})}$ — The number DDOF relevant independent degree of freedom at $\text{CNOD}_{(1)}$ which is coupled to the corresponding dependent degree of freedom at NODENO.

$\beta_{(1,\text{DDOF})}$ — The corresponding contribution of the DDOF relevant degree of freedom at $\text{CNOD}_{(1)}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.

$\text{CNOD}_{(2)}$ — Program defined internal node number for the second independent node.

$\delta_{(2,1)}$ — The first relevant independent degree of freedom at $\text{CNOD}_{(2)}$ which is coupled to the corresponding dependent degree of freedom at NODENO.

$\beta_{(2,1)}$ — The corresponding contribution of the first relevant degree of freedom at $\text{CNOD}_{(2)}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.

$\delta_{(2,2)}$	The second relevant independent degree of freedom at $\text{CNOD}_{(2)}$ which is coupled to the corresponding dependent degree of freedom at NODENO.
$\beta_{(2,2)}$	The corresponding contribution of the second relevant degree of freedom at $\text{CNOD}_{(1)}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.
⋮	
$\delta_{(2,\text{DDOF})}$	The number DDOF relevant independent degree of freedom at $\text{CNOD}_{(2)}$ which is coupled to the corresponding dependent degree of freedom at NODENO.
$\beta_{(2,\text{DDOF})}$	The corresponding contribution of the DDOF relevant degree of freedom at $\text{CNOD}_{(2)}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.
⋮	
⋮	
$\text{CNOD}_{(\text{N})}$	Program defined internal node number for the last independent node - that is independent node *N*.
$\delta_{(\text{N},1)}$	The first relevant independent degree of freedom at $\text{CNOD}_{(\text{N})}$ which is coupled to the corresponding dependent degree of freedom at NODENO.
$\beta_{(\text{N},1)}$	The corresponding contribution of the first relevant degree of freedom at $\text{CNOD}_{(\text{N})}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.
$\delta_{(\text{N},2)}$	The second relevant independent degree of freedom at $\text{CNOD}_{(\text{N})}$ which is coupled to the corresponding dependent degree of freedom at NODENO.
$\beta_{(\text{N},2)}$	The corresponding contribution of the second relevant degree of freedom at $\text{CNOD}_{(\text{N})}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.
⋮	
$\delta_{(\text{N},\text{DDOF})}$	The number DDOF relevant independent degree of freedom at $\text{CNOD}_{(\text{N})}$ which is coupled to the corresponding dependent degree of freedom at NODENO.
$\beta_{(\text{N},\text{DDOF})}$	The corresponding contribution of the DDOF relevant degree of freedom at $\text{CNOD}_{(\text{N})}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

N is the number of independent nodes that NODENO is dependent on.

By simple is meant that f.ex. the first dependent degree of freedom is always coupled to the first independent degree of freedom, the second dependent degree of freedom is always coupled to the second independent degree of freedom, etc.

Note: This data type is *not* available in Sesam.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.28 **BRIGAC**: Rigid Body Acceleration

BRIGAC	→ LLC	ModelNode	COMPLEX	Not Used
	XCOORD	YCOORD	ZCOORD	Not Used
	RACCL$_{(1)}$	RACCL$_{(2)}$	RACCL$_{(3)}$	RACCL$_{(4)}$
	RACCL$_{(5)}$	RACCL$_{(6)}$	IACCL$_{(1)}$	IACCL$_{(2)}$
	IACCL$_{(3)}$	IACCL$_{(4)}$	IACCL$_{(5)}$	IACCL$_{(6)}$

LLC	Local load case number (positive integer number).	
ModelNode	If ModelNode > 0, then it represents the super element assembly origin, or part of the original super element model. That is, ModelNode is equal to the unique tree node identifier created when super element models are assembled to global models for direct analysis in Sestra. If ModelNode $\leq$ 0 it means that the data applies for the whole assembled model.	
COMPLEX	Phase shift definition.	
	= 0	no phase shift.
	= 1	phase shift.
XCOORD	Cartesian X, Y- and Z-co-ordinates of the point of acceleration.	
YCOORD	The co-ordinates are defined relative to the super element	
YCOORD	global co-ordinate system.	
RACCL$_{(i)}$	The real part of the rigid body acceleration with respect to degree of freedom no. 'i'. The three first degrees of freedom are translational accelerations and the next three are angular accelerations (in radians per second squared).	
IACCL$_{(i)}$	The imaginary part of the rigid body acceleration with respect to degree of freedom no. 'i'. The imaginary parts are given only if COMPLEX= 1. The three first degrees of freedom are translational accelerations and the next three are angular accelerations (in radians per second squared).	
Not Used	= 0.	Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

This data type describes an acceleration of a point in a rigid body. It is to be used for calculation of forces of inertia. The position of the point is specified by its co-ordinates on the data type. All rotations are in radians.

NDOF = 6 = number of degrees of freedom at the point in question.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – RACCL$_{(NDOF)}$ – and the first imaginary number – IACCL$_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions IACCL$_{(1)}$, IACCL$_{(2)}$, etc. are left out.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.29 **BRIGDI**: Rigid Body Displacement

BRIGDI	→ LLC	ModelNode	COMPLEX	Not Used
	XCOORD	YCOORD	ZCOORD	Not Used
	RDISP$_{(1)}$	RDISP$_{(2)}$	RDISP$_{(3)}$	RDISP$_{(4)}$
	RDISP$_{(5)}$	RDISP$_{(6)}$	IDISP$_{(1)}$	IDISP$_{(2)}$
	IDISP$_{(3)}$	IDISP$_{(4)}$	IDISP$_{(5)}$	IDISP$_{(6)}$

LLC	Local load case number (positive integer number).
ModelNode	If ModelNode > 0, then it represents the super element assembly origin, or part of the original super element model. That is, ModelNode is equal to the unique tree node identifier created when super element models are assembled to global models for direct analysis in Sestra. If ModelNode $\leq$ 0 it means that the data applies for the whole assembled model.
COMPLEX	Phase shift definition. = 0 no phase shift. = 1 phase shift.
XCOORD	Cartesian X, Y- and Z-co-ordinates of the point of displacement.
YCOORD	The co-ordinates are defined relative to the super element
YCOORD	global co-ordinate system.
RDISP$_{(i)}$	The real part of the rigid body displacement with respect to degree of freedom no. 'i'. The three first degrees of freedom are translations and the next three are rotations (in radians).
IDISP$_{(i)}$	The imaginary part of the rigid body displacement with respect to degree of freedom no. 'i'. The imaginary parts are given only if COMPLEX= 1. The three first degrees of freedom are translations and the next three are rotations (in radians).
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

This data type describes a displacement of a point in a rigid body. It is to be used for calculation of forces of inertia. The position of the point is specified by its co-ordinates on the data type. All rotations are in radians.

NDOF = 6 = number of degrees of freedom at the point in question.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – RDISP$_{(NDOF)}$ – and the first imaginary number – IDISP$_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions IDISP$_{(1)}$, IDISP$_{(2)}$, etc. are left out.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.2.30 **BRIGVE**: Rigid Body Velocity

BRIGVE	→ LLC	ModelNode	COMPLEX	Not Used
	XCOORD	YCOORD	ZCOORD	Not Used
	RVELO$_{(1)}$	RVELO$_{(2)}$	RVELO$_{(3)}$	RVELO$_{(4)}$
	RVELO$_{(5)}$	RVELO$_{(6)}$	IVELO$_{(1)}$	IVELO$_{(2)}$
	IVELO$_{(3)}$	IVELO$_{(4)}$	IVELO$_{(5)}$	IVELO$_{(6)}$

LLC	Local load case number (positive integer number).
ModelNode	If ModelNode > 0, then it represents the super element assembly origin, or part of the original super element model. That is, ModelNode is equal to the unique tree node identifier created when super element models are assembled to global models for direct analysis in Sestra. If ModelNode $\leq$ 0 it means that the data applies for the whole assembled model.
COMPLEX	Phase shift definition. = 0 no phase shift. = 1 phase shift.
XCOORD	Cartesian X, Y- and Z-co-ordinates of the point of velocity.
YCOORD	The co-ordinates are defined relative to the super element
YCOORD	global co-ordinate system.
RVELO$_{(i)}$	The real part of the rigid body velocity with respect to degree of freedom no. 'i'. The three first degrees of freedom are translational velocities and the next three are angular velocities (in radians per second).
IVELO$_{(i)}$	The imaginary part of the rigid body velocity with respect to degree of freedom no. 'i'. The imaginary parts are given only if COMPLEX= 1. The three first degrees of freedom are translational velocities and the next three are angular velocities (in radians per second).
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

This data type describes a displacement, velocity and an acceleration of a point in a rigid body. The rotational part of the velocity yields centrifugal forces of inertia. The position of the point is specified by its co-ordinates on the data type. All rotations are in radians.

NDOF = 6 = number of degrees of freedom at the point in question.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – RVELO$_{(NDOF)}$ – and the first imaginary number – IVELO$_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions IVELO$_{(1)}$, IVELO$_{(2)}$, etc. are left out.

Back to Section 5.2 **Boundary Conditions, Loads and Point Masses**

5.3 Nodal Data and Element Geometry Definition

First level data

GBARM	Cross Section Type Massive Bar	see Section 5.3.1
GBEAMG	General Beam Element Data	see Section 5.3.2
GBOX	Cross Section Type Box Beam	see Section 5.3.3
GCHAN	Cross Section Type Channel Beam	see Section 5.3.4
GCHANR	Cross Section Type Channel Beam with Inside Curvature	see Section 5.3.5
GCOORD	Nodal Co-ordinates	see Section 5.3.6
GCROINT	Specification of Integration Points	see Section 5.3.7
GDOBO	Section Type Double Bottom	see Section 5.3.8
GECC	Local Eccentricities	see Section 5.3.9
GECCEN	Eccentricities	see Section 5.3.10
GELINT	Specification of Integration Stations	see Section 5.3.11
GELMNT1	Element Data Definition	see Section 5.3.12
GELREF1	Reference to Element Data	see Section 5.3.13
GELSTRP	Specification of Stress Points	see Section 5.3.14
GELTH	Thickness of Two-dimensional Elements	see Section 5.3.15
GIORH	Cross Section Type I or H Beam	see Section 5.3.16
GIORHR	Cross Section Type I or H Beam with Inside Curvature	see Section 5.3.17
GLMASS	Modification of Diagonal Mass Matrices	see Section 5.3.18
GLSEC	Cross Section Type L-Section	see Section 5.3.19
GLSECR	Cross Section Type L-Section with Inside Curvature	see Section 5.3.20
GNODE	Correspondence between External and Internal Node Numbering, and Number of Degrees of Freedom at Each Node	see Section 5.3.21
GPGBOX	Cross Section Type Boxed Plate Girder	see Section 5.3.22
GPGDOW	Cross Section Type Double Web Plate Girder	see Section 5.3.23
GPIPE	Cross Section Type Tube	see Section 5.3.24
GSEPSPEC	Specified Separation Description	see Section 5.3.25
GSETMEMB	Set (group) of Nodes or Elements (Members)	see Section 5.3.26
GSLAYER	General Eccentric Sandwich Element	see Section 5.3.27
GSLPLATE	Plate Layer Description	see Section 5.3.28
GSLSTIFF	Stiffener Layer Description	see Section 5.3.29
GTONP	Cross Section T on Plate	see Section 5.3.30
GUNIVEC	Specification of Local Element	see Section 5.3.31

GUSYI	Cross Section Type Unsymmetrical I-Beam	see Section 5.3.32

5.3.1 **GBARM**: Cross Section Type Massive Bar

GBARM	→ GEONO	HZ	BT	BB
	SFY	SFZ	NLOBY	NLOBZ

GEONO	Geometry type number, referenced from the data type **GELREF1**.
HZ	Height of beam.
BT	Width of bar at top. For massive bars which are not able to have different widths at top and bottom this variable is used as the width of the beam.
BB	Width of bar at bottom.
SFY, SFZ	Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively SHARY(MOD) = SHARY(PROG) · SFY SHARZ(MOD) = SHARZ(PROG) · SFZ (The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).
NLOBY	Number of integration points in Y' direction (optional).
NLOBZ	Number of integration points in Z' direction (optional).

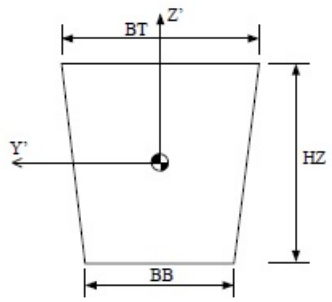


Figure 5.3: Massive bar

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.2 **GBEAMG**: General Beam Element Data

GBEAMG	→ GEONO	COMP	AREA	IX
	IY	IZ	IYZ	WXMIN
	WYMIN	WZMIN	SHARY	SHARZ
	SHCENY	SHCENZ	SY	SZ
	WY	WZ	FABR	Not Used

GEONO	Geometry type number, referenced from the data type **GELREF1**.
COMP	Computation method (0 = computed, 1 = manually overridden, 2 = library values)
AREA	Cross section area.
IX	Torsional moment of inertia about the shear centre.
IY	Moment of inertia about the y-axis $= \int z^2 dA$.
IZ	Moment of inertia about the z-axis $= \int y^2 dA$.
IYZ	Product of inertia about y- and z-axes $= \int yz dA$.
WXMIN	Minimum torsional section modulus about shear centre (= IX$/r_{max}$ for a PIPE element).
WYMIN	Minimum section modulus about y-axis = IY$/z_{max}$.
WZMIN	Minimum section modulus about z-axis = IZ$/y_{max}$.
SHARY	Shear area in the direction of y-axis. If zero, then shear is not included for the BEAS 2-noded beam element (ELTYP = 15), see Section 3.10.
SHARZ	Shear area in the direction of z-axis. If zero, then shear is not included for the BEAS 2-noded beam element (ELTYP = 15), see Section 3.10.
SHCENY	Shear centre location y component.
SHCENZ	Shear centre location z component.
SY	Static area moment about y-axis $= \int z dA$.
SZ	Static area moment about z-axis $= \int y dA$.
WPY	Plastic section modulus about y-axis.
WPZ	Plastic section modulus about z-axis.
FABR	Fabrication method (0 = unknown, 1 = rolled, 2 = welded)
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

If **GBEAMG** is used for the truss element (ELTYP = 10), then it is possible that only the first line with section AREA is on the interface file.

Further, the line with WPY, WPZ and FABR is optional, and is not found on older interface files.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.3 **GBOX**: Cross Section Type Box Beam

GBOX	→ GEONO	HZ	TY	TB
	TT	BY	SFY	SFZ
	NLOBY	NLOBZ		

GEONO	Geometry type number, referenced from the data type **GELREF1**.
HZ	Height of beam at current location.
TY	Thickness of vertical walls (webs) of box section.
TB	Thickness of bottom flange.
TT	Thickness of top flange.
BY	Width of box beam.
SFY, SFZ	Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively SHARY(MOD) = SHARY(PROG) · SFY SHARZ(MOD) = SHARZ(PROG) · SFZ (The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).
NLOBY	Number of integration points in each horizontal wall (flange) of beam (optional).
NLOBZ	Number of integration points in each vertical wall (web) of beam (optional).

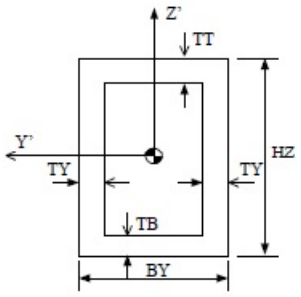


Figure 5.4: Box beam

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.4 **GCHAN**: Cross Section Type Channel Beam

GCHAN	→ GEONO	HZ	TY	BY
	TZ	SFY	SFZ	Not Used
	K	NLOBY	NLOBZ	

GEONO — Geometry type number, referenced from the data type **GELREF1**.

HZ — Height of beam at current location.

TY — Thickness of beam web.

BY — Width of top and bottom flange.

TZ — Thickness of top and bottom flange.

SFY, SFZ — Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively.

$$SHARY(MOD) = SHARY(PROG) \cdot SFY$$

$$SHARZ(MOD) = SHARZ(PROG) \cdot SFZ$$

(The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).

K — Web orientation:

= 0 Web located in the negative local y-direction (and consequently flange in the postitive y'-direction).

= 1 Web located in the positive local y-direction (and consequently flange in the negative y'-direction).

NLOBY — Number of integration points in each flange (optional).

NLOBZ — Number of integration points in beam web (optional).

Not Used — = 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

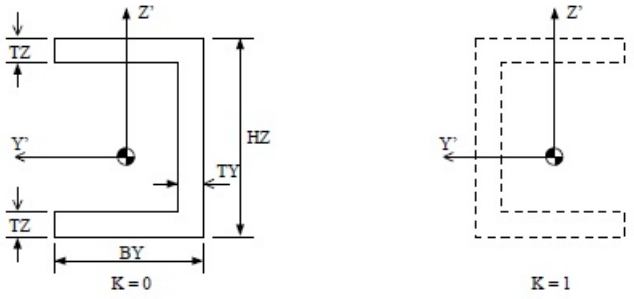


Figure 5.5: Channel beam

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.5 **GCHANR**: Cross Section Type Channel Beam with Inside Curvature

GCHANR	→ GEONO	HZ	TY	BY
	TZ	SFY	SFZ	Not Used
	K	R	NLOBY	NLOBZ

GEONO — Geometry type number, referenced from the data type **GELREF1**.

HZ — Height of beam at current location.

TY — Thickness of beam web.

BY — Width of top and bottom flange.

TZ — Thickness of top and bottom flange.

SFY, SFZ — Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively.

$$SHARY(MOD) = SHARY(PROG) \cdot SFY$$

$$SHARZ(MOD) = SHARZ(PROG) \cdot SFZ$$

(The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).

K — Web orientation:

= 0 Web located in the negative local y-direction (and consequently flange in the postitive y'-direction).

= 1 Web located in the positive local y-direction (and consequently flange in the negative y'-direction).

R — Radius of inside curvature.

NLOBY — Number of integration points in each flange (optional).

NLOBZ — Number of integration points in beam web (optional).

Not Used = 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

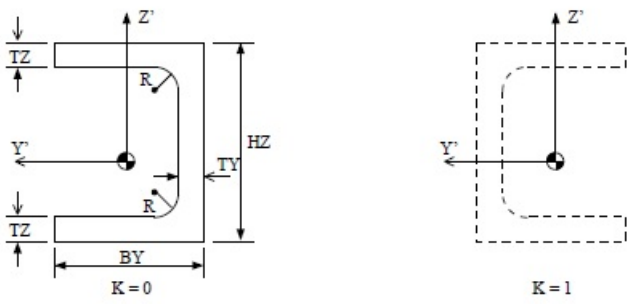


Figure 5.6: Channel beam with inside curvature

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.6 **GCOORD**: Nodal Co-ordinates

GCOORD	→ NODENO	XCOORD	YCOORD	ZCOORD

NODENO	Program defined internal number for the node.
XCOORD	x co-ordinate.
YCOORD	y co-ordinate.
ZCOORD	z co-ordinate.

There will be one data type with the identifier **GCOORD** for each node. The sequence of the data types will correspond to the internal node number, NODENO.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.7 **GCROINT**: Specification of Integration Points

GCROINT	→ CROINO	INTYPE	N1	N2
	N3	Y(1)	Z(1)	W(1)
	. . .	Y(N3)	Z(N3)	W(N3)

For definition of an integration point, see data type **GELINT** (specification of integration stations). This data type should only be given for 1- and 2-dimensional elements.

CROINO — Integration point number, referenced to on the **GELINT** data type.

INTYPE — Integration type number, i.e. information on how to distribute the integration points.

= 0 — The co-ordinates and weights of the points are given, see below. The parameters N1, N2 and N3 have the following interpretation:

- N1 = 0 — The succeeding co-ordinates are specified in curvelinear form (natural co-ordinates).
- N1 = 1 — The succeeding co-ordinates are specified in absolute form.
- N2 — Not employed, i.e. vacant position.
- N3 — Number of integration points.

> 0 — The number of integration points is given and they should be distributed according to the Gaussian integration scheme if INTYPE=1, and the Lobatto integration scheme if INTYPE =2.

The parameters N1, N2 and N3 have the following interpretation for a 2-dimensional element:

- N1 — Number of points to be distributed across the thickness.
- N2 & N3 — Not employed, i.e. vacant positions.

N1, N2 and N3 will have the following interpretation for a 1-dimensional (beam or bar) element:

- N1 & N2 — Specification of number of points in two directions of the cross section. For a tube: circumferential and radial direction respectively. For other cross sections along local element y-axis and z-axis, respectively.
- N3 — Number of integration points. Note that N3 will not necessarily will be the product of N1 and N2 (e.g. for an I-section).

N1, N2, N3 — See explanation of INTYPE above.

Y(i), Z(i) — Coordinates of integration point No. i. Note that Y(i) will not be used, i.e. it is a vacant position, for 2-dimensional elements. Both Y(i) and Z(i) are omitted if INTYPE > 0.

W(i) — Weight of integration point No. i. Omitted if INTYPE > 0.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.8 **GDOBO**: Section Type Double Bottom

GDOBO	→ GEONO	HZ	TY	BY
	TT	TB	SFY	SFZ
	NLOBY	NLOBZ		

GEONO	Geometry type number, referenced from the data type **GELREF1**.
HZ	Height of beam at current location.
TY	Thickness of beam web.
BY	Effective width of plates.
TT	Thickness of top plate.
TB	Thickness of bottom plate.
SFY, SFZ	Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively SHARY(MOD) = SHARY(PROG) · SFY SHARZ(MOD) = SHARZ(PROG) · SFZ (The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).
NLOBY	Number of integration points in each flange (optional).
NLOBZ	Number of integration points in beam web (optional).

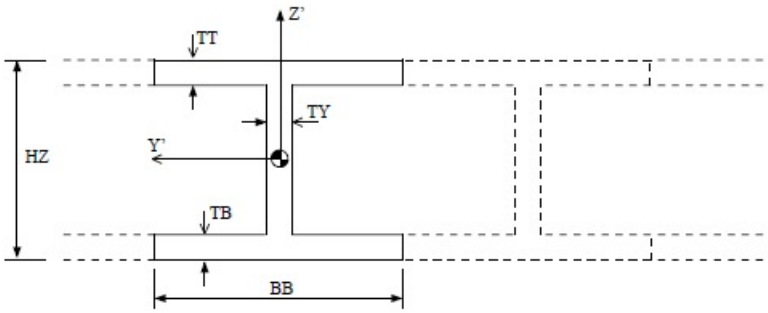


Figure 5.7: Double bottom

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.9 **GECC**: Local Eccentricities

GECC	→ ECCNO	IOPT	EX / EZ	(EY)
	(EZ)			

ECCNO — Eccentricity number, referenced from the **GELREF1** data type. The same ECCNO number may not be on both a **GECC** and a **GECCEN** data type for a super element.

IOPT — Option for number of eccentricity components specified.

- = 1 — Only the local Z-component – EZ – of the eccentricity is specified. The X- and Y-components – EX & EY – are 0.0.
- = 2 — The local X-, Y- and Z-component – EX, EY & EZ – of the eccentricity are specified.

EX, EY, EZ — Eccentricity vector given in the local element co-ordinate system. Positive eccentricity is directed from the system node to the 'element node' (for the specific layer, if more layers).

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.10 **GECCEN**: Eccentricities

GECCEN	→ ECCNO	EX	EY	EZ

ECCNO	Eccentricity number, referenced from the **GELREF1** data type. The same ECCNO number may not be on both a **GECCEN** and a **GECC** data type for a super element.
EX, EY, EZ	Eccentricity vector given in the super element co-ordinate system, the vector points from the global node towards the local element node.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.11 **GELINT**: Specification of Integration Stations

GELINT	→ INTNO	INTYPE	N1	N2
	N3	$CROINO_{(1)}$	...	$CROINO_{(N2)}$
	$X_{(1,1)}$	...	$X_{(1,N3)}$	...
	...	$X_{(N2,1)}$	...	$X_{(N2,N3)}$

An integration station is defined as:

- an assembly of integration points over a cross section of a 1-dimensional (beam or a bar) element,
- an assembly of integration points on a line through the thickness of a 2-dimensional element,
- one single integration point for a 3-dimensional element.

An integration station comprises the properties of the integration points connected to it, and is used to calculate the element matrices (See Figure 5.8).

INTNO — Integration station reference number, referenced to from the data type **GELREF1**, and data types for load description, i.e. from **BEISTE**, **BELLAX**, **BELLO2**, **BELOAD1**, **BEUSLO**, and **BEUVLO**.

INTYPE — Integration type number, i.e. information on how to distribute the integration stations.

= 0 — The co-ordinates of the stations are given, see parameter X below.

The parameters N1, N2 and N3 have the following interpretation:

N1 = 0 — The co-ordinates are specified in curvelinear form.

N1 = 1 — The co-ordinates are specified in absolute form.

N2 — Number of integration stations.

N3 — Number of co-ordinate components needed for the definition of an integration station.

> 0 — The number of integration stations is given and they should be distributed according to the Gaussian integration scheme if INTYPE=1, and the Lobatto integration scheme if INTYPE=2.

The parameters N1, N2 and N3 have the following interpretation:

N1 — Number of stations to be distributed along the 1st local element axis.

N2 — Number of stations to be distributed along the 2nd local element axis (only 2- and 3-dimensional elements).

N3 — Number of stations to be distributed along the 3rd local element axis (only 3-dimensional elements).

N1, N2, N3	Number of integration points in each direction. See also explanation of INTYPE above. For the 3-dimensional 20 noded hexahedron elements the values may be 2, 3, 4, 32 or 42. A two digit number means that different number of integration points are used for normal strain stiffness terms, and shear stiffness terms to avoid 'shear locking'.
CROINO$_{(1)}$	If INTYPE=0: Integration point number, i.e. number referring to specification of integration points in integration station No. 1. If INTYPE>0: As above, only for all integration stations. CROINO(2),...,CROINO(N2) are omitted.
⋮	
CROINO$_{(N2)}$	Integration point number, i.e. number referring to specification of integration points in integration station No. N2 (the last). Omitted if INTYPE > 0.
X$_{(i,j)}$	Coordinate component number j for integration station number i. Omitted if INTYPE > 0.

Note that for 3-dimensional elements CROINO will have no meaning and should be left vacant.

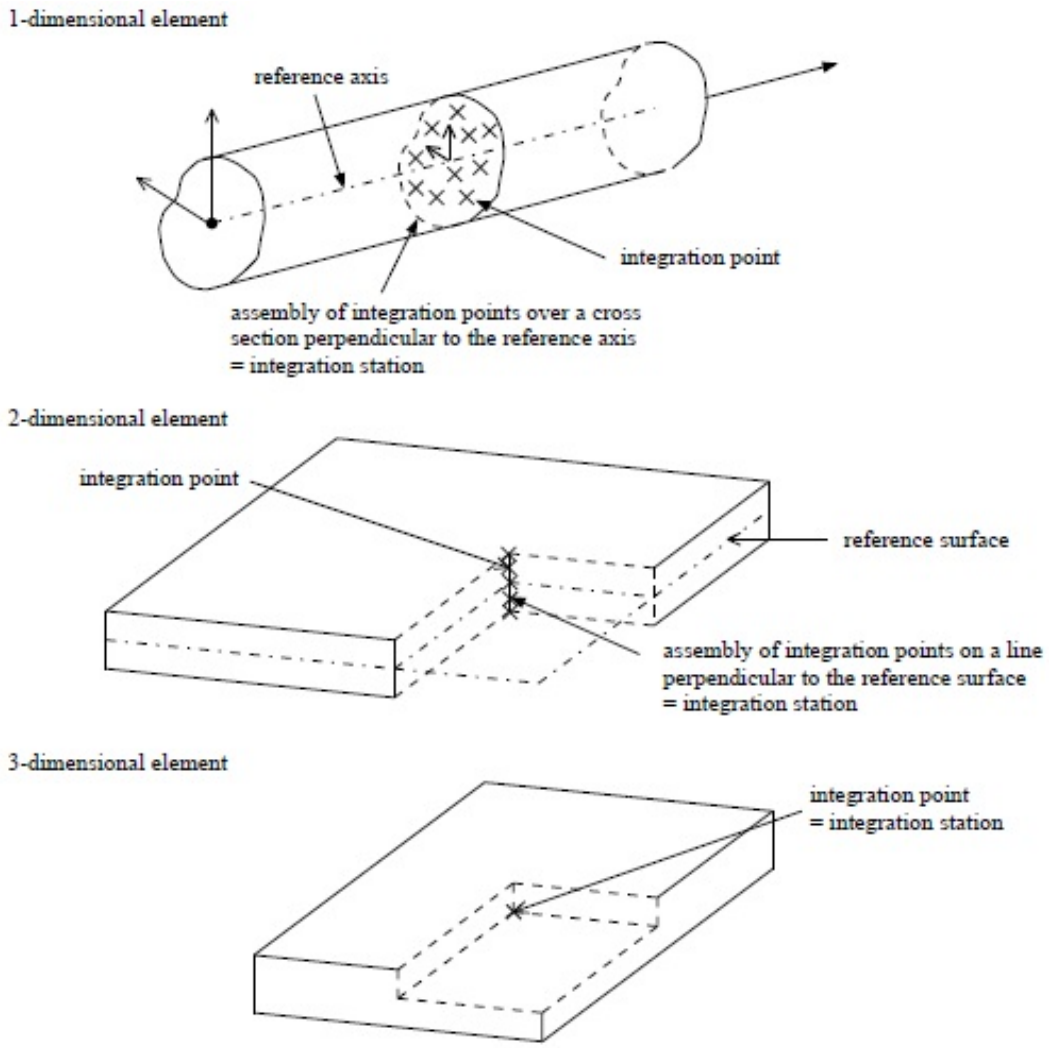


Figure 5.8: Illustration of an integration point and an integration station.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.12 **GELMNT1**: Element Data Definition

GELMNT1	ELNOX	→ ELNO	ELTYP	ELTYAD
	$\text{NODIN}_{(1)}$	$\text{NODIN}_{(2)}$	...	...
	...	$\text{NODIN}_{(\text{NNOD})}$	ModelNode	AddedMass

ELNOX — External element number (specified or controlled by the user). Can be any integer identifier as long as it is unique in the element numbering sequence.

ELNO — Program defined internal number for the element. The internal element numbers range from 1 and continuously - no holes in the numbering - to number of elements in the finite element model.

ELTYP — Element type number. Refer to section 3 for description of legal element type numbers. For element type number 70 ("matrix element") all relevant element data are stored as stiffness, mass, damping matrices, and so on. See the **AMATRIX** data type for more information.

ELTYAD — Additional information related to element type:

For membrane elements used to specify plane stress / plane strain conditions

= 0 Plane stress

= 1 Plane strain

For standard finite elements used to specify structural / non-structural elements:

= 0 Structural elements

= 1 Non structural beam, not including mass in direction of the beam axis

= 2 Non structural elements

For general matrix element (element type number 70) used to specify number of nodes

= NNOD Number of nodes on the matrix element

$\text{NODIN}_{(1)}$

$\text{NODIN}_{(2)}$ — Global internal node numbers of the elements in question. The sequence of the node numbers is in accordance with the local node numbering of the basic elements.

$\text{NODIN}_{(\text{NNOD})}$

ModelNode — If ModelNode > 0, then it represents the super element assembly origin, or part of the original super element model. That is, ModelNode is equal to the unique tree node identifier created when super element models are assembled to global models for direct analysis in Sestra. If ModelNode ≤ 0 it means that the data applies for the whole assembled model.

AddedMass

$\neq 1$ Not added mass element.

= 1 Added mass element.

Note:

NNOD = number of nodes at the element. For element type 70 NNOD = ELTYAD. For the standard Sesam finite elements, the number of nodes is not present at this data type.

The length of the data type is always greater than or equal to 4 + NNOD.

1. If the length of the data type is greater than or equal to 5 + NNOD, then it is assumed that the 5 + NNOD entry is ModelNode.
2. If the length of the data type is greater than or equal to 6 + NNOD, then it is assumed that the 6 + NNOD entry is AddedMass.
3. Further trailing data items should be set to zero.

By global node number is meant the node numbering of the entire super element of which the element ELNOX is a part. The internal node number refers to the node number generated by the program.

The program defined element number ranges from 1 up to number of elements.

The sequence of the data type will correspond to the program defined element numbering, ELNO.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.13 **GELREF1**: Reference to Element Data

GELREF1	→ ELNO	MATNO	ADDNO	INTNO
	MINTNO	STRANO	STRENO	STREPONO
	GEONO/OPT	FIXNO/OPT	ECCNO/OPT	TRANSNO/OPT
	$\text{GEONO}_{(1)}$	. . .	$\text{GEONO}_{(\text{NNOD})}$	$\text{FIXNO}_{(1)}$
	. . .	$\text{FIXNO}_{(\text{NNOD})}$	$\text{ECCNO}_{(1)}$	. . .
	$\text{ECCNO}_{(\text{NNOD})}$	$\text{TRANSNO}_{(1)}$	. . .	$\text{TRANSNO}_{(\text{NNOD})}$

Shortest version for which GEONO, FIXNO, ECCNO, TRANSNO ≥ 0:

GELREF1	ELNO	MATNO	ADDNO	INTNO
	MINTNO	STRANO	STRENO	STREPONO
	GEONO	FIXNO	ECCNO	TRANSNO

ELNO — Program defined internal number for the element

MATNO — Material number.

= 0 — No material data attached to the element.

ADDNO — Additional data type number, i.e. number referring to additional data specification.

= 0 — No additional data attached to the element.

=-1 — I.e when ADDNO = -1, then the item STRANO represents the element type number.

INTNO — Integration station reference number for stiffness matrix, i.e. number referring to the specification of integration stations. An integration station is defined as:

- an assembly of integration points over a cross section of a 1-dimensional (beam or bar) element,
- an assembly of integration points on a line through the thickness of a 2-dimensional element,
- one single integration point for a 3-dimensional element. For further explanation see data type **GELINT**.

= 0 — Default values of the analysis program are employed.

MINTNO — Integration station reference number for mass and damping matrices. Integration station, see INTNO.

= 0 — Default values of the analysis program are employed.

STRANO — Initial strain number, i.e. a number referring to the specification of initial strains. (To be given on data type ASTR which is not yet defined.)

As noted above under item ADDNO, if ADDNO = -1, then STRANO = element type.

STRENO		Initial stress number, i.e. a number referring to the specification of initial stresses. (To be given on data type ASTR which is not yet defined.)
STREPONO		Stress point specification reference number. See data type **GELSTRP** for further information.
GEONO/OPT		Geometry reference number or option for geometry reference number specified later in this data type sequence.
	>0	The geometry reference number (the same for all nodes in the element). $\text{GEONO}_{(1)}$, . . . , $\text{GEONO}_{(\text{NNOD})}$ will not be specified.
	=0	No geometry data is given, i.e. neither here nor on $\text{GEONO}_{(1)}$, . . . , $\text{GEONO}_{(\text{NNOD})}$.
	=-1	Reference numbers to geometry data are specified later in this data type sequence for all nodes, i.e. all $\text{GEONO}_{(1)}$, . . . , $\text{GEONO}_{(\text{NNOD})}$ will be given.
FIXNO/OPT		Fixation reference number or option for fixation reference numbers specified later in this data type sequence. The meaning assigned to the values of FIXNO/OPT corresponds to those for GEONO/OPT.
ECCNO/OPT		Eccentricity reference number or option for eccentricity reference numbers specified later in this data type sequence. The meaning assigned to the values of ECCNO/OPT corresponds to those for GEONO/OPT.
TRANSNO/OPT		Reference number for local co-ordinate system specification or option for specification of local nodal co-ordinate systems later in this data type sequence. Refers to the **GUNIVEC** or **BNTRCOS** data types. The meaning assigned to the values of TRANSNO/OPT corresponds to those for GEONO/OPT.
$\text{GEONO}_{(1)}$		Geometry reference number, i.e. number referring to thickness or cross sectional specification. Not employed for 3-dimensional elements. $\text{GEONO}_{(1)}$ is the reference number for the 1st local node of the element, $\text{GEONO}_{(i)}$ will be the reference number for the i'th local node.
$\text{GEONO}_{(\text{NNOD})}$		Geometry reference number for the last local node of the element.
$\text{FIXNO}_{(1)}$		Number referring to the specification of degree of fixation (data type BELFIX). $\text{FIXNO}_{(1)}$ is the reference number for the 1st local node of the element, $\text{FIXNO}_{(i)}$ will be the reference number for the i'th local node.
$\text{FIXNO}_{(\text{NNOD})}$		Degree of fixation reference number for the last local node of the element.
$\text{ECCNO}_{(1)}$		Eccentricity number for the first local node of the element, i.e. number referring to the specification of eccentricities.
$\text{ECCNO}_{(\text{NNOD})}$		As $\text{ECCNO}_{(1)}$ only for the last local node.
$\text{TRANSNO}_{(1)}$		Number referring to the specification of the local element co-ordinate system for the 1st local node of the element. Refers to **GUNIVEC** or **BNTRCOS** data type depending on element type.
$\text{TRANSNO}_{(\text{NNOD})}$		As $\text{TRANSNO}_{(1)}$ only for the last local node.

NNOD is the number of local nodes at the element.

NOTE: Parameters appear in succeeding order from third line.

The sequence of the data types will be in the program defined element numbering, ELNO.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.14 **GELSTRP**: Specification of Stress Points

GELSTRP	→ STREPONO	STRPTYP	N1	N2
	N3	$X_{(1,1)}$	...	$X_{(1,N3)}$
	...	...	$X_{(N2,1)}$	...
	$X_{(N2,N3)}$			

STREPONO — Stress point specification reference number.

Referenced from the **GELREF1** data type.

STRPTYP — Type of stress point specification

= 0: The co-ordinates of the stress points are given in the $X_{(i,j)}$ data, see below.

N1 = 0: the co-ordinates are specified in curvelinear form.

N1 = 1: the co-ordinates are specified in absolute form.

N2: the number of stress points in the element.

N3: the number of co-ordinate components needed for the definition of a stress point.

> 0: The number of stress points given, and they should be distributed according to the Gaussian integration scheme if STRTYP = 1, and according to the Lobatto integration scheme if STRTYP = 2, i.e. the stress points coincide with the integration points.
If STRTYP = 3, the stress points are distributed according to the default method for the element type considered.

$X_{(i,j)}$ — Co-ordinate component no. j for station no. i. Omitted if STRPTYP > 0.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.15 **GELTH**: Thickness of Two-dimensional Elements

GELTH	→ GEONO	TH	NINT	ISHEAR

GEONO	Geometry type number, referenced from the data type **GELREF1**.
TH	Thickness of the element, measured in a specific node.
NINT	Number of integration points through thickness.
ISHEAR	Option for transversal shear correction 1. order thin shells.[1,2]
	= 0 : No correction.
	= 1 : Correction.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

[1] Disabled in Sestra.
[2] Added to the specification due to usage and a comment in Sestra 8 routine `ESFQ80`.

5.3.16 **GIORH**: Cross Section Type I or H Beam

GIORH	→ GEONO	HZ	TY	BT
	TT	BB	TB	SFY
	SFZ	NLOBYT	NLOBYB	NLOBZ

GEONO	Geometry type number, referenced from the data type **GELREF1**.
HZ	Height of beam at current location.
TY	Thickness of beam web.
BT	Width of top flange.
TT	Thickness of top flange.
BB	Width of bottom flange.
TB	Thickness of bottom flange.
SFY, SFZ	Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively SHARY(MOD) = SHARY(PROG) · SFY SHARZ(MOD) = SHARZ(PROG) · SFZ (The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).
NLOBYT	Number of integration points in top flange (optional).
NLOBYB	Number of integration points in bottom flange (optional).
NLOBZ	Number of integration points in beam web (optional).

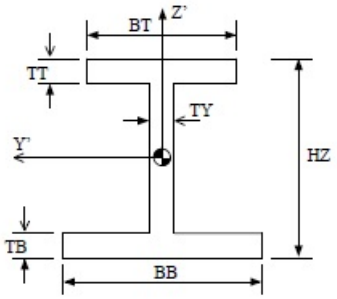


Figure 5.9: I or H beam

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.17 **GIORHR**: Cross Section Type I or H Beam with Inside Curvature

GIORHR	→ GEONO	HZ	TY	BT
	TT	BB	TB	SFY
	SFZ	RT	RB	NLOBYT
	NLOBYB	NLOBZ		

GEONO	Geometry type number, referenced from the data type **GELREF1**.
HZ	Height of beam at current location.
TY	Thickness of beam web.
BT	Width of top flange.
TT	Thickness of top flange.
BB	Width of bottom flange.
TB	Thickness of bottom flange.
SFY, SFZ	Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively SHARY(MOD) = SHARY(PROG) · SFY SHARZ(MOD) = SHARZ(PROG) · SFZ (The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).
RT	Radius of inside curvature at top.
RB	Radius of inside curvature at bottom.
NLOBYT	Number of integration points in top flange (optional).
NLOBYB	Number of integration points in bottom flange (optional).
NLOBZ	Number of integration points in beam web (optional).

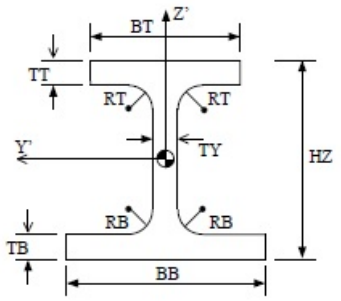


Figure 5.10: I or H beam with inside curvature

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.18 **GLMASS**: Modification of Diagonal Mass Matrices

GLMASS	RFAC			

RFAC — Factor by which the rotational masses of the lumped diagonal mass matrix are multiplied.

Default value = 0.01.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.19 **GLSEC**: Cross Section Type L-Section

GLSEC	→ GEONO	HZ	TY	BY
	TZ	SFY	SFZ	K
	NLOBY	NLOBZ		

GEONO — Geometry type number, referenced from the data type **GELREF1**.

HZ — Height of beam at current location.

TY — Thickness of beam web.

BY — Width of flange.

TZ — Thickness of flange.

SFY, SFZ — Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively

$$SHARY(MOD) = SHARY(PROG) \cdot SFY$$

$$SHARZ(MOD) = SHARZ(PROG) \cdot SFZ$$

(The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).

K — Web orientation:

= 0 web located in the negative local y-direction (and consequently flange in the postitive y'-direction).

= 1 web located in the positive local y-direction (and consequently flange in the negative y'-direction)

NLOBY — Number of integration points in beam flange (optional).

NLOBZ — Number of integration points in beam web (optional).

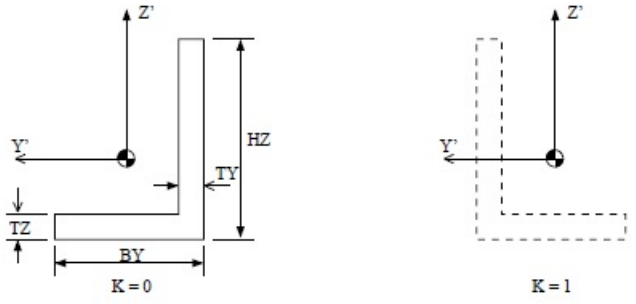


Figure 5.11: L-section

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.20 **GLSECR**: Cross Section Type L-Section with Inside Curvature

GLSECR	→ GEONO	HZ	TY	BY
	TZ	SFY	SFZ	K
	R	NLOBY	NLOBZ	

GEONO — Geometry type number, referenced from the data type **GELREF1**.

HZ — Height of beam at current location.

TY — Thickness of beam web.

BY — Width of flange.

TZ — Thickness of flange.

SFY, SFZ — Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively

$$SHARY(MOD) = SHARY(PROG) \cdot SFY$$

$$SHARZ(MOD) = SHARZ(PROG) \cdot SFZ$$

(The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).

K — Web orientation:

= 0 web located in the negative local y-direction (and consequently flange in the postitive y'-direction).

= 1 web located in the positive local y-direction (and consequently flange in the negative y'-direction)

R — Radius of inside curvature

NLOBY — Number of integration points in beam flange (optional).

NLOBZ — Number of integration points in beam web (optional).

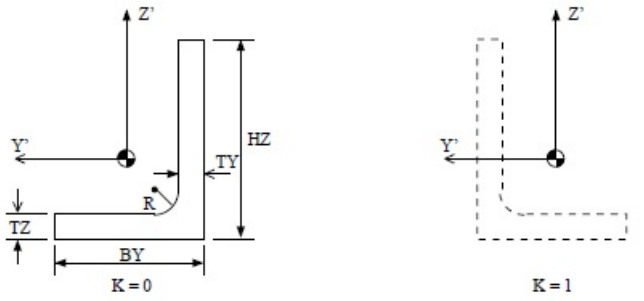


Figure 5.12: L-section with inside curvature

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.21 **GNODE**: Correspondence between External and Internal Node Numbering, and Number of Degrees of Freedom at Each Node

GNODE	NODEX	→ NODENO	NDOF	ODOF

NODEX	External node number (specified or controlled by the user). Can be any integer identifier as long as it is unique in the node numbering sequence.
NODENO	Program defined internal number for the node. The internal node numbers range from 1 and continuously - no holes in the numbering - to number of nodes in the finite element model.
NDOF	Number of degrees of freedom at the node NODENO.
ODOF	Order of the degrees of freedom at node NODENO. NDOF digits. Example NDOF = 3, ODOF = 135 means 3 degrees of freedom. Two translations x, z and and one rotation r_y, respectively in the super element co-ordinate system, unless a local nodal co-ordinate system is specified (see the **BNDOF** and **BNTRCOS** data types).

There will be one data type with the identifier **GNODE** for each node. The sequence of the data type will correspond to the internal node number, NODENO.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.22 **GPGBOX**: Cross Section Type Boxed Plate Girder

GPGBOX	NFIELD	→ GEONO	HZ	TY
	BF	TF	TYA	RF
	SFY	SFZ		

NFIELD	Number of data fields on this data type (including this field).
GEONO	Geometry type number, referenced from the data type **GELREF1**.
HZ	Height of beam at current location.
TY	Thickness of beam web.
BF	Width of beam (top and bottom flanges).
TF	Thickness of flanges (top and bottom).
TYA	Thickness of additional webs (closure plates).
RF	Radius of inside curvature at top and bottom if rolled profile. Fillet weld height if welded profile.
SFY, SFZ	Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively SHARY(MOD) = SHARY(PROG) · SFY SHARZ(MOD) = SHARZ(PROG) · SFZ (The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).

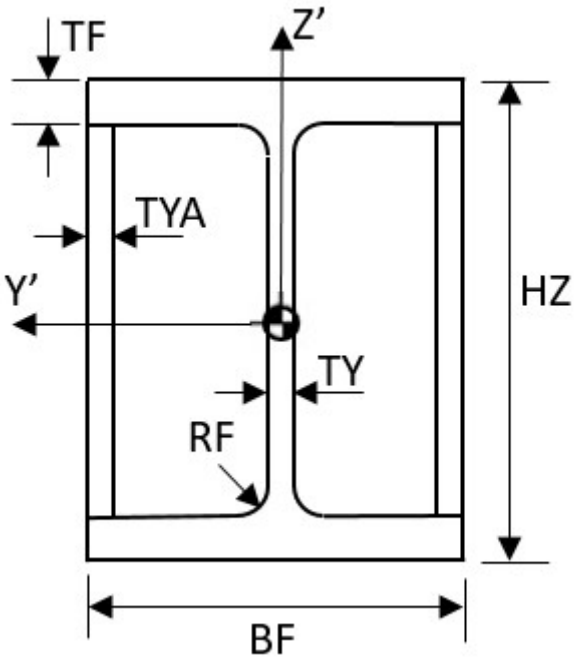


Figure 5.13: Boxed plate girder - rolled profile

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.23 **GPGDOW**: Cross Section Type Boxed Plate Girder

GPGDOW	NFIELD	→ GEONO	HZ	TY
	BF	TF	DY	RF
	RFI	SFY	SFZ	

NFIELD	Number of data fields on this data type (including this field).
GEONO	Geometry type number, referenced from the data type **GELREF1**.
HZ	Height of beam at current location.
TY	Thickness of each beam web.
BF	Width of beam (top and bottom flanges).
TF	Thickness of flanges (top and bottom).
DY	Distance centre-centre webs.
RF	Fillet weld height if welded profile. Fillet radius on the outer side of centre webs if rolled profile.
RFI	Must be set to zero if welded profile. Fillet radius in area between centre webs if rolled profile.
SFY, SFZ	Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively. SHARY(MOD) = SHARY(PROG) · SFY SHARZ(MOD) = SHARZ(PROG) · SFZ (The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).

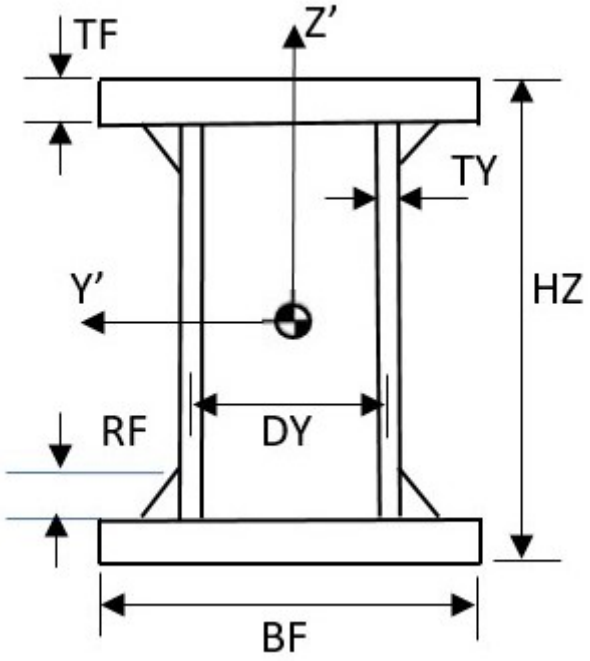


Figure 5.14: Double web plate girder - welded case

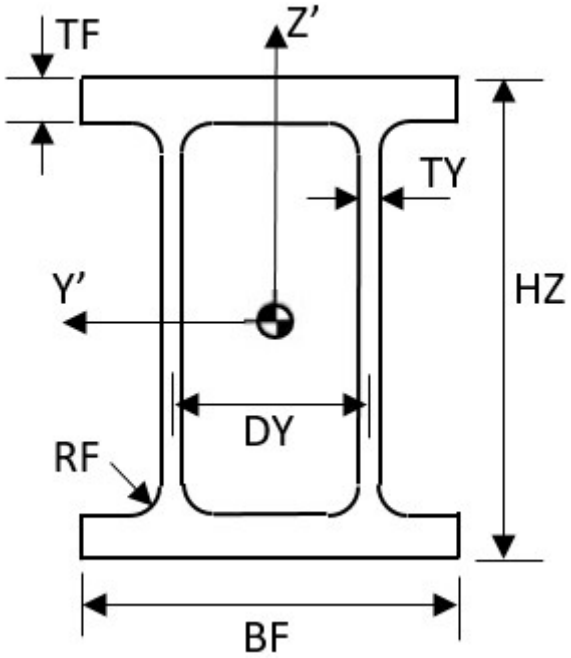


Figure 5.15: Double web plate girder - rolled case

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.24 **GPIPE**: Cross Section Type Tube

GPIPE	→ GEONO	DI	DY	T
	SFY	SFZ	NCIR	NRAD

GEONO	Geometry type number, referenced from the data type **GELREF1**.
DI	Inner diameter of tube.
DY	Outer diameter of tube (mandatory).
T	Thickness of tube (not necessary if DI is given).
SFY, SFZ	Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively SHARY(MOD) = SHARY(PROG) · SFY SHARZ(MOD) = SHARZ(PROG) · SFZ (The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).
NCIR, NRAD	Number of integration points in circumferential and radial direction, respectively (optional).

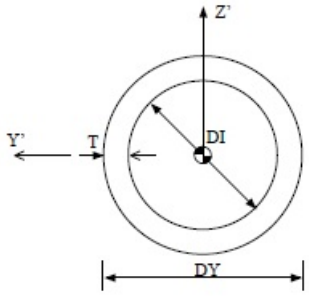


Figure 5.16: Tube

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.25 **GSEPSPEC**: Specified Separation Description

GSEPSPEC	→ SEPARID	OPTION	NDIST	DISTANCE$_0$
	DISTANCE$_1$	...	DISTANCE$_n$	

The 'specified separation' data type is a sub-type of the 'separation' data type. Specified separation type may also specify uniform separation type.

SEPARID	Identification of the separation description. A unique integer value among all the separation description data types (including all sub-types) in the super element. This number is referenced from the **SEPARID** number on the **GSLSTIFF** data type.
OPTION	Usage of DISTANCE$_0$:
	= 0 DISTANCE$_0$ is not specified. Any value of DISTANCE$_0$ is equally good (not used).
	= 1 DISTANCE$_0$ is specified.
NDIST	Number of different distances between stiffeners.
	= 1 indicates that all separations are equal (uniform separation).
DISTANCE$_0$	The distance to the first stiffener from the start of the plate. The start of the plate is the point on the plate which has the smallest y-values (usually negative), where y is in the local beam co-ordinate systems.
DISTANCE$_i$	The distance between stiffener number 'i' and stiffener number '$i+1$'. $i \in [1, \text{NDIST}]$

For a uniform separation (DISTANCE$_0$ is not specified), it will for a stiffener layer result in a smearing (multiplication) factor (F_s) in the stiffener direction for Young's modulus (E_1). This factor is relative to a plate layer with the same heigth as the stiffener layer.

The smearing (multiplication) factor F_s is given as

$$F_s = \frac{b}{d_1},$$

where

$b =$ width of each stiffener.
$d_1 =$ DISTANCE$_i$.

In the other direction, Young's modulus (E_2) will be taken equal to zero.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.26 **GSETMEMB**: Set (group) of Nodes or Elements (Members)

GSETMEMB	NFIELD	→ ISREF	→ INDEX	→ ISTYPE
	ISORIG	$MEMB_1$	$MEMB_2$	$MEMB_3$
	$MEMB_4$	. . .	$MEMB_n$	

This record together with the name and description of a set record (**TDSETNAM**) constitutes the set (group) data type.

NFIELD — Number of data fields on this record (maximum is 1024).

ISREF — Internal set identification number as defined on the name and description of a set record (**TDSETNAM**).

INDEX — Sequential record number for current set (ISREF). Each set may consist of one or more **GSETMEMB** records with same set identification number (ISREF). INDEX must be strictly increasing from 1 and upwards till number of GSETMEMB records for this set of members (nodes or elements).

ISTYPE — Set type:

= 1 set of nodes.

In this case the references $MEMB_i$ refer to program defined internal node numbers.

= 2 set of elements.

In this case the references $MEMB_i$ refer to program defined internal element numbers.

ISORIG — Set origin type:

= 0 undefined origin.

= 1 point.

= 2 line (or curve).

= 3 surface.

= 4 body.

$MEMB_1$ — First set member on this record

$MEMB_2$ — Second set member on this record

⋮

$MEMB_n$ — Set member number n on this record.

n is the number of set members on the current record. n = NFIELD - 5.

Comments:

- The set datatype consists of one name and description of set record (**TDSETNAM**) and one or more set member records (**GSETMEMB**).
- It should be noted that a set may have its set members distributed over several set member records (**GSETMEMB**) all having the same set identification number (ISREF) and consequently also the same **TDSETNAM** record. The total number of set members will then be the sum of the number of set

members (n) for each of the set records.

Restrictions:

- Only one set type (ISTYPE) for same set identification number (ISREF) is allowed.
- If several records for the same set identification number (ISREF), record numbering must be strictly sequential:
  1 < INDEX < NINDEX, where NINDEX is the number of records per set.
- A set member (number) should only be included once in the list.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.27 **GSLAYER**: General Eccentric Sandwich Element

GSLAYER	→ GEONO	NLAYER	LAYERID$_1$	LAYERID$_2$
	LAYERID$_3$	...	LAYERID$_n$	

There are never more than one **GSLAYER** record for an element. The **GSLAYER** data type is referenced from the **GELREF1** record for each element. When this layer 'stack' record is referenced from the **GELREF1** data type, there is no need to refer to any material in the **GELREF1** data type.

GEONO	Geometry type number for this general sandwich (layered) element. The geometry type number is referenced from the data type **GELREF1**.
NLAYER	Number of layers in the general eccentric sandwich (layered) element.
LAYERID$_i$	Identification of layer no. 'i'. LAYERID$_i$ refers to a **GSLPLATE** or **GSLSTIFF** . record with identification LAYERID$_i$. It is a unique integer value among all layers in the super element. $i \in [1, \text{NLAYER}]$

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.28 **GSLPLATE**: Plate Layer Description

GSLPLATE	→ LAYERID	MATNO	SHFACT	NECCNO
	ECCNO$_1$	. . .	ECCNO$_n$	NTHICKID
	THICKID$_1$	. . .	THICKID$_n$	

The plate layer is a sub-type of layer. It identifies that the layer is a plate, i.e. shell element, and references possible eccentricities and mandatory element or element node thickness(es) of the plate.

LAYERID	Identification of the layer. A unique integer value among all the layers in the super element. This number is referenced from the LAYERID numbers on the **GSLAYER** records.
MATNO	Reference to the material.
SHFACT	A factor for calculation of the shear deflection. A commonly used value for square cross sections and plates is 1.2.
NECCNO	Number of eccentricity data for this layer (=1 or number of element nodes).
ECCNO$_i$	Reference to eccentricity description for the layer (=0 if there is no eccentricity for this layer). ECCNO$_i$ refers to a **GECC** or **GECCEN** record. $i \in [1, \text{NECCNO}]$
NTHICKID	Number of thickness data for this layer (=1 or number of element nodes).
THICKID$_i$	Reference to the thickness for the plate or the plate nodes. THICKID$_i$ refers to a **GELTH** record (in this case GEONO on the **GELTH** record should read THICKID$_i$, since the **GELREF1** record does not reference the **GELTH** record directly for the general eccentric sandwich element). $i \in [1, \text{NTHICKID}]$.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.29 **GSLSTIFF**: Stiffener Layer Description

GSLSTIFF	→ LAYERID	MATNO	SHFACT	NECCNO
	ECCNO_1	...	ECCNO_n	NSECTID
	SECTID_1	...	SECTID_n	SEPARID
	ANGLE			

The stiffener layer is a sub-type of layer. It identifies that this layer is a stiffener layer, i.e. beam elements, and specifies the additional information for the beams.

LAYERID — Identification of the layer. A unique integer value among all the layers in the super element. This number is referenced from the LAYERID numbers on the **GSLAYER** records.

MATNO — Reference to the material.

SHFACT — A factor for calculation of the shear deflection. A commonly used value for square cross sections and plates is 1.2.

NECCNO — Number of eccentricity data for this layer (=1 or number of element nodes).

ECCNO_i — Reference to eccentricity description for the layer (=0 if there is no eccentricity for this layer). ECCNO_i refers to a **GECC** or **GECCEN** record. $i \in [1, \text{NECCNO}]$

NSECTID — Number of section references for this layer (=1 or number of element nodes).

SECTID_i — Reference to the section for the beam or the beam nodes. SECTID_i refers to a **GBARM** and a **GBEAMG** record (in this case GEONO on the **GBARM** and **GBEAMG** record should read SECTID_i, since the **GELREF1** data type does not reference the **GBARM** and **GBEAMG** record directly for the general eccentric sandwich element). $i \in [1, \text{NSECTID}]$.

SEPARID — Reference to the separation data description . The separation is the distance between stiffeners in the plane of the element, but orthogonal to the stiffener direction.

ANGLE — Angle with respect to the element's reference direction (local x-axis). The angle is specified in degrees. This angle and the local x-axis of the eccentric sandwich element determines the direction of the local x-axis of the stiffeners. The reference direction (local x-axis) of the eccentric sandwich element are determined by the direction cosine matrix from the **BNTRCOS** data type, referenced on the **GELREF1** record for this element. The direction determined from the first line of this direction cosine matrix is projected down on the element plane and this gives the element's reference direction (local x-axis).

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.30 **GTONP**: Cross Section T on Plate

GTONP	→ GEONO	HZ	TY	BT
	TT	BP	TP	SFY
	SFZ	NLOBYT	NLOBYB	NLOBZ

GEONO	Geometry type number, referenced from the data type **GELREF1**.
HZ	Height of beam at current location.
TY	Thickness of beam web.
BT	Width of top flange.
TT	Thickness of top flange.
BP	Effective width of plate.
TP	Thickness of plate.
SFY, SFZ	Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively SHARY(MOD) = SHARY(PROG) · SFY SHARZ(MOD) = SHARZ(PROG) · SFZ (The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).
NLOBYT	Number of integration points in top flange (optional).
NLOBYB	Number of integration points in bottom plate (optional).
NLOBZ	Number of integration points in beam web (optional).

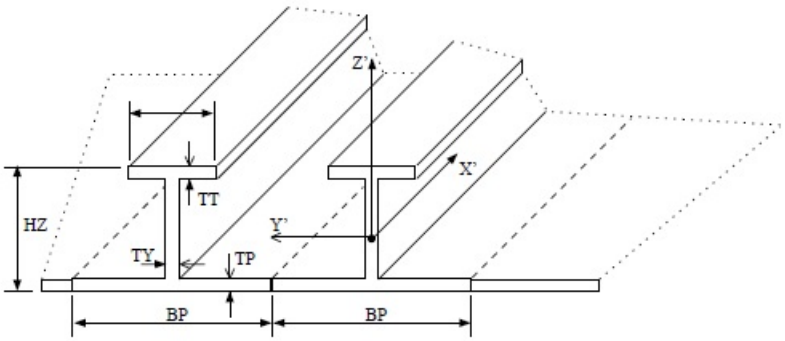


Figure 5.17: I or H beam

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.31 **GUNIVEC**: Specification of Local Element

GUNIVEC	→ TRANSNO	UNIX	UNIY	UNIZ

TRANSNO	Unit vector number, referenced from the data type **GELREF1**.
UNIX	
UNIY	Unit vector given in the super element co-ordinate system along the local z-axis (reference axis in the z-direction) of the element in the particular node.
UNIZ	

The **GUNIVEC** data type is used for beam elements only, i.e. basic element types 2, 15 and 23. Other basic element types may refer to **BNTRCOS** data types. No ambiguity thus exists if both a **GUNIVEC** and a **BNTRCOS** data type have same TRANSNO, but they should preferably have separate numbering (TRANSNO) to avoid possible program problems.

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.3.32 **GUSYI**: Cross Section Type Unsymmetrical I-Beam

GUSYI	→ GEONO	HZ	TY	BT
	B1	TT	BB	B2
	TB	SFY	SFZ	NLOBYT
	NLOBYB	NLOBZ		

GEONO	Geometry type number, referenced from the data type **GELREF1**.
HZ	Height of beam at current location.
TY	Thickness of beam web.
BT	Width of top flange.
B1	Width of half top flange in positive local y-direction.
TT	Thickness of top flange.
BB	Width of bottom flange.
B2	Width of half bottom flange in positive local y- direction
TB	Thickness of bottom flange.
SFY, SFZ	Factors modifying the shear areas calculated by the preprocessor program such that the modified shear areas are respectively SHARY(MOD) = SHARY(PROG) · SFY SHARZ(MOD) = SHARZ(PROG) · SFZ (The shear areas on GBEAMG are SHARY(MOD) and SHARZ(MOD)).
NLOBYT	Number of integration points in top flange (optional).
NLOBYB	Number of integration points in bottom flange (optional).
NLOBZ	Number of integration points in beam web (optional).

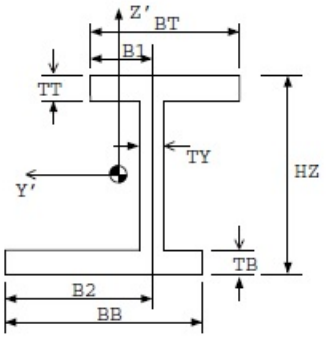


Figure 5.18: I or H beam

Back to Section 5.3 **Nodal Data and Element Geometry Definition**

5.4 Material Data

First level data

MAXDMP	Axial Damper between Two Nodal Points	see Section 5.4.1
MAXSPR	Axial Spring between Two Nodal Points	see Section 5.4.2
MCNT	Material for Non-linear Contact Element	see Section 5.4.3
MGDAMP	Damping Element to Ground	see Section 5.4.4
MGLDAMP	General 2-noded Damping Element	see Section 5.4.5
MGLMASS	General 2-noded Mass Element	see Section 5.4.6
MGMASS	1-Noded Mass element	see Section 5.4.7
MGSPRNG	Spring Element to Ground	see Section 5.4.8
MISOAL	Isotropy, Linear Acoustic Field Problem	see Section 5.4.9
MISOEML	Isotropy, Linear Electromagnetic Field Problem	see Section 5.4.10
MISOHL	Isotropy, Linear Heat Conduction Analysis	see Section 5.4.11
MISOHNL	Isotropy, Non-linear Heat Conduction Analysis	see Section 5.4.12
MISOPL	Non-linear Isotropic Material, Material Types 1-4	see Section 5.4.13
MISOPL	Non-linear Isotropic Material for Grout, Material Type 5	see Section 5.4.14
MISOPL	Non-linear Isotropic Material for De-bonding Material, Material Type 6	see Section 5.4.15
MISOSEL	Isotropy, Linear Elastic Structural Analysis	see Section 5.4.16
MISTEL	Temperature Dependent Isotropic, Linear Elastic Material	see Section 5.4.17
MORSMEL	Anisotropy, Linear Elastic Structural Analysis, 2-D Membrane Elements and 2-D Thin Shell Elements	see Section 5.4.18
MORSSEL	Anisotropy, Linear Elastic Structural Analysis, 3-D One- and Multi-layered Thick Shell Elements	see Section 5.4.19
MORSSOL	Anisotropy, Linear Elastic Structural Analysis, Solid Elements	see Section 5.4.20
MSHGLSP	General 2-noded Spring/Shim Element	see Section 5.4.21
MTEMP	Scaling Curve for Temperature Variation	see Section 5.4.22
MTENONL	Non-linear Material with Temperature Dependency	see Section 5.4.23
MTRMEL	Local Transformation of the Axes of An-isotropy, 2-D Membrane Elements and 2-D Thin Shell Elements	see Section 5.4.24
MTRSEL	Local Transformation of the Axes of An-isotropy, 3-D Multi-layered Thick Shell Element	see Section 5.4.25
MTRSOL	Local Transformation of the Axes of An-isotropy, Solid Elements	see Section 5.4.26

5.4.1 **MAXDMP**: Axial Damper between Two Nodal Points

MAXDMP	→ MATNO	DAMP		

MATNO	Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.
DAMP	Axial damping constant.

The axial damping constant corresponds to the force to be applied in order to get a unit velocity in the direction of the basic element.

Back to Section 5.4 **Material Data**

5.4.2 **MAXSPR**: Axial Spring between Two Nodal Points

MAXSPR	→ MATNO	SCON		

MATNO	Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.
SCON	Axial spring constant.

The axial spring constant corresponds to the force to be applied in order to get a unit displacement in the direction of the basic element.

Back to Section 5.4 **Material Data**

5.4.3 **MCNT**: Material for Non-linear Contact Element

MCNT	→ MATNO	MATYP	EMOD	STIFAC
	FRICOF			

MATNO — Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.

MATYP — Contact surface behaviour.

= 1 Perfect sliding.

= 2 Perfect sticking.

= 3 Linear friction.

EMOD — Typical Young's modulus of surrounding elements.

STIFAC — Factor that EMOD should be multiplied with to obtain contact stiffness.

Default: STIFAC = 10^5.

FRICOF — Linear friction coefficient when MATYP = 3.

Back to Section 5.4 **Material Data**

5.4.4 **MGDAMP**: Damping Element to Ground

MGDAMP	→ MATNO	NDOF	$C_{(1,1)}$	$C_{(2,1)}$
	...	...	$C_{(\text{NDOF},1)}$	$C_{(2,2)}$
	$C_{(3,2)}$	...	...	$C_{(\text{NDOF},2)}$
	$C_{(3,3)}$	...	...	$C_{(\text{NDOF},\text{NDOF})}$

MATNO — Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.

NDOF — Number of degrees of freedom at the node.

$C_{(i,j)}$ — Elements of the damping matrix (only elements on and below the main diagonal are stored, i.e. a symmetric damping matrix is assumed). The elements are referred to a local co-ordinate system if defined (by TRANSNO on the **GELREF1** data type), otherwise to the global co-ordinate system of the super element.

The damper to ground matrix is the viscous damping matrix.

Back to Section 5.4 **Material Data**

5.4.5 **MGLDAMP**: General 2-noded Damping Element

MGLDAMP	→ MATNO		NDOF1	NDOF2
	$D_{(1,1)}$	$D_{(2,1)}$	...	$D_{(\text{NDOF1},1)}$
	$D_{(\text{NDOF1}+1,1)}$	...	$D_{(\text{TDOF},1)}$	$D_{(2,2)}$
	$D_{(3,2)}$	...	$D_{(\text{TDOF},2)}$	$D_{(3,3)}$
	...	$D_{(\text{TDOF},\text{TDOF})}$		

MATNO	Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.
NDOF1	Number of degrees of freedom at local node 1.
NDOF2	Number of degrees of freedom at local node 2 .
$D_{(i,j)}$	Elements of the damping matrix (only elements on and below the main diagonal are stored, i.e. a symmetric damping matrix is assumed). The elements are referred to a local co-ordinate system if defined (by TRANSNO on the **GELREF1** data type), otherwise to the global co-ordinate system of the super element. (TDOF = NDOF1 + NDOF2).

The (i,j)'th element of the damping matrix D corresponds to the force to be given in the i'th degree of freedom in order to get a unit velocity in the j'th degree of freedom.

A **GELMNT1** record with element type = 13 and a **GELREF1** record with reference to this (**MGLDAMP**) record is necessary in order to fulfill the definition of the 2-noded damping element.

Back to Section 5.4 **Material Data**

5.4.6 **MGLMASS**: General 2-noded Mass Element

MGLMASS	→ MATNO		NDOF1	NDOF2
	$M_{(1,1)}$	$M_{(2,1)}$	...	$M_{(\text{NDOF1},1)}$
	$M_{(\text{NDOF1}+1,1)}$	...	$M_{(\text{TDOF},1)}$	$M_{(2,2)}$
	$M_{(3,2)}$	...	$M_{(\text{TDOF},2)}$	$M_{(3,3)}$
	...	$M_{(\text{TDOF},\text{TDOF})}$		

MATNO — Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.

NDOF1 — Number of degrees of freedom at local node 1.

NDOF2 — Number of degrees of freedom at local node 2 .

$M_{(i,j)}$ — Elements of the mass matrix (only elements on and below the main diagonal are stored, i.e. a symmetric mass matrix is assumed). The elements are referred to a local co-ordinate system if defined (by TRANSNO on the **GELREF1** data type), otherwise to the global co-ordinate system of the super element.

(TDOF = NDOF1 + NDOF2).

The (i,j)'th element of the mass matrix D corresponds to the force to be given in the i'th degree of freedom in order to get a unit accleration in the j'th degree of freedom.

A **GELMNT1** record with element type = 13 and a **GELREF1** record with reference to this (**MGLMASS**) record is necessary in order to fulfill the definition of the 2-noded mass element.

Back to Section 5.4 **Material Data**

5.4.7 **MGMASS**: 1-Noded Mass element

MGMASS	→ MATNO	NDOF	$M_{(1,1)}$	$M_{(2,1)}$
	...	...	$M_{(\text{NDOF},1)}$	$M_{(2,2)}$
	$M_{(3,2)}$	...	...	$M_{(\text{NDOF},2)}$
	$M_{(3,3)}$	...	...	$M_{(\text{NDOF},\text{NDOF})}$

MATNO — Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.

NDOF — Number of degrees of freedom at the node.

$M_{(i,j)}$ — Elements of the mass matrix (only elements on and below the main diagonal are stored, i.e. a symmetric stiffness matrix is assumed). The elements are referred to a local co-ordinate system if defined (by TRANSNO on the **GELREF1** data type), otherwise to the global co-ordinate system of the super element.

Back to Section 5.4 **Material Data**

5.4.8 **MGSPRNG**: Spring Element to Ground

MGSPRNG	→ MATNO	NDOF	$K_{(1,1)}$	$K_{(2,1)}$
	...	...	$K_{(\text{NDOF},1)}$	$K_{(2,2)}$
	$K_{(3,2)}$	...	...	$K_{(\text{NDOF},2)}$
	$K_{(3,3)}$	...	...	$K_{(\text{NDOF},\text{NDOF})}$

MATNO — Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.

NDOF — Number of degrees of freedom at the node.

$K_{(i,j)}$ — Elements of the stiffness matrix (only elements on and below the main diagonal are stored, i.e. a symmetric stiffness matrix is assumed). The elements are referred to a local co-ordinate system if defined (by TRANSNO on the **GELREF1** data type), otherwise to the global co-ordinate system of the super element.

The (i, j)'th element of the stiffness matrix corresponds to the force to be given in the i'th degree of freedom to get a unit displacement in the j'th degree of freedom.

Back to Section 5.4 **Material Data**

5.4.9 **MISOAL**: Isotropy, Linear Acoustic Field Problem

MISOAL	→ MATNO	C	CP	CV
	RHO	PRESS	TEMP	R

MATNO	Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.
C	Speed of sound in gas.
CP	Specific heat of gas at constant pressure.
CV	Specific heat of gas at constant volume.
RHO	Specific density of gas.
PRESS	Gas pressure.
TEMP	Gas temperature.
R	Universal gas constant.

Back to Section 5.4 **Material Data**

5.4.10 **MISOEML**: Isotropy, Linear Electromagnetic Field Problem

MISOEML	→ MATNO	PERM		

MATNO — Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.

PERM — Permittivity.

Back to Section 5.4 **Material Data**

5.4.11 **MISOHL**: Isotropy, Linear Heat Conduction Analysis

MISOHL	→ MATNO	RHO	CHEAT	COND

MATNO	Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.
RHO	Density.
CHEAT	Specific heat.
COND	Coefficient of heat conductivity.

Back to Section 5.4 **Material Data**

5.4.12 **MISOHNL**: Isotropy, Non-linear Heat Conduction Analysis

MISOHNL	→ MATNO	RHO	RTEMPNO	CHEAT
	CHTEMPNO	COND	COTEMPNO	

MATNO	Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.
RHO	Density.
RTEMPNO	Reference number to a temperature dependent scaling factor for RHO.
CHEAT	Specific heat.
CHTEMPNO	Reference number to a temperature dependent scaling factor for CHEAT.
COND	Coefficient of heat conductivity.
COTEMPNO	Reference number to a temperature dependent scaling factor for COND. If there is no temperature dependence, the corresponding reference number will be zero.

Back to Section 5.4 **Material Data**

5.4.13 **MISOPL**: Non-linear Isotropic Material, Material Types 1-4

MISOPL	→ MATNO	MATYP	POISS	RHO
	DAMP	ALPH	B1	Not Used
	NP	SIG_1	EPS_1	SIG_2
	EPS_2	...	SIG_{NP}	EPS_{NP}

MATNO — Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.

MATYP — Material type

= 1 Elasto-plastic material.

= 2 Non-linear hyperelastic material.

= 3 Overly technique.

= 4 Isotropic and kinematic hardening.

B1 — Parameter for combining isotropic and kinematic hardening.

= 0.0 Isotropic.

= 1.0 Kinematic.

Any value between 0.0 and 1.0 is legal.

POISS — Poisson's ratio.

RHO — Density.

DAMP — Specific damping.

ALPH — Thermal expansion coefficient.

B1 — Only used for MATYP=4, see above and next page.

NP — Number of points to represent the uniaxial stress-strain curve for increased loading.

Note: NP < 30.

SIG_1 — Stress at the first point representing the stress-strain curve.

EPS_1 — Corresponding strain at the first point representing the stress-strain curve.

(YOUNG $= SIG_1/EPS_1$).

SIG_2 — Stress at the second point representing the stress-strain curve.

EPS_2 — Corresponding strain at the second point representing the stress-strain curve.

(YOUNG $= SIG_2/EPS_2$). See also Figure 5.19

⋮

SIG_{NP} — Stress at the last point representing the stress-strain curve.

EPS_{NP} — Corresponding strain at the last point representing the stress-strain curve.

(YOUNG $= SIG_{NP}/EPS_{NP}$).

Not Used	= 0.	Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

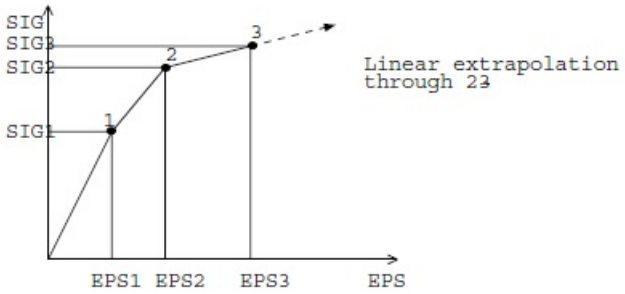


Figure 5.19: Uniaxial stress-strain curve for increased loading

The material parameter B1 may be found from a one-dimensional loading-unloading curve with yielding:

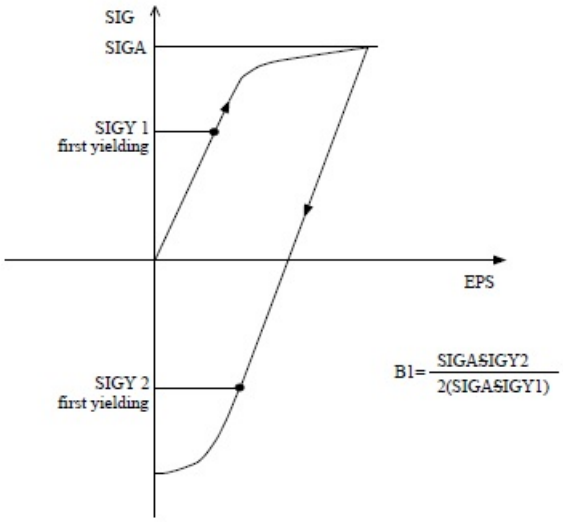


Figure 5.20: Uniaxial stress-strain curve for increased loading

Back to Section 5.4 **Material Data**

5.4.14 **MISOPL**: Non-linear Isotropic Material for Grout, Material Type 5

MISOPL	→ MATNO	MATYP	POISS	RHO
	DAMP	ALPH	Not Used	Not Used
	NP	SIG_1	EPS_1	FCM
	Not Used	Not Used	Not Used	EU
	ET			

MATNO — Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.

MATYP — Material type

= 5 Special theory for grout.

POISS — Poisson's ratio (recommended value 0.15 - 0.2).

RHO — Density.

DAMP — Specific damping.

ALPH — Thermal expansion coefficient.

NP — Number of points to represent the uniaxial stress-strain curve for increased loading.

Note: NP = 1.

SIG_1 — Stress at the first point representing the stress-strain curve.

EPS_1 — Corresponding strain at the first point representing the stress-strain curve.

The relation YOUNG $= SIG_1/EPS_1$ is only used to compute Young's modulus.

FCM — Compression stress (concrete) at which the grout becomes perfectly plastic.

EU — Compressive strain at crushing (uniaxial strain).

ET — Tensile strain at cracking (uniaxial strain).

Not Used = 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

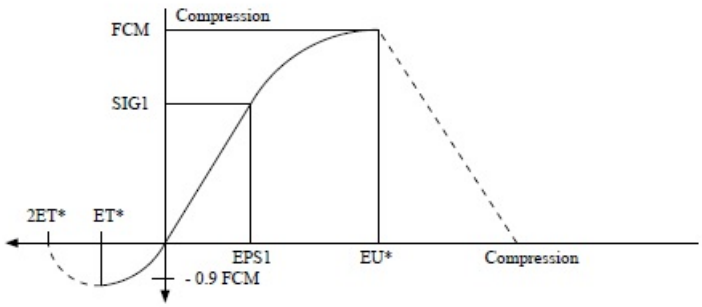


Figure 5.21: Uniaxial stress-strain curve for grout

Back to Section 5.4 **Material Data**

5.4.15 **MISOPL**: Non-linear Isotropic Material for De-bonding Material, Material Type 6

MISOPL	→ MATNO	MATYP	POISS	RHO
	DAMP	ALPH	Not Used	Not Used
	NP	SIG_1	EPS_1	SZMAX
	TAUMAX	EZREF	GREF	

MATNO	Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.
MATYP	Material type
	= 5 Special theory for grout.
POISS	Poisson's ratio (recommended value 0.15 - 0.20).
RHO	Density.
DAMP	Specific damping.
ALPH	Thermal expansion coefficient.
NP	Number of points to represent the uniaxial stress-strain curve for increased loading.
	Note: NP = 1.
SIG_1	Stress at the first point representing the stress-strain curve.
EPS_1	Corresponding strain at the first point representing the stress-strain curve.
	YOUNG $= SIG_1/EPS_1$. Recommended values: same as grout.
SZMAX	Maximum tensile separation stress capacity.
TAUMAX	Maximum shear capacity.
EZREF	Reference strain, i.e. strain where total debonding takes place.
GREF	Friction stiffness for closed gaps, (typical 0.5 times the shear modulus).
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

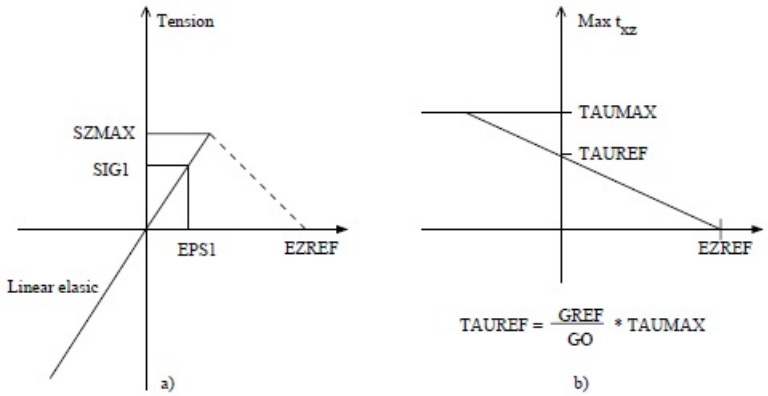


Figure 5.22: a) Uniaxial stress-strain normal to steel surface, b) Maximum shear stress as function of normal strain.

Back to Section 5.4 **Material Data**

5.4.16 **MISOSEL**: Isotropy, Linear Elastic Structural Analysis

MISOSEL	→ MATNO	YOUNG	POISS	RHO
	DAMP	ALPHA	IYIELD	YIELD

MATNO	Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.
YOUNG	Young's modulus.
POISS	Poisson's ratio.
RHO	Density.
DAMP	Specific damping.
ALPHA	Thermal expansion coefficient.
IYIELD	= 0 or blank, then Yield strength is not specified. = 1, then Yield strength is specified.
YIELD	Yield strength of material (required only when IYIELD = 1).

Back to Section 5.4 **Material Data**

5.4.17 **MISTEL**: Temperature Dependent Isotropic, Linear Elastic Material

MISTEL	→ MATNO	YOUNG	YTEMPNO	POISS
	RHO	RTEMPNO	DAMP	DTEMPNO
	ALPHA	ATEMPNO		

MATNO	Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.
YOUNG	Young's modulus.
YTEMPNO	Reference number to a temperature dependent scaling factor of YOUNG. See data type **MTEMP**.
POISS	Poisson's ratio.
RHO	Density.
RTEMPNO	Reference number to a temperature dependent scaling factor for RHO.
DAMP	Specific damping.
DTEMPNO	Reference number to a temperature dependent scaling factor for DAMP.
ALPHA	Thermal expansion coefficient.
ATEMPNO	Reference number to a temperature dependnet scaling factor for ALPHA.

Temperature reference number is zero if no temperature dependence.

Back to Section 5.4 **Material Data**

5.4.18 **MORSMEL**: Anisotropy, Linear Elastic Structural Analysis, 2-D Membrane Elements and 2-D Thin Shell Elements

MORSMEL	→ MATNO	Q_1	Q_2	Q_3
	RHO	D_{11}	D_{21}	D_{22}
	D_{31}	D_{32}	D_{33}	PS1
	PS2	DAMP1	DAMP2	ALPHA1
	ALPHA2			

MATNO	Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.
Q_1-Q_3	Global components of a vector Q indicating axes of anisotropy. The first principal axis of anisotropy is referred to the projection of Q on the membrane plane.
RHO	Density.
D_{11}-D_{33}	Elements of the lower triangular part of the general anisotropic elasticity matrix. In case of orthotropy, only D_{11}, D_{21}, D_{22} and D_{33} are nonzero.
PS1, PS2	Only given for plane strain situation. The stress normal to the membrane plane (σ_n) is calculated as follows: $\sigma_n = PS1 \cdot \sigma_1 + PS2 \cdot \sigma_2$. (For an isotropic material PS1 and PS2 equal Poisson's ratio)
DAMP1, DAMP2	Specific damping along respectively 1. and 2. principal axes of anisotropy.
ALPHA1, ALPHA2	Thermal expansion coefficients along respectively 1. and 2. principal axes of anisotropy.

Note: The vector Q must not be perpendicular to any of the elements.

Back to Section 5.4 **Material Data**

5.4.19 **MORSSEL**: Anisotropy, Linear Elastic Structural Analysis, 3-D One- and Multi-layered Thick Shell Elements

MORSSEL	→ MATNO	Q_1	Q_2	Q_3
	RHO	NLAY	THL^1	$OANG^1$
	D^1_{11}	D^1_{21}	D^1_{22}	D^1_{31}
	D^1_{32}	D^1_{33}	D^1_{41}	D^1_{42}
	D^1_{43}	D^1_{44}	D^1_{51}	D^1_{52}
	D^1_{53}	D^1_{54}	D^1_{55}	$DAMP^1_1$
	$DAMP^1_2$	$ALPHA^1_1$	$ALPHA^1_2$	THL^2
	$OANG^2$	D^2_{11}	...	...
	THL^{NLAY}	...	...	$ALPHA^{NLAY}_2$

MATNO — Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.

Q_1-Q_3 — Global components of a vector Q indicating axes of anisotropy. The first principal axis of anisotropy is referred to the projection of Q on the membrane plane.

RHO — Density.

NLAY — $= 1$ One material type through the thickness.

≥ 2 Number of layers of a multilayered (sandwich) material.

THL^1 — Thickness of first layer in percent of element thickness. If NLAY $= 1$, THL^1 is assumed equal to 100 (%).

$OANG^1$ — Angle in degrees giving rotation of the axes of anisotropy in the shell plane for material layer no. 1.

D^1_{11}-D^1_{55} — Elements of the lower triangular part of the general anisotropic elasticity matrix for material layer no. 1.

In case of orthotropy, only the diagonal terms and D^1_{21} are nonzero.

D is defined by the relation in local layer axes:

$$\sigma = D \cdot \epsilon,$$

where

$$\epsilon = [\epsilon_{xx}, \epsilon_{yy}, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}]^T.$$

$DAMP^1_1$, $DAMP^1_2$ — Specific damping along respectively 1. and 2. principal axes of anisotropy for material no. 1.

$ALPHA^1_1$, $ALPHA^1_2$ — Thermal expansion coefficients along respectively 1. and 2. principal axes of anisotropy.

THL^i – $ALPHA^i_2$ — are repeated for all $i = 1, \cdots,$ NLAY layers of the sandwich material.

THL must add up to 100 (%).

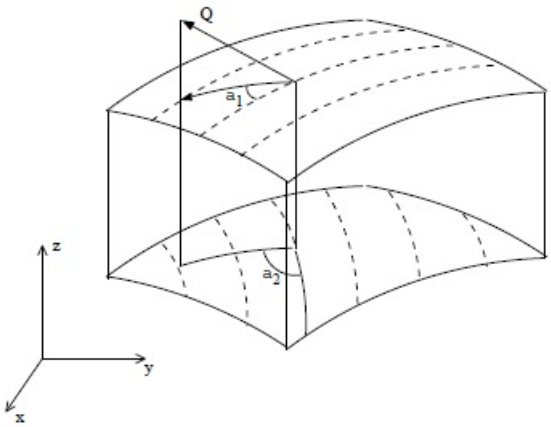


Figure 5.23: 1. principal axis of anisotropy given by the global vector Q and a rotation angle a_1 (OANG[1]) and a_2 (OANG[2]) for a two-layered material.

For each integration point Q is projected on the shell surface and the same angle a is added. This implies that Q must not be perpendicular to the element surface in any of the integration points.

Back to Section 5.4 **Material Data**

5.4.20 **MORSSOL**: Anisotropy, Linear Elastic Structural Analysis, Solid Elements

MORSSOL	→ MATNO	RHO	D^1_{11}	D^1_{21}
	D^1_{22}	D^1_{31}	D^1_{32}	D^1_{33}
	D^1_{41}	D^1_{42}	D^1_{43}	D^1_{44}
	D^1_{51}	D^1_{52}	D^1_{53}	D^1_{54}
	D^1_{55}	D^1_{61}	D^1_{62}	D^1_{63}
	D^1_{64}	D^1_{65}	D^1_{66}	$DAMP_1$
	$DAMP_2$	$DAMP_3$	$ALPHA_1$	$ALPHA_2$
	$ALPHA_3$	TRANSO		

MATNO — Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.

RHO — Density.

D_{11}-D_{66} — Elements of the lower triangular part of the general anisotropic elasticity matrix.

In case of orthotropy, only the diagonal terms and D_{21}, D_{31} D_{32} are nonzero.

D is defined by the relation in local layer axes:

$$\sigma = D \cdot \epsilon,$$

where

$$\epsilon = [\epsilon_{xx}, \epsilon_{yy}, \epsilon_{zz}, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}]^T.$$

$DAMP_1$, $DAMP_2$, $DAMP_3$ — Specific damping along respectively 1. and 2. principal axes of anisotropy for material no. 1.

$ALPHA_1$, $ALPHA_2$, $ALPHA_3$ — Thermal expansion coefficients along respectively 1. and 2. principal axes of anisotropy.

TRANSNO — Reference number to the transformation to the anisotropy axes, defined on data type **BNTRCOS**.

The transformation matrix is referred to the global co-ordinate system ($X_{\text{anis}} = TX_{\text{glob}}$) and is common to all elements of this material type. Additional local rotation is defined on a record of data type **MTRSOL** for each element.

Back to Section 5.4 **Material Data**

5.4.21 **MSHGLSP**: General 2-noded Spring/Shim Element

MSHGLSP	→ MATNO	MATKND	NDOF1	NDOF2
	$K_{(1,1)}$	$K_{(2,1)}$	...	$K_{(\text{NDOF1},1)}$
	$K_{(\text{NDOF1}+1,1)}$	...	$K_{(\text{TDOF},1)}$	$K_{(2,2)}$
	$K_{(3,2)}$	...	$K_{(\text{TDOF},2)}$	$K_{(3,3)}$
	...	$K_{(\text{TDOF},\text{TDOF})}$		

MATNO	Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.
MATKND	Material kind: = 1 Shim-element. = 2 General spring.
NDOF1	Number of degrees of freedom at local node 1.
NDOF2	Number of degrees of freedom at local node 2 .
$K_{(i,j)}$	Elements of the stiffness matrix (only elements on and below the main diagonal are stored, i.e. a symmetric stiffness matrix is assumed). The elements are referred to a local co-ordinate system if defined (by TRANSNO on the **GELREF1** data type), otherwise to the global co-ordinate system of the super element. (TDOF = NDOF1 + NDOF2).

The (i,j)'th element of the stiffness matrix K corresponds to the force to be given in the i'th degree of freedom in order to get a unit displacement in the j'th degree of freedom.

A **GELMNT1** record with element type = 40 and a **GELREF1** record with reference to this (**MSHGLSP**) record is necessary in order to fulfill the definition of the 2-noded shim- or spring element.

Back to Section 5.4 **Material Data**

5.4.22 **MTEMP**: Scaling Curve for Temperature Variation

MTEMP	→ TEMPNO	NPOINT	$SCAL_1$	$TEMP_1$
	...	...	$SCAL_{NPOINT}$	$TEMP_{NPOINT}$

TEMPNO	Temperature reference number of this curve.
NPOINT	Number of points on this curve.
$SCAL_i$	Scaling factor for point 'i' on the curve.
$TEMP_i$	Temperature at point 'i' on the curve.

Back to Section 5.4 **Material Data**

5.4.23 **MTENONL**: Non-linear Material with Temperature Dependency

MTENONL	→ MATNO	MATYP	POISS	PTEMPNO
	RHO	RTEMPNO	DAMP	DTEMPNO
	ALPHA	ATEMPNO	B1	Not Used
	MTEMPNO	YOUNG	YTEMPNO	NPOINT
	SIG_1	EPS_1	SIG_2	EPS_2
	. . .	. . .	SIG_{NPOINT}	EPS_{NPOINT}

MATNO — Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.

MATYP — Material type

= 1 Elasto-plastic material.

= 2 Non-linear hyperelastic material.

= 3 Overly technique.

= 4 Isotropic and kinematic hardening.

B1 Parameter for combining isotropic and kinematic hardening.

= 0.0 Isotropic.

= 1.0 Kinematic.

Any value between 0.0 and 1.0 is legal.

= 5 Special theory for concrete.

POISS — Reference Poisson's ratio.

PTEMPNO — Reference number to a temperature dependent scaling factor of POISS.

RHO — Reference Density.

RTEMPNO — Reference number to a temperature dependent scaling factor of RHO.

DAMP — Specific damping.

DTEMPNO — Reference number to a temperature dependent scaling factor of DAMP.

ALPHA — Thermal expansion coefficient.

ATEMPNO — Reference number to a temperature dependent scaling factor of ALPHA.

B1 — Only used for MATYP=4, see above.

MTEMPNO — Reference number to a temperature dependent scaling factor to the uniaxial stress-strain curve.

YOUNG — Number of points to represent the uniaxial stress-strain curve for increased loading.

YTEMPNO — Reference number to a temperature dependent scaling factor of YOUNG.

NPOINT — Number of points to represent the uniaxial stress-strain curve for increased loading.

SIG_1 — Stress at the first point representing the stress-strain curve.

EPS_1	Corresponding strain at the first point representing the stress-strain curve. (YOUNG $= SIG_1/EPS_1$).
SIG_2	Stress at the second point representing the stress-strain curve.
EPS_2	Corresponding strain at the second point representing the stress-strain curve. (YOUNG $= SIG_1/EPS_1$).
⋮	
SIG_{NPOINT}	Stress at the last point representing the stress-strain curve.
EPS_{NPOINT}	Corresponding strain at the last point representing the stress-strain curve. (YOUNG $= SIG_{NPOINT}/EPS_{NPOINT}$).
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

Note:
*X*TEMPNO (*X* = P, R, D, A or M) is given on record **MTEMP**.
*X*TEMPNO = 0 means no temperature dependency of the actual parameter.

Back to Section 5.4 **Material Data**

5.4.24 **MTRMEL**: Local Transformation of the Axes of An-isotropy, 2-D Membrane Elements and 2-D Thin Shell Elements

MTRMEL	→ ELNO	OANG		

ELNO — Program defined internal number for the element.

OANG — Angle in degrees giving local rotation of the axes of anisotropy in the element plane (see Figure 5.24). OANG is referred to the projection of Q defined on the material record of data type **MORSMEL**.

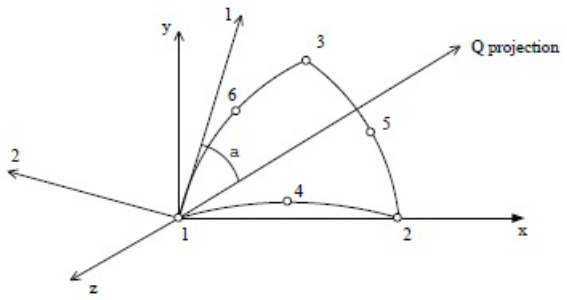


Figure 5.24: Axes of anisotropy (1, 2) given by Q (see material record of data type **MORSMEL**) and the angle a (OANG for a six-noded membrane in local co-ordinate system (x,y,z).

Back to Section 5.4 **Material Data**

5.4.25 **MTRSEL**: Local Transformation of the Axes of An-isotropy, 3-D Multi-layered Thick Shell Element

MTRSEL	→ ELNO	QROT	NLAY	$OANG_1$
	...	$OANG_{NLAY}$		

ELNO	Program defined internal number for the element.
QROT	Additional rotation in degrees of all material layers of this element.
NLAY	Number of layers of sandwich material.
	If NLAY = 0, then $OANG_i$ for $i \in [1, \ldots, NLAY]$ are assumed equal to zero.
	If NLAY > 0, then $OANG_i$ for $i \in [1, \ldots, NLAY]$ are explicitly given.
$OANG_i$	Additional rotation in the ("drilling") degree of material layer no. i.

This record is only read if **MORSSEL** records are given. If **MTRSEL** is not found for the actual element, no local rotation is assumed for this element.

Back to Section 5.4 **Material Data**

5.4.26 **MTRSOL**: Local Transformation of the Axes of An-isotropy, Solid Elements

MTRSOL	→ ELNO	TRANS		

ELNO	Program defined internal number for the element.
TRANS	Reference number to the transformation of the axes of anisotropy defined on data type **BNTRCOS**. This transformation is added to the transformation defined on material record **MORSSOL**, i.e. $X_{\text{TRANS}} = T \cdot X_{\text{TRANSNO}}$.

This record is only read if **MORSSOL** records are given. If **MTRSOL** is not found for the actual element, no local transformation is assumed for this element.

Back to Section 5.4 **Material Data**

6 HIGHER LEVEL DATA

6.1 Additional Sub Element Data

Higher level data

ADDATA	Additional User defined Sub Element Data	see Section 6.1.1
AMATRIX	Matrix control Data for Stiffness, Mass, Damping, Load and Resulting Displacement Matrix / Vector	see Section 6.1.2
AMDACCL	Vector Data for Matrix Element Acceleration Vector	see Section 6.1.3
AMDDAMP	Matrix Data for Matrix Element Damping Matrix	see Section 6.1.4
AMDDISP	Vector Data for Matrix Element Displacement Vector	see Section 6.1.5
AMDFREQ	Frequency Definition for **AMATRIX** data types	see Section 6.1.6
AMDLOAD	Vector Data for Matrix Element Load Vector	see Section 6.1.7
AMDMASS	Matrix Data for Matrix Element Mass Matrix	see Section 6.1.8
AMDSTIFF	Matrix Data for Matrix Element Stiffness Matrix	see Section 6.1.9
AMDVELO	Vector Data for Matrix Element Velocity Vector	see Section 6.1.10

6.1.1 **ADDATA**: Additional User defined Sub Element Data

ADDATA	→ ADDNO	NPAR	$PAR_{(1)}$	$PAR_{(2)}$
	...	...	$PAR_{(NPAR)}$	

ADDNO	Additional data type number, i.e. reference number referring to additional data specifications.
NPAR	Number of parameters specified by the user.
$PAR_{(1)}$	
$PAR_{(2)}$	Values for the different terms of the matrix input. The sequence is according to the convention of the analysis program. Relevant only if UNIT=0.
⋮	
$PAR_{(NPAR)}$	

Whenever the analysis program requires data that are particular to a sub element of the super element type in question (and which are not defined elsewhere) this data type may be employed to assign the data.

The data assigned above are data intended for a lower level sub element. In the data type labelled **GELMNT2** of the particular sub element an ADDNO must be included which corresponds with the ADDNO on the **ADDATA** data type.

The definition of the various parameters is depending on the analysis program to be used and must be revised when switching to another analysis program.

Back to Section 6.1 **Additional Sub Element Data**

6.1.2 **AMATRIX**: Matrix control Data for Stiffness, Mass, Damping, Load and Resulting Displacement Matrix / Vector

AMATRIX	NFIELD	→ MATNO	Not Used	NNOD
	NSUB	NODGEN	Not Used	Not Used
	MATRTYP	MATRREF	MATRFORM	IFREQ
	MCOMPL	Not Used	Not Used	Not Used

	. . .	. . .	. . .	. . .
	. . .	. . .	. . .	. . .

	MATRTYP	MATRREF	MATRFORM	IFREQ
	MCOMPL			

NFIELD — Number of data fields on this data type (including this field and embedded not used fields).

MATNO — Reference number ("material number") for this **AMATRIX** data type.

NNOD — Number of "normal" nodes on this element, not including possible generalised degrees of freedom (see description of NODGEN below) from e.g. component mode synthesis dynamic analysis. NNOD must correspond to specification on the **GELMNT1** data type.

NSUB — No. of data fields in each sub-record (= 8 in present version of the **AMATRIX** data type).

NODGEN — Number of nodes with generalised degrees of freedom. These extra "nodes" are counted after the "normal" nodes.

MATRTYP — Matrix / vector type indicator:

= 1 Stiffness matrix

= 2 Damping matrix

= 3 Mass matrix

= 4 Load vector

= 5 Resulting Displacement vector

= 6 Resulting Velocity vector

= 7 Resulting Acceleration vector

MATRREF — Matrix reference no. for the physical matrix. This no. is pointing to the reference MATRREF on the corresponding **AMDSTIFF**, **AMDDAMP**, **AMDMASS**, **AMDLOAD**, **AMDDISP**, **AMDVELO**, **AMDACCL** and / or **AMDFREQ** data types. MATRREF is zero (0) if MATRFORM below is equal to -1.

MATRFORM — = -1 Element result vectors (displacement, velocity or acceleration) are stored in Result File Format as described in "SIF, Results Interface File, File Description", ref [**?**].

= 0 Element vectors (load, displacement, velocity or acceleration) are stored.

	= 1 Element matrix is symmetric and only upper triangle is stored. For sub-matrices on the diagonal, all terms are stored and the diagonal sub-matrices must be symmetric. = 2 Element matrix is diagonal and only the diagonal nodal matrices are stored. All terms within the nodal sub-matrices are stored, also terms being zero. = 3 Element matrix is non-symmetric and the full matrix is stored. = 4 Element matrix is a null matrix, is uniquely defined and no elements need be stored. Hence: No storing of nodal matrices! = 5 Element matrix is a unit matrix, is uniquely defined and no elements need be stored. Hence: No storing of nodal matrices!
IFREQ	Additional attribute reference number for the matrix of type MATRTYP (e.g. frequency no. for which a stiffness, mass or damping matrix is valid). This means that more than one matrix of same type may be stored for the same element. It is also possible that only one of the matrix types is e.g frequency dependent, while the other types are e.g. frequency independent (only one stiffness, mass and / or damping matrix stored). The correspondence between the frequency number and the frequency is stored on an **AMDFREQ** data type.
MCOMPL	Indicator of matrix being real or complex. = 0 Real values in matrix = 1 Complex values in matrix
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

The matrices are stored as sparse block data, which means that nodal matrices and / or vectors with all terms being zero are not stored.

The flag for load / displacement / velocity / acceleration vectors being real or complex are on each nodal vector data type **AMDLOAD**, **AMDDISP** and so on, since there may be a mixture of real and complex vectors.

A matrix element may only have one **AMATRIX** data type.

Back to Section 6.1 **Additional Sub Element Data**

6.1.3 **AMDACCL**: Vector Data for Matrix Element Acceleration Vector

AMDACCL	NFIELD	→ MATRREF	→ MNODI	→ LLC
	COMPLEX	NDOF	$RACCL_{(1)}$	$RACCL_{(2)}$
	...	...	...	$RACCL_{(NDOF)}$
	$IACCL_{(1)}$	$IACCL_{(2)}$	...	...
	...	$IACCL_{(NDOF)}$		

This data type contains acceleration terms for a nodal sub vector of an element (resulting) acceleration vector. It may be a "reduced" acceleration vector of a super element or an element acceleration vector of a basic element. Each record contains the acceleration terms of one node for one load case.

NFIELD	Number of data fields on this data type (including this field and embedded not used fields)..
MATRREF	Reference number for this acceleration vector data type.
MNODI	Local matrix element node number.
LLC	Local acceleration vector number (positive integer number).
COMPLEX	Phase shift definition.
	= 0 no phase shift.
	= 1 phase shift.
NDOF	Number of degrees of freedom at the node MNODI.
$RACCL_{(1)}$	The real part of the acceleration vector with respect to the first degree of freedom at MNODI.
$RACCL_{(2)}$	The real part of the acceleration vector with respect to the second degree of freedom at MNODI.
⋮	
$RACCL_{(NDOF)}$	The real part of the acceleration vector with respect to the last degree of freedom at MNODI.
$IACCL_{(1)}$	The imaginary part of the acceleration vector with respect to the first degree of freedom at MNODI.
$IACCL_{(2)}$	The imaginary part of the acceleration vector with respect to the second degree of freedom at MNODI.
⋮	
$IACCL_{(NDOF)}$	The imaginary part of the acceleration vector with respect to the last degree of freedom at MNODI.

The matrices / vectors are stored as sparse block data, which means that nodal matrices and / or vectors with all terms being zero are not stored.

RACCL and IACCL refer to the transformed co-ordinate system if the node MNODI is transformed, else to the global co-ordinate system of the super element.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $RACCL_{(NDOF)}$ – and the first imaginary number – $IACCL_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions $IACCL_{(1)}$, $IACCL_{(2)}$, etc. are left out.

Back to Section 6.1 **Additional Sub Element Data**

6.1.4 **AMDDAMP**: Matrix Data for Matrix Element Damping Matrix

AMDDAMP	NFIELD	→ MATRREF	→ MNODI	→ MNODJ
	CODDOF	$C_{(1,1)}$	$C_{(2,1)}$	$C_{(3,1)}$
	$C_{(4,1)}$	...	$C_{(IDOF,1)}$	$C_{(1,2)}$
	$C_{(2,2)}$	...	$C_{(IDOF,2)}$	...
	$C_{(1,JDOF)}$	...	$C_{(IDOF,JDOF)}$	

This data type contains damping terms for a nodal sub matrix of an element damping matrix. It may be a "reduced" damping matrix of a super element or an element damping matrix of a basic element. Each record contains the damping terms connecting one node with another, or with itself. For sub matrices on the diagonal, all terms are always stored. Each **AMDDAMP** data type is indexed by the nodes which is connected by the damping terms of the sub matrix.

NFIELD	Number of data fields on this data type (including this field and embedded not used fields)..
MATRREF	Reference number for this damping matrix data type.
MNODI	Local matrix element node number.
MNODJ	Local matrix element node number.
CODDOF	Coded form of sub matrix dimension:
	CODDOF = IDOF*1000 + JDOF. The inverse relation will then be:
	IDOF = integer part of (CODDOF/1000)
	JDOF = remaindering of (CODDOF/1000)
	IDOF = Number of degrees of freedom at the node MNODI.
	JDOF = Number of degrees of freedom at the node MNODJ.
$C_{(i,j)}$	The (i,j)'th element of the damping matrix C.

If this is a complex damping matrix, the imaginary terms are stored after all the real terms.

The matrices are stored as sparse block data, which means that nodal matrices and / or vectors with all terms being zero are not stored.

Back to Section 6.1 **Additional Sub Element Data**

6.1.5 **AMDDISP**: Vector Data for Matrix Element Displacement Vector

AMDDISP	NFIELD	→ MATRREF	→ MNODI	→ LLC
	COMPLEX	NDOF	$RDISP_{(1)}$	$RDISP_{(2)}$
	...	...	...	$RDISP_{(NDOF)}$
	$IDISP_{(1)}$	$IDISP_{(2)}$	...	...
	...	$IDISP_{(NDOF)}$		

This data type contains displacement terms for a nodal sub vector of an element (resulting) displacement vector. It may be a "reduced" displacement vector of a super element or an element displacement vector of a basic element. Each **AMDDISP** data type contains the displacement terms of one node for one load case.

NFIELD	Number of data fields on this data type (including this field and embedded not used fields)..
MATRREF	Reference number for this displacement vector data type.
MNODI	Local matrix element node number.
LLC	Local displacement vector number (positive integer number).
COMPLEX	Phase shift definition.
	= 0 no phase shift.
	= 1 phase shift.
NDOF	Number of degrees of freedom at the node MNODI.
$RDISP_{(1)}$	The real part of the displacement vector with respect to the first degree of freedom at MNODI.
$RDISP_{(2)}$	The real part of the displacement vector with respect to the second degree of freedom at MNODI.
⋮	
$RDISP_{(NDOF)}$	The real part of the displacement vector with respect to the last degree of freedom at MNODI.
$IDISP_{(1)}$	The imaginary part of the displacement vector with respect to the first degree of freedom at MNODI.
$IDISP_{(2)}$	The imaginary part of the displacement vector with respect to the second degree of freedom at MNODI.
⋮	
$IDISP_{(NDOF)}$	The imaginary part of the displacement vector with respect to the last degree of freedom at MNODI.

The matrices / vectors are stored as sparse block data, which means that nodal matrices and / or vectors with all terms being zero are not stored.

RDISP and IDISP refer to the transformed co-ordinate system if the node MNODI is transformed, else to the global co-ordinate system of the super element.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $\text{RDISP}_{(\text{NDOF})}$ – and the first imaginary number – $\text{IDISP}_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions $\text{IDISP}_{(1)}$, $\text{IDISP}_{(2)}$, etc. are left out.

Back to Section 6.1 **Additional Sub Element Data**

6.1.6 **AMDFREQ**: Frequency Definition for **AMATRIX** data types

AMDFREQ	NFIELD	→ MATRREF	NFREQ	Not Used
	$\text{IFREQ}_{(1)}$	$\text{FREQ}_{(1)}$	$\text{IFREQ}_{(2)}$	$\text{FREQ}_{(2)}$
	...	...	$\text{IFREQ}_{(\text{NFREQ})}$	$\text{FREQ}_{(\text{NFREQ})}$

This data type contains the frequencies referred to by the IFREQ field on the **AMATRIX** data type with the same MATRREF as this data type.

NFIELD	Number of data fields on this data type (including this field and embedded not used fields)..
MATRREF	Reference number for this frequency definition data type.
NFREQ	Number of frequencies defined on this **AMDFREQ** data type.
$\text{IFREQ}_{(1)}$	The first frequency reference number.
$\text{FREQ}_{(1)}$	The frequency referred to by $\text{IFREQ}_{(1)}$ on the **AMATRIX** data type.
	The dimension of the frequency is Herz $(1/second)$, or in other words oscillations per second.
$\text{IFREQ}_{(2)}$	The second frequency reference number.
$\text{FREQ}_{(2)}$	The frequency referred to by $\text{IFREQ}_{(2)}$ on the **AMATRIX** data type.
⋮	
$\text{IFREQ}_{(\text{NFREQ})}$	The last frequency reference number.
$\text{FREQ}_{(\text{NFREQ})}$	The frequency referred to by $\text{IFREQ}_{(\text{NFREQ})}$ on the **AMATRIX** data type.
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

All the IFREQ numbers referred to on the **AMATRIX** records must be defined on the **AMDFREQ** data type.

Back to Section 6.1 **Additional Sub Element Data**

6.1.7 **AMDLOAD**: Vector Data for Matrix Element Load Vector

AMDLOAD	NFIELD	→ MATRREF	→ MNODI	→ LLC
	COMPLEX	NDOF	$\text{RLOAD}_{(1)}$	$\text{RLOAD}_{(2)}$
	...	...	...	$\text{RLOAD}_{(\text{NDOF})}$
	$\text{ILOAD}_{(1)}$	$\text{ILOAD}_{(2)}$	...	...
	...	$\text{ILOAD}_{(\text{NDOF})}$		

This data type contains load terms for a nodal sub vector of an element load vector. It may be a "reduced" load vector of a super element or an element displacement vector of a basic element. Each **AMDLOAD** data type contains the load terms of one node for one load case.

NFIELD	Number of data fields on this data type (including this field and embedded not used fields)..
MATRREF	Reference number for this load vector data type.
MNODI	Local matrix element node number.
LLC	Local load vector number (positive integer number).
COMPLEX	Phase shift definition.
	= 0 no phase shift.
	= 1 phase shift.
NDOF	Number of degrees of freedom at the node MNODI.
$\text{RLOAD}_{(1)}$	The real part of the load vector with respect to the first degree of freedom at MNODI.
$\text{RLOAD}_{(2)}$	The real part of the load vector with respect to the second degree of freedom at MNODI.
⋮	
$\text{RLOAD}_{(\text{NDOF})}$	The real part of the load vector with respect to the last degree of freedom at MNODI.
$\text{ILOAD}_{(1)}$	The imaginary part of the load vector with respect to the first degree of freedom at MNODI.
$\text{ILOAD}_{(2)}$	The imaginary part of the load vector with respect to the second degree of freedom at MNODI.
⋮	
$\text{ILOAD}_{(\text{NDOF})}$	The imaginary part of the load vector with respect to the last degree of freedom at MNODI.

The matrices / vectors are stored as sparse block data, which means that nodal matrices and / or vectors with all terms being zero are not stored.

RLOAD and ILOAD refer to the transformed co-ordinate system if the node MNODI is transformed, else to the global co-ordinate system of the super element.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $RLOAD_{(NDOF)}$ – and the first imaginary number – $ILOAD_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions $ILOAD_{(1)}$, $ILOAD_{(2)}$, etc. are left out.

Back to Section 6.1 **Additional Sub Element Data**

6.1.8 **AMDMASS**: Matrix Data for Matrix Element Mass Matrix

AMDMASS	NFIELD	→ MATRREF	→ MNODI	→ MNODJ
	CODDOF	$M_{(1,1)}$	$M_{(2,1)}$	$M_{(3,1)}$
	$M_{(4,1)}$	...	$M_{(IDOF,1)}$	$M_{(1,2)}$
	$M_{(2,2)}$	...	$M_{(IDOF,2)}$	...
	$M_{(1,JDOF)}$	...	$M_{(IDOF,JDOF)}$	

This data type contains mass terms for a nodal sub matrix of an element mass matrix. It may be a "reduced" mass matrix of a super element or an element mass matrix of a basic element. Each data type contains the mass terms connecting one node with another, or with itself. For sub matrices on the diagonal, all terms are always stored. Each data type of this type is indexed by the nodes which is connected by the mass terms of the sub matrix.

NFIELD	Number of data fields on this data type (including this field and embedded not used fields)..
MATRREF	Reference number for this mass matrix data type.
MNODI	Local matrix element node number.
MNODJ	Local matrix element node number.
CODDOF	Coded form of sub matrix dimension:
	CODDOF = IDOF*1000 + JDOF. The inverse relation will then be:
	IDOF = integer part of (CODDOF/1000)
	JDOF = remaindering of (CODDOF/1000)
	IDOF = Number of degrees of freedom at the node MNODI.
	JDOF = Number of degrees of freedom at the node MNODJ.
$M_{(i,j)}$	The (i,j)'th element of the mass matrix M.

If this is a complex mass matrix, the imaginary terms are stored after all the real terms.

The matrices are stored as sparse block data, which means that nodal matrices and / or vectors with all terms being zero are not stored.

Back to Section 6.1 **Additional Sub Element Data**

6.1.9 **AMDSTIFF**: Matrix Data for Matrix Element Stiffness Matrix

AMDSTIFF	NFIELD	→ MATRREF	→ MNODI	→ MNODJ
	CODDOF	$K_{(1,1)}$	$K_{(2,1)}$	$K_{(3,1)}$
	$K_{(4,1)}$	...	$K_{(IDOF,1)}$	$K_{(1,2)}$
	$K_{(2,2)}$	...	$K_{(IDOF,2)}$	...
	$K_{(1,JDOF)}$	...	$K_{(IDOF,JDOF)}$	

This data type contains stiffness terms for a nodal sub matrix of an element stiffness matrix. It may be a "reduced" stiffness matrix of a super element or an element stiffness matrix of a basic element. Each data type contains the stiffness terms connecting one node with another, or with itself. For sub matrices on the diagonal, all terms are always stored. Each data type of this type is indexed by the nodes which is connected by the stiffness terms of the sub matrix.

NFIELD	Number of data fields on this data type (including this field and embedded not used fields)..
MATRREF	Reference number for this stiffness matrix data type.
MNODI	Local matrix element node number.
MNODJ	Local matrix element node number.
CODDOF	Coded form of sub matrix dimension:
	CODDOF = IDOF*1000 + JDOF. The inverse relation will then be:
	IDOF = integer part of (CODDOF/1000)
	JDOF = remaindering of (CODDOF/1000)
	IDOF = Number of degrees of freedom at the node MNODI.
	JDOF = Number of degrees of freedom at the node MNODJ.
$K_{(i,j)}$	The (i,j)'th element of the stiffness matrix K.

If this is a complex stiffness matrix, the imaginary terms are stored after all the real terms.

The matrices are stored as sparse block data, which means that nodal matrices and / or vectors with all terms being zero are not stored.

Back to Section 6.1 **Additional Sub Element Data**

6.1.10 **AMDVELO**: Vector Data for Matrix Element Velocity Vector

AMDVELO	NFIELD	→ MATRREF	→ MNODI	→ LLC
	COMPLEX	NDOF	RVELO$_{(1)}$	RVELO$_{(2)}$
	...	...	...	RVELO$_{(NDOF)}$
	IVELO$_{(1)}$	IVELO$_{(2)}$	...	...
	...	IVELO$_{(NDOF)}$		

This data type contains velocity terms for a nodal sub vector of an element (resulting) velocity vector. It may be a "reduced" velocity vector of a super element or an element velocity vector of a basic element. Each data type contains the velocity terms of one node for one load case.

NFIELD	Number of data fields on this data type (including this field and embedded not used fields)..
MATRREF	Reference number for this velocity vector data type.
MNODI	Local matrix element node number.
LLC	Local velocity vector number (positive integer number).
COMPLEX	Phase shift definition. = 0 no phase shift. = 1 phase shift.
NDOF	Number of degrees of freedom at the node MNODI.
RVELO$_{(1)}$	The real part of the velocity vector with respect to the first degree of freedom at MNODI.
RVELO$_{(2)}$	The real part of the velocity vector with respect to the second degree of freedom at MNODI.
⋮	
RVELO$_{(NDOF)}$	The real part of the velocity vector with respect to the last degree of freedom at MNODI.
IVELO$_{(1)}$	The imaginary part of the velocity vector with respect to the first degree of freedom at MNODI.
IVELO$_{(2)}$	The imaginary part of the velocity vector with respect to the second degree of freedom at MNODI.
⋮	
IVELO$_{(NDOF)}$	The imaginary part of the velocity vector with respect to the last degree of freedom at MNODI.

The matrices / vectors are stored as sparse block data, which means that nodal matrices and / or vectors with all terms being zero are not stored.

RVELO and IVELO refer to the transformed co-ordinate system if the node MNODI is transformed, else to the global co-ordinate system of the super element.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $\text{RVELO}_{(\text{NDOF})}$ – and the first imaginary number – $\text{IVELO}_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions $\text{IVELO}_{(1)}$, $\text{IVELO}_{(2)}$, etc. are left out.

Back to Section 6.1 **Additional Sub Element Data**

6.2 Boundary Conditions, Loads and Point Masses

Higher level data

BLDEP	Node with Linear Dependency	see Section 6.2.1
BNBCD	Node with Boundary Condition	see Section 6.2.2
BNDISPL	Node with Displacement, Velocity and/or Acceleration	see Section 6.2.3
BNDOF	Node with Transformation	see Section 6.2.4
BNINCO	Node with Initial Condition If Arbitrary Time Dependent Loading	see Section 6.2.5
BNLOAD	Node with Load	see Section 6.2.6
BNMASS	Node with Point Mass	see Section 6.2.7
BNTRCOS	Transformation from Global to Local Co-ordinate System, Direction Cosines	see Section 6.2.8
BQDP	Node with Simple Quadratic Dependence	see Section 6.2.9
BSELL	Sub Element Load Description	see Section 6.2.10

6.2.1 **BLDEP**: Node with Linear Dependency

BLDEP	→ SLAVE	MASTER	NDDOF	NDEP
	s_1	m_1	β_1	Not Used
	s_2	m_2	β_2	Not Used
	...	...	...	...
	s_{NDEP}	m_{NDEP}	β_{NDEP}	Not Used

SLAVE	Program defined internal number for the node which is dependent.
MASTER	Program defined internal number for the node which is independent.
NDDOF	Number of dependent degrees of freedom at node SLAVE. When not specified, NDDOF is equal to NDEP.
NDEP	Number of triplets with s_i, m_i and β_i.
s_i	Dependent node's degree of freedom ("slave" degree of freedom).
m_i	Independent node's degree of freedom ("master" degree of freedom).
β_i	The contribution of the degree of freedom m_i of the master (independent) node to degree of freedom s_i on the slave (dependent) node.
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

Each line specifies one dependent degree of freedom which is dependent on the independent node's specified degree of freedom with the factor β_i. The degrees of freedom must also be specified on **BNBCD** data types as linear dependent, i.e. $\text{FIX}_{(i)}$ = 3 for the dependent node; and as retained, i.e. $\text{FIX}_{(i)}$ = 4 for the independent node if the current analysis is a super element analysis where only the Sestra super-matrix solver is used.

If linear dependencies are restricted to the first-level super elements and the Sestra multi-frontal solver is used, then the independent node does not need to be set as retained. It can, however, be a super node but is not required to be a super node only to resolve linear dependency relations. This does also apply for direct analyses — i.e. only one super element— when the Sestra multi-front solver is used. An implication is that the Sestra super-matrix solver cannot be used when linear dependencies are specified unless there are at least two super elements involved.

A node may be dependent on many nodes. For each combination of SLAVE and MASTER a new data type, starting with the identifier **BLDEP**, is given.

The **same** combination of SLAVE and MASTER may occur only once.

When node transformations have been specified for any of the nodes implicated in the linear dependence, the degrees of freedom refer to the transformed local co-ordinate system.

Multipoint constraints (2nd and higher order dependence) may be specified through more **BLDEP** data types with the same linear dependent node and different independent nodes. The factors β_i may be found as Lagrange multipliers or coefficients (Lagrange interpolation polynomial). For 2nd-order dependence this may as well be specified on one **BQDP** data type.

Back to Section 6.2 **Boundary Conditions, Loads and Point Masses**

6.2.2 **BNBCD**: Node with Boundary Condition

BNBCD	→ NODENO	NDOF	$FIX_{(1)}$	$FIX_{(2)}$
	...	...	...	$FIX_{(NDOF)}$

NODENO — Program defined internal number for the node with specified boundary condition.

NDOF — Number of degrees of freedom at the node NODENO.

$FIX_{(1)}$

$FIX_{(2)}$

⋮ — Specification of boundary condition codes for the relevant degrees of freedom.

$FIX_{(NDOF)}$

The following boundary condition codes, i.e. the values for $FIX_{(1)}$, $FIX_{(2)}$, ..., $FIX_{(NDOF)}$ - are defined for the status of the degrees of freedom:

$FIX_{(i)}$	= 0	no boundary condition is applied to the degree of freedom.
$FIX_{(i)}$	= 1	fixed at zero displacement, temperature, etc.
$FIX_{(i)}$	= 2	prescribed displacement, temperature, velocity, acceleration, etc. different from zero.
$FIX_{(i)}$	= 3	linearly dependent.
$FIX_{(i)}$	= 4	retained degree of freedom, i.e. super node.

The code $FIX_{(i)}$ = 2 just indicates specified condition for the relevant degree of freedom. Whether it is displacement, first time derivative of the displacement etc. is defined on the **BNDISPL** data type.

Degrees of freedom with $FIX_{(i)}$ = 2 which are not defined on the **BNDISPL** data type will be fixed (have zero displacement, velocity and acceleration).

The node numbers (degrees of freedom) with $FIX_{(i)}$ = 4 are called super nodes. The super node numbering is according to the increasing order of their internal node number.

Back to Section 6.2 **Boundary Conditions, Loads and Point Masses**

6.2.3 **BNDISPL**: Node with Displacement, Velocity and/or Acceleration

BNDISPL	→ LLC	DTYPE	COMPLEX	Not Used
	→ NODENO	NDOF	$\text{RDISP}_{(1)}$	$\text{RDISP}_{(2)}$
	...	...	...	$\text{RDISP}_{(\text{NDOF})}$
	$\text{IDISP}_{(1)}$	$\text{IDISP}_{(2)}$	...	...
	...	$\text{IDISP}_{(\text{NDOF})}$		

LLC	Local load case number (positive integer number).
DTYPE	Type of boundary condition.
	= 1 specified displacement, temperature, etc.
	= 2 specified velocity, first time derivative of the temperature, etc.
	= 3 specified acceleration, etc.
COMPLEX	Phase shift definition.
	= 0 no phase shift.
	= 1 phase shift.
NODENO	Program defined internal number for the node with specified boundary condition.
NDOF	Number of degrees of freedom at the node NODENO.
$\text{RDISP}_{(1)}$	The real part of the specified boundary condition with respect to the first degree of freedom at NODENO.
$\text{RDISP}_{(2)}$	The real part of the specified boundary condition with respect to the second degree of freedom at NODENO.
⋮	
$\text{RDISP}_{(\text{NDOF})}$	The real part of the specified boundary condition with respect to the last degree of freedom at NODENO.
$\text{IDISP}_{(1)}$	The imaginary part of the specified boundary condition with respect to the first degree of freedom at NODENO.
$\text{IDISP}_{(2)}$	The imaginary part of the specified boundary condition with respect to the second degree of freedom at NODENO.
⋮	
$\text{IDISP}_{(\text{NDOF})}$	The imaginary part of the specified boundary condition with respect to the last degree of freedom at NODENO.
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

RDISP and IDISP refer to the transformed co-ordinate system if the node NODENO is transformed, else to the global co-ordinate system of the super element.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $\text{RDISP}_{(\text{NDOF})}$ – and the first imaginary number – $\text{IDISP}_{(1)}$. If no phase shift is specified

(that is COMPLEX= 0), then the fields or positions $IDISP_{(1)}$, $IDISP_{(2)}$, etc. are left out.

Back to Section 6.2 **Boundary Conditions, Loads and Point Masses**

6.2.4 **BNDOF**: Node with Transformation

BNDOF	→ NODENO	TRANSD	TRANSR	

NODENO	Program defined internal number for the node with transformation.
TRANSD	Reference number to the transformed co-ordinate system of the displacements, given on **BNTRCOS** for NODENO.
TRANSR	Reference number to the transformed co-ordinate system of the rotations, given on **BNTRCOS** for NODENO.

If no reference number is given, no transformation is relevant to the relevant type of degree of freedom, i.e. translations or rotations for NODENO.

Back to Section 6.2 **Boundary Conditions, Loads and Point Masses**

6.2.5 **BNINCO**: Node with Initial Condition If Arbitrary Time Dependent Loading

BNINCO	→ INCONO	DTYPE	Not Used	Not Used
	→ NODENO	NDOF	RVALUE$_{(1)}$	RVALUE$_{(2)}$
	...	...	...	RVALUE$_{(NDOF)}$

INCONO	Initial condition number.
DTYPE	Type of initial condition. = 1 displacement. = 2 velocity. Both initial displacements and velocities may be specified for a node, but then on separate **BNINCO** data types.
NODENO	Program defined internal number for the node with initial condition.
NDOF	Number of degrees of freedom at the node NODENO.
RVALUE$_{(1)}$	The initial condition with respect to the first degree of freedom at NODENO.
RVALUE$_{(2)}$	The initial condition with respect to the second degree of freedom at NODENO.
⋮	
RVALUE$_{(NDOF)}$	The initial condition with respect to the last degree of freedom at NODENO.
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

Back to Section 6.2 **Boundary Conditions, Loads and Point Masses**

6.2.6 **BNLOAD**: Node with Load

BNLOAD	→ LLC	LOTYP	COMPLEX	Not Used
	→ NODENO	NDOF	$\text{RLOAD}_{(1)}$	$\text{RLOAD}_{(2)}$
	...	...	...	$\text{RLOAD}_{(\text{NDOF})}$
	$\text{ILOAD}_{(1)}$	$\text{ILOAD}_{(2)}$	...	...
	...	$\text{ILOAD}_{(\text{NDOF})}$		

LLC	Local load case number (positive integer number).	
LOTYP	Type of load at the node NODENO.	
	= 0	not decided whether conservative or non-conservative load.
	= 1	conservative load.
	= -1	non-conservative load.
COMPLEX	Phase shift definition.	
	= 0	no phase shift.
	= 1	phase shift.
NODENO	Program defined internal number for the node with load condition.	
NDOF	Number of degrees of freedom at the node NODENO.	
$\text{RLOAD}_{(1)}$	The real part of the load condition with respect to the first degree of freedom at NODENO.	
$\text{RLOAD}_{(2)}$	The real part of the load condition with respect to the second degree of freedom at NODENO.	
⋮		
$\text{RLOAD}_{(\text{NDOF})}$	The real part of the load condition with respect to the last degree of freedom at NODENO.	
$\text{ILOAD}_{(1)}$	The imaginary part of the load condition with respect to the first degree of freedom at NODENO.	
$\text{ILOAD}_{(2)}$	The imaginary part of the load condition with respect to the second degree of freedom at NODENO.	
⋮		
$\text{ILOAD}_{(\text{NDOF})}$	The imaginary part of the load condition with respect to the last degree of freedom at NODENO.	
Not Used	= 0.	Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

The imaginary numbers follow immediately after the real numbers, i.e. there are no blank fields between the last real number – $\text{RLOAD}_{(\text{NDOF})}$ – and the first imaginary number – $\text{ILOAD}_{(1)}$. If no phase shift is specified (that is COMPLEX= 0), then the fields or positions $\text{ILOAD}_{(1)}$, $\text{ILOAD}_{(2)}$, etc. are left out.

Back to Section 6.2 **Boundary Conditions, Loads and Point Masses**

6.2.7 **BNMASS**: Node with Point Mass

BNMASS	→ NODENO	NDOF	$\text{MASS}_{(1)}$	$\text{MASS}_{(2)}$
	...	...	...	$\text{MASS}_{(\text{NDOF})}$
	ModelNode			

NODENO	Program defined internal number for the node with point mass.
NDOF	Number of degrees of freedom at the node NODENO.
$\text{MASS}_{(1)}$	The point mass with respect to the first degree of freedom at NODENO.
$\text{MASS}_{(2)}$	The point mass with respect to the second degree of freedom at NODENO.
⋮	
$\text{MASS}_{(\text{NDOF})}$	The point mass with respect to the last degree of freedom at NODENO.
ModelNode	If ModelNode > 0, then it represents the super element assembly origin, or part of the original super element model. That is, ModelNode is equal to the unique tree node identifier created when super element models are assembled to global models for direct analysis in Sestra. If ModelNode $\leq$ 0 it means that the data applies for the whole assembled model. For the BNMASS datatype ModelNode is optional. If ModelNode is not given the data applies for the whole model.

The point mass contributions in $\text{MASS}_{(i)}$ refer to the transformed co-ordinate system if the node NODENO is transformed, else to the global co-ordinate system of the super element.

A more general method for specifying mass is the mass element **GMAS**(Sesam element type number 11), specified with the **GELMNT1** and the **MGMASS** data type.

Back to Section 6.2 **Boundary Conditions, Loads and Point Masses**

6.2.8 **BNTRCOS**: Transformation from Global to Local Co-ordinate System, Direction Cosines

BNTRCOS	→ TRANSNO	$c_{1,1}$	$c_{2,1}$	$c_{3,1}$
	$c_{1,2}$	$c_{2,2}$	$c_{3,2}$	$c_{1,3}$
	$c_{2,3}$	$c_{3,3}$		

TRANSNO	Reference number to the transformed co-ordinate system.
$c_{1,1}$	
$c_{2,1}$	
$c_{3,1}$	
$c_{1,2}$	
$c_{2,2}$	Terms (9 direction cosines) of the transformation matrix C.
$c_{3,2}$	
$c_{1,3}$	
$c_{2,3}$	
$c_{3,3}$	

The transformation matrix C describes the transformation defined by

$$r' = Cr \tag{6.1}$$

where r' refers to the local co-ordinate system and r to the global (super element) co-ordinate system.

C is applied to both translations and rotations.

For 2D models all 9 values should be given and the program reading the data type must extract the values $c_{1,1}$, $c_{2,1}$, $c_{1,2}$ and $c_{2,2}$ from this matrix.

The **GUNIVEC** data types are used for beam elements only, i.e. basic element types 2, 15 and 23. Other basic element types may refer to **BNTRCOS** data types. No ambiguity thus exists if both a **GUNIVEC** and **BNTRCOS** data type have same TRANSNO, but they should preferably have separate numbering (TRANSNO) to avoid possible program problems.

Back to Section 6.2 **Boundary Conditions, Loads and Point Masses**

6.2.9 **BQDP**: Node with Simple Quadratic Dependence

BQDP	→ NODENO	DDOF	Not Used	Not Used
	$\text{CNOD}_{(1)}$	$\delta_{(1,1)}$	$\beta_{(1,1)}$	$\delta_{(1,2)}$
	$\beta_{(1,2)}$	...	...	$\delta_{(1,\text{DDOF})}$
	$\beta_{(1,\text{DDOF})}$	Not Used	Not Used	Not Used
	$\text{CNOD}_{(2)}$	$\delta_{(2,1)}$	$\beta_{(2,1)}$	$\delta_{(2,2)}$
	$\beta_{(2,2)}$	...	...	$\delta_{(2,\text{DDOF})}$
	$\beta_{(2,\text{DDOF})}$	Not Used	Not Used	Not Used

⋮

	$\text{CNOD}_{(\text{N})}$	$\delta_{(\text{N},1)}$	$\beta_{(\text{N},1)}$	$\delta_{(\text{N},2)}$
	$\beta_{(\text{N},2)}$	...	...	$\delta_{(\text{N},\text{DDOF})}$
	$\beta_{(\text{N},\text{DDOF})}$			

NODENO — Program defined internal number for the node with dependency.

DDOF — Number of dependent degrees of freedom at the node NODENO. (Must correspond with the number of degrees of freedom with the code FIX = 3 referenced in **BNBCD** for NODENO.)

$\text{CNOD}_{(1)}$ — Program defined internal node number for the first independent node.

$\delta_{(1,1)}$ — The first relevant independent degree of freedom at $\text{CNOD}_{(1)}$ which is coupled to the corresponding dependent degree of freedom at NODENO.

$\beta_{(1,1)}$ — The corresponding contribution of the first relevant degree of freedom at $\text{CNOD}_{(1)}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.

$\delta_{(1,2)}$ — The second relevant independent degree of freedom at $\text{CNOD}_{(1)}$ which is coupled to the corresponding dependent degree of freedom at NODENO.

$\beta_{(1,2)}$ — The corresponding contribution of the second relevant degree of freedom at $\text{CNOD}_{(1)}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.

⋮

$\delta_{(1,\text{DDOF})}$ — The number DDOF relevant independent degree of freedom at $\text{CNOD}_{(1)}$ which is coupled to the corresponding dependent degree of freedom at NODENO.

$\beta_{(1,\text{DDOF})}$ — The corresponding contribution of the DDOF relevant degree of freedom at $\text{CNOD}_{(1)}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.

$\text{CNOD}_{(2)}$ — Program defined internal node number for the second independent node.

$\delta_{(2,1)}$ — The first relevant independent degree of freedom at $\text{CNOD}_{(2)}$ which is coupled to the corresponding dependent degree of freedom at NODENO.

$\beta_{(2,1)}$ — The corresponding contribution of the first relevant degree of freedom at $\text{CNOD}_{(2)}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.

$\delta_{(2,2)}$	The second relevant independent degree of freedom at $\text{CNOD}_{(2)}$ which is coupled to the corresponding dependent degree of freedom at NODENO.
$\beta_{(2,2)}$	The corresponding contribution of the second relevant degree of freedom at $\text{CNOD}_{(1)}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.
⋮	
$\delta_{(2,\text{DDOF})}$	The number DDOF relevant independent degree of freedom at $\text{CNOD}_{(2)}$ which is coupled to the corresponding dependent degree of freedom at NODENO.
$\beta_{(2,\text{DDOF})}$	The corresponding contribution of the DDOF relevant degree of freedom at $\text{CNOD}_{(2)}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.
⋮	
⋮	
$\text{CNOD}_{(\text{N})}$	Program defined internal node number for the last independent node - that is independent node *N*.
$\delta_{(\text{N},1)}$	The first relevant independent degree of freedom at $\text{CNOD}_{(\text{N})}$ which is coupled to the corresponding dependent degree of freedom at NODENO.
$\beta_{(\text{N},1)}$	The corresponding contribution of the first relevant degree of freedom at $\text{CNOD}_{(\text{N})}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.
$\delta_{(\text{N},2)}$	The second relevant independent degree of freedom at $\text{CNOD}_{(\text{N})}$ which is coupled to the corresponding dependent degree of freedom at NODENO.
$\beta_{(\text{N},2)}$	The corresponding contribution of the second relevant degree of freedom at $\text{CNOD}_{(\text{N})}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.
⋮	
$\delta_{(\text{N},\text{DDOF})}$	The number DDOF relevant independent degree of freedom at $\text{CNOD}_{(\text{N})}$ which is coupled to the corresponding dependent degree of freedom at NODENO.
$\beta_{(\text{N},\text{DDOF})}$	The corresponding contribution of the DDOF relevant degree of freedom at $\text{CNOD}_{(\text{N})}$ to the quadratic dependence of the corresponding dependent degree of freedom at NODENO.
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

N is the number of independent nodes that NODENO is dependent on.

By simple is meant that f.ex. the first dependent degree of freedom is always coupled to the first independent degree of freedom, the second dependent degree of freedom is always coupled to the second independent degree of freedom, etc.

Note: This data type is *not* available in Sesam.

Back to Section 6.2 **Boundary Conditions, Loads and Point Masses**

6.2.10 **BSELL**: Sub Element Load Description

BSELL	→ LC	→ SUBNO	Not Used	Not Used
	$LLC_{(1)}$	$FACT_{(1)}$	$LLC_{(2)}$	$FACT_{(2)}$
	...	...	$LLC_{(NLOAD)}$	$FACT_{(NLOAD)}$

LC	Global load case number (positive integer number).
SUBNO	Sub element number of the super element in question. Notice that `SUBNO` is the number of the current super element in the two level basic assembly in which it is a part and not the `SELTYP` number at the **IDENT** data type. `SUBNO` is generated by Presel and is the first item at the **GELMNT2** data type.
$LLC_{(1)}$	First local load case included in the global load case LC.
$FACT_{(1)}$	Scaling factor for the first local load case.
$LLC_{(2)}$	Second local load case included in the global load case LC.
$FACT_{(2)}$	Scaling factor for the second local load case.
⋮	
$LLC_{(NLOAD)}$	Last local load case included in the global load case LC.
$FACT_{(NLOAD)}$	Scaling factor for the last local load case.
Not Used	= 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

NLOAD is the number of local load cases included in the global load case LLC.

Back to Section 6.2 **Boundary Conditions, Loads and Point Masses**

6.3 Nodal Data and Element Geometry Definition

Higher level data

GCOORD	Nodal Co-ordinates	see Section 6.3.1
GELMNT1	Element Data Definition	see Section 6.3.2
GELMNT2	Sub Element Description with Simple Correspondence between Degrees of Freedom of Sub Element and Relevant Assembly	see Section 6.3.3
GELREF1	Reference to Element Data	see Section 6.3.4
GNODE	Correspondence between External and Internal Node Numbering, and Number of Degrees of Freedom at Each Node	see Section 6.3.5

6.3.1 **GCOORD**: Nodal Co-ordinates

GCOORD	→ NODENO	XCOORD	YCOORD	ZCOORD

NODENO	Program defined internal number for the node.
XCOORD	x co-ordinate.
YCOORD	y co-ordinate.
ZCOORD	z co-ordinate.

There will be one data type with the identifier **GCOORD** for each node. The sequence of the data types will correspond to the internal node number, NODENO.

Back to Section 6.3 **Nodal Data and Element Geometry Definition**

6.3.2 **GELMNT1**: Element Data Definition

GELMNT1	ELNOX	→ ELNO	ELTYP	ELTYAD
	$\text{NODIN}_{(1)}$	$\text{NODIN}_{(2)}$	. . .	. . .
	. . .	$\text{NODIN}_{(\text{NNOD})}$	Not Used	Not Used

ELNOX — External element number (specified or controlled by the user). Can be any integer identifier as long as it is unique in the element numbering sequence.

ELNO — Program defined internal number for the element. The internal element numbers range from 1 and continuously - no holes in the numbering - to number of elements in the finite element model.

ELTYP — Element type number. Refer to section 3 for description of legal element type numbers. For element type number 70 ("matrix element") all relevant element data are stored as stiffness, mass, damping matrices, and so on. See the **AMATRIX** data type for more information.

ELTYAD — Additional information related to element type:

= IPLANE for membranes

= ISTRUCT for standard elements, not including membranes

= IMATRX for "matrix elements"

IPLANE Used to specify plane stress / plane strain conditions

- 0 Plane stress
- 1 Plane strain

ISTRUCT Used to specify structural / non-structural elements

- 0 Structural elements
- 1 Non structural beam, not including mass in direction of the beam axis
- 2 Non structural elements

IMATRX Reference no. to the corresponding **AMATRIX** data type.

- n Referring to the the **AMATRIX** data type with IMATRX = n.

$\text{NODIN}_{(1)}$

$\text{NODIN}_{(2)}$ — Global internal node numbers of the elements in question. The sequence of the node numbers is in accordance with the local node numbering of the basic elements.

$\text{NODIN}_{(\text{NNOD})}$

Not Used — = 0. Notice that trailing blanks or zeros at the end of the data type to pad the line are not required.

Note:

NNOD = number of nodes at the element.

The length of the data type is always greater than or equal to 4 + NNOD.

1. If the length of the data type is greater than 4 + NNOD, then the trailing data items should be set to zero.

By global node number is meant the node numbering of the entire super element of which the element ELNOX is a part. The internal node number refers to the node number generated by the program.

The program defined element number ranges from 1 up to number of elements.

The sequence of the data type will correspond to the program defined element numbering, ELNO.

Back to Section 6.3 **Nodal Data and Element Geometry Definition**

6.3.3 **GELMNT2**: Sub Element Description with Simple Correspondence between Degrees of Freedom of Sub Element and Relevant Assembly

GELMNT2	→ SUBNO	SLEVEL	SELTYP	ADDNO
	$T_{(1,1)}$	$T_{(2,1)}$	$T_{(3,1)}$	$T_{(1,2)}$
	$T_{(2,2)}$	$T_{(3,2)}$	$T_{(1,3)}$	$T_{(2,3)}$
	$T_{(3,3)}$	$T_{(1,4)}$	$T_{(2,4)}$	$T_{(3,4)}$
	NNOD	$\text{NOD}_{(1)}$	$\text{NOD}_{(2)}$	...
	...	$\text{NOD}_{(\text{NNOD})}$		

SUBNO — Sub element number within the relevant assembly which this super element is part of.

SLEVEL — Super element level.

Should be identical to SLEVEL on the **IDENT** data type for this sub element.

The level of a super element is defined as the highest level number among its sub elements plus 1. (Basic elements, i.e. beams, shells, springs, etc. have level zero.)

SELTYP — Super element type number.

That is, the identification number of the sub element in question.

Should be identical to SELTYP on the **IDENT** data type for this sub element.

ADDNO — Additional data type number, i.e. reference number referring to additional data specifications.

$T_{(1,1)}$
$T_{(2,1)}$
$T_{(3,1)}$
$T_{(1,2)}$
$T_{(2,2)}$
$T_{(3,2)}$ — Elements of the general transformation matrix T (see below).
$T_{(1,3)}$
$T_{(2,3)}$
$T_{(3,3)}$
$T_{(1,4)}$
$T_{(2,4)}$
$T_{(3,4)}$

NNOD — Number of nodes of the sub element in question.

$\text{NOD}_{(1)}$
$\text{NOD}_{(2)}$

$\vdots$ — Node numbers of the sub element in question.

NOD$_{(NNOD)}$

Note: The sequence of the nodes defined on this data type, i.e. NOD$_{(1)}$, NOD$_{(2)}$, . . . , NOD$_{(NNOD)}$, must be in ascending order of the internal node numbers of the super nodes on the level below. If this is not the case the coupling between the super elements will be wrong.

The general transformation matrix T is defined through the following relation between the sub element co-ordinate system x' and the assembly, or basic super element assembly co-ordinate system x

$$x' = Tx,$$

where

$$\begin{bmatrix} x' \\ 1 \end{bmatrix} = \begin{bmatrix} t_{(1,1)} & t_{(1,2)} \\ t_{(2,1)} & t_{(2,2)} \end{bmatrix} \begin{bmatrix} x \\ 1 \end{bmatrix},$$

and where the sub-matrices are defined as follows

$$\begin{bmatrix} x'_1 \\ x'_2 \\ x'_3 \\ 1 \end{bmatrix} = \begin{bmatrix} T_{(1,1)} & T_{(1,2)} & T_{(1,3)} & T_{(1,4)} \\ T_{(2,1)} & T_{(2,2)} & T_{(2,3)} & T_{(2,4)} \\ T_{(3,1)} & T_{(3,2)} & T_{(3,3)} & T_{(3,4)} \\ 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ x_3 \\ 1 \end{bmatrix}. \tag{6.2}$$

The 9 terms (cosines) of the first sub matrix $t_{(1,1)}$, i.e.

$$\left[t_{(1,1)}\right] = \begin{bmatrix} T_{(1,1)} & T_{(1,2)} & T_{(1,3)} \\ T_{(2,1)} & T_{(2,2)} & T_{(2,3)} \\ T_{(3,1)} & T_{(3,2)} & T_{(3,3)} \end{bmatrix},$$

are due to a possible rotation and/or mirroring of the sub element in question.

The three terms of the second sub matrix $t_{(1,2)}$, i.e.

$$\left[t_{(1,2)}\right] = \begin{bmatrix} T_{(1,4)} \\ T_{(2,4)} \\ T_{(3,4)} \end{bmatrix},$$

are the co-ordinates of the origin of the global (assembly) co-ordinate system in the sub element co-ordinate system.

Notice: as can be seen from equation 6.2 the explicit matrix zeros and unity element, i.e. $T_{(4,1)} = 0$, $T_{(4,2)} = 0$, $T_{(4,3)} = 0$, and $T_{(4,4)} = 1$ are not given on the **GELMNT2** data type. This definition differs from the **HSUPTRAN** data type where they are explicitly given.

Back to Section 6.3 **Nodal Data and Element Geometry Definition**

6.3.4 **GELREF1**: Reference to Element Data

GELREF1	→ ELNO	MATNO	ADDNO	INTNO
	MINTNO	STRANO	STRENO	STREPONO
	GEONO/OPT	FIXNO/OPT	ECCNO/OPT	TRANSNO/OPT
	$\text{GEONO}_{(1)}$	. . .	$\text{GEONO}_{(\text{NNOD})}$	$\text{FIXNO}_{(1)}$
	. . .	$\text{FIXNO}_{(\text{NNOD})}$	$\text{ECCNO}_{(1)}$	. . .
	$\text{ECCNO}_{(\text{NNOD})}$	$\text{TRANSNO}_{(1)}$	. . .	$\text{TRANSNO}_{(\text{NNOD})}$

Shortest version for which GEONO, FIXNO, ECCNO, TRANSNO ≥ 0:

GELREF1	ELNO	MATNO	ADDNO	INTNO
	MINTNO	STRANO	STRENO	STREPONO
	GEONO	FIXNO	ECCNO	TRANSNO

ELNO — Program defined internal number for the element

MATNO — Material number.

= 0 — No material data attached to the element.

ADDNO — Additional data type number, i.e. number referring to additional data specification.

= 0 — No additional data attached to the element.

=-1 — I.e when ADDNO = -1, then the item STRANO represents the element type number.

INTNO — Integration station reference number for stiffness matrix, i.e. number referring to the specification of integration stations. An integration station is defined as:

- an assembly of integration points over a cross section of a 1-dimensional (beam or bar) element,
- an assembly of integration points on a line through the thickness of a 2-dimensional element,
- one single integration point for a 3-dimensional element. For further explanation see data type **GELINT**.

= 0 — Default values of the analysis program are employed.

MINTNO — Integration station reference number for mass and damping matrices. Integration station, see INTNO.

= 0 — Default values of the analysis program are employed.

STRANO — Initial strain number, i.e. a number referring to the specification of initial strains. (To be given on data type ASTR which is not yet defined.)

As noted above under item ADDNO, if ADDNO = -1, then STRANO = element type.

STRENO		Initial stress number, i.e. a number referring to the specification of initial stresses. (To be given on data type ASTR which is not yet defined.)
STREPONO		Stress point specification reference number. See data type **GELSTRP** for further information.
GEONO/OPT		Geometry reference number or option for geometry reference number specified later in this data type sequence.
	>0	The geometry reference number (the same for all nodes in the element). $\text{GEONO}_{(1)}$, . . . , $\text{GEONO}_{(\text{NNOD})}$ will not be specified.
	=0	No geometry data is given, i.e. neither here nor on $\text{GEONO}_{(1)}$, . . . , $\text{GEONO}_{(\text{NNOD})}$.
	=-1	Reference numbers to geometry data are specified later in this data type sequence for all nodes, i.e. all $\text{GEONO}_{(1)}$, . . . , $\text{GEONO}_{(\text{NNOD})}$ will be given.
FIXNO/OPT		Fixation reference number or option for fixation reference numbers specified later in this data type sequence. The meaning assigned to the values of FIXNO/OPT corresponds to those for GEONO/OPT.
ECCNO/OPT		Eccentricity reference number or option for eccentricity reference numbers specified later in this data type sequence. The meaning assigned to the values of ECCNO/OPT corresponds to those for GEONO/OPT.
TRANSNO/OPT		Reference number for local co-ordinate system specification or option for specification of local nodal co-ordinate systems later in this data type sequence. Refers to the **GUNIVEC** or **BNTRCOS** data types. The meaning assigned to the values of TRANSNO/OPT corresponds to those for GEONO/OPT.
$\text{GEONO}_{(1)}$		Geometry reference number, i.e. number referring to thickness or cross sectional specification. Not employed for 3-dimensional elements. $\text{GEONO}_{(1)}$ is the reference number for the 1st local node of the element, $\text{GEONO}_{(i)}$ will be the reference number for the i'th local node.
$\text{GEONO}_{(\text{NNOD})}$		Geometry reference number for the last local node of the element.
$\text{FIXNO}_{(1)}$		Number referring to the specification of degree of fixation (data type BELFIX). $\text{FIXNO}_{(1)}$ is the reference number for the 1st local node of the element, $\text{FIXNO}_{(i)}$ will be the reference number for the i'th local node.
$\text{FIXNO}_{(\text{NNOD})}$		Degree of fixation reference number for the last local node of the element.
$\text{ECCNO}_{(1)}$		Eccentricity number for the first local node of the element, i.e. number referring to the specification of eccentricities.
$\text{ECCNO}_{(\text{NNOD})}$		As $\text{ECCNO}_{(1)}$ only for the last local node.
$\text{TRANSNO}_{(1)}$		Number referring to the specification of the local element co-ordinate system for the 1st local node of the element. Refers to **GUNIVEC** or **BNTRCOS** data type depending on element type.
$\text{TRANSNO}_{(\text{NNOD})}$		As $\text{TRANSNO}_{(1)}$ only for the last local node.

NNOD is the number of local nodes at the element.

NOTE: Parameters appear in succeeding order from third line.

The sequence of the data types will be in the program defined element numbering, ELNO.

Back to Section 6.3 **Nodal Data and Element Geometry Definition**

6.3.5 **GNODE**: Correspondence between External and Internal Node Numbering, and Number of Degrees of Freedom at Each Node

GNODE	NODEX	→ NODENO	NDOF	ODOF

NODEX	External node number (specified or controlled by the user). Can be any integer identifier as long as it is unique in the node numbering sequence.
NODENO	Program defined internal number for the node. The internal node numbers range from 1 and continuously - no holes in the numbering - to number of nodes in the finite element model.
NDOF	Number of degrees of freedom at the node NODENO.
ODOF	Order of the degrees of freedom at node NODENO. NDOF digits. Example NDOF = 3, ODOF = 135 means 3 degrees of freedom. Two translations x, z and and one rotation r_y, respectively in the super element co-ordinate system, unless a local nodal co-ordinate system is specified (see the **BNDOF** and **BNTRCOS** data types).

There will be one data type with the identifier **GNODE** for each node. The sequence of the data type will correspond to the internal node number, NODENO.

Back to Section 6.3 **Nodal Data and Element Geometry Definition**

6.4 Super Element Hierarchy Information in Highest Level T-File

Higher level data

HIERARCH	Super Element Hierarchy Description	see Section 6.4.1
HSUPSTAT	Super Element Statistical Information	see Section 6.4.2
HSUPTRAN	Super Element Transformation	see Section 6.4.3

6.4.1 **HIERARCH**: Super Element Hierarchy Description

HIERARCH	NFIELD	→ IHREF	ISELTY	INDSEL
	ISLEVL	ITREF	IHPREF	NSUB
	$\text{IHSREF}_{(1)}$	$\text{IHSREF}_{(2)}$	. . .	$\text{IHSREF}_{(\text{NSUB})}$

This data type identifies a super element in the hierarchy. All the **HIERARCH** data types are written in the highest level (top level) T-file.

The set of **HIERARCH** data types stored will define the super element hierarchy, see Figure 6.1. Note that the reference IHREF is unique for all "nodes" in the super element tree, i.e. unique for every **HIERARCH** data type. If super elements are repeated, each repetition will have a unique hierarchy reference IHREF, although the super element type number is identical.

NFIELD	Number of data fields on this data type (including this field and embedded not used fields).
IHREF	Hierarchy reference number. Number 1 is reserved for the top level super element. In the Sesam system, the super element pre-processor Presel is writing the **HIERARCH** data types and defining a unique number (IHREF) for each appearance of the different super elements. See also Figure 6.1 below.
ISELTY	Super element type number.
INDSEL	Super element index number. Super element index in case of repeated super elements. If super element is not repeated, INDSEL=1 must be used.
ISLEVL	Super element level.
ITREF	Reference to data type **HSUPTRAN**, defining super element transformation between actual super element and parent super element.
IHPREF	Reference to **HIERARCH** data type of parent super element.
NSUB	Number of sub elements in this super element.
$\text{IHSREF}_{(i)}$	Reference to **HIERARCH** data type for sub element number i.

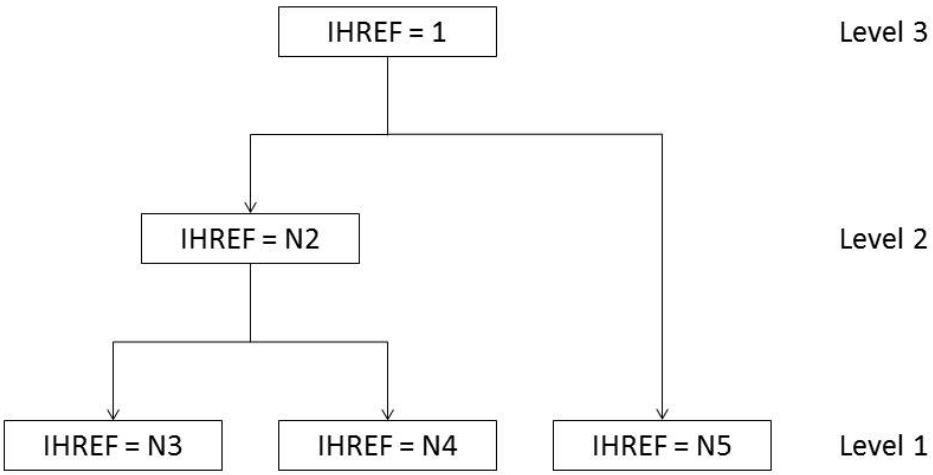


Figure 6.1: Super element hierarchy with 3 levels.

In Figure 6.1 above, please notice that N2, N3, N4, and N5 may take any values as long as they are unique in the hierarchy. Further notice that the top level super element has IHREF = 1 as required for the top level super element.

Back to Section 6.4 **Super Element Hierarchy Information in Highest Level T-File**

6.4.2 **HSUPSTAT**: Super Element Statistical Information

HSUPSTAT	NFIELD	→ ISELTY	NIDOF	NRDOF
	NBAND	NELT	LINDEP	RELOADC
	COMPLC			

This data type lists statistical information about super elements. All the **HSUPSTAT** data types are written in the highest level (toplevel) T-file. The **HSUPSTAT** data type is referenced from the **HIERARCH** data type through the super element type number (ISELTY).

NFIELD	Number of data fields on this data type (including this field and embedded not used fields).
ISELTY	Super element type number.
NIDOF	Estimated number of internal degrees of freedoms.
NRDOF	Estimated number of retained degrees of freedoms.
NBAND	Estimated bandwidth of the internal degrees freedoms. The estimated bandwidth shall be equal to -1 if no bandwidth information exists.
NELT	Estimated number of elements. The estimated number of elements is only required for first level super elements.
LINDEP	If LINDEP > 0, this super element has linear dependent nodes.
RELOADC	Number of real load cases.
COMPLC	Number of complex load cases.

Back to Section 6.4 **Super Element Hierarchy Information in Highest Level T-File**

6.4.3 **HSUPTRAN**: Super Element Transformation

HSUPTRAN	NFIELD	→ ITREF	$T_{(1,1)}$	$T_{(2,1)}$
	$T_{(3,1)}$	$T_{(4,1)}$	$T_{(1,2)}$	$T_{(2,2)}$
	$T_{(3,2)}$	$T_{(4,2)}$	$T_{(1,3)}$	$T_{(2,3)}$
	$T_{(3,3)}$	$T_{(4,3)}$	$T_{(1,4)}$	$T_{(2,4)}$
	$T_{(3,4)}$	$T_{(4,4)}$		

This data type is defining the super element transformation between actual super element and parent super element. All the **HSUPTRAN** data types are written in the highest level (the top level) T-file. The **HSUPTRAN** data type is referenced from the **HIERARCH** data type through the super element transformation reference number, ITREF.

NFIELD	Number of data fields on this data type (including this field and embedded not used fields).
ITREF	Reference to the **HSUPTRAN** data type (from the **HIERARCH** data type).
$T_{(i,j)}$	Term with indices i, j of the transformation matrix between actual super element and parent super element, defined by: $X'_{actual} = T \cdot X_{parent}$

The general transformation matrix T is defined through the following relation between the sub element co-ordinate system x' and the assembly, or basic super element assembly co-ordinate system x

$$x' = Tx,$$

where

$$\begin{bmatrix} x' \\ 1 \end{bmatrix} = \begin{bmatrix} t_{(1,1)} & t_{(1,2)} \\ t_{(2,1)} & t_{(2,2)} \end{bmatrix} \begin{bmatrix} x \\ 1 \end{bmatrix},$$

and where the sub-matrices are defined as follows

$$\begin{bmatrix} x'_1 \\ x'_2 \\ x'_3 \\ 1 \end{bmatrix} = \begin{bmatrix} T_{(1,1)} & T_{(1,2)} & T_{(1,3)} & T_{(1,4)} \\ T_{(2,1)} & T_{(2,2)} & T_{(2,3)} & T_{(2,4)} \\ T_{(3,1)} & T_{(3,2)} & T_{(3,3)} & T_{(3,4)} \\ 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ x_3 \\ 1 \end{bmatrix}. \tag{6.3}$$

The 9 terms (cosines) of the first sub matrix $t_{(1,1)}$, i.e.

$$\left[t_{(1,1)}\right] = \begin{bmatrix} T_{(1,1)} & T_{(1,2)} & T_{(1,3)} \\ T_{(2,1)} & T_{(2,2)} & T_{(2,3)} \\ T_{(3,1)} & T_{(3,2)} & T_{(3,3)} \end{bmatrix},$$

are due to a possible rotation and/or mirroring of the sub element in question.

The three terms of the second sub matrix $t_{(1,2)}$, i.e.

$$\left[t_{(1,2)}\right] = \begin{bmatrix} T_{(1,4)} \\ T_{(2,4)} \\ T_{(3,4)} \end{bmatrix},$$

are the co-ordinates of the origin of the global (assembly) co-ordinate system in the sub element co-ordinate system.

Notice: as can be seen from equation 6.3 the explicit matrix zeros and unity element, i.e. $T_{(4,1)} = 0$, $T_{(4,2)} = 0$, $T_{(4,3)} = 0$, and $T_{(4,4)} = 1$ are given on the **HSUPTRAN** data type. This definition differs from the **GELMNT2** data type where they are not given.

Back to Section 6.4 **Super Element Hierarchy Information in Highest Level T-File**

6.5 Material Data

Higher level data

MAXDMP	Axial Damper between Two Nodal Points	see Section 6.5.1
MAXSPR	Axial Spring between Two Nodal Points	see Section 6.5.2
MGDAMP	Damping Element to Ground	see Section 6.5.3
MGSPRNG	Spring Element to Ground	see Section 6.5.4

6.5.1 **MAXDMP**: Axial Damper between Two Nodal Points

MAXDMP	→ MATNO	DAMP		

MATNO	Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.
DAMP	Axial damping constant.

The axial damping constant corresponds to the force to be applied in order to get a unit velocity in the direction of the basic element.

Back to Section 6.5 **Material Data**

6.5.2 **MAXSPR**: Axial Spring between Two Nodal Points

MAXSPR	→ MATNO	SCON		

MATNO	Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.
SCON	Axial spring constant.

The axial spring constant corresponds to the force to be applied in order to get a unit displacement in the direction of the basic element.

Back to Section 6.5 **Material Data**

6.5.3 **MGDAMP**: Damping Element to Ground

MGDAMP	→ MATNO	NDOF	$C_{(1,1)}$	$C_{(2,1)}$
	...	...	$C_{(\text{NDOF},1)}$	$C_{(2,2)}$
	$C_{(3,2)}$	...	...	$C_{(\text{NDOF},2)}$
	$C_{(3,3)}$	...	...	$C_{(\text{NDOF},\text{NDOF})}$

MATNO — Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.

NDOF — Number of degrees of freedom at the node.

$C_{(i,j)}$ — Elements of the damping matrix (only elements on and below the main diagonal are stored, i.e. a symmetric damping matrix is assumed). The elements are referred to a local co-ordinate system if defined (by TRANSNO on the **GELREF1** data type), otherwise to the global co-ordinate system of the super element.

The damper to ground matrix is the viscous damping matrix.

Back to Section 6.5 **Material Data**

6.5.4 **MGSPRNG**: Spring Element to Ground

MGSPRNG	→ MATNO	NDOF	$K_{(1,1)}$	$K_{(2,1)}$
	...	...	$K_{(\text{NDOF},1)}$	$K_{(2,2)}$
	$K_{(3,2)}$	...	...	$K_{(\text{NDOF},2)}$
	$K_{(3,3)}$	...	...	$K_{(\text{NDOF},\text{NDOF})}$

MATNO — Material number referred to in the element specification, see the **GELMNT1** and **GELREF1** data types.

NDOF — Number of degrees of freedom at the node.

$K_{(i,j)}$ — Elements of the stiffness matrix (only elements on and below the main diagonal are stored, i.e. a symmetric stiffness matrix is assumed). The elements are referred to a local co-ordinate system if defined (by TRANSNO on the **GELREF1** data type), otherwise to the global co-ordinate system of the super element.

The (i, j)'th element of the stiffness matrix corresponds to the force to be given in the i'th degree of freedom to get a unit displacement in the j'th degree of freedom.

Back to Section 6.5 **Material Data**

References

[1] Finite element results data types. Technical report 94-7099, DNV, 2021. 1

[2] O. Aamlid. Implementation of new membrane formulation for the low order flat shell elements in sestra. Technical Report 97-2056, Det Norske Veritas, 1997. 3.45, 3.46

[3] Aa. Berdal. Trsi – transition element between the twenty node isoparametric solid element and the eight node subparametric curved thick shell element. Technical report VT87-3260, DNV, 1984. 3.31

[4] T.J.R. Hughes and T.E. Tezduyar. Finite elements based upon mindlin plate theory with particular reference to the four-node bilinear isoparametric element. *Journal of Applied Mechanics*, September 1981(3):587–596, 1981. 3.48.1, 3.49.1

[5] R. Karlsson. Contact element. theoretical manual. VSS Note TN091989, DNV, 1989. 3.41, 3.42, 3.43, 3.44

[6] R. Karlsson. Contact element. users manual. VSS Note TN081989, DNV, 1989. 3.41, 3.42, 3.43, 3.44

[7] H.F. Klem et al. Finite element library (ellib) – users manual. Technical report VT87-3501, DNV, 1987. 3.2, 3.3, 3.4, 3.5, 3.6, 3.10, 3.15, 3.16, 3.17, 3.18, 3.19, 3.20, 3.21, 3.23, 3.25, 3.26, 3.27, 3.28, 3.45, 3.46

About DNV

We are the independent expert in risk management and quality assurance. Driven by our purpose, to safeguard life, property and the environment, we empower our customers and their stakeholders with facts and reliable insights so that critical decisions can be made with confidence. As a trusted voice for many of the world’s most successful organizations, we use our knowledge to advance safety and performance, set industry benchmarks, and inspire and invent solutions to tackle global transformations.

Digital Solutions

DNV is a world-leading provider of digital solutions and software applications with focus on the energy, maritime and healthcare markets. Our solutions are used worldwide to manage risk and performance for wind turbines, electric grids, pipelines, processing plants, offshore structures, ships, and more. Supported by our domain knowledge and Veracity assurance platform, we enable companies to digitize and manage business critical activities in a sustainable, cost-efficient, safe and secure way.

DNV | digital@dnv.com | dnv.com/digital