2.5 UFO INPUT RECORDS

UFO is special designed for being an efficient **U**ser friendly structural file **FO**rmat in connection with modelling of framed structures.

The user may give all input on one file, or distribute the data on several files. All control parameters are specified in the Analysis Control File. Structure data can also be read from this file, but is usually given on one separate file. The specific content of these files is not important, as long as all data are present.

UFO do not use any **internal** numbers, all data are referred to **user defined external** numbers.

These files may be written with a text editor, or generated by preprocessors

The input records is presented in a standard frame with the following format:

RECORD IDENTIFICATOR	**Parameter No. 1**	**Parameter No. 2**	**Parameter No. 3**	**...**
Parameter	*Description*			
1	Description of contents and default values			
2	Description of contents and default values			
3	Description of contents and default values			
Optional box for comments, notes, exceptions etc.				

NOTE ! Parameters written in **bold** set are mandatory.
Parameters in regular set are optional, and default values will be used if omitted.

2.5.1 Analysis identification

HEAD	<	>
	< Three lines of text identifying the analysis	>
	<	>

Character 9 to 80 from each line are stored as text strings.

This record is given only once.

2.5.2 Nodal Data

NODE **Node ID** **x** **y** **z** ix iy iz irx iry irz

Parameter	*Description*
Node ID	User defined (external) node number
x,y,z	X, Y and Z coordinate of the node
ix	Boundary condition code for X-direction of the actual coordinate system used at the node: **0: Free, 1: Fixed**
iy, iz, irx, iry, irz	Similar boundary condition codes for the remaining 2 translation degrees of freedom and 3 rotation degrees of freedom. **NOTE !** Zeros at the end of the record may be omitted. This implies that by omitting the 6 boundary codes, the node is free in all 6 degrees of freedom.

With this record, the user defines a nodal point to be used in the finite element analysis.

Example 1:

```
NODE   10200   1.0 0.0 4.0   0 0 0 0 0 0
NODE   10200   1.0 0.0 4.0
```

define both a node with ID = 10200, Coordinates (1.0, 0.0, 4.0) with all degrees of freedom free.

Example 2:

```
NODE   10300   1.30 0.0 133.0   1 1 0 0 0 0
NODE   10300   1.30 0.0 133.0   1 1
```

define both a node with ID = 10300, Coordinates (1.3, 0.0, 133.0) with x- and y- degrees of freedom fixed, the rest are free.

This record may be repeated.

NODELOAD **Load Case** **Node ID** **fx fy fz** **mx my mz**

Parameter	*Description*
Load Case	Load Case Number
Node ID	User defined (external) node number
fx,fy,fz	Concentrated force in X-, Y- and Z-direction.
mx,my,mz	Concentrated bending moment about X-, Y- and Z-axis.
	NOTE ! Zeros at the end of the record may be omitted and will be treated as zero load.

With this record, the user defines a concentrated load.

Example 1:

```
NODELOAD   3  10200   1000.0
NODELOAD   3  10200   1000.0  0.0  0.0  0.0  0.0 0.0
```

define both a force in X-direction acting at node 10200 with load case number 3.

Example 2:

```
NODELOAD   4  10500   0.0  0.0  1000.0
NODELOAD   4  10500   0.0  0.0  1000.0  0.0  0.0 0.0
```

define both a force in Z-direction acting at node 10500 with load case number 4.

This record may be repeated.

NODEMASS **Node ID** **M_x** M_y M_z M_Rx M_Ry M_Rz	
Parameter	*Description*
Node ID	User defined (external) node number
fx,fy,fz	Concentrated Mass in X-, Y- and Z-direction.
mx,my,mz	Concentrated Rotation Mass about X-, Y- and Z-axis. **NOTE !** Zeros at the end of the record may be omitted and will be treated as zero mass with following exception : If only M_x is specified, M_y and M_z are assigned the same value.

With this record, the user defines a concentrated mass.

Example 1:

```
NODEMASS   10200   1000.0
```

defines a mass in X-, Y and Z-direction at node 10200.

Example 2:

```
NODEMASS   10500   0.0  0.0  1000.0
NODEMASS   10500   0.0  0.0  1000.0   0.0  0.0  0.0
```

define both a mass in Z-direction at node 10500.

This record may be repeated.

NODTRANS **Trans ID** T_{11} T_{12} T_{13} / T_{21} T_{22} T_{23} / T_{31} T_{32} T_{33}	
Parameter	*Description*
Trans ID	User defined (external) node transformation number
T_{11} - T_{33}	The 9 terms in the 3x3 Transformation matrix **T**.

With this record, the user defines a rotation transformation matrix to be used to define skew boundary conditions at nodes or defining local coordinate system for the 1 node spring to ground.

This record may be repeated.

2.5.3 Element Data

BEAM	**Elem ID** **Node1** **Node2** **Material** **Geom** L_Coor Ecc1 Ecc2
Parameter	*Description*
Elem ID	User defined (external) element number
Node1 Node2	Node 1 of the beam is connected to the **user defined (external)** node number. Node 2 of the beam is connected to the **user defined (external)** node number.
Material	User defined **material** number defining the material properties of the element
Geom	User defined **geometry** number defining the geometry of the element
L_Coor	User defined **unit vector** number defining the local coord. system of the element. If omitted, a default local coordinate system will be used, (local zx-plane is parallel with global zx-plane except for vertical members which have the local zx-plane parallel with the global xy-plane).
Ecc1 Ecc2	Node 1 of the beam has an eccentricity defined by **user defined** Ecc1. Node 2 of the beam has an eccentricity defined by **user defined** Ecc2. **NOTE !** Zeros at the end of the record may be omitted.

With this record, the user defines a beam element to be used in the finite element analysis.

Example 1:

```
BEAM   100200   100  200   1 17 4
```

defines a beam element with ID = 100200, connected to the two nodes with ID = 100 and 200. Material with ID = 1 defines the material properties and geometry with ID = 17 defines the cross section. The local coordinate system is defined by the unit vector with ID=4. No eccentricities.

Example 2:

```
BEAM   100200   100  200   1 17 4  33 34
```

defines the beam in Example 1 with following difference:

End 1 of the beam has an eccentricity defined by an eccentricity vector with ID = 33.
End 2 of the beam has an eccentricity defined by an eccentricity vector with ID = 34.

This record may be repeated.

SPRNG2GR	**Elem ID** **Node** **Material** L_Coor Ecc
Parameter	*Description*
Elem ID	User defined (external) element number of the spring to ground.
Node	The spring is connected to the **user defined (external)** node number.
Material	User defined **material** number defining the spring characteristic.
L_Coor	User defined **transformation matrix** number defining the local coordinate system of the spring. If omitted, local coordinate system and global system are equal.
Ecc	The spring to ground has an eccentricity defined by **user defined** Ecc. **NOTE !** Zeros at the end of the record may be omitted.

With this record, the user defines a 1 node spring to ground element to be used in the finite element analysis.

Example 1:

SPRNG2GR 1020 85 1020

defines a spring to ground element with ID = 1020, connected to the node with ID = 85. The spring characteristic is defined by a material with ID = 1020.
The local coordinate system is parallel to the global system, and no eccentricity is defined.

Example 2:

SPRNG2GR 1020 85 1020 200 33

defines the spring in Example 1 with following difference:

The local coordinate system of the spring is defined by the *NODTRANS* record with ID = 200. The spring has an eccentricity defined by an eccentricity vector with ID = 33.

This record may be repeated.

MEMBRANE **Elem ID** **nod1** **nod2** **nod3** **nod4** **mat** **geom** e1 e2 e3 e4

Parameter	*Description*
Elem ID	User defined (external) element number
nod1 nod2 nod3 nod4	Node 1 of the membrane is connected to the **user defined (external)** nod1. Node 2 of the membrane is connected to the **user defined (external)** nod2. Node 3 of the membrane is connected to the **user defined (external)** nod3. Node 4 of the membrane is connected to the **user defined (external)** nod4.
mat	User defined **material** number defining the material properties of the element
geom	User defined **geometry** number defining the thickness of the element
e1 e2 e3 e4	Node 1 of the membrane has an eccentricity defined by **user defined** e1. Node 2 of the membrane has an eccentricity defined by **user defined** e2. Node 3 of the membrane has an eccentricity defined by **user defined** e3. Node 4 of the membrane has an eccentricity defined by **user defined** e4. **NOTE !** Zeros at the end of the record may be omitted.

With this record, the user defines a membrane element to be used in the finite element analysis.

Example 1:

MEMBRANE 1234 1 2 3 4 100 88

defines a 4 node membrane element with ID = 1234, connected to the four nodes with ID = 1, 2, 3 and 4. Material with ID = 100 defines the material properties and geometry with ID = 88 defines the thickness.
No eccentricities.

This record may be repeated.

QUADSHEL Elem ID nod1 nod2 nod3 nod4 mat geom e1 e2 e3 e4

Parameter	*Description*
Elem ID	User defined (external) element number
nod1 nod2 nod3 nod4	Node 1 of the shell is connected to the **user defined (external)** nod1. Node 2 of the shell is connected to the **user defined (external)** nod2. Node 3 of the shell is connected to the **user defined (external)** nod3. Node 4 of the shell is connected to the **user defined (external)** nod4.
mat	User defined **material** number defining the material properties of the element
geom	User defined **geometry** number defining the thickness of the element
e1 e2 e3 e4	Node 1 of the shell has an eccentricity defined by **user defined** e1. Node 2 of the shell has an eccentricity defined by **user defined** e2. Node 3 of the shell has an eccentricity defined by **user defined** e3. Node 4 of the shell has an eccentricity defined by **user defined** e4. **NOTE !** Zeros at the end of the record may be omitted.

With this record, the user defines a 4 node quadrilateral shell element to be used in the finite element analysis.

Example 1:

```
QUADSHEL   1234   1 2 3 4  100  88  100
```

defines a 4 node shell element with ID = 1234, connected to the four nodes with
ID = 1, 2, 3 and 4. Material with ID = 100 defines the material properties and geometry with ID = 88 defines the thickness.
Node 1 of the shell has an eccentricity defined by Ecc. vector with ID = 100. The other 3 nodes have no eccentricity.

This record may be repeated.

TRISHELL	**Elem ID nod1 nod2 nod3 mat geom** e1 e2 e3
Parameter	*Description*
Elem ID	User defined (external) element number
nod1 nod2 nod3	Node 1 of the shell is connected to the **user defined (external)** nod1. Node 2 of the shell is connected to the **user defined (external)** nod2. Node 3 of the shell is connected to the **user defined (external)** nod3.
mat	User defined **material** number defining the material properties of the element
geom	User defined **geometry** number defining the thickness of the element
e1 e2 e3	Node 1 of the shell has an eccentricity defined by **user defined** e1. Node 2 of the shell has an eccentricity defined by **user defined** e2. Node 3 of the shell has an eccentricity defined by **user defined** e3. **NOTE !** Zeros at the end of the record may be omitted.

With this record, the user defines a 3 node triangular shell element to be used in the finite element analysis.

Example 1:

```
TRISHELL   123   10 20 30   100  88
```

defines a 3 node shell element with ID = 123, connected to the three nodes with
ID = 10, 20 and 30.
Material with ID = 100 defines the material properties and geometry with ID = 88 defines the thickness.

No eccentricities.

This record may be repeated.

SOLID8 Elem ID n1 n2 n3 n4 n5 n6 n7 n8 mat

Parameter	*Description*
Elem ID	User defined (external) element number
n1-n8	Node 1-8 of the solid-element is connected to the **user defined (external)** nodes.
mat	User defined **material** number defining the material properties of the element

With this record, the user defines a 8 node hexahedron solid element to be used in the finite element analysis.

This record may be repeated.

BEAMLOAD	**Load Case** **Elem ID** **qx1 qy1 qz1** qx2 qy2 qz2
Parameter	*Description*
Load Case	Load Case Number
Elem ID	User defined (external) element number, (beam element)
qx,qy,qz_1	Intensity in X-, Y- and Z-direction at **end 1** of the beam, (Global direction).
qx,qy,qz_2	Intensity in X-, Y- and Z-direction at **end 2** of the beam, (Global direction). If end 2 intensities are omitted, end 1 intensities are used, i.e. uniform distributed load. **NOTE !** Zeros at the end of the record may be omitted and will be treated as zero load.

With this record, the user defines a distributed beam element load.

Example 1:

```
BEAMLOAD   5  1020   1000.0
BEAMLOAD   5  1020   1000.0  0.0  0.0
BEAMLOAD   5  1020   1000.0  0.0  0.0   1000.0  0.0 0.0
```

define all a uniform distributed load with intensity 1000 in global X-direction acting at element 1020 with load case number 5.

Example 2:

```
BEAMLOAD   5  1021   0.0  1000.0  0.0    0.0 2000.0
BEAMLOAD   5  1021   0.0  1000.0  0.0    0.0 2000.0  0.0
```

define both a distributed load with intensity 1000 in global Y-direction at end 1 of element 1021 and intensity 2000 at end 2 of the element. The load case number is 5.

This record may be repeated.

PRESSURE	**Load Case** **Elem ID** **p1** p2 p3 p4
Parameter	*Description*
Load Case	Load Case Number
Elem ID	User defined (external) element number, (2-D element)
p1-p4	Normal pressure intensity at node 1-4 of the element. **NOTE !** If only p1 is specified, p2-p4 are given the same intensity, i.e constant pressure over the entire element surface.

With this record, the user defines a normal (*non conservative*) pressure load.

Example 1:

```
PRESSURE   2  1234   100
PRESSURE   2  1234   100  100  100  100
```

define both a constant normal pressure with intensity 100 acting at element 1234 with load case number 2.

Example 2:

```
PRESSURE   4  1234   100  200  300  400
```

defines a normal pressure with intensity 100 at node 1, 200 at node 2, etc. of element 1234. The load case number is 4.

This record may be repeated.

SHELLOAD Load Case qx qy qz <List Type> [Data]

Parameter	*Description*
Load Case	Load Case Number
qx,qy,qz	Intensity in X-, Y- and Z-direction, (Global direction).
<List Type>	Definition of the ID list (next parameter(s)), two different list types are available: *Element* : The actual shell load is assigned to the elements given under [Data] *Material* : The actual shell load is assigned to shell elements with material ID's given in [Data]
[Data]	List of Element ID's (if type=element) or Material ID's (if type=material) **NOTE**! : *No* ID's specified has same meaning as specifying *all* actual ID's

With this record, the user defines a distributed (*conservative*) shell element load referred to Global coordinate system.

Example 1:

SHELLOAD 5 0.0 0.0 -1000.0 *Element* 1020 2020 3030

Defines a uniform distributed load (load case number 5) with intensity -1000 [N/m^2] in global Z-direction for elements 1020, 2020 and 3020.

Example 2:

SHELLOAD 3 0.0 0.0 -2200.0 *Material* 245 355

Defines a uniform distributed load (load case number 3) with intensity –2200 [N/m^2] in global Z-direction for all shell elements referred to material ID's 245 or 355.

Example 3:

SHELLOAD 4 0.0 0.0 -2000.0 *Element*
SHELLOAD 4 0.0 0.0 -1000.0 *Element* 1020 2020 3030

Defines a uniform distributed load (load case number 4) with intensity –2000 [N/m^2] in global Z-direction for all shell elements in the model, except for the elements 1020, 2020 and 3020, which get an intensity of –1000.

This record may be repeated.

REFINE	**N_divide** Elem 1 Elem 2 Elem 3 ....
Parameter	*Description*
N_divide	Each of the specified beam elements should be divided into 'N_divide' elements.
Elem 1,...	User defined (external) element numbers of the elements to be refined.
	NOTE ! 1: If **no** elements are specified, all beam elements are refined as specified. 2: If an element is defined with **initial deformations**, (GIMPER, GELIMP), the generated nodes between the original beam ends will follow the actual imperfection curve.

With this record, the user specifies elements to be sub-divided.

Example 1:

```
REFINE   4   1020   1030  1040
REFINE  10   1050
```

defines that elements 1020, 1030 and 1040 should be divided into 4 elements and element 1050 into 10.

Example 2:

```
REFINE   3
REFINE   8   1050
```

defines that all beam elements in the model should be divided into 3 elements except element 1050 which should be divided into 8.

This record may be repeated.

UNITVEC	**Trans ID unix uniy uniz**
Parameter	*Description*
Trans ID	Unit vector number (external) referred to in f.inst record *BEAM*.
unix uniy uniz	Unit in global coordinates. The vector specifies the direction along local element z-axis.

With this record, the user specifies a unit vector to be used in connection with f.inst. defining local coordinate system of beam elements.

This record may be repeated.

ECCENT	**Ecc ID ex ey ez**
Parameter	*Description*
Ecc ID	Eccentricity number (external) referred to in f.inst record *BEAM*.
ex ey ez	Eccentricity vector in global coordinates. The vector points from the global node towards the local element node.

With this record, the user specifies an eccentricity vector.

This record may be repeated.

ADDMBEAM mx my mx Elem_1 ID Elem_2 ID .....

Parameter	*Description*
mx	Added mass intensity of the local X-direction of the beam element.
my	Added mass intensity of the local Y-direction of the beam element.
mz	Added mass intensity of the local Z-direction of the beam element.
Elem_1 ID	User defined (external) element number 1, (beam element)
Elem_2 ID	User defined (external) element number 2
.	.
.	.
	NOTE ! If **no** elements are specified, all beam elements are specified with the actual added mass.

With this record, the user defines added mass data for beam elements.

Example 1:
ADDMBEAM 0 1000 1000

defines added mass in X-direction = 0.0, Y-and Z direction : 1000 applied on all beam elements.

This record may be repeated.

BANANA Offset Angle

Parameter	*Description*
Offset	Maximum offset divided by element length. (default = 0.0015)
Angle	Orientation of offset. Specified in degrees counter-clockwise from the local element z-axis, (default = 0°)
	NOTE ! If no parameters are defined, all beam elements will get an initial out of straightness of 0.15%.

With this record, the user defines the out of straightness of the beam elements.

NOTE ! For physical members divided into more than one beam element, the coordinates of the nodes along the physical member are moved according to the specified offset and orientation.
The updated coordinates are printed on the .out file.

This record is given once!

2.5.4 Cross Section Data

PIPE Geom ID D_o T Shear_Y Shear_Z

Parameter	*Description*
Geom ID	User defined (external) geometry number
D_o	Outer Diameter of the pipe.
T	Wall thickness of the pipe.
Shear_Y Shear_Z	Shear area factor of Y-axis. Shear area = Shear_Y * Calculated shear area. Shear area factor of Z-axis. Shear area = Shear_Z * Calculated shear area If Shear_Y and Shear_Z are omitted or equal to zero, Shear_Z and Shear_Z are both set equal to 1.0.

With this record, the user defines a pipe cross section to be used in the finite element analysis.

Example 1:

```
PIPE   45025   0.450  0.025
```

defines a pipe identified by the ID: 45025 with outer diameter = 0.45 and wall thickness = 0.025.
The shear area is not scaled, (calculated shear area is used directly).

This record may be repeated.

BOX **Geom ID** **H** **T_side** **T_bott** **T_top** **Width** Shear_Y Shear_Z	
Parameter	*Description*
Geom ID	User defined (external) geometry number
H	Height of the profile.
T_side	Thickness of the side walls of the RHS-profile.
T_bott	Thickness of the bottom "flange" of the profile.
T_top	Thickness of the top "flange" of the profile.
Width	Width of the box profile.
Shear_Y Shear_Z	Shear area factor of Y-axis. Shear area = Shear_Y * Calculated shear area. Shear area factor of Z-axis. Shear area = Shear_Z * Calculated shear area If Shear_Y and Shear_Z are omitted or equal to zero, Shear_Z and Shear_Z are both set equal to 1.0.
With this record, the user defines a rectangular hollow cross section to be used in the finite element analysis. This record may be repeated.	

IHPROFIL **Geom ID** **H** **T_web** **W_top** **T_top** **W_bott** **T_bott** Shear_Y Shear_Z	
Parameter	*Description*
Geom ID	User defined (external) geometry number
H T_web	Height of the profile. Thickness of the web.
W_top T_top	Width of the top flange Thickness of the top flange
W_bott T_bott	Width of the bottom flange Thickness of the bottom flange
Shear_Y Shear_Z	Shear area factor of Y-axis. Shear area = Shear_Y * Calculated shear area. Shear area factor of Z-axis. Shear area = Shear_Z * Calculated shear area If Shear_Y and Shear_Z are omitted or equal to zero, Shear_Z and Shear_Z are both set equal to 1.0.
With this record, the user defines a I/H profile to be used in the finite element analysis. This record may be repeated.	

PLTHICK	**Geom ID Thick**
Parameter	*Description*
Geom ID	User defined (external) geometry number
Thick	Plate Thickness

With this record, the user defines a plate thickness to be used in the finite element analysis.

This record may be repeated.

LSECTION	**Geom ID H T_side** Width T_bott I_symm
Parameter	*Description*
Geom ID	User defined (external) geometry number
H	Height of the profile.
T_side	Thickness of the vertical part.
Width	Width of the profile
T_bott	Thickness of the horizontal part of the profile
I_symm	Symmetry switch: 0 or omitted : "opposite L" 1 : L
	NOTE ! If thickness of horizontal part is omitted, thickness for the vertical part is used. If both width and thickness of the horizontal part are omitted, the width is set equal to the height and the thickness for the vertical part is used.

With this record, the user defines an open angle section to be used in the finite element analysis.

Example 1 :

```
LSECTION   15015   0.150   0.015
```

defines a regular angle with thickness 0.015 and width = height = 0.150

This record may be repeated.

GENBEAM Geom ID Area It Iy Iz Wpx Wpy Wpz Shy Shz

Parameter	*Description*
Geom ID	User defined (external) geometry number
Area	Cross sectional area
It	Torsion moment of inertia
Iy	Moment of inertia about y-axis
Iz	Moment of inertia about z-axis
Wpx	Plastic torsional section modulus
Wpy	Plastic sectional modulus about y-axis
Wpz	Plastic sectional modulus about z-axis
Shy	Shear area in direction of y-axis
Shz	Shear area in direction of z-axis

With this record, the user defines a general cross section to be used in the finite element analysis.

This record may be repeated.

2.5.5 Material Data

SPRIDIAG	**Mat ID** S_{11} S_{22} S_{33} S_{44} S_{55} S_{66}
Parameter	*Description*
Mat ID	User defined (external) material number
S_{11} - S_{66}	Diagonal terms of the linear 6x6 spring to ground stiffness matrix **NOTE !** Zeros at the end of the record may be omitted.

With this record, the user defines a diagonal spring to ground stiffness matrix to be used in the finite element analysis.

Example 1:

SPRIDIAG 33 1.0E4

Defines a spring to ground stiffness identified by the ID: 33 with stiffness 1.0E4 in global X-direction. The 5 omitted diagonal terms are all set equal to zero

This record may be repeated.

SPRIFULL	**Mat ID** S_{11} S_{12} S_{13} S_{14} S_{15} S_{16} S_{21} S_{22} S_{23} S_{24} S_{25} S_{26} ...... S_{61} S_{62} S_{63} S_{64} S_{65} S_{66}
Parameter	*Description*
Mat ID	User defined (external) material number
S_{11} - S_{66}	36 terms of the linear 6x6 spring to ground stiffness matrix **NOTE !** All 36 stiffness terms must be given.

With this record, the user defines a full spring to ground stiffness matrix to be used in the finite element analysis.

This record may be repeated.

HYPELAST Mat ID P_1 δ_1 P_2 δ_2 ... P_n δ_n

Parameter	*Description*
Mat ID	User defined (external) material number
P_1 δ_1	Definition of the first point of the force/displacement curve
P_2 δ_2	Definition of the second point of the force/displacement curve
P_n δ_n	Definition of the last point of the force/displacement curve
	NOTE ! Only points in 1. and 3. quadrant are legal, and the first point defines the 'bottom left' end of the curve. Max 25 points available.

With this record, the user defines a hyperelastic P-δ curve to be used in connection with definition of a nonlinear spring matrix.

Example 1:

```
HYPELAST   444      -1200 -1.00
                    -1000 -0.01
                     1000  0.01
                     1200  1.00
```

defines a hyperelastic P-δ curve by 4 discrete points identified by the material number 444.

This record may be repeated.

ELPLCURV **Mat ID** **P_1 δ_1 P_2 δ_2 ... P_n δ_n**

Parameter	*Description*
Mat ID	User defined (external) material number
P_1 δ_1	Definition of the first point of the force/displacement curve
P_2 δ_2	Definition of the second point of the force/displacement curve
P_n δ_n	Definition of the last point of the force/displacement curve
	NOTE ! Only points in 1. and 3. quadrant are legal, and the first point defines the 'bottom left' end of the curve. Max 25 points available.

With this record, the user defines an elastoplastic P-δ curve to be used in connection with definition of a nonlinear spring matrix.

Example 1:

```
ELPLCURV   555     -1200 -1.00
                   -1000 -0.01
                    1000  0.01
                    1200  1.00
```

defines an elastoplastic P-δ curve by 4 discrete points identified by the material number 555.

This record may be repeated.

ELASTIC	**Mat ID E Poiss Rho Therm**
Parameter	*Description*
Mat ID	User defined (external) material number
E Poiss Rho Therm	E modulus Poison ratio Density Thermal expansion coefficient **NOTE !** Zeros at the end of the record may be omitted.
With this record, the user defines an elastic isotropic material. This record may be repeated.	

2.5.6 Misc Data

GRAVITY Load Case Ax Ay Az

Parameter	*Description*
Load Case	Load Case Number
Ax	Acceleration field in global X-direction
Ay	Acceleration field in global Y-direction
Az	Acceleration field in global Z-direction
	NOTE ! Zeros at the end of the record may be omitted and will be treated as zero acceleration.

With this record, the user defines a distributed beam element load.

Example 1:

GRAVITY 1 0.0 0.0 -9.81

defines an acceleration field of 9.81 in negative Z-direction. The load case number is 1.

This record may be repeated.

2.5.7 UFO Examples

Zayas Frame

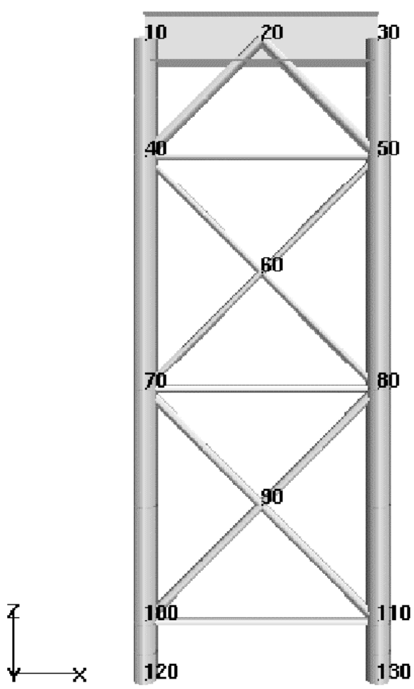
The plane frame described in the figures below consists of 13 nodes and 23 beam elements.

A concentrated force is acting at node 10 in positive X-direction in addition to the gravity load.

All nodes are free except the two bottom nodes, 120 and 130.

The frame consists of pipes and one I/H profile.

The structure is described in UFO-format at the next page.



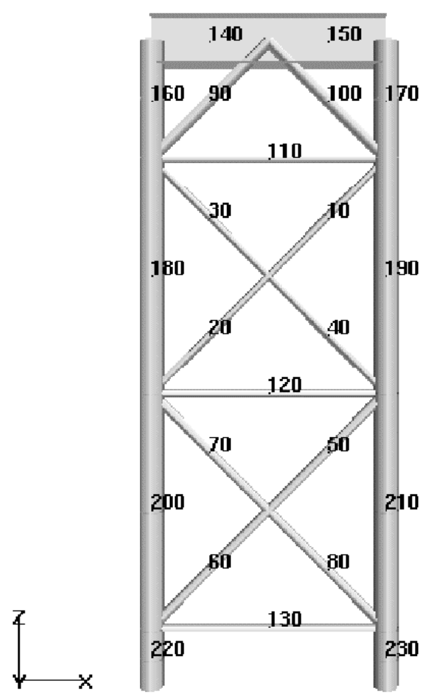


Figure ***Error! No text of specified style in document.****-1 Node Numbers* *Figure* ***Error! No text of specified style in document.****-2 Element Numbers*

UFO input file describing the Zaya's Frame:

```
HEAD                 Z A Y A S   F R A M E
                         described in
                        U F O - Format
'
' -----------------------------------------------------------------
'       N O D E
' -----------------------------------------------------------------
'
'           Node ID           X             Y             Z    Boundary code
 NODE          10            .000          .000         8.382
 NODE          20           1.524          .000         8.382
 NODE          30           3.048          .000         8.382
 NODE          40            .000          .000         6.858
 NODE          50           3.048          .000         6.858
 NODE          60           1.524          .000         5.334
 NODE          70            .000          .000         3.810
 NODE          80           3.048          .000         3.810
 NODE          90           1.524          .000         2.286
 NODE         100            .000          .000          .762
 NODE         110           3.048          .000          .762
 NODE         120            .000          .000          .000   1 1 1 1 0 1
 NODE         130           3.048          .000          .000   1 1 1 1 0 1
'
' -----------------------------------------------------------------
'       E L E M E N T
' -----------------------------------------------------------------
'
'           Elem ID     np1      np2   material   geom    lcoor    ecc1     ecc2
 BEAM          10        60       50         1       4      1
 BEAM          20        70       60         1       4      1
 BEAM          30        40       60         1       4      2
 BEAM          40        60       80         1       4      2
 BEAM          50        90       80         1       3      1
 BEAM          60       100       90         1       3      1
 BEAM          70        70       90         1       3      2
 BEAM          80        90      110         1       3      2
 BEAM          90        40       20         1       2      1
 BEAM         100        20       50         1       2      2
 BEAM         110        40       50         1       4      3
 BEAM         120        70       80         1       4      3
 BEAM         130       100      110         1       4      3
 BEAM         140        10       20         3       5      3
 BEAM         150        20       30         3       5      3
 BEAM         160        10       40         2       1      4
 BEAM         170        30       50         2       1      4
 BEAM         180        40       70         2       1      4
 BEAM         190        50       80         2       1      4
 BEAM         200        70      100         2       1      4
 BEAM         210        80      110         2       1      4
 BEAM         220       100      120         2       1      4
 BEAM         230       110      130         2       1      4
'
' -----------------------------------------------------------------
'       G E O M E T R Y
' -----------------------------------------------------------------
'
'           Geom ID         Do         Thick
 PIPE            1       0.32385     0.007137
 PIPE            2       0.15240     0.003175
 PIPE            3       0.12700     0.003048
 PIPE            4       0.10160     0.002108
'
'           Geom ID     H      T-web    W-top   T-top    W-bot   T-bot Sh_y Sh_z
 IHPROFIL        5     1.40   0.027     1.22   0.050     1.22   0.050
'
'           Loc-Coo          dx             dy             dz
 UNITVEC       1           -.707           .000           .707
 UNITVEC       2            .707           .000           .707
 UNITVEC       3            .000           .000          1.000
 UNITVEC       4            .000          1.000           .000
'
' -----------------------------------------------------------------
'       L O A D
' -----------------------------------------------------------------
'
'           Load Case  Node ID      Fx        Fy   Fz
 NODELOAD          1       10     4.00E+04      0    0
'
'           Load Case   Acc_X       Acc_Y       Acc_Z
GRAVITY          5        0           0       -9.81
```