Table of Contents

6 INPUT DESCRIPTION

USFOS reads input from symbolic files.

The user may give all input on one file, or distribute the data on two or three 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 or two separate files. The specific content of these files is not important, as long as all data are present. For convenience, these files are labelled "Structure file" and "Load file", cfr Figure 2.1.

USFOS has its own file format, (see section 6.5), which may be written with a text editor, or generated by interactive pre-processor and load generation programs. Utility tools (StruMan etc.) could be used to convert structural model data from different program systems (Sesam, SACS and STAAD) to USFOS input.

In addition to the USFOS structure model file format, USFOS also reads the SESAM FEM-files generated by GeniE.

The input records specific for USFOS nonlinear analysis are presented in Section 6.3. Structure input and load input are presented in Section 6.4.

6.1 GENERAL INFORMATION

The input data are organized in records, each record starting with a record identifier of four to eight characters. Each record may consist of one or more lines of data, terminating on the next record identifier. Each line may be **up to 132 characters long**. The data items may be integer on real data.

The data records may be given in an arbitrary order.

In this manual, each record is presented in a standard frame. Each frame represents either one single record or a sequence of similar records.

RECORD_IDENTIFIER Param_1 Param_2 Param_3 ... Param_n		
Parameter	*Description*	*Default*
Param_1	Description of 1st parameter	Default
Param_2	Description of 2nd parameter	values
Param_3	Description of 3rd parameter	if
...		Applicable
Param_n	Description of n'th parameter	
Optional box for comments, notes, exceptions etc.		

6.2 FORMATS

SAM/FII, a FORTRAN free-format reader and decoder read nearly all input data. This means that the data items may be written anywhere on the line, **as long as the specified order is satisfied.** The data items must be separated with at least one blank (exceeding blanks are ignored). Note that blank is exclusively interpreted as a delimiter, and cannot be used to specify a zero value as accepted by standard FORTRAN READ.

Important:

All digits, letters and/or special symbols in a data item must be given consecutively without blanks.

6.2.1 Comments

Lines with following characters in the first column are interpreted as comments, and are simply ignored. Comments may occur anywhere in the input data stream.

'
*
#
%
!

Example:

'THIS IS A COMMENT
NOTE! COMMENTS ARE IGNORED BY THE PROGRAM

In-Line Comments:

The character ! may be used to terminate a data line, and data items on the rest of the line are just ignored.

Example:

```
#
#        Mat_ID   E-mod    poiss   F_yield   Density   therm.exp.
 MISOIEP    1     2.10E+5   0.3     355      8235.7      0.0
 MISOIEP    2     2.10E+5   0.3     340      8235.7      0.0      ! NEW 96-02-13
#MISOIEP    2     2.10E+5   0.3     300      8235.7      0.0      ! OLD 96-02-13
```

This is a useful option in connection with modification/correction of input files.

Alphanumeric Data Items

An alphanumeric data item may consist of one or more characters. The first character is always a letter (A-Z), while the remaining ones may be letters, digits or special symbols (except /, $, & and blank).

There is no upper limit to the number of characters in an alphanumeric data item. However, only the first 8 characters will be decoded, and all characters in excess of this are simply ignored.

Integer Data Items

- All characters must be digits
- The first digit may be preceded by + or -

Example:

0 1 -27 +66

Real Number Data Items

Real numbers data entry may consist of up to 3 components, i.e. an integer part i, a decimal part d, and exponent part e. The following 4 basic forms are accepted:

(±)i (±)i. (±)i.d (±).d

These may all be combined with exponent parts yielding the forms:

(±)iE(±)e (±)i.E(±)e (±)i.dE(±)e (±).dE(±)e

Example:

0 +1. -0.2E14 +17.E-3 1.8E+3

Text Strings

Text strings may consist of one or more characters, which may be letters (A-z), digits or special symbols.

72 characters are stored in a text string, beginning at the 9th character of the line.

<u>Example:</u>

This text string uses special characters & \ #

and will be stored as

t string uses special characters & \ #

Numerical Operations on the Input

The FII input reader interprets some simple numerical operations. This means that the user may define mathematical expressions in the input, for example in order to scale parameters. The expressions are:

- Adding (+)
- Subtraction (-)
- Multiplication (*)
- Division (/)
- Trig (SIN(ang) and COS(ang), ang in radians)

<u>Example 1, Scaling the yield stress:</u>

```
MISOIEP 10 210000E6 0.3  355E6/1.15 7850 1.4E-5  ! Material coeff 1.15 (ULS)
```

<u>Example 2, Adding value to coordinate :</u>

```
'     ID     X      Y         Z
NODE  10     0.0   10.0+1.23   10.0   ! Add 1.23 to the Y-coordinate
```

<u>Example 3, Create a load vector (1MN) depending on angle (here 30 degrees)</u>

```
'         lCase  NodeID          Fx                   Fy               Fz
NODELOAD    3      10    1.0E6*COS(30*PI/180)  1.0E6*SIN(30*PI/180)  0.0
```

6.3 USER INPUT DESCRIPTION

This section describes the input records specific for USFOS nonlinear analysis. The following information MUST be supplied

• inelastic material properties	**MISOIEP**	(if the yield stress is not specified in the structure file)	
• control node(s)	**CNODES**		
• load control	**CUSFOS**		(static analysis)
	...or... **CICYFOS**		(cyclic analysis)
	...or... **DYNAMIC + LOADHIST + TIMEHIST**		(dynamic analysis)

Remaining input records are optional.

In general, it is recommended to locate the "USFOS control parameters" in the "control file" (the "head file")

General

Page 6.3-6

Analysis identification	* **HEAD**	1st line of text identifying the analysis
		2nd line of text identifying the analysis
		3rd line of text identifying the analysis
Print control	**CPRINT**	inprint outprint termprint
Data storage	**CSAVE**	restart result print
Save additional data for XACT presentation	* **XFOSFULL**	

Load control

Page 6.3-8

Static analysis	**CUSFOS**	nloads npostp mxpstp mxpdis
		lcomb lfact mxld nstep minstp
		lcomb lfact mxld nstep minstp
		: : : : :
		: : : : :
		lcomb lfact mxld nstep minstp
Cyclic (static) analysis	**CICYFOS**	nloads npostp mxpstp mxpdis
		lcomb lfact mxld mxdisp nstep minstp
		lcomb lfact mxld mxdisp nstep minstp
		: : : : : :
		: : : : : :
		lcomb lfact mxld mxdisp nstep minstp
Load combination	**COMBLOAD**	new_case oldcase1 fac1 oldcase2 fac2 ……
Control displacement	**CNODES**	ncnods
		nodex idof dfact
		nodex idof dfact
		: : :
		: : :
		nodex idof dfact
Static analysis, time history format	**STATIC**	end_time inc dT_res dT_term mxdisp nstep minstp
Dynamic analysis, time history format	**DYNAMIC**	end_time delta_t dT_res dT_term
Time history definition by points	**TIMEHIST**	histno Point time1 factor1 time2 factor2 ...
Time history definition by Switch	**TIMEHIST**	histno Switch dTime Factor T_start
Time history definition by S_Curve	**TIMEHIST**	histno S_Curve T1 T2 factor Power
Time history definition by Sine Curve	**TIMEHIST**	histno Sine Amp Per Phase (tStart nPer)
Time history definitions by points with const dT	**TIMEHIST**	histno ConstInc dTime fac1 fac2 fac_n
Time history definition by Two Levels	**TIMEHIST**	histno TwoLevel T1 T2 fac1 fac2 Power
Dynamic analysis, load specification	**LOADHIST**	l_case time_hist
Initialization Time	**INI_TIME**	T_ini

Dynamic Modelling Parameters

Page 6.3-19

Structural damping - Rayleigh	**RAYLDAMP**	alpha1	alpha 2					
Structural damping - time dependent	**DAMPRATIO**	Ratio 1	Ratio 2	Freq1	Freq2	History		
Damping Parameters. General	**DAMPDATA**	Damp_ID	Type	(data)				
Element Damping	**ELEMDAMP**	Damp_ID	ListType(ID-list)					
Mass formulation - lumped	**LUMPMASS**	rotmas						
Mass formulation - consistent	**CONSIMAS**							
System Damping Formulation Switch	**SYSDAMP**	Switch						
Initial velocity	**INI_VELO**	Type	Time Vx Vy Vz rVx rVy rVz Id_1 Id_2					
Dynamic result. Nodal data	**DYNRES_N**	Type	Node_ID		Dof	(Node_2		Dof_2)
Dynamic result. Element data	**DYNRES_E**	Type	Elem_ID		End	Dof	(opt)	
Dynamic result. Global data	**DYNRES_G**	Type						
Dynamic result. Xtra data	**DYNRES_X**	Type	Data					

Analysis Control Parameters

Page 6.3-27

Program parameters	**CPROPAR**	epssol	gamstp	ifunc	pereul	ktrmax	dentsw	cmax	ifysw	detersw
Switch off 2 Surface model	**SURF2OFF**									
Switch off Arc Length Control	**ARC_OFF**									
Switch off Determinant check	**DETEROFF**									
Local Dent Formulation	**DENT_TYPE**	type								
Switch off Local Dent formulation	**DENT_OFF**									
Override default max number of steps	**CMAXSTEP**	max_step								
Eigenvalue analysis	**EIGENVAL**	KeyWord		Value						
Bifurcation analysis	**CBIFURC**	idcomb	idstep	ibifsw						

Numerical Procedure Parameters

Page 6.3-31

Iteration control	**CITER**	cmin	cneg	itmax	isol	epsit	cmineg
Iteration control "Light Version"	**LITER**	itmax					
Control of repeated plastification/unloading	**CUNFAL**	max_on/off					
Time integration parameters	**CDYNPAR**	alpha	beta	gamma			
Predictor-corrector method	**PCOR_ON**						
Direct implicit time integration	**PCOR_OFF**						

Material/Plasticity Modelling

Page 6.3-33

Elasto-plastic material (beam-column)	**MISOIEP**	matno	E-mod	poiss	yield	density	therm.exp	
Control of cross section yield surface size	**GBOUND**	geono	z_y^m	z_b^m	z_y^c	z_b^c		
Nonlinear spring definition (spring property references)	**MREF**	matno	refx	refy	refz	refrx	refry	refrz
Elastic-to-plastic transition parameters	**MPLASMON**	matno	c1	a1	c2	a2	c3	a3
and hardening parameters			c4	a4	c5	a5	c6	a6
Elastic-to-plastic transition parameters	**MPLASCYC**	matno	c1	a1	c2	a2	c3	a3
and hardening parameters			c4	a4	c5	a5	c6	a6
Strain Model Data	**STRAINMOD**		KeyWord	ListTyp	IDs			

Member Modelling

Page 6.3-38

Control of plastic hinges in members	**CELHINX**	elnox	ihin1	ihin2	ihinmid				
Control of plastic hinge (alt. input)	**PLASTHIN**	ihin1	ihin2	ihinm	elnox1	elnox2	.....		
Internal Hinge of Beam Elements	**BEAMHING**	ix1	iy1	iz1	irx1	iry1	irz1		
		ix2	iy2	iz2	irx2	iry2	irz2	elnox1	elnox2 ....
Define member initial imperfection groups	**GIMPER**	impgrp	impshpe	angle	offset	dent1	dent2	dentmid	
Assign initial imperfection to element	**GELIMP**	elnox	impgrp						
Assign initial imperfection to members	**MEMBER**	imper	impgrp	ListType	IDs				
Linear dependencies/ Shim elements	**BLINDP2**	insl	iem	ix	iy	iz	irx	iry	irz
Non-structural members	**NONSTRU**	ListType			[List]				
Structural members (override NONSTRU)	**STRUCTEL**	ListType			[List]				
Linear Elements	**LIN_ELEM**	Form	ListType		[List]				
Beam type definition	**BEAMTYPE**	Type	ListType		[List]				

Analysis Calibration - Initial deformations	**CINIDEF**	Size/ColumnCurve		Pattern	LoadCase
Initial deformation from Eigenvalue analysis	**BuckMode**	EigenMode no		Scale Factor	
Grouted Members	**Grouted**	GroutMatID	ListType	{ IDList }	
Local Element Transformation Update	**ElmTrans**	Type	nod/end ListType	{ IDList }	

Joint Modelling

Page 6.3-49

Joint flexibility	**SHELL**	nodex elnox1 elnox2 d t
Overlapping braces at joint with local flexibility specified	**OVERLAP**	nodex elnob1 elnob2 ....
Joint capacity check - minimum input	**CHJOINT**	nodex elnox1 elnox2
Joint capacity check - extended input	**CHJOINT**	nodex elnox1 elnox2 geono CapRule alpha1 alpha2 alpha3 alpha4 alpha5 nbraces brace1 axial torsion Mipb Mopb brace2 axial torsion Mipb Mopb : : : : : bracen axial torsion Mipb Mopb
Joint capacity check – *MSL* char.	**CHJOINT**	nodex elnox1 elnox2 geono CapRule CapLevel Q_f
Surface Size for Joints	**JSURFSIZ**	Size_Y Size_B nodex1 nodex2 ...
User Defined Joint Gap	**JOINTGAP**	Gap NodeID (brace1 brace2 ..)
Joint Capacity Formulation	**JNT_FORM**	form_no { NodeID_List }
Joint Classification Interval	**JNTCLASS**	interval
Control of plastic hinges at nodes	**CNOHINX**	nodex hinmx

Foundation Modelling

Page 6.3-65

Spud can element	**SPUDMAT**	matno Type
	MSPUD	matno R ß γ φ c ν V_{pre} G_V G_H G_R YFSW C_8
Pile elements	**PILE**	ID Nodex1 Nodex2 Soil_ID Pile Mat Pile Geo Local_coord Imper
Pile geometries (Single Pile)	**PILEGEO**	ID Type Do T
Pile geometries (Pile Group)	**PILEGEO**	ID Type Do T nPile y_loc_1 z_loc_1 y_loc_2 z_loc_2 .... y_loc_n z_loc_n
Depth varying Pile Diam. and Thick.	**PILE_D-T**	ID Z_Mud Z_top_1 Z_bott_1 $(Do\ \ T)_1$ Z_top_2 Z_bott_2 $(Do\ \ T)_2$ .... Z_top_n Z_bott_n $(Do\ \ T)_n$
Depth varying Soil Diam.(override default)	**SOILDIAM**	ID Z_Mud Z_top_1 Z_bott_1 $(Do\ \ Dummy)_1$ Z_top_2 Z_bott_2 (Do $Dummty_2$ .... Z_top_n Z_bott_n $(Do\ \ Dummy)_n$
Soil characteristics	**SOILCHAR**	ID Type Z_Mud D_ref F_fac L_fac Z_top_1 Z_bott_1<Data>$_1$ Z_top_2 Z_bott_2<Data>$_2$ Z_top_3 Z_bott_3<Data>$_3$ .... Z_top_n Z_bott_n<Data>$_n$
Nonlinear Soil Spring Model	**SPRI_MOD**	model
Spring Damping (Dashpot)	**SPRIDAMP**	Dof C Elem_1 Elem_2……..
Automatic calculation of P-Y, T-Z and Q-Z	**API_SOIL**	ID Soiltype LoadType {Data}
Scaling of P-Y, T-Z and Q-Z for one pile	**PILEOPT**	KeyWord OptID Type {Data}

Fracture/Ductility Control

Page 6.3-80

Fracture check switch	**CFRACT**	
Fracture criterion definition	**MFRACT**	matno crit-ctod σ_u ε_u ε_sa
User defined member fracture	**USERFRAC**	elnox Type <Crit.>
User defined Damage of elements	**DAMAGE**	KeyWord {Data} ListType IDs…

Fire (Temperature Response) Analysis

Page 6.3-84

Member temperature fields	**BELTEMP**	llc elnox t_0 t_{ygrad} t_{zgrad} (PFPCrackAng)
Member temperature fields	**ELEMTEMP**	llc type [data_set] iel1 iel2 iel3 .....
Temperature fracture check switch	**CTFRACT**	
Temperature fracture criterion definition	**TFRACT**	matno T-fract Sy-fract E-fract
Temperature dependent material properties,		
- steel	**STEELTDEP**	curve_no mat no 1 mat no 2 ....
- aluminium	**ALUMTDEP**	curve_no mat no 1 mat no 2 ....
- user-defined reduction curve	**TDEPFUNC**	curve_no type [Data]
- user-defined material	**USERTDEP**	mat_no dep_E dep_yield dep_plasticdep_exp.
Load-case vs. time definition	**LCASETIM**	l_case time
Fire Degradation analysis	**PUSHDOWN**	KeyWord [Data]

Ship Impact analysis

Page 6.3-90

Ship impact	**BIMPACT**	ldcs elnox elpos energy extent xdir ydir zdir ship
Ship indentation characteristics	**MSHIP**	ship p1 p2 p3 d1
Dynamic ship impact	**DYNIMPCT**	ldcs elnox elpos V_ship Mass xdir ydir zdir NL_ship Time
Multiple impact Switch	**MULT_IMP**	(opt)

External pressure Effects

Page 6.3-95

External hydrostatic pressure	**EXTPRES**	elnox1 elnox2 elnox3 .....
Sea surface elevation	**SURFLEV**	hisurf losurf density gravity

Super-element / Sub-structure Modelling

Page 6.3-97

Super elements	**SUPERELM**	Elem ID nNodes Nodex_1 Nodex_2 .... Nodex_n Material
Sub-structure analysis	**SUBSTRU**	Matno
Shell sub-structure generation	**SUBSHELL**	Elem_ID (opt)
Shell sub structure load	**SSH_LOAD**	Lcase Elem_ID Fx Fy Fz Typ Xc Phi_c ExtentX ExtentArc
Define mesh density for pipe	**MESHPIPE**	nLength nCirc ElemID(s)
Define mesh density for box	**MESHBOX**	nLength nSide nTop nBott ElemID(s)
Define mesh density for I/H profile	**MESHIPRO**	nLength nWeb nTop nBott ElemID(s)

Miscellaneous

Page 6.3-105

'Mating' analysis/activation of members	**ACTIVELM**	ListType l_case elnox1 elnox2 ......
Total load -> Incr. load pre-processing	**TOTL2INC**	lc-start lc-end
Distributed load in local element system	**COROLOAD**	L_Case Elem_1 Elem_2 ....
Sliding Interface (contact search)	**SI**	Type nMst nSlv { Mst ID's } {Slv ID's}.
Make non-linear springs invisible	* **INVISIBLE**	ListType Id_1 Id_2 ......
Element Group definition	**GROUPDEF**	GroupID ListType [List]
Add node(s) to a given group	**GROUPNOD**	GroupID Nod_1 Nod_2 ……. Nod_n
Redefine element material	**CHG_MAT**	MatID Typ {ID list}
Redefine boundary conditions	**CHG_BOUN**	ix iy iz irx iry irz Typ {ID list}
Special Switches	**SWITCHES**	KeyWord SubKey { Value }
Scaling Spring Properties	**SPRISCAL**	KeyWord LoadCase Mat_1 Mat_2 ….
Illegal element mesh, bypass	**ILLEGAL**	KeyWord SubKey Data….

Hydrodynamics

Page 6.3-117

Wave Definition	**WAVEDATA**	l_case type height period dir phase surflev depth n
		X1 f1
		....
		Xn fn.
Current depth-profile	**CURRENT**	L_Case speed dir surflev depth Z1 f1
		....
		Zn fn.
Current *time* dependency	**CURRHIST**	Time_1 f1
		... ...
		Time_n fn.
Account for relative velocity	**REL_VELO**	nAvrg
Print of wave loads	**WAVCASE1**	l_case1 iTotal
Marine Growth profile	**M_GROWTH**	Z1 Add T1 Z2 Add T2 ......
Definition of Cd and Cm element by elem.	**HYD_CdCm**	Cd Cm Elm_1 Elm_2 ......
Defining Cd by depth profile	**HYDRO_Cd**	Z1 Cd1 Z2 Cd2 ......
Defining Cm by depth profile	**HYDRO_Cm**	Z1 Cm1 Z2 Cm2 ......
Wave Kinematics Reduction Factor	**Wave_KRF**	Factor
	Wave_KRF	Factor ListType ID's ….
	Wave_KRF	Factor Profile….
Buoyancy Switch	**BUOYANCY**	L_Case Write
Flooded Members	**FLOODED**	Elm_1 Elm_2 Elm_3 ....
	FLOODED	ListType ID's …
Wave load pre-processing	**MAXWAVE**	Criterion dT EndT Write
	MAXWAVE	Keyword LoadCase(s)
Wave load scaling (due to units)	**WAVMXSCL**	Scale
Switch ON hydro checking for all elem.	**WET_ELEM**	All
Wave load integration points	**WAVE_INT**	NIS Elem_1 Elem_2 Elem_3 ....
	WAVE_INT	NIS ListType ID's ….
	WAVE_INT	NIS Profile….
	WAVE_INT	Mesh "Z. Dist"-profile….
Current Blockage Factor	**CURRBLOC**	Type [Data]
	CURRBLOC	Profile….
Buoyancy History	**BUOYHIST**	Hist_ID ListType {IDList}
Buoyancy Formulation	**BUOYFORM**	Form ListType {IDList}
Internal Fluid / Free Surface Calculation	**INTFLUID**	Density FillTyp Hist_ID ListType {IDList}
Spool Irregular Wave to Peak	**SPOOLWAV**	TimeBeforePeak Order dT StormLength Crit
Non Hydrodynamic Elements	**NONHYDRO**	ListType {IDList}
Hydrodynamic Parameters (Powerful)	**HYDROPAR**	KeyWord Value ListType {IDList}
Hydrodynamic Nodal Mass	**HYDMASS**	Type Opt. Value. ListType {IDList}
Hydrodynamic Nodal Drag	**HYDROXTR**	Key Opt. valx valy valz ListType {IDList}

Aerodynamics

Page 6.3-146

Wind Definition New Syntax	**WINDFIELD**	L_Case Type Ux Uy Uz Zo Zbott Rho power
Wind History	**WINDHIST**	DOF Hist_ID
Wind Cross Sect Basic Coefficients	**W_COEFFS**	ID Type { Data }
Wind Cross Sect Coefficients. Combine	**W_COEFFS**	ID Combine ID_drag ID_lift ID_mom
Assign Coefficients.to Element	**ELMCOEFF**	ID ListType IDs
Wind Definition Old syntax	**WINDFIELD**	L_Case T_ini Zbott U10 power rho
		WO_x WO_y WO_z
		alpha beta gamma n_ini
Aero dynamical parameters for cross sections	**WINDPAR**	GeoID Type height width np_t np_d np_l np_m
		Ct ......
		Cd .....
		Cl ......
		Cm ...
Compute Max Wind	**MAXWIND**	Criterion dT EndT Write
Max Wind options	**MAXWIND**	Keyword lCase .

Earthquake

Page 6.3-153

Prescribed nodal displacement	**NODEDISP**	l_case node dof_code [Prescribed values for the dofs]
Prescribed nodal velocity (dynamic only)	**NODEVELO**	l_case node dof_code [Prescribed values for the dofs]
Prescribed nodal acceleration (dynamic only)	**NODEACC**	l_case node dof_code [Prescribed values for the dofs]
Prescribed Soil displacement	**SOILDISP**	l_case Type(=2) Pile_ID dof_code [values for the dofs]
Prescribed Soil acceleration (dynamic only)	**SOILACC**	l_case Type(=1) z_Top z_Bott dof_code [values for the dofs]
Prescribed spring displacement	**SPRIDISP**	l_case elem dof_code [Prescribed values for the dofs]

6.3.1 General

HEAD	1st line of text identifying the analysis 2nd line of text identifying the analysis 3rd line of text identifying the analysis
Character 9 to 80 from each line are stored as text strings This record (of three lines) is given only once	

CPRINT	inprint outprint termprint	
Parameter	*Description*	*Default*
inprint	Code to specify amount input verification print 0: Analysis identification, Key parameters and Load control data 1: + Structural data 2: + Input load data 3: + Calculated load data (i.e. gravity) 4: + Echo of unprocessed input data 5: + Internal FEM parameters	1
outprint	Code to specify amount of analysis print-out 0: Global history output 1: Results of each load step. Control node displacements Status of heavily stressed elements (Γ > -0.20) 2: + Status of heavily stressed elements (Γ > -0.50) 3: + Status of all elements (Γ > -1.00) 4: + All nodal displacements	2
termprint	Code to specify output to terminal / batch log file 0: Screen output adapted to interactive running 1: Output adapted to batch running. The global history output is written to the batch log file	1

This record controls the amount and format of print to terminal and to the Analysis Print File (Sect 8.1)

The data available for post-processing are unaffected by this record

This record is given only once

CSAVE n m k

Parameter	Description	Default
n	Restart data is stored every "n'th" load step of each load case/combination	0
m	Result data is stored every "m'th" load step of each load case/combination	-10
k	Result data is printed at the 'out-file' every "k'th" step of each load case/combination	1

This record is used to specify storage of restart and result data

If this record is skipped, restart + result data are stored for every step

If *n* = 0 then no restart data will be stored
If *n* = -i then restart data will be stored every i'th step plus at the end of each load specification, (CUSFOS/CICYFOS-line)

If *m* = 0 then no result data will be stored
If *m* = -j then result data will be stored every j'th step plus at the end of each load specification, (CUSFOS/CICYFOS-line)

If *k* = 0 then no result data will be printed at the out-file
If *k* = -k then result data will be printed every k'th step plus at the end of each load specification, (CUSFOS/CICYFOS-line)

NOTE ! In connection with **dynamic** analysis the parameters n, m and k are 0/1 switches. The time increment between result saving is controlled on the DYNAMIC record.
By default the switches are set to [0 1 1].

This record is given only once.

XFOSFULL

Parameter	Description	Default
	If this record is specified, (no parameters), <u>all</u> available data are stored on the RAF-file for inspection by XACT. By default, (to save disc space), some result types are not stored.	

With this record the user may specify that all available result data should be stored on the RAF-file.
This record is given once.

6.3.2 Load control

```
CUSFOS    nloads   npostp  mxpstp  mxpdis
          lcomb    lfact   mxld    nstep   minstp
          lcomb    lfact   mxld    nstep   minstp
            :        :       :       :       :
            :        :       :       :       :
          lcomb    lfact   mxld    nstep   minstp
```

Parameter	*Description*	*Default*
nloads	Number of load specifications (lcomb + lfact + mxld + nstep + minstp). Used in connection with restart of analyses (Sect 4.7)	
npostp	Number of load steps in the post collapse range	
mxpstp	Max load step size in the post collapse range	
mxpdis	Max incremental displacements in the post collapse range. Suggested value is 1.0	
lcomb	Load combination or load case number	
lfact	Load factor. Size of the initial (un scaled) load increment specified as a factor of the reference load	
mxld	The current load vector is repeated until the accumulated load reaches the relative load level *mxld*, specified as a factor of the reference load	
nstep	... OR until the current load vector has been incremented *nstep* times. Both *mxld* and *nstep* may be specified at the same time; the load is incremented until either of the conditions is satisfied. For a zero value of *mxld* or *nstep* no test of upper limit is performed	
minstp	Minimum load increment for automatic load step scaling. Specified as a fraction of the total load combination or load case. (Not multiplied by *lfact*). Suggested value is 0.001**mxld*, where *mxld* is the maximum of the *mxld*'s within the actual load combination	

This record is used to specify the loading history, with load and displacement control parameters (Sect 4.1 and Sect 4.2.2)

Note! In a restart analysis the load history up to the step from which calculations resumes must NOT be included in the CUSFOS record

This record is given only once and no default values exist

CICYFOS	nloads	npostp	mxpstp	mxpdis		
	lcomb	lfact	mxld	mxdisp	nstep	minstp
	lcomb	lfact	mxld	mxdisp	nstep	minstp
	:	:	:	:	:	:
	:	:	:	:	:	:
	lcomb	lfact	mxld	mxdisp	nstep	minstp

Parameter	*Description*	*Default*
nloads	Number of load specifications (lcomb + lfact + mxld + nstep + minstp). Used in connection with restart of analyses (Sect 4.7)	
npostp	Number of load steps in the post collapse range	
mxpstp	Max load step size in the post collapse range	
mxpdis	Max incremental displacements in the post collapse range. Suggested value is 1.0	
lcomb	Load combination or load case number	
lfact	Load factor. Size of the initial (un scaled) load increment specified as a factor of the reference load	
mxld	Max load level for current load vector (Sect 4.1)	
mxdisp	Max total displacement for current load vector	
nstep	Max number of load steps for current load vector	
minstp	Minimum load increment for automatic load step scaling. Specified as a fraction of the reference load combination or load case. Suggested value is 0.0001**mxld*, where mxld is the max in the actual load combination	

This record is used to specify the loading history, with load and displacement control parameters (Sect 4.1 and Sect 4.2.2). The current load vector (Sect 4.1) is incremented until the accumulated load reaches the load level defined by *mxld*, **OR** until the total displacement reaches the displacement level defined by *mxpdis*, **OR** when the load vector has been incremented *nstep* times

If a zero value is given for one (but not all) of the *mxld*, *mxdisp* or *nstep* parameters, that parameter is disregarded

To run an input check, simply specify zero values for *mxld*, *mxdisp* and *nstep*. That is, the specified load vector will be incremented **zero** times

Note! In a restart analysis the load history up to the step from which analysis resumes must be excluded. This record is given only once and no default values exist

COMBLOAD	**Comb_Case**	
	L_Case_1	$Factor_1$
	L_Case_2	$Factor_2$
	L_Case_3	$Factor_3$
	L_Case_n	$Factor_n$

Parameter	*Description*	*Default*
comb-case	The result of the combination is collected in a load case with number (ID) = Comb_Case. This case is referred to in USFOS (CUSFOS or LoadHist). Original definition of this case (if any) is override.	
L_case_1 $Factor_1$	First Load Case contributing to the combination. The loads in this load case are multiplied with $Factor_1$ etc.	

With this record, the user defines a load combination to be generated and referred to in a USFOS analysis. This option replaces the old CCOMB option, (which had limitation on number of cases and no individual scaling available).

CombLoad is executed during reading of the user input before the analysis begins.

NOTE! :

When CombLoad is used, all ***read-in*** load cases are ***deleted*** after this operation. Also the Load case ID's are cleared and cannot be referred to from other commands.

CombLoad *cannot* be used for *generated* loads (f ex hydrodynamic loads)

CombLoad operates on *basic, read-in* loads only, (NodeLoad, BeamLoad, Pressure, Gravity).

Example:

```
'          New Case  OldCase  Factor
COMBLOAD       3        4       1.3
                        3       2.0
                       10       0.7
                       13      -0.55
```

Generates a *new* load case 3 as follows:

l_case3 = (l_case4 * 1.3 + l_case3*2.0 + l_case10*0.7 - l_case13 *0.55)

The ***original*** loads, (case 4, 3, 10 and 13), are ***cleared*** after all combinations are processed. **Only Load Case 3 will exist** in the USFOS analysis.

This record may be repeated.

CCOMB lcomb ldcs1 ldcs2 ldcs3

Parameter	*Description*	*Default*
lcomb	Load combination number	
ldcs1	First load case to enter the load combination	
ldcs2	Second load case to enter the load combination	
ldcs3	Third load case to enter the load combination	

This record is used to combine input load cases into load combinations. In nonlinear analyses, the load responses may **NOT** be superposed. Any superposition must apply to the loads themselves (Sect 4.1)

If no **CCOMB** records are given, the program will use the input load case numbers. However, if this record is used once, then all load specification will refer to load combination numbers and not to input load case numbers

This record may be repeated

Note! Maximum 3 load cases per load combination

NOTE 2!! This command should be replaced by the new CombLoad option.

```
CNODES   ncnods
         nodex   idof   dfact
           :       :      :
         nodex   idof   dfact
```

Parameter	*Description*	*Default*
ncnods	Number of load control nodes. Used in connection with restart analyses (Sect 4.7)	
nodex	External (user-specified) number of control node	
idof	Global degree of freedom to be used in the control displacement 1: X - displacement 2: Y - displacement 3: Z - displacement	
dfact	Weight factor of the specified DOF	

This record is used to specify the Control Displacement of the structure. The Control Displacement is calculated as a balanced average of the specified DOFs multiplied with their respective weight factors (Sect 4.2.2)

Also, the global displacements of these nodes are printed to the Analysis Control File (outprint > 0) (Sect 8.1.4)

If only print is required at selected nodes, the nodes may be specified with *dfact* = 0.0
This record is given only once

STATIC End_Time Delta_T dT_res dT_term mxdisp nstep minstp

Parameter	*Description*	*Default*
End_Time	Continue the static analysis with the specified time increment to the specified 'End_Time' is reached.	
Delta_T	Time (load) increment to be used until 'End_Time' is reached.	
dT_res	Time between saving of results on the 'RAF'-file	10*D_T
dT_term	Time between print to terminal	D_T
mxdisp	Max total displacement	0.0
nstep	Max number of steps	0
minstp	Minimum load increment	0.001

This record is used to specify static initialization of a dynamic analysis with all loading controlled through time.

NOTE ! This option should be used only when the structure responds almost *linearly* (without yielding / step scaling)
and ***should not*** be used for pushover analysis.

Example:

```
STATIC     1.0    0.100   0.50   0.1
DYNAMIC   10.0    0.020   0.50   0.1
```

means that the first second is used to apply for example deadweight load cases with no inertia effects accounted for. From time = 1.0 the analysis is transferred to an ordinary dynamic analysis.

Mxdisp is not defined which means that this criterion is not active.
Nstep is not defined which means that 'infinite' number of steps could be used to reach time=1.0.
Minimum step size is 0.001.

This record may be repeated.

DYNAMIC End_Time Delta_T dT_res dT_term

Parameter	*Description*	*Default*
End_Time	Continue the dynamic analysis with the specified time increment to the specified 'End_Time' is reached.	
Delta_T	Time increment to be used until 'End_Time' is reached.	
dT_res	Time between saving of results on the 'RAF'-file	10*D_T
dT_term	Time between print to terminal	D_T

This record is used to specify dynamic analysis with all loading controlled through time.
This record may be repeated.

Example:

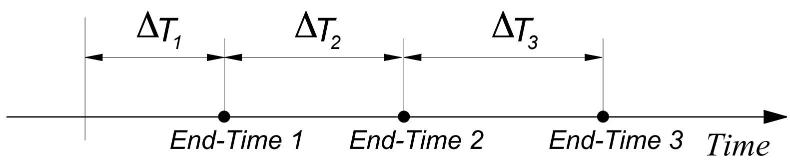
```
DYNAMIC   0.1    0.001   0.10   0.1
DYNAMIC   1.0    0.010   0.50   0.1
DYNAMIC  10.0    0.020   0.50   0.1
```

means that the dynamic analysis is started with a time increment of 0.001s which is kept until time = 0.1s is reached. Then the time increment is increased to 0.010s, which again is changed to 0.020 at time 1.0. The analysis will terminate at time = 10.0s.

Terminal print is updated each 0.1'th second, and results are saved each 0.5'th second.

In connection with the DYNAMIC option, the global step counter is updated each time the results are saved, and the limitation of maximum steps (ref CMAXSTEP record) is affected by the total number of **saved** steps only. In the example, the total number of saved steps is 1+2+18=**21** (default maximum steps to be saved is 512).
However, total number of (internal) analysis steps is: 100+90+450 = 640.

Reduced saving of results reduces both the disc space requirements and the CPU time.

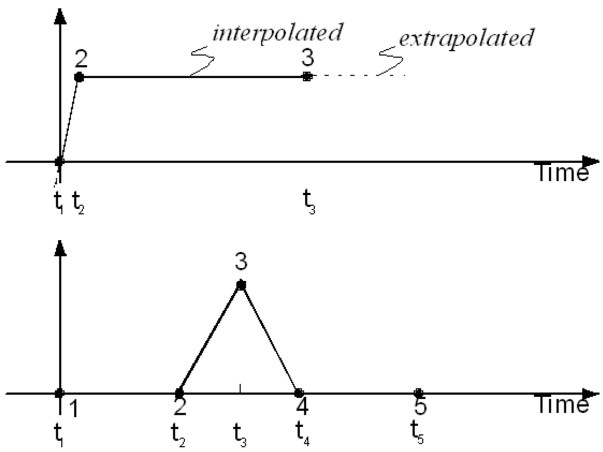


TIMEHIST histno Points time1 factor1 time2 factor2 ...	
Parameter	*Description*
histno	Time history number (user defined ID)
Points	Time History Type: Discrete points
time1 factor1	time (seconds) Scale factor 1
time2 factor2	time (seconds) Scale factor 2

With this record, the user specifies a time history by discrete points.

Values between the tabulated points are interpolated. Values outside the specified range are extrapolated as indicated in the figure below.

This record may be repeated.



TIMEHIST histno Switch dTime factor Tstart

Parameter	*Description*
histno	Time history number (user defined ID)
Switch	Time History Type "switch"
dTime	Time between calculations of forces, (see WAVEDATA). dTime=0 means that loads are calculated every analysis time step.
factor	Forces calculated from wave/current/wind are multiplied with this factor (default=1.0)
Tstart	Time for switching ON the actual environmental load (default Tstart=0)

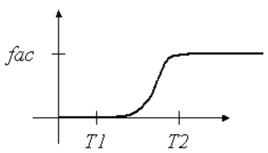
With this record, the user specifies a time history used to control the time for applying the wave/current/wing loads. The time between recalculation of the forces is controlled in this command as well as force scaling factors.

This record may be repeated.

TIMEHIST histno *S_Curve* T1 T2 (factor power)

Parameter	*Description*	*Def*
histno	Time history number (user defined ID) TimeHist Type = 4 (S_Curve)	
S_Curve	History Type: S shaped curve	
T1	Time for activation this history. (Curve value = Zero for time < T1)	
T2	Time for full value . (Curve value = Factor for time > T2)	
factor	Actual curve value for time > T2	1
power	power = 1 : Linear curve (straight line) from T1 to T2 power = 2 : Second order (s-shaped) curve from T1 to T2 power = 3 : 3rd order (s-shaped) curve from T1 to T2	2

With this record, the user specifies a time history used to control the time for applying for example deadweight and buoyancy gradually.



This record may be repeated.

TIMEHIST histno *ConstInc* dTime Fac1 Fac2 Facn …

Parameter	*Description*
histno	Time history number (user defined ID)
ConstInc	TimeHist Type: Constant increment
dTime	The *constant* time increment between the tabulated values
Fac 1 Fac 2	Scale factor 1 Scale factor 21
Fac n	Scale factor n

With this record, the user specifies a time history by discrete points, where the time between each point is *constant*. (This is a special version of the "Points" time history type) and is recommended for extreme long time series, (reduced memory requirement and faster simulation).

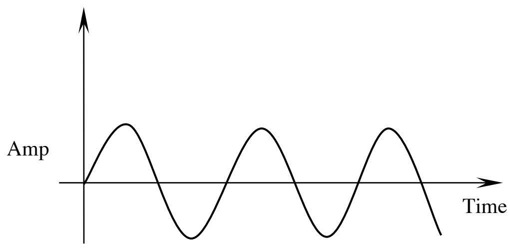
Values between the tabulated points are interpolated, values outside the specified range are extrapolated..

This record may be repeated.

TIMEHIST histno *Sine* Amp Period Phase (tStart nPer)

Parameter	*Description*	*Def*
histno	Time history number (user defined ID) TimeHist	
Sine	History type: Sine Function	
Amp	Amplitude	
Period	Period	
phase	Phase (degrees)	
tStart	Start time for sine function. (Function value = 0 for T< tStart)	0
nPer	Number of Periods to apply	∞

With this record, the user specifies a time history following a sine function.



This record may be repeated.

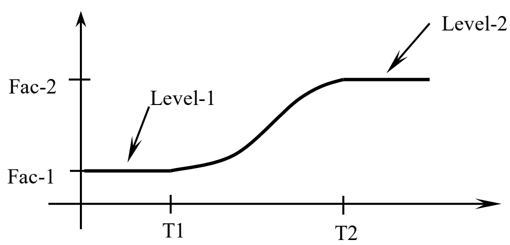
TIMEHIST histno *TwoLevel* T1 T2 fac1 fac2 (power)

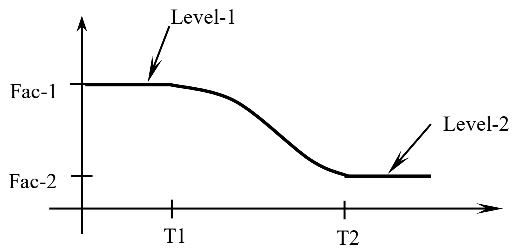
Parameter	*Description*	*Def*
histno	Time history number (user defined ID) TimeHist Type = 7 (S_Curve)	
TwoLevel	History type: Two Level with smooth transition	
T1	Time for activation of transition. (Curve value = *fac1* for time < T1)	
T2	Time for completed transition. (Curve value = *fac2* for time > T2)	
fac1	Curve value for time < T1	
fac2	Curve value for time > T2	
power	power = 1 : Linear curve (straight line) from T1 to T2 power = 2 : Second order (s-shaped) curve from T1 to T2 power = 3 : 3^{rd} order (s-shaped) curve from T1 to T2	2

With this record, the user specifies a time history, which has a smooth transition from one level to another. The transition curve is by default a 2^{nd} order curve. The levels may change from low to high or opposite as shown in the figures.

This curve is typically used for changing material properties (scaling of springs), etc.

NOTE!
This curve *cannot* be used to control forces since the force histories have to start with value=zero at time zero.





This record may be repeated.

LOADHIST L_Case TimeHist

Parameter	*Description*	*Default*
L_Case	Load case number of the load vector(s) to be activated according to the specified 'Time History'.	
TimeHist	Time History ID, which is used to scale the specified load vector(s).	

This record is used to specify the loads to be activated during a dynamic analysis with all loading controlled through **time**. The different time histories define when the loads connected to these time histories should be activated/scaled/deactivated.

The load vectors are multiplied, (scaled) with the actual time history-scaling factor at the time. If the scaling factor is 0, the actual load is not activated, (see 'TIMEHIST' definition).

This record may be repeated.

Example:

```
LOADHIST    1    10
LOADHIST    5    10
LOADHIST    7    1000
```

means that both load vectors 1 and 5 are activated according to time history with ID = 10 while load vector 7 is controlled through time history 1000.

INI_TIME Time

Parameter	*Description*	*Default*
Time	Initialization time.	0.0

This record is used to specify an initialisation phase used f ex in connection with analyses of slender structures. The “USFOS clock” is reset to zero when the time is reached, together with the structural displacements, which also are set to zero. The initialisation phase is removed from the result file (raf file), and the initialised position becomes the un-deformed reference position.

.
This record is given once.

6.3.3 Dynamic Modelling Parameters

RAYLDAMP alpha1 alpha 2

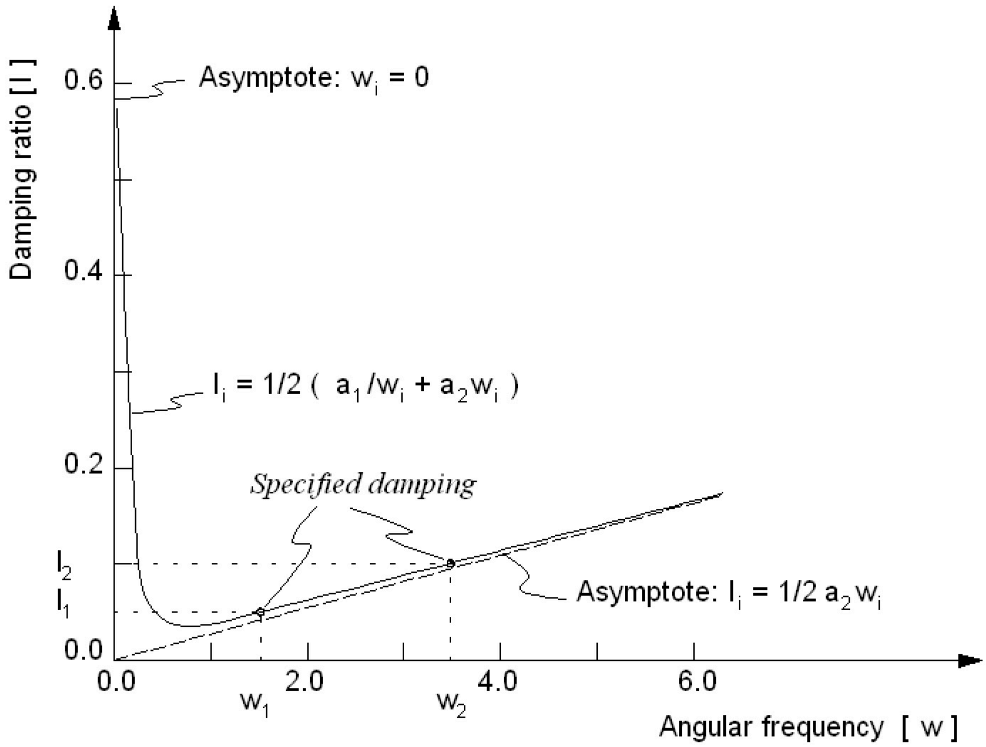
Parameter	*Description*	*Default*
alpha1	Mass proportional damping coefficient	
alpha2	Stiffness proportional damping coefficient	

This record is used to control structural damping according to the Rayleigh damping model

If the damping ratio is known for two individual structural frequencies, the damping parameters may be chosen as follows:

$$\alpha_1 = \frac{2\,\omega_1\omega_2}{\omega_2^2 - \omega_1^2}(\lambda_1\omega_2 - \lambda_2\omega_1)$$

$$\alpha_2 = \frac{2(\omega_2\lambda_2 - \omega_1\lambda_1)}{\omega_2^2 - \omega_1^2}$$



DampRatio Ratio_1 Ratio_2 Freq_1 Freq_2 History

Parameter	*Description*	*Default*
Ratio_1 Ratio_2	Damping ratio at frequency 1 (1% damping is given as 0.01) Damping ratio at frequency 2	
Freq_1 Freq_2	Frequency 1 (given in Hz) Frequency 2 (given in Hz)	
History	Time history (optional) scaling the actual damping ratios as a function of time. If omitted (or set equal to 0) the damping is kept constant during the analysis.	0

With this record the user may specify the damping to be used in a Dynamic Analysis. Rayleigh damping coefficients (alpha_1 and alpha_2) are calculated on basis on the given damping ratios and associated frequencies.

Example 1

```
'           d1     d2      fq1    fq2
DampRatio   0.02   0.01    0.1    10
```

means 2% damping at 0.1 Hz (T=10s) and 1% damping at 10 Hz. Constant damping ratios during the analysis.

Example 2

```
'            d1      d2      fq1     fq2     Hist
DampRatio    0.01    0.01    0.1     10      101
‘
'        ID   Typ      t1    f1       t2    f2  ……......................
TimeHist 101  points   0.0   1.0      10    1.0  11.0  5.0   12 .0  1.0   100.0  1.0
```

means 1% damping at 0.1 Hz (T=10s) and 1% damping at 10 Hz. Damping ratios are scaled according to time history with id = 101, which means increased damping between time = 10 and time = 12. Peak damping is 5% at time=11s.

NOTE ! This record is an extended alternative to the RAYLDAMP input, and override data specified under the RAYLDAMP input.

This record is given only once.

DampData Damp_ID Type [Data]

Parameter	Description	Default
Damp_ID Type	ID to be referred to from ELEMDAMP Kind of Damping data. Actual Types: Rayl_All : Common data for all 6 dofs. Data: alpha1 & alpha2 Rayl_Ind: Individual data for the dofs. Data: alpha1 & alpha2 (Dof1) alpha1 & alpha2 (Dof6) Alfa1 and 2: Rayleigh Coefficients (see under RaylDamp for more info)	

With this record the user may specify general damping to be referred to from individual elements used in a Dynamic Analysis.

<u>*Example 1*</u>

```
'            DampID   DampType      alpha1   alpha2
DampData     10010     Rayl_All  .    0.0      3.0E-4
DampData     10011     Rayl_All  .    1.0E-2   3.0E-4
```

Two damping data packets are defined. One with stiffness proportional damping only and one with both mass- and stiff proportional damping.

This record could be repeated.

ElemDamp Damp_ID ListType [Id-List]

Parameter	Description	Default
Damp_ID ListType ID-List	Actual Damping Property ID Definition of ID list *Element* : Element ID list *Material* : Material ID list *All* : All elements are given the actual damping properties Actual List of ID's	

With this record the user may specify general damping to be referred to from individual elements used in a Dynamic Analysis..

<u>*Example 1*</u>

```
'            DampID   ListType   IDList
ElemDamp     10010     All                          ! Assign Damp Prop 10010 to all elements
ElemDamp     10011     Mat        200       300     ! Assign to special thereafter
```

With this record, damping for each element is defined.

NOTE ! This record is an extended alternative to the RAYLDAMP input, and overrides data specified under the RAYLDAMP input.

This record could be repeated.

LUMPMASS rotmas

Parameter	*Description*	*Default*
rotmas	Rotational mass scaling factor	0.01

This record is used to prescribe use of a lumped mass formulation for the structural finite element in connection to dynamic analysis

If *rotmas* is set to zero the mass terms associated with rotation at the element nodes are zero, ref USFOS Theory Manual Sect 14

This record is given only once.

CONSIMAS

This record is used to describe use of the consistent mass formulation ref USFOS Theory Manual Sect 14

This record is given only once.

SYSDAMP [Switch]

Parameter	*Description*	*Default*
switch	Switch ON/OFF: Switch = 0 : Switch OFF system damping Switch = 1 : Switch ON system damping	0

This record is used to switch ON system damping formulation, (forming a separate $\mathbf{C}_0$ matrix). By default (if SYSDAMP is not specified in the input), $\mathbf{C}_0$ is not established, and only Rayleigh system damping is available.

It is recommended to switch SYSDAMP ON, if discrete dashpot dampers are defined, (the dashpot damping terms will be added to $\mathbf{C}_0$ matrix on top of the usual Rayleigh coefficients).

This record is given only once.

Ini_Velo Type Time Vx Vy Vz rVx rVy rVz Id_1 Id_2 ....

Parameter	*Description*	*Default*
Type	Data type used to specify initial velocity at nodes: Node : The specified Id's are node numbers which all should be assigned to the initial velocity at the specified time. Mat : The specified Id's are material numbers. All nodes "in contact" with elements with the specified material numbers will all be assigned to the initial velocity at the specified time.	
Time	Time for the velocity to be initialized	
Vx Vy Vz rVx rVy rVz	Velocity X-component Velocity Y-component etc.... Velocity component: rotation about X-axis etc...	
Id_1 Id_2	Node number 1 (if Type is Node) or Material no. 1 (if Type is Mat) Node number 2 (if Type is Node) or Material no. 2 (if Type is Mat)	

With this record the user may specify initial velocity for a Dynamic Analysis.

Example 1 :

```
'         Typ  Time        Velo            Mat_ID
Ini_Velo  Mat  0.0  1.5   0  0   0  0  0   1001
```

will assign a X-velocity of 1.5 m/s to all nodes on the "body" with material ID 1001 from time= 0.0.

Example 2:

```
'         Typ  Time        Velo            Mat_ID
Ini_Velo  Mat  10.0 0  2  0   0  0  0      2002
```

Will assign a Y-velocity of 2.0 m/s to all nodes on the "body" with material ID 2002 at time 10 seconds.

This record may be repeated.

Dynres_N Type Node_ID Dof (Node_2 Dof_2)

Parameter	*Description*	*Default*
Type	Actual Node result type: Disp : Nodal Displacement Vel : Nodal Velocity Acc : Nodal Acceleration RelDis : Relative displacement between 2 nodes/dofs, (extra node and dof required)	
Node_ID	Actual Node ID of which dynamic results should be saved	
Dof	Actual Nodal degree of freedom: 1 : X – displacement 2 : Y – displacement etc...	
Node_2	Second node (relevant if RelDis only)	
Dof_2	Second node degrees of freedom (relevant if RelDis only)	

With this record the user may specify *nodal* quantities to be saved **every** step during a Dynamic Analysis independent on the 'RAF'-file saving interval. Results are stored on a separate file named <prefix>**.dyn**, and these time histories are accessed from XACT, (or dynres/dynmax).

This record may be repeated.

Dynres_E Type Elem_ID End Dof (opt)

Parameter	*Description*	*Default*
Type	Actual Element result type: Disp : Element Displacement Force : Element Force	
Elem_ID	Actual Element ID of which dynamic results should be saved	
End	Element end	
Dof	Actual *local element* degree of freedom: 1 : X - displacement / Axial force 2 : Y - displacement / Shear force, y-direction If the element is a shell element, see description below.	
Opt	If the element is a Shell Element, the forces could be projected into the local coordinate system of a beam element. "Opt" is then the beam element ID.	0

With this record the user may specify *element* quantities to be saved **every** step during a Dynamic Analysis independent on the 'raf'-file saving interval. Results are stored on a separate file named <prefix>.dyn, and these time histories are accessed from XACT.

Dynres forces for Beam Elements always refer to local beam coordinate system.

Dynres forces for Shell Elements could be referred as follows:
'
- Forces are referring to Global Coord system (default)
- Forces are referring to the Local Beam axis of a reference Beam Element

'
When forces are referring to a Beam, the specified DOF refers to The Beam's Local coordinate system:
'
Example :
```
'              Type    elem    end  Dof  (BeamID)
 DynRes_Elem    Force    100      3   1       1   ! Shell Force Projected on Beam
```

The force component in the beam X-axis is used. '
Element 'end' means local shell element node.

This record may be repeated.

Dynres_G Type

Parameter	*Description*	*Default*
Type	Actual Global result type: Wint : Internal Energy Wext : External Energy Wplast : Internal Plastic Energy Wkin : Kinetic Energy Wtot : Total Energy WaveLoad : Total Wave Load Forces WaveOVTM : Wave Overturning moment. ReacBSH : Base Shear Reaction. ReacOVTM : Reaction Overturning Moment WaveElev : Surface Elevation ReacXDir : Reaction force. Global X- Direction ReacYDir : Reaction force. Global Y- Direction ReacZDir : Reaction force. Global Z- Direction ReacXMom : Reaction Moment about Global X axis ReacYMom : Reaction Moment about Global Y axis SysMassX : System Mass. X-degree of freedom. Similar for Y, Z.	

With this record the user may specify *global* quantities to be saved **every** step during a Dynamic Analysis independent on the 'raf'-file saving interval. Results are stored on a separate file named <prefix>.dyn, and these time histories are accessed from XACT for time history and frequency distribution presentation.

This record may be repeated.

Dynres_X Type Param

Parameter	*Description*	*Default*
Type	Actual Xtra Data: *TimeHist : Plot of Time History definitions (user's input)*	
Param	Actual parameter	

With this record the user may specify *special extra* data to be saved **every** step during a Dynamic Analysis independent on the 'raf'-file saving interval. Results are stored on a separate file named <prefix>.dyn, and these time histories are accessed from XACT for time history and frequency distribution presentation.

This record may be repeated.

Example-1

```
'              Type      ID
 Dynres_X   TimeHist   101   !  Store time history 101
 Dynres_X   TimeHist   201   !  Store time history 201
```

6.3.4 Analysis Control Parameters

NOTE !

This record is "Historic". Use the specific Commands on the next pages instead !

CPROPAR epssol gamstp ifunc pereul ktrmax dentsw cmax ifysw detersw

Parameter	*Description*	*Default*
epssol	Numerical accuracy of the equation solver (Sect 4.6)	1.0E-20
gamstp	Accepted value for overshooting the yield surface (Sect 4.2.2)	0.10
ifunc	FEM beam shape function (Sect 3.1) 1: Sine/cosine shape functions 3: 3rd degree polynomials 2: 3rd degree polynomials used when $P_x \le P_{Euler}$*_pereul_ Sine/cosine functions used when $P_x > P_{Euler}$*_pereul_	2
pereul	Level of transition from 3rd degree polynomial shape function to sine/cosine shape function. Specified as a factor of the Euler buckling load (Sect 3.1)	0.05
ktrmax	Max number recalculations of one load step due to element unloading (Sect 4.1)	5
dentsw	Formulation of dented tubes and local buckling (Sect 3.4) 0: Only initially perfect cross section included 1: Local damages and dent growth included for tubes. Local buckling of initially perfect rectangular and tube cross section included	1
cmax	Elastic spring-back is introduced when the current Stiffness Parameter exceeds *cmax* (Sect 4.2.1)	999
ifysw	Parameter for elasto-plastic transition 0: Elasto-plastic transition performed 1: Elasto-plastic transition not performed	0
detersw	Determinant criterion for identification of load limit points or bifurcation points 0: No determinant check 1: Determinant criterion active. Load increment is reversed when the determinant changes sign	1

This record were used to change default program parameters on early versions of USFOS.

NOTE!
This command is **not** recommended. Instead, use the specific commands defined on the next pages.

This record is given only once

SURF2OFF		
Parameter	*Description* **Switch off** two-surface model, (Elasto-plastic transition not performed) This record overrides default data and data specified on CPROPAR record	*Default* ON

ARC_OFF		
Parameter	*Description* **Switch off** arc length control. This record overrides default data and data specified on CPROPAR record	*Default* ON

DETEROFF		
Parameter	*Description* **Switch off** determinant check. This record overrides default data and data specified on CPROPAR record	*Default* ON

DENT_OFF		
Parameter	*Description* **Switch off** formulation of dented tubes and local buckling. This record overrides default data and data specified on CPROPAR record	*Default* ON

DENT_TYPE Type

Parameter	*Description*	*Default*
Type	Dent formulation type. 0 : Default dent formulation Cyclic dent formulation	0

This record may be used to switch on the cyclic dent formulation.

This record is given once

CMAXSTEP Max_step

Parameter	*Description*	*Default*
Max_step	Max number of steps	2048

This record may be used to increase the max number of analysis steps available.
This record is given once.

EIGENVAL KeyWord Value

Parameter	*Description*	*Default*
Keyword	Keyword describing the actual "Value":	
	Time : Definition of the time when the eigenvalue is performed.	-
	NumberOf : Specification of number of eigenvector to compute.	20
	ModeScale : Scaling of eigenvectors in connection with visualization.	1.0
	Algorithm : Actual algorithm to use. (SubSpace or Lanczos)	SSIT
	Shift : Shift parameter used if singular systems.	0.0
Value	Actual value	

This record is used to perform an eigenvalue analysis for calculation of vibration frequencies and corresponding modes, (Ref: *EigenSolvers for structural problems* by ***K.Bell***). The eigenvalues/modes are calculated, and the mode-shapes may be inspected in XACT. If the modes are difficult to see (small displacements), specify the modescale, f ex wit ha value=10.

NOTE ! Specification of time is important. In order to get correct stresses (and thus correct nonlinear stiffness), the eigenvalue calculations should be performed when the self-weight etc. is introduced in the structure.

Available for **DYNAMIC** analysis only

This record could be repeated.

CBIFURC idcomb idstep ibifsw

Parameter	*Description*	*Default*
idcomb	Load combination to be reached before the bifurcation analysis becomes active	
idstep	Load step to be reached before the bifurcation analysis becomes active	
ibifsw	= 0 or <blank>: Eigenvectors and eigenvalues are calculated and written to the print file at the load step where bifurcation is detected. The .bif file is generated with nodal loads corresponding to the eigenvectors = 1: In addition the eigenvectors are written to the result file for XACT presentation and the analysis is then terminated. Each eigenvector is stored as global displacements for dummy load steps which follow the last real load step in consecutive order	0

This record may be used switch on the bifurcation analysis

6.3.5 Numerical Procedure Parameters

CITER cmin cneg itmax isol epsit cmineg

Parameter	*Description*	*Default*
cmin	Iterations omitted for current stiffness value between *cmin* and *cmineg*	0.000
cneg	Definition of "large" negative current stiffness	-2.0
itmax	Max number of iterations	10
isol	Number of iterations between updating of stiffness matrix	1
epsit	Convergence criterion for iterations	0.0001
cmineg	Iterations are omitted for current stiffness value between *cmin* and *cmineg*	-cmin

This record may be used to switch on iteration, and to change default program parameters

Note! Iterations should **not** be performed if the SHELL option is activated.

This record is given once

LITER [itmax]

Parameter	*Description*	*Default*
itmax	Max number of iterations	1

This record may be used to switch on "Light" iteration, with possibility to change the itmax only.
The other parameters are fixed, ("CITER" default parameters, see above).

Note! Iterations should **not** be performed if the SHELL option is activated.
This record is given once, and if CITER is specified (before LITER), these parameters are kept, (except itmax).

CUNFAL max on/off

Parameter	*Description*	*Default*
max on/off	Max number of subsequent load steps with plastification/elastic unloading of a single element	

This record impose restrictions on repeated plastification/elastic unloading of single elements

If an element unloads/re-plastifies in more than *max on/off* **subsequent** load steps, elastic unloading will be suppressed in the remaining steps

The restriction is removed the first time the element goes through a load step **without** trying to unload. The restriction is also removed on the first load step of each new load vector (Sect 4.2), e.g. if the external load is reversed

This record is given only once

CDYNPAR alpha beta gamma		
Parameter	*Description*	*Default*
alpha	Integration parameter controlling high frequency numerical damping	-0.1
beta	Newmark integration parameter	0.3*)
gamma	Newmark integration parameter	0.6*)
	To obtain high frequency numerical damping, select alpha in the range -0.3 to 0.0. The accuracy in the integration of the equation of motion is maintained for this case by setting both beta and gamma negative. USFOS will then automatically select the Newmark integration parameters to: $\gamma = 1/2(1-2\alpha)$ $\beta = 1/4(1-\alpha)^2$ $-0.3 \leq \alpha \leq 0.0$ **) Computed values based on the default alpha*	
This record may be used to specify numerical integration parameters for the time integration of the equation of motion according to the Hilber, Hughes and Taylor alpha-dissipation method, ref USFOS Theory Manual, Sect 14		

PCOR_ON		
Parameter	*Description*	*Default*
	Predictor-Corrector approach used in the numerical integration of the equation of motion. The solution of the equation of motion at a typical time-step is then performed as a two-step procedure: 1. Predict increments in velocity and acceleration 2. Perform corrector steps. One or several corrector steps may be specified on the CITER record	
This record may be used to activate the Predictor-Corrector approach in the numerical time integration of the equation of motion *Note!* This record may only be used in combination with the CITER record		

PCOR_OFF		
Parameter	*Description*	*Default*
	Predictor-Corrector approach NOT used when solving the equation of motion by means of numerical integration.	

6.3.6 Material/Plasticity Modelling

MISOIEP	matno	E-mod	poiss	yield	density	term.exp
	c1	a1	c2	a2	c3	a3
	c4	a4	c5	a5	c6	a6

Parameter	*Description*	*Default*
matno	Material reference number	
E-mod	Elastic modulus	
poiss	Poisson's ratio	
yield	Yield strength	
density	Mass density	
term.exp	Thermal expansion ratio	
c1	Hardening parameter for x-force	$2.0{\cdot}10^{-3}$
a1	Elasto-plastic growth parameter for x-force	0.25
c2	Hardening parameter for y-force	$2.0{\cdot}10^{-3}$
a2	Elasto-plastic growth parameter for y-force	0.25
.		
.		
.	Ditto for z-force	
c6	x-moment	
a6	y-moment	
	z-moment	

This record is used to specify material properties when USFOS is used as a stand alone program or if the "host" analysis system does not supply all parameters (i.e. yield strength), or if hardening properties will be changed

The default hardening properties refer to the condition where elasto-plastic transition is performed, see record **Surf2Off**. If elasto-plastic transition is **not** performed, the default values are zero

c2-c6 are not given if they are equal to c1
a2-a6 are not given if they are equal to a1

This record may be repeated

GBOUND geono z_y^m z_b^m z_y^c z_b^c

Parameter	*Description*	*Default*
geono	Geometry number	
z_y^m	Extension ("size") of the yield surface, relative to the bounding surface (Sect 4.3.2). Material behaviour under monotonous loading	
z_b^m	Extension ("size") of the bounding surface ("full plastic capacity") (Sect 4.3.3). Material behaviour under monotonous loading	
z_y^c	Extension of the "cyclic" yield surface, relative to the "cyclic" bounding surface. (Material behaviour under cyclic loading)	
z_b^c	Extension of the "cyclic" bounding surface. (Material behaviour under cyclic loading)	

This record is mandatory for general beams. For other cross sections it may be used to change default values

Default z-yield values are determined by the ratio of elastic and plastic section modulus of typical cross sections

		z_y^m	z_b^m	z_y^c	z_b^c
I or H:	def =	0.87	1.0	0.65	1.00
Box:	def =	0.83	1.0	0.65	1.00
Pipe:	def =	0.79	1.0	0.60	1.00

The record is only relevant if elasto-plastic transition is performed, see record **SURF2OFF**

MREF matno refx refy refz refrx refry refrz

Parameter	*Description*	*Default*
matno	Material number	
refx	Properties in local x-direction is defined by material with ID number *refx*	
refy-refrz	Similar to *refx* for the other degrees of freedom	

This record is used to specify material properties for the **nonlinear spring** element (both 1 node and 2 node)

This record may be repeated

Properties in the 6 DOFs are specified by referring to other material input (SESAM record MISOPL or UFO records HYPELAST and ELPLCURV). Material numbers equal to zero means that the spring has no stiffness in the actual degrees of freedom

If element type no 18 (spring to ground) refers to a "MREF" material, the element will be handled by USFOS as a 1 node **nonlinear** spring to ground

If element type no 15 (2 node beam), refers to a "MREF" material, the element will be handled by USFOS as a 2 node nonlinear spring

MPLASMON matno	c1	a1	c2	a2	c3	a3
	c4	a4	c5	a5	c6	a6

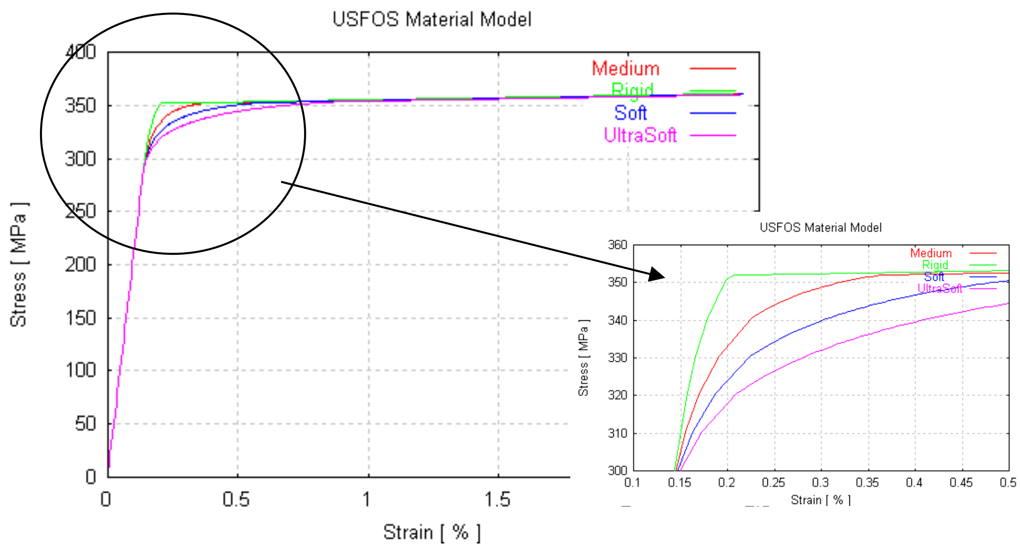
Parameter	*Description*	*Default*
matno	Material reference number	
c1	Hardening parameter for x-force	$2.0 \cdot 10^{-3}$
a1	Elasto-plastic transition parameter for x-force	0.25
c2	Hardening parameter for y-force	$2.0 \cdot 10^{-3}$
a2	Elasto-plastic transition parameter for y-force	0.25
.		
.c6	Ditto for z-force	
a6	x-moment	
	y-moment	
	z-moment	

This record is used to specify material hardening parameters and elasto-plastic transition parameters for the material under monotonous loading conditions

The default hardening properties refer to the condition where elasto-plastic transition is performed, see record **SURF2OFF**. If elasto-plastic transition is not performed, the default values are zero

c2-c6 need not be given if they are equal to c1
a2-a6 need not be given if they are equal to a1

Below, and example describes the "a"-parameter's impact on the stress-strain curve. ((Typical Steel material with E=210.000Mpa and Yield=355Mpa). The hardening is fixed to 0.2% in the example.



- ❑ Rigid (Green curve) a = 2.0
- ❑ Medium (Red curve) a = 1.0
- ❑ Soft (Blue Curve) a = 0.25 (Default)
- ❑ UltraSoft (Pink curve) a = 0.15

This record may be repeated

MPLASCYC matno c1 a1 c2 a2 c3 a3
c4 a4 c5 a5 c6 a6

Parameter	*Description*	*Default*
matno	Material reference number	
c1	Hardening parameter for x-force	0.014
a1	Elasto-plastic growth parameter for x-force	0.84
c2	Hardening parameter for y-force	0.024
a2	Elasto-plastic growth parameter for y-force	0.34
.		
.		
.	Ditto for z-force	
c6	x-moment	
a6	y-moment	
	z-moment	

This record is used to specify material hardening parameters and elasto-plastic transition parameters for the material under cyclic loading conditions

The default hardening properties refer to the condition where elasto-plastic transition is performed, see record **SURF2OFF**. If elasto-plastic transition is not performed, the default values are zero

c2-c6 need not be given if they are equal to c1
a2-a6 need not be given if they are equal to a1

This record may be repeated

STRAINMOD KeyWord ListType IDs

Parameter	*Description*	*Default*
KeyWord	S235 : Use 235 Steel data (built-in)	S355
	S310 : Use 310 Steel data	
	S355 : Use 355 Steel data	
	S420 : Use 420 Steel data	
	S460 : Use 460 Steel data	
	UserDef : User Defined data. More data required	
ListType	Definition of ID list (has to be "material")	
IDs	Material IDs	

This record is used to specify strain model data used in connection with strain calculation. The calculation is based on the plastic hinge displacements and rotations.

This record may be repeated

6.3.7 Member Modelling

CELHINX elnox ihin1 ihin2 ihinmid

Parameter	*Description*	*Default*
elnox	External (user-specified) element number	
ihin1	Restriction code for plastic hinge development at first node 0: Plastic hinge free to develop 1: Plastic hinge is suppressed	
ihin2	Restriction code for plastic hinge development at second node	
ihinmid	Restriction code for plastic hinge development at midspan	

This record may be used to suppress development of plastic hinges on specified elements (Sect 4.5)

This record may be repeated

PLASTHIN ihin1 ihin2 ihinmid elnox1 elnox2 ...

Parameter	*Description*	*Default*
ihin1	Restriction code for plastic hinge development at first node 0: Plastic hinge free to develop 1: Plastic hinge is suppressed	
ihin2	Restriction code for plastic hinge development at second node	
ihinmid	Restriction code for plastic hinge development at midspan	
elnox1 elnox2 . .	External (user-specified) element number 1 External (user-specified) element number 2 . . NOTE ! If no elements are specified, all beam elements get the actual restriction code.	

This record may be used to suppress development of plastic hinges on specified elements.

This record may be repeated

Example: PLASTHIN 0 0 1
PLASTHIN 0 0 0 1010 1020 1030

Will suppress plastic hinge at midspan for all elements except for elements 1010, 1020 and 1030.
This record is an alternative to the *CELHINX* record.

BEAMHING ix1 iy1 iz1 irx1 iry1 irz1
ix2 iy2 iz2 irx2 iry2 irz2 elnox1 elnox2 .... elnoxn

Parameter	*Description*	*Default*
ix1	Fixation Code for Local X-Direction of node 1 of element : 0 : Fully Released 1 : Fully Connected	
iy1	Fixation Code for Local Y-Direction of node 1 of element	
iz1	Fixation Code for Local Z-Direction of node 1 of element	
irx1	Fixation Code for Torsion Moment of node 1 of element	
iry1	Fixation Code for Moment about Local Y-axis of node 1 of element	
irz1	Fixation Code for Moment about Local Z-axis of node 1 of element	
ix2	Fixation Code for Local X-Direction of node 2 of element etc...	
elnox1	External element number 1 to be defined with the actual internal hinge	
elnox2	External element number 2 to be defined with the actual internal hinge .	
.	.	
.		
elnoxn	External element number n to be defined with the actual internal hinge **NOTE !** If no elements are specified, all beam elements will be defined with the actual internal hinge definition.	

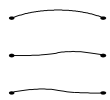
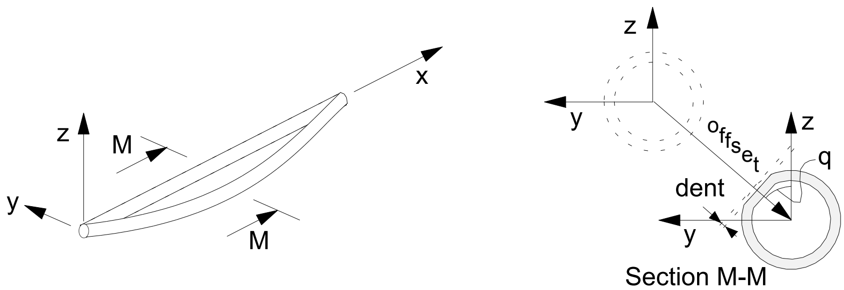
This record is used to assign internal hinges to beam elements.
This record may be repeated.

Example: BEAMHING 1 1 1 1 0 0 1 1 1 1 0 0
BEAMHING 1 1 1 1 1 1 1 1 1 1 1 1 1010 1020 1030 2010 2020 2030

Default internal hinge is changed from fully connected(default) to released bending degrees of freedom at both ends of the elements, (applied to all beam elements).
However, elements 1010 1020 1030 2010 2020 and 2030 are defined with fully connected joints at both ends, (the latest BEAMHING definition overrides previous definitions).

NOTE ! This option should be used with care!
Due to nonlinear geometrical effects, the release of some degrees of freedom could be valid at the first step only.
This is obvious in connection with release of the two shear degrees of freedom (no 2 and 3) where the vertical component of the axial stiffness will become more and more significant as the beam axis rotates.

GIMPER impgroup impshape angle offset dent1 dent2 dentmid

Parameter	*Description*	*Default*
impgroup	Reference number of imperfection group	
impshape	Shape of initial deformation (Sine-functions) 0: Both ends rotated 1: End 1 undeformed 2: End 2 undeformed 	
angle	Orientation of offset and dent. Specified in degrees counter-clockwise from the local element z-axis  Section M-M	
offset	Maximum offset divided by element length	
dent1	Depth of dent divided by pipe diameter. First node	
dent2	Depth of dent divided by pipe diameter. Second node	
dentmid	Depth of dent divided by pipe diameter. Element midspan	

This record is used to define initial deformations and damage conditions for tubes (Sect 3.4)

Note! The effect of local **dents** is only included if the dent switch parameter *dentsw* = 1 (default) (Sect 6.3.C)

This record may be repeated

GELIMP elnox impgroup

Parameter	*Description*	*Default*
elnox	External (user-defined) element number	
impgroup	Reference number of imperfection group	

This record is used to assign initial deformations or damage conditions to specified elements

This record may be repeated

MEMBER keyword impgroup ListType IDs….

Parameter	*Description*	*Default*
keyword	Command keyword: -Imperfection : Define member imperfection	
Impgroup	Reference number of imperfection group	
ListType	Definition of ID list: -Group : Specify Group IDs	
IDs	Actual Element Groups	

This record is used to assign user defined initial deformations or damage conditions to specified members. A member has to be defined as a group of elements.

This record may be repeated

MEMBER keyword auto ListType IDs….

Parameter	*Description*	*Default*
keyword	Command keyword: -Imperfection : Define member imperfection	
Impgroup	Reference number of imperfection group	
ListType	Definition of ID list: -Group : Specify Group IDs	
IDs	Actual Element Groups	

This record is used to assign initial deformations or damage conditions to specified members. A member has to be defined as a group of elements. The actual initial deformation is specified using the CINIDEF command.

Example:

```
MEMBER   Imperfect  Auto   Group 101  102  103
CINIDEF  API_WSD  MembLoad  5
```

This record may be repeated

BLINDP2 insl iem ix iy iz irx iry irz		
Parameter	*Description*	*Default*
insl	Slave (dependent) node	
iem	Element with master (independent) node(s). The coupled degrees of freedom (DOFs) are referred to this local element coordinate system	
ix iy iz irx iry irz	Specification of linear dependent DOFs in local master element coordinate system: i.. = 0 No coupling i.. = 1 The slave DOF is linearly dependent on the corresponding master DOF(s) (including torsional effects for iy and iz, Ref. /1/) i.. = -1 Only valid for iy and iz. Same as i.. = 1, but not including torsional effects	
This record is used to define dependent degrees of freedom (Sect 3.8) The record is given once for every node with dependent degrees of freedom		

NONSTRU ListType { List }

Parameter	*Description*	*Default*
ListType	Definition Type: *Element* : Element ID list *Material* : Material ID list *Geometry*: Geometry ID list *Group* : Group ID list *Visible* : The non-structural elements are visualized in Xact, (default invisible)	
{ List }	List of actual ID's	

This record is used to specify non-structural beam elements (passive elements), and such elements are excluded from the global stiffness formulation *while the elements' attracted loads are kept*.

Example 1: NonStru Element 1001 1002 2001 2002
NonStru Geom 27304 27305 27306

All beam elements referring to one of the cross section geometry ID's: 27304, 27305 or 27306 will become non-structural. In addition, the elements 1001, 1002, 2001 and 2002 get "nonstru".

Example 2: NonStru Mat 210355 210345

All beam elements referring to one of the material ID's: 210355 or 210345 will become non-structural.

Example 3: NonStru Group 1111 2222 3333

All beam elements "member of" one of the Group ID's: 1111, 2222 or 3333 will become non-structural.

Example 4: NonStru Visible

This command switches *ON* the visualization of the non-structural members. By default, only structural members are visualized in Xact.

All parameters may be abbreviated, but enough characters to ensure non-ambiguous definition is required, (Geometry could be specified as Ge, Geo, Geom, etc, material as M, Ma, Mat, etc).

NOTE:

The command: "*STRUCTEL*" (structural elements) will override the NonStru definitions.

This record may be repeated

STRUCTEL ListType { List }

Parameter	*Description*	*Default*
ListType	Definition Type: *Element* : Element ID list *Material* : Material ID list *Geometry*: Geometry ID list *Group* : Group ID list	
{ List }	List of actual ID's	

This record is used to ***override*** *non*-structural beam element(s) specified under NONSTRU. This option is useful in connection with models where for example most elements with given geometry ID's should become non-structural except some few, which should become structural.

Example 1: NonStru Geom 27304 27305 27306
StructEl Element 1001 1002

All beam elements referring to one of the cross section geometry ID's: 27304, 27305 or 27306 will become non-structural. However, the elements 1001 and 1002 (which refer to one of the specified geometry ID's) should *remain structural*.

This record may be repeated

LIN_ELEM Type ListType { List }

Parameter	*Description*	*Default*
Type	For conventional beam, type = 0. (for riser beam, type=1).	
ListType	Definition Type: *Element* : Element ID list *Material* : Material ID list *Geometry*: Geometry ID list *All* : All elements become elastic, no further input	
{ List }	List of actual ID's	

This record is used to specify elements, which should be linear and the command is an alternative to the more detailed PLASTHIN command. Development of plastic hinges is suppressed in all 3 nodes.

Example 1: Lin_Elem 0 Geom 27304 27305 27306
Lin_Elem 0 Element 1001 1002

All beam elements referring to one of the cross section geometry ID's: 27304, 27305 or 27306 will become elastic (yielding in all 3 hinges are suppressed) in addition the elements 1001 and 1002.

Example 2: Lin_Elem 1 Mat 210355 210345

All beam elements referring to one of the material ID's: 210355 or 210345 will become elastic. The beam formulation is the "integration point based" riser element, which also suppresses elastic buckling if one element per member is used.

This record may be repeated

CINIDEF Size/ColumnCurve Pattern LoadCase

Parameter	*Description*	*Default*
Size	Magnitude of initial imperfection or column curve number Size < 0.1 : e/L given directly Size > 1 : Column curve name (or number). See below	
Pat	MembLoad : Direction following member line load BaseShear : Direction following global base shear	
LoadCase	Load case to determine direction of initial imperfection	

This record is used to switch **on** automatic calculation of initial member imperfection for calibration of analysis model to buckling curves.

Implemented column curves are:

KeyWord	*Number*	*Curve*	
API_WSD	10	API WSD / SSRC WSD	
NORSOK	15	NORSOK	
AISC	20	AISC	
API_LRFD	30	API LRFD	
Euro3_A	41	Eurocode 3 Curve A	
Euro3_B	42	Eurocode 3 Curve B	
Euro3_C	43	Eurocode 3 Curve C	(Fire)
Euro3A0	44	Eurocode 3 Curve A0	
Euro3_D	45	Eurocode 3 Curve D	
DnV_A	51	DNV Curve A	
DnV_B	52	DNV Curve B	
DnV_C	53	DNV Curve C	
DnVA0	54	DNV Curve A0	
DnV_E	55	DNV Curve E	
SSRCCSA1	61	SSRC / CSA Curve 1	(Lehigh characteristic curve 1)
SSRCCSA2	62	SSRC / CSA Curve 2	(Lehigh characteristic curve 2)
SSRCCSA3	63	SSRC / CSA Curve 3	(Lehigh characteristic curve 3)
SSRCmea1	66	SSRC mean column curve 1	(Lehigh mean curve 1)
SSRCmea2	67	SSRC mean column curve 2	(Lehigh mean curve 2)
SSRCmea3	68	SSRC mean column curve 3	(Lehigh mean curve 3)
Chen	70	Chen column curve	

Example

CINIDEF API_WSD MembLoad 3

Means that initial imperfections are assigned on all tubular members according to API WSD column curve, in direction of the distributed loading for each member under load case 3.

This record is given only once.

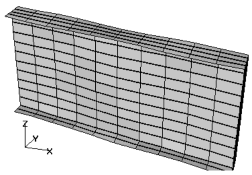
BuckMode Mode No ScaleFact

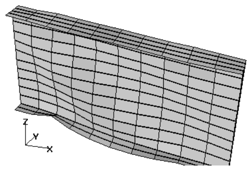
Parameter	*Description*	*Default*
Mode No	Apply the specified Eigen mode as initial imperfection	
ScaleFact	Scale the selected Eigen values and add to the user defined structural nodal coordinates.	

This record is used to apply initial imperfections based on an eigen value analysis performed automatically as a pre processing task.

Example : **BuckMode 5 0.1**

An eigen value analysis *automatically* will be initiated prior to the ordinary collapse analysis, and the 5'th eigenmode is scaled by 0.1 and added to the initial nodal coordinates. The stress free structural shape is shown in the figure below (to the left), while the final collapse mode is shown to the right.





Initial Mode ***Collapse Mode***

This record is given only once.

GROUTED GroutMat ListType { List }

Parameter	*Description*	*Default*
GroutMat	Material ID defining the Grout material parameters. (see MATERIAL)	
ListType	Definition Type:	
List	*Element* : Element ID list *Material* : Material ID list *Geometry*: Geometry ID list	
{ List }	List of actual ID's	

This record is used to specify elements with grout.

<u>Example 1:</u>
Grouted 1020 Geom 27304 27305 27306
All elements referring to cross section (must be pipes) 27304, 27305 or 27306 will be "filled" with grout wit material properties defined by material ID 1020

This record may be repeated

ElmTrans *KeyWord* node (or ID) SubKey ID's		Def
KeyWord *Node(or ID)* SubKey ID's	Node Reference Type: ☐ GlobNode : "Node" refers to Node ID ☐ LocNode : "Node" refers to the element's local node (end) ☐ MainBeam : Direct reference to MainBeam ☐ Tolerance : Specify tolerance for switch to ZL-spring Node Id of node or local element node (see above). Main Beam ID if Keyword "MainBeam" is selected Keyword for interpreting of the succeeding ID- list ☐ "elem" Element ID's are specified ☐ "Mat" Material ID's are listed. Elements referring to mat are selected ☐ ZeroSpri ID of actual spring is given under ID's. Actual ID list	 0.05

With this record, the user defines how to define and update an element's local-global transformation matrix. This command is needed when zero length non-linear springs are defined, since the orientation of the local coordinate system cannot be defined in the normal way. (having local x-axis going from end 1 to 2 etc).

Example1:

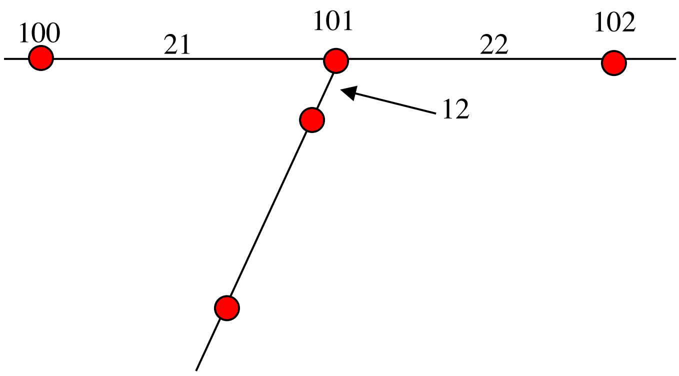
```
`           Type       NodeID  ListType    ID's
 ElmTrans  GlobNode    101      Elem        12
```

Element no 12 will then update it's transformation according to rotations of node 101.

If Element 12 is a 2-node non linear spring, with ~zero length, (less than "Tol"), or a friction spring, the following will be done in addition:

- ☐ The initial local-global transformation matrix will be set equal to the transformation matrix of the first *beam* element connected to node 101.

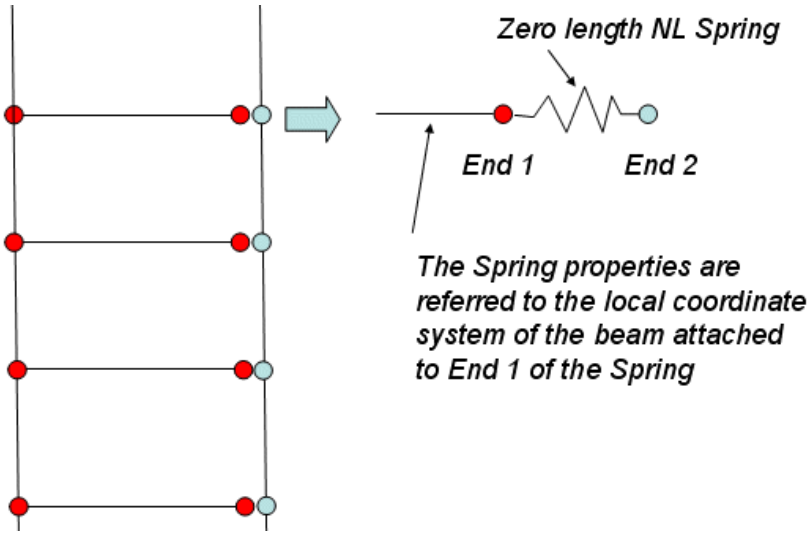
This option is typically used in connection with modelling of bearings, where the properties should follow the structure's large displacements and rotations.



Example2:

```
`           Type       End  ListType     ID's
 ElmTrans   LocNode     1     Mat         3 5 8
```

Means that all spring elements referring to materials 3, 5 or 8 will have the local coordinate system defined by the beam element attached to end 1 of the spring in question. If the user is planning the modelling of Zero length spring, the amount of input to define the local coordinate system could be limited to just one line as described in this example.



Example3:

```
`           Keyword     MainElem  SubKey     SpriElem
 ElmTrans   MainBeam     101      ZeroSpri    700030
```

Means that zero-length spring element 700030 should use the coordinate system of beam-element 101. This option is recommended to ensure that the ZeroSpri gets correct reference element, (for cases where more than one beam element is attached to the common spring-beam node).

This record could be repeated..

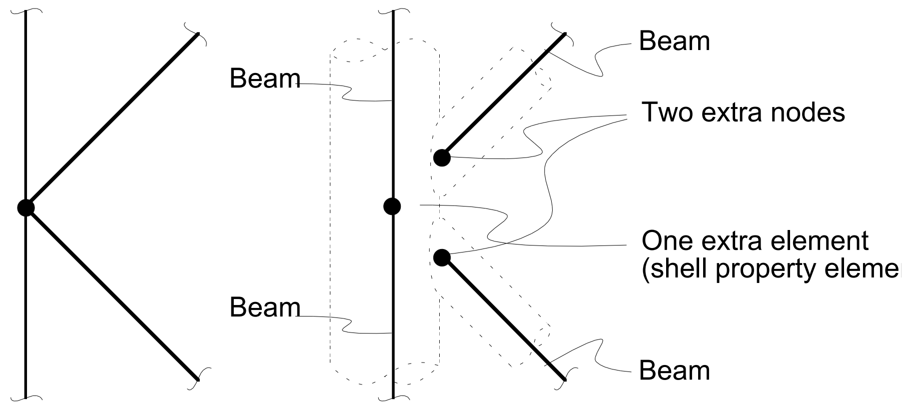
6.3.8 Joint Modelling

SHELL nodex elnox1 elnox2 d t

Parameter	*Description*	*Default*
nodex	External node number referring to the joint where shell effects should be considered	
elnox1	External element number defining one of the two elements connected to the node	
elnox2	External element number defining the second **chord** element	
d , t	Diameter and thickness of the chord at the joint, respectively (canned joint). If omitted, the data for *elnox1* is used	

With this record, a complete shell analysis of the specified tubular joint will be included in the analysis.

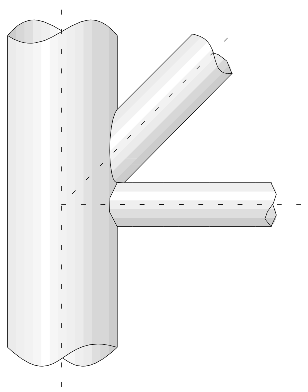
Note! Internally in USFOS one extra element and extra nodes are introduced into the finite element model, and the CPU consumption of the analysis increases somewhat.



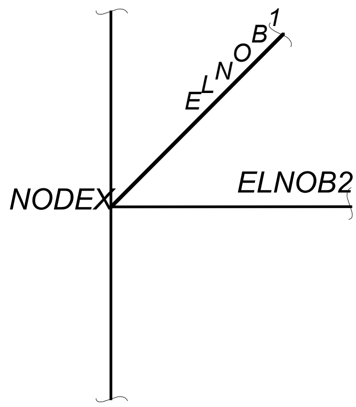
OVERLAP nodex elnob1 elnob2 . . . . .

Parameter	*Description*	*Default*
nodex	External node number for the actual tubular joint	
elnob1	External element number for overlapping brace no 1	
elnob2	External element number for overlapping brace no 2	
.	.	
.	.	
.	.	

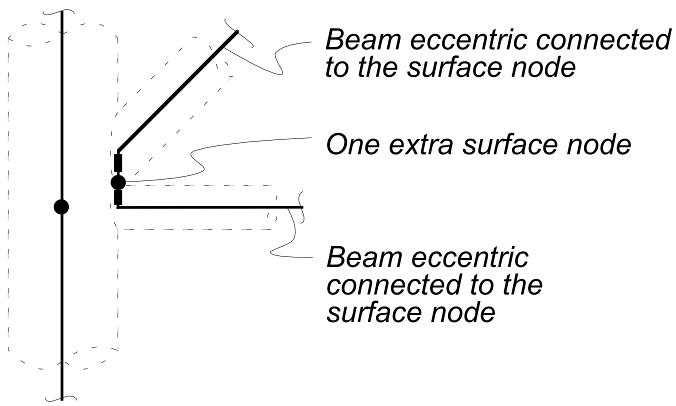
This record defines overlapping braces at the tubular joints where shell effects are specified



Real structure



Users beam model

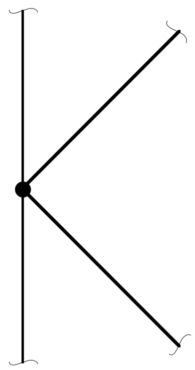


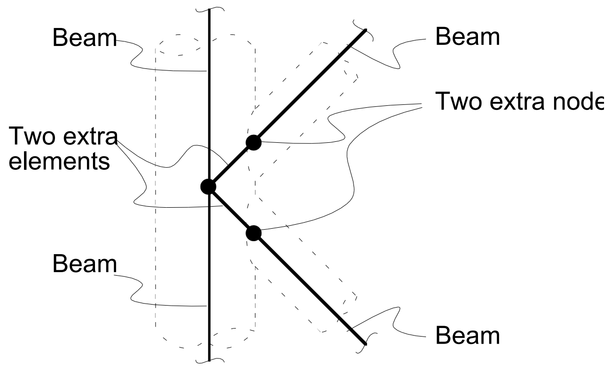
Model with local flexibility and overlapping included

CHJOINT	Node Chord1 Chord2 geono CapRule	
Parameter	*Description*	*Default*
Node	Node ID referring to the joint where joint capacity and non-linear joint behaviour should be considered.	
Chord1	Element ID of one of the two **Chord**-elements connected to the node	
Chord2	Element ID defining the second **Chord** element	
geono	Geometry reference number defining the diameter and thickness of the chord at the joint (canned joint). If omitted or equal to 0, the data for *Chord1* is used.	0
CapRule	Capacity rule switch: MSL : MSL non-linear joint characteristics, (see next pages). NORSOK: Latest implemented Norsok, (currently R4- 2021).. NOR_R3 : Norsok N-004, rev 3 non-linear joint characteristics NOR_2021: Norsok N-004, rev 4, 2021 ISO19902: Latest implemented ISO, (currently 2020) ISO_2007: ISO 19902:2007 non-linear joint characteristics. ISO_2020: ISO 19902:2020 non-linear joint characteristics. APIWSD: API RP 2A WSD ES3 (non-linear joint characteristics). Historic, ***not*** recommended: OldAPI : API-LRFD, 1993. (no more data required) OldDoE : DoE (no more data required) UserSurf : User defined capacity and surface definition, more data required. UserSpri : User defined Joint Springs, more data required.	NOR_2021

With this record, the capacity of each brace/chord connection at the tubular joint will be checked according to a selected joint capacity equation.
This check will impose restrictions on the load transfer through each brace/chord connection at the specified joint

The generated curves are printed in the '.jnt' file or '.out' – file (Historic options), and the **peak capacities** will be printed using the **Verify/Element/Information** option in **Xact**.





Conventional joint model *Joint with capacity check included*

CHJOINT	node	Chord1	Chord2	geono	rule	nbraces
				$brace_1$	$Mref\_ID_1$	
				$brace_2$	$Mref\_ID_2$	
				$brace_n$	$Mref\_ID_n$	

Parameter	*Description*	*Default*
node	Node ID referring to the joint where joint capacity and non-linear joint behaviour should be considered.	
Chord1	Element ID of one of the two **Chord**-elements connected to the node	
Chord2	Element ID defining the second **Chord** element	
geono	When user defined capacity is used, this parameter is dummy, but a dummy parameter must be given	
rule	Capacity rule switch = **-101**, **User defined Joint Springs.**	
nbraces	Number of Braces connected to the actual joint	
$brace_1$	Brace element ID of first brace	
$Mref\_ID_1$	ID of MREF-record defining the nonlinear spring properties of brace 1 connection. Data for **all** braces connected to the joint are required !	

With this record, the capacity of each brace/chord connection of the tubular joint will be limited by the user defined P_d curves defining the behaviour of the joint.

This check will impose restrictions on the load transfer through each brace/chord connection at the specified joint

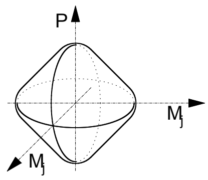
CHJOINT	node	Chord1	Chord2	geono	rule
	alpha1	alpha2	alpha3	alpha4	alpha5
	nbraces				
	brace1	axial	torsion	Mipb	Mopb
	......				

Parameter	*Description*	*Default*
node	Node ID referring to the joint where joint capacity and non-linear joint behaviour should be considered.	
Chord1	Element ID of one of the two **Chord**-elements connected to the node	
Chord2	Element ID defining the second **Chord** element	
geono	Dummy when rule = -3	
rule	Capacity rule switch=-3 : User defined capacity and surface definition.	
alpha1-5	Parameters used to define the actual surface	
nbraces	Number of braces connected to the actual joint	
brace1	User defined element number of first brace element	
axial torsion Mipb Mopb	Axial capacity: *brace1* - chord connection Torsion capacity: *brace1* - chord connection In plane bending capacity: *brace1* - chord connection Out of plane bending capacity: *brace1* - chord connection Data for all braces connected to the joint are required	

With this record, the capacity of each brace/chord connection at the tubular joint will be checked according to the specified rule

This check will impose restrictions on the load transfer through each brace/chord connection at the specified joint

When the USER-defined capacity is used (rule = -3), the following surface definition is applied:



$$\left(\frac{P}{P_M}\right)^{\alpha 1} + \left(\left(\frac{M_z}{M_{xu}}\right)^{\alpha 2} + \left(\frac{M_i}{M_{iu}}\right)^{\alpha 3} + \left(\frac{M_0}{M_{ou}}\right)^{\alpha 4}\right)^{\alpha 5} = 1$$

CHJOINT node Chord1 Chord2 geono CapRule CapLevel Q_f_SafetyCoeff

Parameter	*Description*	*Default*
node	Node number referring to the joint where joint capacity and non-linear joint behaviour should be considered	
Chord1	Element ID of one of the two **CHORD** elements connected to the node	
Chord2	Element ID defining the second **CHORD** element	
Geono	Geometry reference number defining the diameter and thickness of the chord at the joint (canned joint). If omitted or equal to 0, the data for *elnox1* is used.	
CapRule	Capacity rule: **MSL**: *MSL* non-linear joint characteristics	
CapLevel	Capacity level or capacity multiplier. *mean* : use mean value joint capacities *char* : use characteristic joint capacities *fcrack* : use the "first crack" curve *scalfact* : joint capacities are set to mean value capacities multiplied by *scalfact*, (where "*scalfact*" is a positive real number). This option is only available with the MSL joint formulation.	mean
Q_f_Safety Coeff	The Q_f factor for joint capacities includes a safety factor (or partial safety coefficient) in the chord stress utilisation factor.	1.0

With this record, the capacity of each brace/chord connection at the tubular joint will be checked (modelled) according to a selected joint capacity equation

This joint model will impose restrictions on the load transfer through each brace/chord connection at the specified joint, and the non-linear joint characteristics will be included in the USFOS analysis.

Extra nodes and elements will be introduced in the FE model, and the behaviour of these elements are assigned according to the selected joint capacity rule or specified joint capacity

The Finite Element formulation for these "joint elements" is selected automatically, (JNT_FORM record not needed).

NOTE !
This record is "Historic" and does not work for the recommended joint capacity models.

*Use **JntOption "Scale" and "Duct"** options if the capacity curves should be modified.*

JSURFSIZ z_y^m z_b^m nodex1 nodex2 ....

Parameter	*Description*	*Default*
z_y^m	Extension ("size") of the yield surface, relative to the reference-bounding surface (Sect 4.3.2).	0.50
z_b^m	Extension ("size") of the bounding surface ("full plastic capacity") relative to the reference-bounding surface (Sect 4.3.3).	1.00
nodex1	External node number of joint 1	
nodex2	External node number of joint 2	
.	.	
.	.NOTE ! If no nodes are specified, **all** joints defined get the actual surface ratios.	

With this record the user may change the default values of the ratio of elastic and plastic "section" modulus of the joint model. It is then possible to scale the actual joint capacities, (e.g.. calculated according to API, see CHJOINT).

This is an useful option in connection with:
- Sensitivity studies where joint capacities are important
- Cases where some joints are damaged (reducing the bounding surface)
- Cases with reinforces joints (expanding the bounding surface)
- etc...

This record may be repeated.

```
Example:    JSURFSIZ   0.95  1.5
            JSURFSIZ   0.50  1.0     101  102  103
```

will change the yield surface size to 0.95 and the bounding surface size to 1.5 of the initial, reference-bounding surface for all joints except for nodes 101, 102 and 103.

JOINTGAP Gap NodeID Brace1 Brace2 ……

Parameter	*Description*	*Default*
Gap	Actual Gap specified in current length unit	
NodeID	Actual Joint, (CHJOINT must be specified for this joint)	
Brace_i . .	The actual gap should be applied to the Braces (Elem ID's) specified. *NOTE 1*: If **no** node is specified, **all** joints defined get the actual gap. *NOTE 2*: If **no** braces are specified, ***all*** braces get the actual gap.	

With this record the user may override the gaps between braces, which are computed based on the structural model, (member coordinates and offsets).

This is a useful option if the structural model does not describe correct gaps (f ex a model without member eccentricities/offsets).

This record may be repeated.

	'	Gap	(NodeID)
***Example 1*:**	**JOINTGAP**	**0.050**	

This will force a gap = 0.050 to be used in all relevant capacity calculations for all joints with CHJOINT applied (does not influence T/Y capacities).

	'	Gap	NodeID
***Example 2*:**	**JOINTGAP**	**0.050**	
	JOINTGAP	**0.070**	**1010**
	JOINTGAP	**0.070**	**2010**
	JOINTGAP	**0.070**	**3010**

This will force a gap = 0.050 to be used in all relevant capacity calculations for all joints except for joints 1010, 2010 and 3010, which will use a gap of 0.070.

	'	Gap	NodeID	Brace1	Brace2	Brace3
***Example 3*:**	**JOINTGAP**	**0.050**				
	JOINTGAP	**0.070**	**1010**	**20100**	**20101**	**20102**

This will force a gap = 0.050 to be used in all relevant capacity calculations for all joints except for joint *1010*, where the braces *20100, 20101* and *20102*, which will use a gap of 0.070. The other braces connected to joint 1010 will use a gap of 0.050.

	'	Gap	NodeID	Brace1	Brace2	Brace3
***Example 4*:**	**JOINTGAP**	**0.070**	**1010**	**20100**	**20101**	**20102**

This will force a gap=0.070 to be used for joint 1010, braces 20100, 20101 and 20102. The other braces connected to joint 1010 as well as all other joints with ChJoint specified will use the gap computed on basis on the structural model (coordinates and eventual offsets).

NOTE !
This record is "Historic" and is not needed for the recommended joint capacity models, (correct FE-formulation is selected automatically).

JNT_FORM form NodeIDs

Parameter	*Description*	*Default*
Form	FE formulation for the "joint elements" introduced by the CHJOINT option. Capacity level or capacity multiplier. 0 : Beam-column representation. Default on old versions of USFOS, but no longer recommended. The three-hinge mechanism in the beam-column element may introduce numeric instabilities for the small "joint elements". Spring : P-δ spring representation. Un-coupled P-d curves with ductility limits will be generated automatically for each joint degree of freedom, based on the capacities specified under the **CHJOINT** record(s). Plast : Plasticity formulation including brace load interaction and joint (re)classification as specified by the **JntClass** record.	0
NodeIDs	Node(s) for which the formulation should be applied. NOTE If no nodes are specified, all joints get the actual formulation	

Use of the **CHJOINT** option will introduce extra elements in the FE model. The behaviour of these elements will be assigned according to the selected joint capacity rule or specified joint capacity. The FE formulation for the "joint elements" is selected by the **JNT_FORM** record.

P-δ curves (option 1) are derived from the actual capacities as follows:

Displacement = 0.1% of Chord diameter defines 'yielding' (confer JSURFSIZ)
Displacement = 1.0% of Chord diameter defines maximum force/moment
Displacement = 5.0% of Chord diameter defines end of maximum capacity
Displacement = 10 % of Chord diameter defines fracture of joint

The generated curves are printed in the '.out' - file, and the **peak capacities** will be printed using the **Verify/Element/Information** option in **Xact**.

NOTE!

If JNT_FORM ***is not*** specified, following defaults are used for the various joint capacity rules:

- API : formulation no. 0
- UserSurface : formulation no. 0
- UserSpring : formulation no. 1
- MSL : formulation no. 3

It is possible to mix different Capacity rules and FE formulations in same analysis.

This record may be repeated

JNTCLASS interval

Parameter	*Description*	*Default*
interval	Joints will be (re)classified according to geometry and force state at specified intervals during the analysis. The joint capacities will be updated according to the revised classification, P-δ curves for each joint degree of freedom and the Q_f factor will be updated. 0 : No joint classification. 1 : Continuous joint (re)classification. Joint capacities, non-linear joint characteristics and the Q_f factor will be updated at every step in the USFOS analysis. n >1 : Joints will be (re)classified at every n'th step. Joint capacities, non-linear joint characteristics and the Q_f factor will be updated at every n'th step in the USFOS analysis.	1

If the record is not given, the default value of one (classified every step) is used for the MSL, Norsok, ISO and API-WSD variants.

This record is given once

JNTOPTION Keyword Joint ID's ………

Parameter	*Description*	*Default*
Keyword	Subcommand defining the actual option: *Grouted* : The actual joint(s) uses grouted capacity formulation *NoGrouted* : Grout option is switched Off for the actual joints	

This record is only valid in combination with CHJOINT option.

Example 1:

JntOption Grouted 1001 2001 3001

Means that the joints 1001, 2001 and 3001 are using grouted joint formulation. The formulation described under CHJOINT is disregarded.
NOTE: ChJoint must be defined for the actual joints.

Example 2:

JntOption NoGrouted 133 144

Means that the joints 133 and 144 are using the formulation described under CHJOINT.

NOTE: If the chord members are defined as Grouted and default CHJOINT parameters are used, (minimum input), the joint formulation is changed to Grouted automatically.

This record may be repeated

JNTOPTION Keyword ScaleFact *ListType* { Data } ....

Parameter	*Description*	*Default*
Keyword	Subcommand defining the actual scaling option: *ScaleAll* : All force components are scaled with specified factor *ScaleTens* : Axial, tension component is scaled with specified factor *ScaleComp* : Axial, compression is scaled with specified factor *ScaleIPB* : In-Plane Bending is scaled with specified factor *ScaleOPB* : Out-of-Plane Bending is scaled with specified factor	
ScaleFact	Actual Scaling factor to scale the force components.	
ListType	Keyword defining how to define the actual joint connections: *Joint* : All connections at the specified joint(s) are scaled *Connection* : Only one connection is scaled.	
Data	Input defining the connection(s). Depends on "ListType": "joint" : Actual Node ID(s). (Several nodes could be specified). "connection" : Actual Node and Brace ID defining connection	

This record is only valid in combination with CHJOINT option, and is used to scale the force- and moment components of the joint capacity curves.

Example 1:

'	*Key*	*fact*	*ListOpt*	*Joint-IDs…*
JntOption	ScaleAll	0.8	joint	1001 2001 3001

Means that *all* force and moment components for *all connections* at the joints 1001, 2001 and 3001 are scaled with the factor 0.8.

Example 2:

'	*Key*	*fact*	*ListOpt*	*Joint-IDs…*
JntOption	ScaleTens	0.6	joint	1001 2001 3001

Means that Axial tension forces for *all connections* at the joints 1001, 2001 and 3001 are scaled with the factor 0.6. If this record is given after "Example-1", the values 0.8 will be replaced with 0.6.

Example 3:

'	*Key*	*fact*	*ListOpt*	*Joint-ID*	*Brace-ID*
JntOption	ScaleTens	0.1	Connection	2001	101

Means that Axial tension forces for *one single connection* (joint 2001, brace element 101) is scaled with the factor 0.1.

If this record is given after "Example-1" and "Example-2", the value 0.6 will be replaced with 0.1, (I.e: Start with the general scaling and specify the exceptions thereafter).

This record may be repeated

JNTOPTION Keyword ScaleFact *ListType* { Data } ....

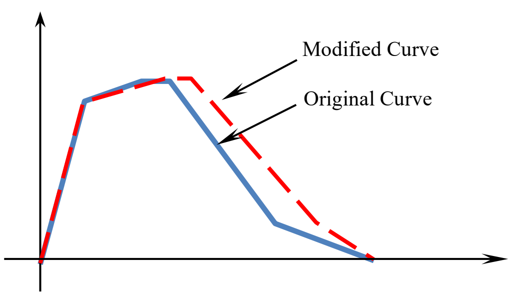
Parameter	*Description*	*Default*
Keyword	Subcommand defining the actual scaling option: *DuctAxT* : Axial, tension disp component is scaled with specified factor *DuctAxC* : Axial, compression disp. is scaled with specified factor *DuctIPB* : In-Plane rotation is scaled with specified factor *DuctOPB*: Out-of-Plane rotation is scaled with specified factor	
ScaleFact	Actual Scaling factor to "stretch" the displacement/rotation components. Minimum factor : 0.8 Maximum factor : 2.0	1.0
ListType	Keyword defining how to list the actual joint connections: *Joint* : All connections at the specified joint(s) are scaled *Connection* : Only one connection is scaled.	
Data	Input defining the connection(s). Depends on "ListType": "joint" : Actual Node ID(s). (Several nodes could be specified). "connection" : Actual Node and Brace ID defining connection	

This record is only valid in combination with CHJOINT option, and is used to scale the displacement and rotation components of the joint capacity curves. I.e. make the joint more ductile (fact > 1) or brittle (fact < 1).

Example:

'	*Key*	*fact*	*ListOpt*	*Joint-IDs…*	
JntOption	DuctAxT	1.5	joint	1001 2001 3001	
'	*Key*	*fact*	*ListOpt*	*Joint-ID*	*Brace-ID*
JntOption	DuctAxT	1.0	Connection	2001	101

Means that *all* Axial tension displacement values for *all connections* at the joints 1001, 2001 and 3001 are "stretched" with the factor 1.5 and thus makes the connections more ductile in axial tension, (elastic properties are unchanged). The other components (axial comp and bending) are unchanged. *One single connection* (joint 2001, brace element 101) is scaled with the factor 1.0, (using the original curve).



This record may be repeated.

JNTOPTION Keyword Value *ListType* { Data } ………

Parameter	*Description*	*Default*
Keyword	Subcommand defining the actual can geometry parameters: *CanLength* : Specify Lc and account for short can reduction *CanThick* : Specify Can Thickness. Override Default (Chord-1) thick *CanDiam* : Specify Can Diameter. Override Default (Chord-1) diam	
Value	Actual geometry value.	
ListType	Keyword defining how to list the actual joint connections: *Joint* : All connections at the specified joint(s) are scaled *Connection* : Only one connection is scaled.	
Data	Input defining the connection(s). Depends on "ListType": "joint" : Actual Node ID(s). (Several nodes could be specified). "connection" : Actual Node and Brace ID defining connection	

This record is only valid in combination with CHJOINT option, and is used to define geometry input parameters used to compute the joint capacity curves. I.e. override the data derived from the FE-model.

Example 1:

```
'           Key         Value    ListOpt    JointID …
JntOption  CanLength    0.400     joint     100      110
JntOption  CanThick     0.010     joint     100      110
JntOption  CanDiam      0.500     joint     100      110

'           KeyWord     Value     ListType     JointID    BraceID
JntOption  CanLength    0.350   Connection       100        130
```

Means that *all connections* at the joints 100 and 110 will account for "short can reduction" and use the specified geometry data. Exception for one connection (joint-100, brace 130).

The data used for the joint calculations are printed in the "out" file:

```
 -----------------------------
  Joint  :       100
  Connect  Brace   Can_Lc  D_Chord T_Chord  T_Nom  d_Brace t_Brace  (CanRed)
    1        60     0.400   0.500   0.010   0.007   0.127   0.003  0.67-1.0
    2       130     0.350   0.500   0.010   0.007   0.150   0.005  0.65-1.0

  -----------------------------
  Joint  :       110
  Connect  Brace   Can_Lc  D_Chord T_Chord  T_Nom  d_Brace t_Brace  (CanRed)
    3        80     0.400   0.500   0.010   0.007   0.127   0.003  0.67-1.0
    4       130     0.400   0.500   0.010   0.007   0.102   0.002  0.67-1.0
```

Print from the USFOS "out" file.

Data are also printed in the .jnt file:

```
   NODE          Capacity                        Chord       Chord       Chord
   ID              rule                        diameter   thickness   yield str.
   100        Norsok N-004 Rev2                5.000E-01   1.000E-02   3.240E+08

 Brace    Angle    Conn     Facing    Gap         Axial       MipB        MopB
   ID     (deg)    Type      brace               Cap/Qf      Cap/Qf      Cap/Qf

    60     45        Y                         2.054E+05   3.326E+04   2.066E+04
                                                    1.00        1.00        1.00
   130     90        T                         1.593E+05   3.281E+04   1.798E+04
                                                    1.00        1.00        1.00


   NODE          Capacity                        Chord       Chord       Chord
   ID              rule                        diameter   thickness   yield str.
   110        Norsok N-004 Rev2                5.000E-01   1.000E-02   3.240E+08


 Brace    Angle    Conn     Facing    Gap         Axial       MipB        MopB
   ID     (deg)    Type      brace               Cap/Qf      Cap/Qf      Cap/Qf

    80     45        Y                         2.054E+05   3.326E+04   2.066E+04
                                                    1.00        1.00        1.00
   130     90        T                         1.244E+05   1.505E+04   1.126E+04
                                                    1.00        1.00        1.00
```

Print from the USFOS "jnt" file.

NOTE 1.
If "**`SWITCHES Joint ShortCan ON`**" is specified, all connections are checked for short can reduction. The geometry data are found in the FE-model unless specified explicitly.

NOTE 2.
A tubular joint has only **one** diameter and **one** thickness. I.e. D and T are Joint data.
The chord diameter and thickness are found and stored in the following sequence:

1. From the FE-model, chord element no 1
2. The chord geometry specified under CHJOINT (if $\neq 0$)
3. Specified CanDiam and/or CanThick.
4. The ***latest*** defined CanDiam & CanThick are used for ***all connections*** to this joint.

This record may be repeated.

SWITCHES Joint EccUpdate Switch

Parameter	*Description*			*Default*
Switch	ON	:	Joint eccentricities are “repaired”	OFF
	OFF	:	No checking	

If the switch is ON, the joints defined using CHJOINT is checked for “bad shaped” eccentricities, and the following is done for braces with eccentricities:

- The eccentricity is moved from the chord surface to the chord centre
- The brace keeps the same position as before
- The new joint capacity element gets a reasonable length
- The new node is located on the chord surface (same as for EyeLift = 1.0)

This record is given once

SWITCHES Joint ShortCan Switch

Parameter	*Description*			*Default*
Switch	ON	:	Automatic short can detection	
	OFF	:	Switch OFF all short can including manually defined	

If the switch is ON, the FE-model is checked for cans:

- If CAN is found, the short can parameters are computed
- If no can is found for a connection, this is printed in the out file
- The label file, ***short_can_info.usl*** is created, and info could be displayed on the model.

This record is given once

SWITCHES Joint EyeLift Value

Parameter	*Description*	*Default*
Value	The radial distance from chord centre to the location of the new joint node. Value=1.0 means that the node is located on the chord surface	1.2

If the joints are defined as recommended (means that the brace does not end on the chord surface, but goes into the chord axis), it is possible to define where the new nodes are located.

This record is given once

CNOHINX nodex hinmx

Parameter	*Description*	*Default*
Nodex	External (user-specified) node number	
Hinmx	Max number of plastic hinges to develop at this node	

This record may be used to control the number of plastic hinges to be developed at specified nodes (Sect 4.5)

Number of plastic hinges can only be modified by the user when an elastic perfectly plastic model is selected, i.e. *ifysw* = 1 and C_i = 0

This record may be repeated

6.3.9 Foundation Modelling

SPUDMAT	matno	Type	Vpre	R_tot	β	G_v	G_H	G_R	ν	φ
	c	γ	[C8	C1	C7]					

Parameter	*Description*	*Default*
matno	Material number	
Type	Soil Type (= **Sand**)	
V_{pre}	Vertical preload	
R_tot	Total Spudcan Radius	
β	Apex angle [deg]	
G_V	Shear module (vertical)	
G_H	Shear module (horizontal)	
G_R	Shear module (rotational)	
ν	Poisson number(soil)	
φ	Friction angle (soil) [deg]	
c	Cohesion (soil)	
γ	Effective unit weight (soil)	
C_8	Degree of "associated flow": 1.0 = Associated flow 2.0 = Non-uplift sliding	1.0
C_1	"Inner yield surface" parameter	0.3
C_7	Yield surface Transition parameter	1.4

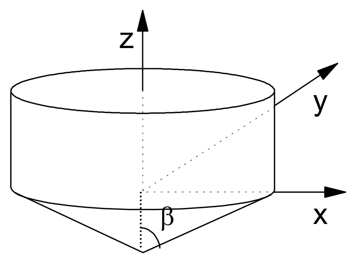
This record is used to specify material properties for the **nonlinear jackup foundation** element (1 node element). This material model is based on SNAME "Recommended Practice"

If the apex angle is 90 degrees, the preload is a dummy variable , if the apex angle is less than 90 an effective radius is calculated based on the input.

Local coordinate system is defined with the **BNTRCOS** card.

Element coordinate system is shown in the figure below.

This record may be repeated.



SPUDMAT	matno	Type	V_{pre}	R_{eff}	D_{emb}	A_s	G_v	G_H	G_R	ν
			c_{uo}	α	Suct	[Bfill	C_8	C_1	C_7]	

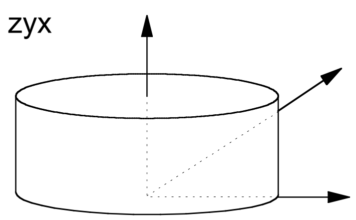
Parameter	*Description*	*Default*
matno	Material number	
Type	Soil Type (= **Clay**)	
V_{pre}	Vertical preload	
R_{eff}	Effective spudcan radius	
D_{emb}	Embedment depth	
A_S	Spudcan laterally projected embedded area	
G_V	Shear module (vertical)	
G_H	Shear module (horizontal)	
G_R	Shear module (rotational)	
ν	Poisson number(soil)	
c_{UO}	Undrained cohesive shear strength at max bearing area (soil)	
α	Adhesion factor, 1.0 = soft clay -> 0.5 = stiff clay	
Suct	Suction flag: 1 = Suction, 0 = No suction	
Bfill	Backfill flag : 1 = Backfill, 0 = No backfill	0
C_8	Degree of associated flow: 1.0 = Associated flow 2.0 = Non-uplift sliding	1.0
C_1	"Inner yield surface" parameter	0.5
C_7	Yield surface transition parameter	1.0

This record is used to specify material properties for the **nonlinear jackup foundation** element (1 node element)

Local coordinate system is defined with the **BNTRCOS** card.

Element coordinate system is shown in the figure below.

This record may be repeated.

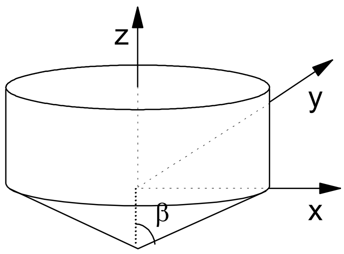


MSPUD	matno R β γ φ c ν V_{pre} G_V G_H G_R YFSW C_8	
Parameter	*Description*	*Default*
matno	Material number	
R	Total radius of spudcan	
β	Apex angle [deg]	
γ	Effective unit weight (soil)	
φ	Friction angle (soil) [deg]	
c	Cohesion (soil)	
ν	Poisson number(soil)	
V_{pre}	Vertical preload	
G_V	Shear module (vertical)	
G_H	Shear module (horizontal)	
G_R	Shear module (rotational)	
YFSW	Yield Function switch: 0 = Yield function according to SNAME RP 1 = Old Yield function (SINTEF Geoteknikk)	0
C_8	Degree of associated flow(apply to SNAME model only): 1.0 = Associated flow 2.0 = Non-uplift sliding	1.0

This record describes the old format used to specify material properties for the **nonlinear jackup foundation** element (1 node element).

Only valid for Sand. Kept for backward compatibility only, the SPUDMAT record is recommended for new Models!

If the apex angle is 90 deg., the preload is a dummy variable.
Local coordinate system is defined with the **BNTRCOS** card.
Element coordinate system is shown in the figure below.
This record may be repeated.

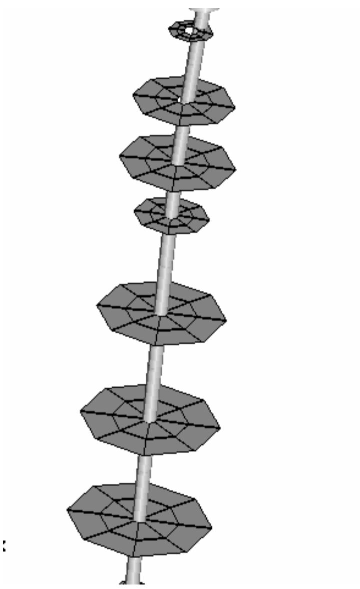


PILE ID Nodex1 Nodex2 Soil ID Pile Mat Pile Geo Local_coord Imper

Parameter	*Description*	*Default*
ID	User defined number used to identify the Pile	
Nodex1	External (user-defined) node number defining end 1 of the pile (top)	
Nodex2	External (user-defined) node number defining end 2 of the pile (tip)	
Soil ID	Soil characteristics are defined by *SOILCHAR* record with this ID	
Pile Mat	Pile Material Properties are defined by the referred *MISOIEP* record	
Pile Geo	Pile Geometry is defined by the *PILEGEO* record with this ID	
Loc Coord	The referred unit vector defines local Coordinate System for the pile. If omitted or equal to zero, default local coord system is used.	0
Imper	Pile Imperfection, reference to GIMPER record	0

This record is used to define a pile element.

This record may be repeated.



In XACT the pile is visualized with discs representing the soil defined in the referred SOILCHAR record.

The relative size of the discs reflects the relative strength of the soil weighting the T-Z capacity with a factor 100, and the P-Y capacity with 1. The user could modify this. See SWITCHES command.

Max disc diameter is 10 times max pile diameter.

Minimum disc diameter is 2 times pile diameter

By pointing on any soil disc using the option
verify - element - information
The peak soil capacities are printed.

Relative movement of the pile and the surrounding soil is visualized as well as the utilization of the soil.

The soil displacements are easier to observe with the mesh applied.

PILEGEO ID Type Do T

Parameter	*Description*	*Default*
ID	User defined number used to identify the Pile Geometry referred to on the PILE record	
Type	Pile Type. Pile type = 1 means a single Pipe with specified outer diameter and thickness.	
Do	Outer Diameter of the pile	
T	Wall Thickness of the pile	

This record is used to define "single-pile" geometry, (type=1).

This record may be repeated.

PILEGEO ID Type Do T nPile y_loc_1 z_loc_1
y_loc_2 z_loc_2
y_loc_3 z_loc_3

Parameter	*Description*	*Default*
ID	User defined number used to identify the Pile Geometry referred to on the PILE record	
Type	Pile Type. Pile type = 2 means a pile group with specified data	
Do	Outer Diameter of piles	
T	Wall thickness of piles	
nPile	Number of identical piles in this group	
y_loc_1	Y-coord from PILE element axis to pile-1 centre, (local pile coord).	
z_loc_1	Z-coord from PILE element axis to pile-1 centre, (local pile coord).	
y_loc_2	Y-coord from PILE element axis to pile-2 centre, (local pile coord).	
z_loc_2	Z-coord from PILE element axis to pile-2 centre, (local pile coord).	
etc ...		

This record is used to define one **Pile Group** geometry, (type=2).

This record may be repeated.

PILE_D-T	ID	Z_mud	{ Z_1	Z_2	Do	T $\}_1$
			{ Z_1	Z_2	Do	T $\}_2$
			. . . .			
			{ Z_1	Z_2	Do	T $\}_n$

Parameter	*Description*	*Default*
ID	User defined number used to identify the Pile Geometry referred to on the PILE record	
Z_mud	Z-coordinate of Mudline. Pipe data are given relative to Mudline	
Z_1 , Z_2	Z-coordinates of the part of the piles to apply the specified Do and T, (relative to Mudline defined above).	
Do	Outer Diameter of piles within the actual Z-coordinate range.	
T	Wall thickness of piles within the actual Z-coordinate range.	

This record is used to define Diameter and thickness of piles varying as function of depth.

NOTE 1 : This record overrides the diameter and thickness defined under PILEGEO.
NOTE 2 : The Z-axis is pointing **upwards**.

This record may be repeated.

PILEGEO	ChgCross	Pile_ID	{ Z_1	Z_2	GeoID $\}_1$
			{ Z_1	Z_2	GeoID $\}_2$
			. . . .		
			{ Z_1	Z_2	GeoID $\}_n$

Parameter	*Description*	*Default*
ChgCross	The Keyword "ChgCross" changes the interpretation of PILEGEO	
Pile_ID	Pile ID	
Z_1 , Z_2	Z-coordinates of the part of the piles to apply the specified Geometry, (relative to Mudline defined above).	
GeoID	Actual Geometry to use for the pile btw Z1 and Z2	

This record is used to define different geometry properties of as function of depth.

NOTE 1 : This record overrides the geometry defined under the conventional PILEGEO and PILE_D-T described above.
NOTE 2 : The Z-axis is pointing **upwards**.

This record may be repeated.

PILEMAT	Pile_ID	{ Z_1	Z_2	MatID $\}_1$
		{ Z_1	Z_2	MatID $\}_2$
		. . . .		
		{ Z_1	Z_2	MatID $\}_n$

Parameter	*Description*	*Default*
Pile_ID	Pile ID	
Z_1 , Z_2	Z-coordinates of the part of the piles to apply the specified Material, (relative to Mudline defined above).	
MatID	Actual Material to use for the pile btw Z1 and Z2	

This record is used to define different material properties of as function of depth.

NOTE 1 : This record overrides the material defined under PILE.
NOTE 2 : The Z-axis is pointing **upwards**.

This record may be repeated.

SOILDIAM	ID	Z_mud	{ Z_1	Z_2	D_soil	Dummy $\}_1$
			{ Z_1	Z_2	D_soil	Dummy $\}_2$
			. . . .	. . . .		
			{ Z_1	Z_2	D_soil	Dummy $\}_n$

Parameter	*Description*	*Default*
ID	User defined number used to identify the Pile Geometry referred to on the PILE record	
Z_mud	Z_coordinate of Mudline. Pipe data are given relative to mudline	
Z_1 , Z_2	Z-coordinates of the part of the piles to apply the specified Do and T, (relative to mudline defined above).	
D_soil	Diameter to be used in the soil capacity calculations within the actual Z-coordinate range.	
Dummy	Dummy parameter, which must be typed in	

This record is used to define Diameter to be used for the soil capacity calculations if the diameter is different from the actual steel pile's outer diameter

NOTE 1 : This record will not influence the diameter and thickness used for the piles itself.

NOTE 2 : The Z-axis is pointing **upwards**.

This record may be repeated.

Example

'	*ID*	*Nod1*	*Nod2*	*Soil_ID*	*Pile_Mat*	*Pile_Geo*
PILE	1001	1	101	99	100	***10***
'	*ID*	*Type*	*Diam*	*Thick*		
PILEGEO	***10***	Single	1.22	0.030		
'	*ID*	*Z_Mud*	*Z_Top*	*Z_Bott*	*SoilDiam*	*Dummy*
SOILDIAM	***10***	Single	0.0	-50.0	1.22	0.0
			-50.0	-55.0	5.00	0.0

In this example, the steel pile has a diameter of 1.22, but from depth 50m below mudline, the soil capacity should be calculated based on a diameter = 5.0m, (the pile capacity is still based on D/T=1.22/0.030).

SOILCHAR ID Type Z_Mud D_ref F_fac L_fac Z_top_1 Z_bott_1 P-Y T-Z Q-Z
Z_top_2 Z_bott_2 P-Y T-Z Q-Z
. . . .
Z_top_n Z_bott_n P-Y T-Z Q-Z

Parameter	*Description*	*Default*
ID	User defined number used to identify the Soil Characteristics referred to on the PILE record	
Type	Soil Data Type= 1. Soil is defined through predefined curves.	
Z_Mud	Z_coordinate of Mudline. The soil data are given relative to mudline. **The Z-axis is pointing upwards!**	
D_ref	Reference Diameter, Soil data has been generated for this diameter	
F_fac	Scaling factor for the force unit used in the soil curves	
L_fac	Scaling factor for the length unit used in the soil curves	
Z_top_1 Z_bott_1	Z_coordinate to the top of soil layer 1, (relative to mudline, typical=0.0) Z_coordinate to the bottom of soil layer 1, (relative to mudline)	
P-Y	P-Y data is defined by the referred curve, (e.g. MISOPL, ELPLCURV) P is force per unit length of pile, Y is deformation of soil.	
T-Z	T-Z data is defined by the referred curve, (e.g. MISOPL, ELPLCURV) T is force per unit length of pile, Z is deformation of soil	
Q-Z	Q-Z data is defined by the referred curve, (e.g. MISOPL, ELPLCURV) Q is Tip force, Z is tip deformation.	

This record is used to define the **Soil Characteristics** referred to on the PILE record, and at the **middle of each soil layer a discrete spring** representing the soil is inserted automatically. Confer record **SPRI_MOD** below for selecting spring model to be used.

Calculation of discrete spring **Force / Displacement Characteristics** representing the pile resistance for a given thickness of a soil layer is performed according to the following formulas:

$$P-Y : Force = \left(\frac{F_{fac}}{L_{fac}}\right) * \left(\frac{D_{pile}}{D_{ref}}\right) * Thickness * p$$

$$T-Z : Force = \left(\frac{F_{fac}}{L_{fac}}\right) * \left(\frac{D_{pile}}{D_{ref}}\right) * Thickness * T$$

$$Q-Z : Force = F_{fac} * Q * \frac{D_{pile}}{D_{ref}}$$

$$All : Displacement = L_{fac} * (Soil\ deformation)$$

This record may be repeated.

SOILCHAR ID Type Z_Mud D_ref F_fac L_fac Z_top_1 Z_bott_1 Soil_ID
Z_top_2 Z_bott_2 Soil_ID
. . . .
Z_top_n Z_bott_n Soil_ID

Parameter	*Description*	*Default*
ID	User defined number used to identify the Soil Characteristics referred to on the PILE record	
Type	Soil Data Type= 2. Soil is defined through soil parameters, (e.g.: API_SOIL).	
Z_Mud	Z_coordinate of Mudline. The soil data are given relative to mudline.	
D_ref	Reference Diameter, Soil data is generated for this diameter. D_ref does not have any influence on the results *), but the generated curves will be printed for this diameter.	
F_fac	Scaling factor for the force unit. Soil curves are **always** generated using **SI units** (N and m). If e.g. the force unit is MN, F_fac=10^{-6}	
L_fac	Scaling factor for the length unit. Soil curves are **always** generated using **SI units** (N and m). If e.g. the length unit is mm, L_fac=1000	
Z_top_1 Z_bott_1	Z_coordinate to the top of soil layer 1, (relative to mudline, typical=0.0) Z_coordinate to the bottom of soil layer 1, (relative to mudline)	
Soil_ID	The actual layer is defined by a API_SOIL record with this ID	

This record is used to define the **Soil Characteristics** referred to on the PILE record, and at the middle of each soil layer a discrete spring representing the soil is inserted automatically.

This record may be repeated.

**) D_ref is used during API_SOIL curve generation and upper layers are somewhat influenced. Use D_Ref equal to the actual pile diameter if possible.*

SPRI_MOD Soilmod (pilemod)

Parameter	*Description*	*Default*
soilmod	Spring model to be used to model the soil nonlinear behaviour	1
pilemod	Beam model to be used to model the pile	0

This record is used to select the type of spring element to be used in connection with modelling of the soil nonlinear characteristics. Two alternatives are available:

soilmod = 0 : "old" nonlinear spring model, step scaling, **not recommended**.
soilmod = 1 : "new" plasticity formulation, no step scaling, iterations are needed, (is switched on automatically)

This record is given once

SPRIDAMP <Dof> < Cdamp> < Elem1> <Elem2> …….

Parameter	*Description*	*Default*
Dof	Actual Spring degree of freedom *Axial (or 1) : Axial degree of freedom of spring is damped* *ShearY (or 2) : Y translation degree of freedom is damped* *ShearZ (or 3) : Z translation degree of freedom is damped* *Torsion (or 4) : Torsion degree of freedom is damped* *Ymoment (or 5) : Y rotation degree of freedom is damped* *Zmoment (or 6) : Z rotation degree of freedom is damped*	:
Cdamp	Damping Characteristics given in consistent units. (Damper force = Cdamp * translation speed Damper moment = Cdamp * angular velocity)	
Elem1,…	List of 2 node Spring Element ID's **NOTE** : If **no** elements are specified, **all springs** are defined with the actual damping	

This record is used to define 'dashpot' damping of spring elements to be used in connection with dynamic analysis.

Example 1:

SPRIDAMP Axial 50000 10210 10220 10230

means that spring elements 10210, 10220 and 10230 get axial damping of 50000. If the units are N and m, the damping force: Force [N] = 50000 * Axial Speed [m/s]

Example 2:

SPRIDAMP Axial 50000

means that all spring elements get axial damping of 50000

Example 3:

SPRIDAMP Axial 50000
SPRIDAMP Axial 30000 10210 10220

means that all spring elements get axial damping of 50000, except for elements 10210 and 10220, which get a damping of 30000.

This record may be repeated

API_SOIL ID Soiltype LoadType { Data }		
Parameter	*Description*	*Default*
ID	User defined number used to identify the API Soil Characteristics referred to on the SOILCHAR record	
Soiltype	Soiltype SoftClay : Soft Clay StifClay : Stiff Clay Sand : Sand	
LoadType	Load type Static : Static Loading, (monotonic). Cyclic : Cyclic Loading, (stabilized cyclic behaviour of pile during complete loading history).	
{ Data }	**Clay** (Soil type 1 and 2) : { γ pl s_u ε_{50} J TResF Q_pLim } , where *unit* γ : Effective unit weight of soil. [N/m^3] pl : pl=0 : unplugged pl=1 : plugged s_u : Soil undrained shear strength. [N/m^2] ε_{50} : Strain which occurs at 50% of maximum stress on laboratory undrained compression test. J : Experimental coefficient, can be taken to 0.5 for Mexican Gulf sediments, 0.25 otherwise. TResF : Tres/TPeak ratio (ref Sec. G API RP2A 1993) Typical values : 0.7 - 0.9 Q_pLim : Limit of pile tip resistance [N/m^2] **Sand** (Soil type 3) : {γ pl ϕ δ N_q Q_pLim }, where *unit* γ : Effective unit weight of soil. [N/m^3] pl : pl=0 : unplugged pl=1 : plugged ϕ : Angle of internal friction (typical 10-45)[Deg] δ : Pile-soil interface friction (typical 15-45)[Deg] N_q : End bearing factor (typical 8-50) Q_pLim : Limit of pile tip resistance [N/m^2]	

This record is used to define the properties of the **API Soil** referred to on the SOILCHAR record.

NOTE!:
The Reference Pile Diameter (D_Ref) specified under SOILCHAR is used when the soil curves are generated. This has impact on zone with reduced strength of the upper layers, (larger pile diameter gives a deeper zone). Use D_Ref equal to the actual pile diameter, (or the larger if several diameters).

This record may be repeated.

PILEOPT	KeyWord	OptID	Type	{ Z Fac }$_1$ { Z Fac }$_2$ . . . . { Z Fac }$_n$

Parameter	*Description*	*Default*
KeyWord	Actual Pile option: - *SoilScal* : Scale actual soil parameter - *CyclDegr* : Cyclic degradation. See next page - *SoilDamp* : Soil Damping. See next page	
OptID	Option ID	
Type	Actual Parameters: - P-Y : Scale P-Y properties for specified pile(s) - T-Z : Scale T-Z properties for specified pile(s) - Q-Z : Scale Q-Z properties for specified pile(s) - Assign : Assign actual Option to one or more piles	
Z	Z_coordinate relative to mudline, (Positive Z goes upwards)	
Fac	Actual Scaling factor	

This record is used to scale soil properties used by one or more piles.

This record may be repeated.

Example, SoilScaling:

Define Pile Option 100 and assign to pile 1001. The Pile Option means that the soil properties used by pile 1001 will be scaled by a factor 0.5 for all depths. Same factor is used for P-Y, T-Z and Q-Z.

```
' ---------------------------------------------------------
'    Define Pile Options (ID=100) and Assign to Pile 1001
' ---------------------------------------------------------
'           KeyWord      ID      Type       Z     Fac
 PileOpt    SoilScal    100      P-Y        0     0.5
                                           -1     0.5
                                           -2     0.5
                                          -80     0.5

'           KeyWord      ID      Type       Z     Fac
 PileOpt    SoilScal    100      T-Z        0     0.5
                                           -1     0.5
                                           -2     0.5
                                          -80     0.5

'           KeyWord      ID      Type       Z     Fac
 PileOpt    SoilScal    100      Q-Z        0     0.5
                                           -1     0.5
                                           -2     0.5
                                          -80     0.5

'           KeyWord      ID      Type      PileID ....
 PileOpt    SoilScal    100      Assign      1001
```

PILEOPT SoilDamp OptID Type { Z Fac $\}_1$
{ Z Fac $\}_2$
. . . .
{ Z Fac $\}_n$

Parameter	*Description*	*Default*
KeyWord	Actual Pile option: - *SoilDamp* : Soil Damping	
OptID	Option ID	
Type	Actual Parameters: - P-Y : Damping of P-Y for specified pile(s) - T-Z : Damping of T-Z for specified pile(s) - Assign : Assign actual Option to one or more piles	
Z	Z_coordinate relative to mudline, (Positive Z goes upwards)	
C	Actual damping Coefficient (ref SPRIDAMP)	

This record is used to define soil damping properties used by one or more piles.

This record may be repeated.

Example Soil Damping:

Define Pile Option 100 and assign to pile 1001. The Pile Option means that the soil properties used by pile 1001 will get damping.

```
' --------------------------------------------------------
'    Define Pile Options (ID=100) and Assign to Pile 1001
' --------------------------------------------------------
'           KeyWord      ID     Type      Z    C
 PileOpt    SoilDamp    100     P-Y       0    1E4
                                         -1    1E4
                                         -2    2E4
                                        -80    2E4

'           KeyWord      ID     Type      Z    Fac
 PileOpt    SoilDamp    100     T-Z       0    2E5
                                         -1    2E5
                                         -2    3E5
                                        -80    3E5


'           KeyWord      ID     Type      Pile_ID ....
 PileOpt    SoilDamp    100     Assign      1001
```

PILEOPT	CyclDegr	OptID	Type	{ nCyc Fac }$_1$ { nCyc Fac }$_2$ . . . . { nCyc Fac }$_n$

Parameter	*Description*	*Default*
KeyWord	Actual Pile option: - *CyclDegr* : Cyclic degradation of the soil	
OptID	Option ID	
Type	Actual Parameters: - P-Y : Degradation of P-Y for specified pile(s) - T-Z : Degradation of T-Z for specified pile(s) - Assign : Assign actual Option to one or more piles	
nCyc	Number of cycles	
Fac	Degradation (damage) factor. 1 is un-damaged. 0 is completely damaged.	

This record is used to define soil degradation properties used by one or more piles.

This record may be repeated.

Example Cyclic Degradation:

Define Pile Option 100 and assign to pile 1001. The Pile Option means that the soil properties used by pile 1001 will be damaged for increasing plastic work.

```
' -------------------------------------------------------
'    Define Pile Options (ID=100) and Assign to Pile 1001
' -------------------------------------------------------
'           KeyWord      ID     Type     ncyc  Degr
 PileOpt     SoilDamp    100     P-Y        0     1
                                            1    0.9
                                            5    0.5
                                           10    0.5

'           KeyWord      ID     Type     ncyc  Degr
 PileOpt     SoilDamp    100     T-Z        0     1
                                            1    0.9
                                            5    0.4
                                           10    0.4


'           KeyWord      ID     Type      Pile_ID ....
 PileOpt     SoilDamp    100     Assign      1001
```

After one cycle, the soil is degraded to 90% of the initial strength. The soil is degraded to minium 50% for P-Y and 40% for T-Z after 5 cycles.

6.3.10 Fracture/Ductility Control

CFRACT

If **CFRACT** is specified, all beam elements are checked for fracture (plastic tension exceed a specified limit)

If **CFRACT** is not specified, no fracture checking will be performed (**MFRACT**-records are not fracture check switch)

The fracture criteria of the materials should be user defined (using **MFRACT**-records), because the local conditions (flaw size etc.), can not be predicted generally

This record is given once

NOTE ! : This option is no longer recommended!. Use the UserFrac instead

MFRACT matno crit-CTOD σ_u ε_u ε_s a

Parameter	*Description*	*Default*
matno	Material number	
Crit-CTOD	Critical crack tip opening displacement	
σ_u	Ultimate stress (typical value: 1.3 σ_y)	
ε_u	Ultimate strain corresponding to σ_u (typical value: 0.15)	
ε_s	Strain level at the beginning of hardening (typical value: 20.0 ε_y)	
a	Flaw size	

Critical CTOD and the flaw size parameter a should have the same dimension

This record specifies additional information to the **MISOIEP**-record with the same material number

Note! The value of the fracture check is strongly dependent on the input parameters. It is the responsibility of the user to give adequate data for the critical CTOD and the flaw size. Confer Sect 3.15 - "Element Fracture" for a discussion of this. The following condition should also be satisfied if the fracture check is to be used

$$\text{crit CTOD} < \pi\, \varepsilon_u \frac{\sigma_u}{\sigma_y} \cdot a$$

where σ_y = yield stress

NOTE ! : This option is no longer recommended!. Use the UserFrac instead

USERFRAC	ListType	CritType	[Criterion]	IDList

Parameter	*Description*	*Default*
ListType	Definition of the ID_List specified. Actual List Types: *Element* : The "ID_List" contains Element ID's *Material* : The "ID_List" contains Material ID's, and all beam elements referring to the actual material ID's get the actual UserFrac Parameters. *Geometry* : The "ID_List" contains Geometry ID's, and all beam elements referring to the actual Geometry ID's get the actual UserFrac Parameters.	 Cut
CritType	Type of criterion to be used to remove the element: *Cut* : just remove *Time* : at given *time* *Lcomb* : at the specified loadcomb/loadlevel *Util* : if exceeding specified utilization *Strain* : if the estimated tensile strain exceeds the specified limit. *PlastWork*: if the accumulated plastic work (normalized) exceeds the limit	
Criterion	Criterion for element removal: *Type* *[Criterion]* *Description* Cut : -- N/A. -- Time : *time* : Time for element removal Lcomb : *Lcomb Level* : Loadcomb and Loadlevel Util : *Util* : Member utilization (f ex 1.05) Strain : *Strain* : Strain level (f ex 0.15) PlastWork: Work : Accum. work (f ex 0.4)	
IDList	List of ID's for which the UserFrac should be applied. The Id's are interpreted as defined in *ListType* (Elem, Mat or Geo).	

This record may be used to force elements to be fractured.

The 'fracture' will become active the step after the step detected the fracture, and the forces in the specified element will be removed. The element will not contribute to the global stiffness in the succeeding steps

Example1

'	*ListTyp*	*CritType*	*Crit*	*ID_List*
UserFrac	**Elem**	**Strain**	0.15	1001 1002 1003

Will define a strain fracture criterion (15%) on elements 1001 1002 and 1003.

Example2

'	*ListTyp*	*CritType*	*Crit*	*IDList*
UserFrac	**Geo**	**PlastWork**	0.4	51012 51015

Will define a plastic strain energy fracture criterion (0.4) on all elements referring to Geometry ID's 51012 and 51015.

This command may be repeated.

DAMAGE KeyWord {Data} ListType IDList

Parameter	*Description*	*Default*
KeyWord	Keyword defining criterion for damage: *LoadCase* : Damage starts after specified load case / step *History* : Damage follows a time history *PlastWork* : Damage depends on the plastic work level.	
Data	If "*LoadCase*", four parameters are given: Param-1 : Load Case no Param-2 : Load Level Param-3 : Max damage on stiffness (E-mod). Range 0 – 0.95 Param-4 : Max damage on capacity (Yield) Range 0 – 0.95 If "*History*", three parameters are given: Param-1 : History ID for timehist controlling the damage Param-2 : Max damage on stiffness (E-mod). Range 0 – 0.95 Param-3 : Max damage on capacity (Yield) Range 0 – 0.95 If "*PlastWork*", two parameters are given: Param-1 : Curve ID defining stiffness degradation vs. work Param-2 : Curve ID defining capacity degradation vs. work	
ListType	Definition of the ID_List specified. Actual List Types: *Element* : The "ID_List" contains Element IDs *Material* : The "ID_List" contains Material ID's *Geometry*: The "ID_List" contains Geometry IDs *Group* : The "ID_List" contains Group IDs	
IDList	List of ID's for which the Damage should be applied.	

This record may be used to define elements to degrade.

This command may be repeated.

Example 1, Load Case

```
'         Type     Lcase  LoadLev  DamE  DamY   Incr   ListTyp  Ids
 Damage  LoadCase    1      4.0      0.9   0.9   0.05    Elem     10
```

Will start the element-damage for element 10, load case 1, level 4.0. The maximum damage is 0.9 for both stiffness (E-mod) and capacity (yield). The damage increases with 5% per step until max damage is reached.

Example 2, History

```
'         Type     HistID  DamE  DamY  ListTyp   Ids
 Damage  History    1001   0.9   0.9    Elem       7

`          ID     Type    T1   T2    Fac
 TimeHist 1001  S_Curve   5    8    1   ! Introduction of damage
```

Will start the element-damage for elem 7, at time = 5s. The maximum damage is 0.9 for both stiffness (E-mod) and capacity (yield) and is reached for time=8s.

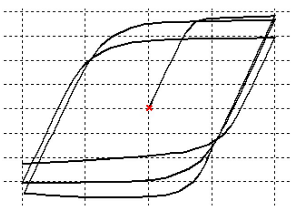
Example 3, Plastic Work (cyclic degradation)

```
'         Type      DamE  DamY  ListTyp  Ids
 Damage  PlastWork   101   102   Mat      2   3   4

'         MatID   Type    Curv     W1  W2    Fac
 Material 101   General  S_Curve  0.1 0.5  0.10  ! E-Mod
 Material 102   General  S_Curve  0.1 0.5  0.90  ! Yield
```

All elements using material 2, 3 and 4 will be checked for degradation due to accumulated plastic work (normalized). Stiffness and Capacity follow different degradation curves:

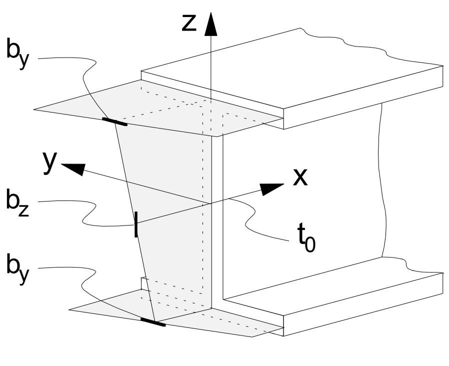
- E-mod : Degradation starts for Work=0.1 and ends for 0.5. Max degr = 0.1
- Yield : Degradation starts for Work=0.1 and ends for 0.5. Max degr = 0.9

The accumulated plastic work increases for every load cycle, and this option could be used in connection with for example earthquake analysis.



cyclic degradation

6.3.11 Fire (Temperature Response) Analysis

BELTEMP llc elnox t_0 t_{ygrad} t_{zgrad} (pfp_{crack_ang})		
Parameter	*Description*	*Default*
llc	Load case number	
elnox	External (user-defined) element number	
t_0	Element mean temperature	
t_{ygrad}	Temperature gradient in local y-direction of beam section (°C per unit length)	
t_{zgrad}	Temperature gradient in local z-direction of beam section (°C per unit length) 	
pfp_{crack_ang}	Maximum accepted plastic rotation of the beam element before the passive fire protection is assumed to be cracked/damaged. If the plastic rotation exceeds this limit, the element is removed (fractured).	

This record contains specification of temperature increments with temperature gradients over the beam element cross-section. (Mean temperature increments for the plate elements. Gradients are not given for the plates).

This record may be repeated

ELEMTEMP llc type [Dataset] Iel1 iel2 iel3 .....

Parameter	*Description*	*Default*
llc	Load case number	
type	Definition of the Dataset, see below.	
[Dataset]	Actual temperature Dataset, see below.	
Iel1, iel2 ...	External (user-defined) element number,(s). If no elements are specified, all elements get the specified temperature data.	

This record contains specification of detailed temperature increments generated by FAHTS.

This record may be repeated.

The definition of the **Dataset** is as follows:

Type = 1 : Dataset consists of 3 values: t_mean, t_grad_Y and t_grad_Z

Type = 2 : Dataset consists of an arbitrary number of values:

n x1 y1 z1 t1 x2 y2 z2 t2 .... Xn Yn Zn tn, where
n is number of points with temperature data
x,y,z are coordinates to the actual point, (local cross-sect coords)
t is actual temperature increment.

CTFRACT

If **CTFRACT** is specified, all beam elements are checked for fracture due to elevated temperature in the material

If **CTFRACT** is not specified, no temperature fracture checking will be performed

Note! The **TFRACT**-record do not activate fracture checking

This record is given once

NOTE ! : This option is no longer recommended! Use the UserFrac or PushDown instead.

TFRACT matno T-fract Sy-fract E-fract

Parameter	*Description*	*Default*
matno	Material number	
T-fract	Critical element temperature at reference axis for element fracture	
Sy-fract	Yield stress degradation factor for element fracture. Defined by the ratio: Yield stress at temperature T to the yield stress at 0^{o} C	
E-fract	Elastic modulus degradation factor for element fracture. Defined by the ratio: Elastic modulus at temperature T to the elastic modulus at 0^{o} C	

This record specifies additional information to the **MISOIEP**-record with same material number

Note! For the case when all elements connected to a node fractures, numerical problems will occur in the solution algorithm

NOTE ! : This option is no longer recommended! Use the UserFrac or PushDown instead.

STEELTDEP Curve no Mat no 1 Mat no 2 ....

Parameter	*Description*	*Default*
Curve no	Actual Steel Reduction Curve No, see below.	
Mat no	Material numbers, (referring to MISOIEP), to be defined temperature dependent. If no material numbers are specified, all materials (MISOIEP) get this temperature curve.	

With this record, the user specifies the temperature dependent material properties by pre-defined Steel curves.

ALUMTDEP Curve no Mat no 1 Mat no 2 ....

Parameter	*Description*	*Default*
Curve no	Actual Aluminium Reduction Curve No, see below.	
Mat no 1	Material numbers, (referring to MISOIEP), to be defined temperature dependent. If no material numbers are specified, all materials get this temperature curve.	

With this record, the user specifies the temperature dependent material properties by pre-defined Aluminium curves.

TDEPFUNC Curve no Type [Data]

Parameter	*Description*	*Default*
Curve no	Curve ID no to be referred to from the records *STEELTDEP*, *ALUMTDEP*, and *USERTDEP.	
Type	Function type: 1: Polynomial expression 2: (Vacant)	
[Data]	Parameters required for the actual function type. For function type = 1 the required data are: The coefficients: A_0, A_1, A_2,..., A_n to be used in the expression: $f(T) = A_0 + A_1T + A_2T^2 + ... + A_nT^n$	

With this record, the user specifies a temperature-reduction curve used to control the material properties at different temperature levels.

This record may be repeated.

USERTDEP Mat no Dep E Dep Yield Dep Yield2 Dep Exp.

Parameter	*Description*	*Default*
Mat no	Actual Material ID no to be defined temperature dependent, (ref to MISOIEP with same ID no).	
Dep E	**E_mod** defined in MISOIEP record is scaled according to the defined TEMPDEPY curve number. Omitted or 0 means default curve.	0
Dep Yield Dep Yield2	**Effective Yield Stress** referring to the Yield Stress defined in MISOIEP record is scaled according to the defined TEMPDEPY curve. If omitted or 0, the curve for initial yielding is used. (DepYield2 is dummy and should be set equal to Dep Yield).	0
Dep Exp.	Thermal Expansion Coefficient defined in MISOIEP record is scaled according to the defined TEMPDEPY curve number. Omitted or 0 means default, (temperature independent).	0

With this record, the user specifies the temperature dependent material properties by referring to user defined temperature-reduction curves.

Example:
```
'           MatID  Dep E  Dep Yield   Dep Yield2   DepExp
 USERTDEP   210355   1       2            2          0
```

The Stiffness (E-Module) is scaled according to TEMPDEPY curve no 1 and the capacity (Yield) is scaled according to TEMPDEPY curve no 2.

This record may be repeated.

LCASETIM L-Case Time

Parameter	*Description*	*Default*
L_Case	Actual load case number to be connected to a time.	
Time	Time corresponding to completed load case.	

With this record the user may connect a time to each load case. This is f. inst. used in connection with fire analyses where the different load cases, (defined through BELTEMP), correspond to a time, (duration of the fire).

This record may be repeated.

PUSHDOWN KeyWord Value

Parameter	*Description*	*Default*
Keyword	Keyword describing the actual "Value":	
	LoadCase : Actual Fire Load Case to use. Positive LCase : Temp at actual state. Negative LCase : "All-time-high" up to actual state.	-
	AtTime : Actual Fire Time *) to use.	
	UpToTime : "All-time-high" up to specified Fire Time.	
	Sy_Degrad : Lowest accepted Yield reduction factor.	0.001
	E_Degrad : Lowest accepted E-Mod reduction factor	0.001
	MeltTemp : Highest accepted Element temperature.	2000
Value	Actual parameter value	

*) The **exact** fire time must exist on the beltemp-file defined by the LCASETIM command.

This record is used for Fire Degradation Analysis, where the RSR_{FIRE} should be computed.

A normal static analysis is performed, but with the temperature at a given temperature load step. The material, and section properties are adjusted for every element according to the actual temperature.

If the Yield, or E-mod or Temp criterions are exceeded for an element, the element is removed (becomes NonStru).

The PUSHDOWN option replaces the previous ¯K option.

This record could be repeated.

6.3.12 Ship Impact analysis

BIMPACT ldcs elnox elpos energy extent. xdir ydir zdir ship		
Parameter	*Description*	*Default*
ldcs	Load case number	
elnox	External (user-defined) element number	
elpos	Position of impact 1: Impact on member end 1 2: Impact on member end 2	
energy	Impact energy (energy unit) = (yield stress unit) · (length unit)^3 If σ_y in MN/m^2 and length in m ==> give energy in MNm	
extent.	Extension of the impact area along the beam length (given in the actual length unit)	
xdir ydir zdir	Direction of the impact, referred to global coordinates	
ship	Reference number for local ship indentation characteristics (record MSHIP) 0: All energy is absorbed by the structure >0: Local indentation and energy absorption in the ship is calculated according ship "material" number, ship (MSHIP data)	

This record is used to define ship impact load and may be repeated

The impact energy must be specified in units consistent to external loads and nodal deformations

$$W = F * u$$

When the total impact energy has been dissipated, the impact load will be unloaded in a separate, program-defined load case

Note that both excessive member straining (fracture) and joint failure must be evaluated in connection with the impact simulation. These checks can be included in the USFOS analysis (records **CHJOINT**, **CFRACT**, **MFRACT**), but the user should subject the results to a separate validation. In particular the fracture criterion parameters should be carefully evaluated before application

Local denting under the impact load is only calculated for tubular sections

The impact analysis will be terminated if fracture occurs

The "CUSFOS" line is somewhat different from the 'normal' for the ship impact loadcase

Example follows (see CUSFOS description):

<lcase>	*<lfact>*	*<mxld>*	*<nstep>*	*<minstp>*
3	1.0	0.0	50	0.01

lcomb: Ship impact load case no

lfact: Load factor for the nodal load suggested by USFOS, *lfact* = 1.0 is recommended

mxld: Max value of the suggested nodal load. The maximum is unknown, and is set equal to 0

nstep: Max number of steps used to reach the defined energy

minstp: Min step, see "CUSFOS"

MSHIP ship p1 p2 p3 d1

Parameter	*Description*	*Default*
ship	Ship "material" number	
p1	Collision load at zero indentation	
p2	Collision load at 50% of maximum indentation	
p3	Collision load at maximum indentation	
d1	Maximum indentation	

This record is used to define local ship indentation characteristics

This record may be repeated

The ship indentation and corresponding forces must be specified in units corresponding to external loads and nodal deformations. As default, the values recommended in /9/ are implemented, defined in (MN) (MegaNewtons) and (m) (meters), respectively

If the impact force exceeds the maximum indentation force (*p3*), no further indentation of ship energy absorption is calculated

Default ship-indentation data are used if *p1*, *p2*, *p3* and *d1* are left out

MULT_IMP

Parameter	*Description*	*Default*

This record is used to switch ON multiple impact analysis, which means that several BIMPACT record could be executed in sequence. The option is used in connection with fracture control of impacted members, and makes it possible to let the impact energy be absorbed by several members.

This record is given once

Example:

```
'              LCase   Elem   Pos    Energy   extent xdir ydir zdir ship
BIMPACT          2      11     2     14E6      0.0    1    0    0    0
BIMPACT          3     121     2      0.0      0.0    1    0    0    0
MULT_IMP
             Elem    TypCrit       Crit
USERFRAC      11     Strain        0.15

CUSFOS   10   15  1.0  1.0
'            LCase   lFact    mxld    nStep    MinStp
               2      1.0      0.0     200      0.01
               3      1.0      0.0     200      0.01
```

It is specified an impact on element 11, and for strain above 15%, the element becomes fractured. If element 11 failed before all 14 MJ was absorbed, the remaining energy will be applied as a new impact on element 121. (Load case 2 is terminated when elem 11 fails, and next Cusfos "line" is activated. Load Case 3 is also a BIMPACT load, and the remaining energy from loadcase 2 is added to the energy specified for BIMPACT 3 (which here is 0.0).

BIMPDATA Type { Data }

Parameter	*Description*	*Default*
Type	Data Type to be specified. *PileThick* : Specification of Pile-In-Leg data. "Data" contains the thickness and a list of Elements to which the pile in leg data should be applied.	
Data	On or more numbers defining the Type. .	

This record is used to specify additional data to be used for Boat impact analysis.

This record is given once

Example:

```
‘              Type       Tick    Legelm1  LegElem2 ,, …
BIMPDATA       PileThick  0.035     103      203      303
```

Inside leg elements 102, 203 and 303, piles with thickness 0.035m exist. The pile will have impact on the local (wall) dent growth caused by the concentrated boat impact. For boat impact on elements 103 – 303 the effect from the inner pile will be accounted for.

DYNIMPCT	ldcs	elnox	elpos	V_ship	Mass	xdir	ydir	zdir	NL_ship	Time

Parameter	*Description*	*Default*
ldcs	Load case number	
elnox	External (user-defined) element number	
elpos	Position of impact 1: Impact at member end 1 2: Impact at member end 2 3: Impact at member midspan, (element is subdivided)	
V_ship	Ship Speed in the actual direction	
Mass	Ship Mass	
xdir ydir zdir	Direction of the impact, referred to global coordinates	
NL_ship	Reference number for local ship indentation characteristics (record *Material*, type "Curve" or *ElPlCurv/HypElast*) 0: Default elastic spring with 'infinite' stiffness. >0: Resulting hit member and ship indentation and energy is calculated according to the nonlinear spring material <0: No spring is inserted between the mass and the structure.	0
Time	Time for ship mass rebound. If omitted, the ship mass is detached from the structure when the inserted spring force changes from compression to tension. NOTE: Mandatory if NL_ship < 0!	

This record is used to define ship impact load and is given once.

The Ship **speed** and **mass** should be specified in SI-units, (kg, m and s).

This record is **not** available together with **BIMPACT** and **MSHIP**.

The option is available together with Dynamic Analysis only!

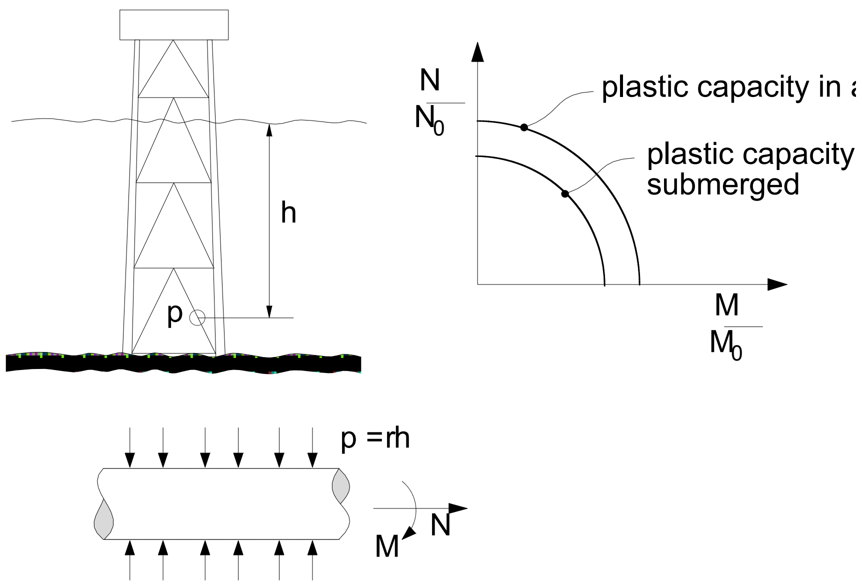
6.3.13 External pressure Effects

EXTPRES elnox1 elnox2 elnox3		
Parameter	*Description*	*Default*
elnox<i>	External element number i	

This record is used to identify tubular beam elements with external hydrostatic pressure

On basis of the element position, the hydrostatic pressure is calculated and imposed on the element at the initiation of the analysis, i.e. prior to imposing loads specified in the CUSFOS or CICYFOS records. The external pressure option affects the plastic axial and moment capacities of the tubular cross section

Note! Sea surface level must be specified by use of the SURFLEV record



SURFLEV hisurf losurf density gravity

Parameter	*Description*	*Default*
hisurf	Sea surface level defined by the global reference system z-coordinate	
losurf	Sea bottom level defined by the global reference system z-coordinate	
density	Fluid density (use consistent units!)	
gravity	Acceleration of gravity in global z-direction (use consistent units!)	

This record is used to identify the sea surface level and sea bottom position relative to the structure, as well as properties of the surrounding fluid.

NOTE ! This record does **not** define the data required by the wave load routines !
To be used together with the EXTPRES option only !

6.3.14 Super-element / Sub-structure Modelling

SUPERELM Elem ID nNodes Nodex_1 Nodex_2 .... Nodex_n Material

Parameter	*Description*	*Default*
Elem ID	Element ID of the super element	
nNodes	Number of Element **Super Nodes**, (nodes to be connected to the global frame model).	
Nodex_1	User defined node ID in the global frame model to which super node 1 is connected to	
Nodex_2	User defined node ID in the global frame model to which super node 2 is connected to	
Nodex_n	User defined node ID in the global frame model to which the last super node is connected to	
Material	Material ID number referring to the *MISOIEP* record defining the nonlinear material properties of the super element. If material ref. is omitted, Material defined on the local super element file is used.	

With this record, the user defines a superelement to be included in the frame model, (e.g. a shell model of a tubular joint). *(Not yet implemented)*.

For each substructure, following USFOS prompt will appear:

    Super Element no .....   prefix :

The super element file (with extension **.fem**), contains all necessary FEM-data.

The super element data might be defined in any of the available file formats.

This record may be repeated.

SUBSTRU matno

Parameter	*Description*	*Default*
matno	Material number for the super element stiffness matrix which is generated on the basis of the input substructures	

This record may be used to perform a substructure analysis in order to calculate the super element stiffness matrix (by means of static condensation of the internal nodes) for the structure in the geometry file. The super element stiffness matrix which is identified by the material number *matno* specified by the user, may the be input to a subsequent USFOS analysis

The super element stiffness matrix is written to the print file according to the "AMATRIX" and "ADMSTIFF" record formats

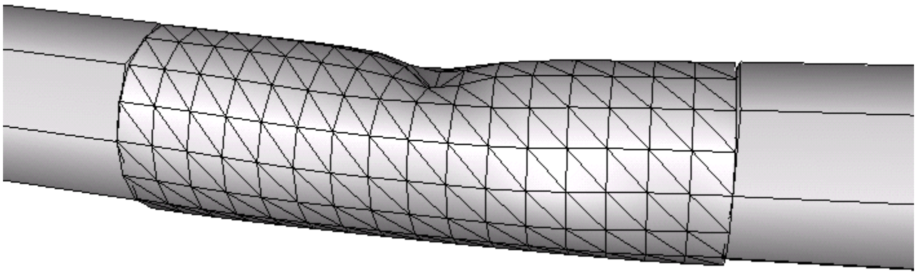
The element super nodes have to be specified on the BNBCD records

The super element matrix is output in the global reference system

SUBSHELL Elem ID

Parameter	*Description*	*Default*
Elem ID	Element ID of beam element to be replaced by an *automatically* generated *shell element substructure*.	

With this record, the user specifies the beam element, which will be replaced by an automatically generated shell element substructure. The beam element properties, (material, wall thickness) are transferred to the shell model.



Results for the shell substructure are presented in Xact together with the rest of the structure.

This record may be repeated.

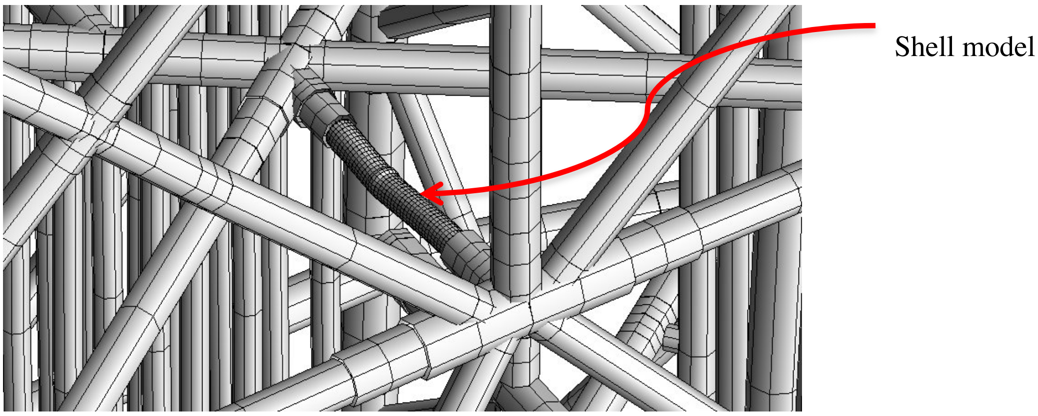
SUBSHELL Elem ID DumpFEM (IOP IDAdd iTranFEM)

Parameter	*Description*	*Default*
Elem ID	Element ID of beam element to be replaced by an *automatically* generated *shell element substructure*.	
DumpFEM	Keyword defining "Dump FE-model"	
IOP	Option for FE-model. IOP=1, shell only. IOP=2, connection from shell to to beam axis included.	2
IDAdd	Number to be added to the generated shell IDs	40E6
iTranFEM	Switch for transforming local shell model to global system.	1

With this record, the user specifies automatic generating of shell elements to be included in a subsequent analysis. The shell element is based on the properties of the specified beam element.

```
'            ID   KeyWord   IOP    IDAdd    iTranFEM
 SUBSHELL     2   DumpFEM    2      40E6       1         ! Full input
```
or
```
'            ID   KeyWord
 SUBSHELL     2   DumpFEM                                ! Minimum input
```

Will **both** generate a local shell model of element 2. The generated shell model gets IDs like: 40 000 001, 40 000 002 etc. to avoid overlap with user's model. The generated shell model will be transformed to the global coordinate system and is therefore "ready-to-use" in the global analysis.



The user is asked for the file prefix to store the generated shell model.

```
     ======================================
     ====  Writing  UFO  File ...      ====
     ======================================

 UFO structural file prefix        :
```

Answer for example shell, and the file ***shell_ufo.fem*** is generated. The file contains a conventional FE-model to be used as the "opt" file in USFOS. The file contains also the command:

```
 NONSTRU Element 2
```

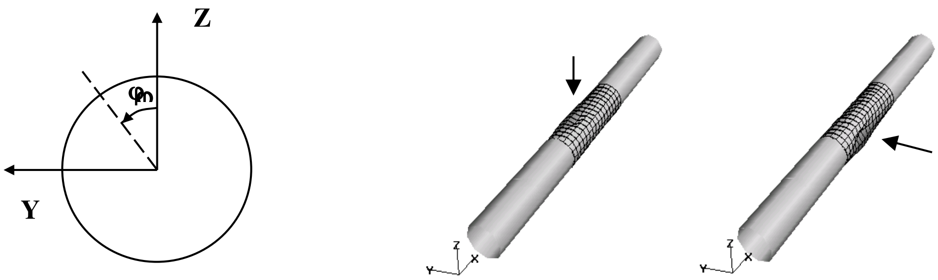
and the original beam element is set "NONSTRU" automatically. Loads are transferred to the original end nodes.

This record is given once.

SSH_LOAD Lcase Elem ID Fx Fy Fz Typ X_c φ_c Ext_X Ext_Arch

Parameter	*Description*	*Default*
Lcase	Load Case number	
Elem ID	Element ID of the beam element (subshell must have been defined)	
Fx, Fy, Fz	Total force to be applied referring to **Global** Coordinate System	
Typ	Cross section type of the beam	
X_c	Centre of the load in x-direction referring to **local** beam coord system	
φ_c	Centre of the load in φ-direction referring to **local** beam coord system	
Ext_X	Extent of load in X-direction	
Ext_Arch	Extent of load in φ-direction	

With this record, the user specifies a local load to be applied on a Shell Beam element.

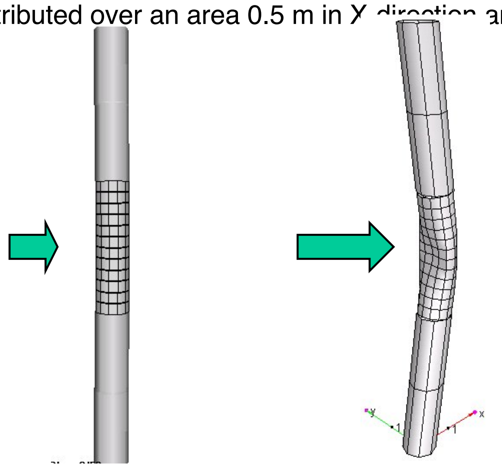


Definition of φ ***φ_c = 0°*** ***φ_c = -90°***

Example :

'	*LCase*	*Elem_ID*	*Fx*	*Fy*	*Fz*	*Typ*	*Xc [m]*	*Phi [Deg]*	*Ext_X[m]*	*Ext_Arch [Deg]*
SSH_Load	*2*	*23*	*10.0E3*	*0*	*0*	*Pipe*	*1.5*	*-90*	*0.5*	*30*

In the example, a force in **global** X-direction is applied on beam element no 23. The load is attacking the beam at position x=1.5 m and φ = 90° (referring to **local** beam coordinate system). The total Fx force is distributed over an area 0.5 m in X direction and 30° in circumferential direction.

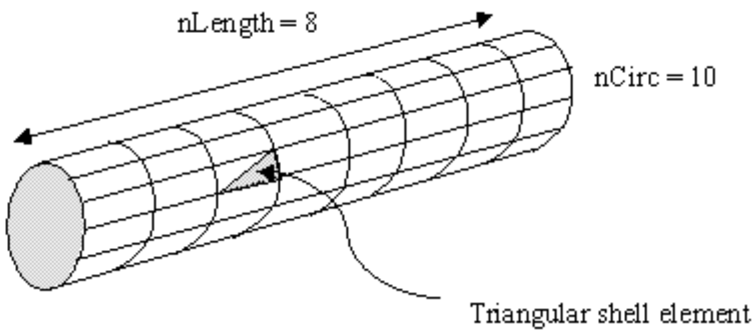


This record may be repeated.

MESHPIPE	nLength nCirc Elem ID_1 Elem ID_2 Elem ID_3 ………..	
Parameter	*Description*	*Default*
nLength nCirc	Number of shell elements in longitudinal direction Number of shell elements in circumferential direction	
Elem ID	Element ID of beam element(s) with pipe cross-section.	

With this record, the user specifies the mesh density of beam element which will be replaced by an automatically generated shell element substructure

Note that the *nLength*/*nCirc* parameters define the quadrilateral mesh, and if triangular elements are used, two elements are defined per quad as seen in the figure below.



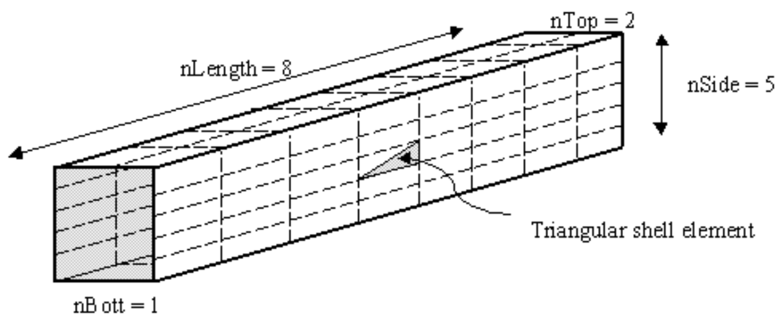
This record may be repeated.

MESHBOX nLength nSide nTop nBott Elem ID_1 Elem ID_2 .

Parameter	*Description*	*Default*
nLength nSide nTop nBott	Number of shell elements in longitudinal direction Number of shell elements on the two profile sides Number of shell elements on top "flange" Number of shell elements on bottom "flange"	
Elem ID	Element ID of beam element(s) with ***box*** cross-section.	

With this record, the user specifies the mesh density of beam element which will be replaced by an automatically generated shell element substructure

Note that the *nLength*/*nSide/nTop/nBott* parameters define the quadrilateral mesh, and if triangular elements are used, two elements are defined per quad as seen in the figure below.

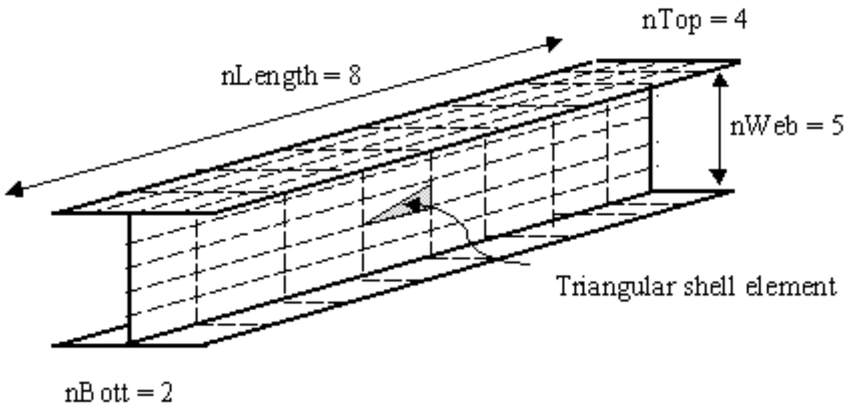


This record may be repeated.

MESHIPRO nLength nWeb nTop nBott Elem ID_1 Elem ID_2 .		
Parameter	*Description*	*Default*
nLength nWeb nTop nBott	Number of shell elements in longitudinal direction Number of shell elements on the profile web Number of shell elements on top “flange” Number of shell elements on bottom “flange”	
Elem ID	Element ID of beam element(s) with I - cross section.	

With this record, the user specifies the mesh density of beam element which will be replaced by an automatically generated shell element substructure

Note that the *nLength*/*nWeb/nTop/nBott* parameters define the quadrilateral mesh, and if triangular elements are used, two elements are defined per quad as seen in the figure below.



This record may be repeated.

6.3.15 Miscellaneous

ACTIVELM type l-comb elnox1 elnox2 ......
(time)

Parameter	Description	Default
type	Activating type = 1	
l-comb (time)	Static analysis : The elements are activated at the specified *loadcomb* Dynamic analysis : The elements are activated at the specified ***time***	
elnoxi	External (user defined) element numbers of elements to be activated.	

The user may use this record to control the activation of elements (that is, when one or more elements should contribute to the system stiffness)

Example 1 , Static analysis:

```
'           Typ  Lcomb    Elem_ID....
ACTIVELM     1     5      1001   1002
```

means that elements 1001 and 1002 are activated at load combination 5.

Example 2 , Dynamic analysis:

```
'           Typ  Time     Elem_ID....
ACTIVELM     1    25      1001   1002
```

means that elements 1001 and 1002 are activated at time = 25 seconds.

This record may be repeated

TOTL2INC lc-start lc-end

Parameter	Description	Default
lc-start	First load-case of the total load	
lc-end	Last load-case of the total load	

With this record, the user defined **total** load-vectors are transformed to **incremental** load vectors.

Load-cases with loadcase-number less than *lc-start* or greater then *lc-end* are not changed

Nodal loads, element-loads and temperature loads are processed

Note! The load cases must be given successively within the defined loadcase interval.

This record is given only once.

COROLOAD L_Case Elem_1 Elem_2 . . . .

Parameter	*Description*	*Default*
L_Case	Actual load case number	
Elem_1	Element 1 having a co-rotated load applied	
Elem_2	Element 2 having a co-rotated load applied	

With this record the user may specify load cases and elements for which the distributed load should be treated as co-rotated loads defined in the ***local beam co-ordinate*** systems.

NOTE! The distributed loads are defined in the local element co-ordinate system for the beams/load cases defined. A z-component, will f. ex. be applied in the beam local z-axis.

This record may be repeated.

SI Type nMst nSlv { Mst_1 mst_2 …mst_{nMst}}. { Slv_1 slv_2 …slv_{nSlv}}.

Parameter	*Description*	*Default*
Type	Sliding Interface Type: MatBeam : Beam/Beam contact specified through material ID's	
nMSt	Number of master surfaces	
nSlv	Number of slave surfaces	
Mst_1 -Mst_n	Master Surface material ID's	
Slv_1 - Slv_n	Slave Surface material ID's	

With this record the user may specify activation of the contact search procedure, which automatically detects contact between structural members.

This record may be repeated.

Invisible Type Id_1 Id_2 ....

Parameter	*Description*	*Default*
Type	Data type used to specify invisible elements: Material : The specified Id's are material numbers. All non-linear spring elements referred to these materials are not visualized in Xact	
Id_1 Id_2	Nonlinear spring Material number l Nonlinear spring Material number 2 etc...	

With this record the user may specify nonlinear spring elements to become invisible in XACT.

Example 1 :

Assuming materials 101 and 102 are of type MREF. The command:

Invisible Material 101 102

will make all non-linear spring elements referred to any of the two materials invisible in XACT.

This record may be repeated.

GROUPDEF ID ListType { List }

Parameter	*Description*	*Default*
ID	Group ID	
ListType	Definition Type: *Element* : Element ID list *Material* : Material ID list *Geometry*: Geometry ID list *Group* : Group ID list	
{ List }	List of actual ID's	

This record is used to define an element group, which could be referred to in connection with specification of f ex NONSTRU input.

Example 1: GroupDef 88 Geom 27304 27305 27306

All elements referring to one of the cross section geometry ID's: 27304. 27305 or 27306 will become a 'member' in "Geometry Group 88".

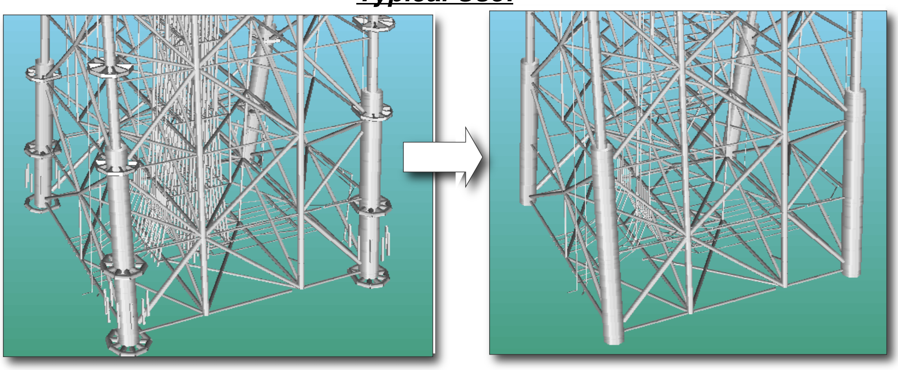
Example 2: GroupDef 66 Mat 210355 210345

All elements referring to one of material ID's: 210355 or 210345 will become a 'member' in "Material Group 66".

Example 3: GroupDef 1001 Group 66 88

All elements, which are members in one of the groups with ID's: 66 or 88 will become a 'member' in "Group of groups 1001".

Typical Use:



Original Model

Using a few commands, a large number of elements are defined "Non Structural".

This record may be repeated

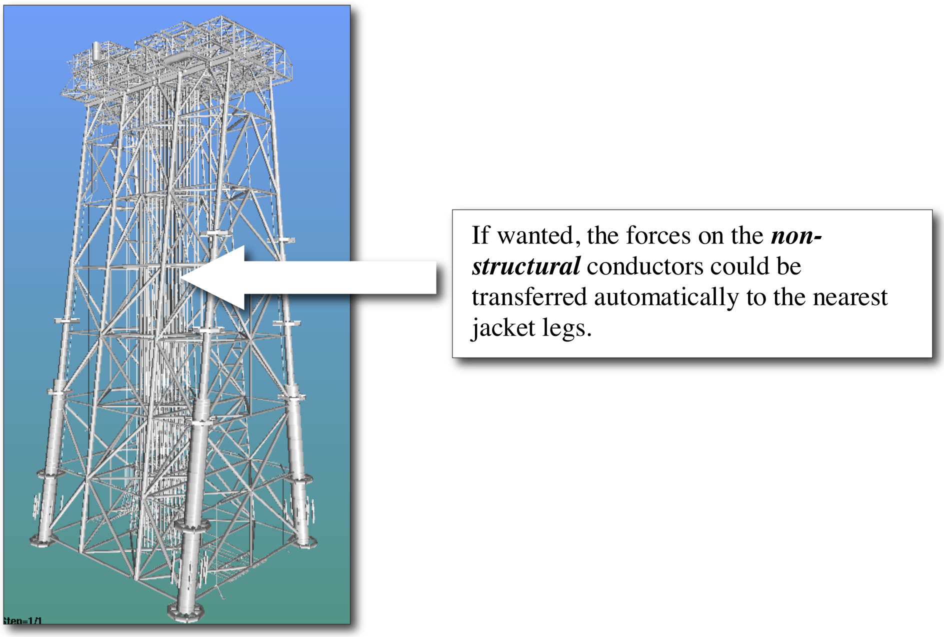
GROUPNOD	GroupID	$Node_1$.... $Node_2$.... $Node_3$........ $Node_i$

Parameter	*Description*	*Default*
GroupID	Actual Group ID	
$Node_i$	Nodes to be included in the actual group.	

With this command the user may specify nodes to become members in the actual group.

By default, the nodes to which the elements in a given group are attached to become members in the actual group. In connection with the NONSTRU command, loads on non-structural members are transferred to nearest node *within same group* with at least one *structural* member attached.

If the loads should be “guided” towards special nodes, (if, for example, all nodes in a given group are attached to non structural members only) including these nodes in the actual group will ensure correct load transfer.



This record may be repeated.

Chg_Mat **Mat_ID** Type {Id_List}

Parameter	*Description*	*Default*
Mat_ID	Material ID to be used (to override existing reference)	
Type	Data type used to specify the elements: Element : The specified Id's are element numbers. Geo : The specified Id's are geometry numbers. Group : The specified Id's are group numbers.	
{Id_List}	One or several id's separated by space	

With this record the user may override the material referred to on the structural file. In the analysis the specified material is used for all listed elements. The elements are listed either directly (element list) or in directly (using geometry or group references).

Example 1 *:*

Chg_Mat 210420 Elem 101 102 103

The command implies that elements 101 102 103 will used material property 210420 in the analysis independent on what materials the elements are referred to on the structural file.

Example 2 *:*

Chg_Mat 210420 Geo 52012 55013 60015

The command implies that all elements referring to one of the geometries 52012 55013 60015 will used material property 210420 in the analysis independent on what materials the elements are referred to on the structural file

This record may be repeated.

Chg_Boun ix iy iz irx iry irz Type {Id_List}

Parameter	*Description*	*Default*
ix, iy,…	Boundary cond code for the 6 dofs. 0: Free, 1: Fixed	
Type	Data type used to specify the node(s): Node : The specified Id's are node numbers. All : All nodes get the actual boundary code	
{Id_List}	One or several id's separated by space	

With this record the user may override the boundary conditions defined on the structural file. In the analysis the specified boundary codes are applied to all listed nodes. The nodes are listed directly (node ID list) or all in once.

Example 1 :

Chg_Boun 1 1 1 0 0 0 ***Node*** 101 102 103

The command implies that nodes 101 102 103 will be *translation fixed* and *rotation free* in the analysis, (independent on how the boundary conditions are defined in the structural file).

Example 2 :

Chg_Boun 0 0 0 0 0 0 ***All***
Chg_Boun 1 1 1 1 1 1 ***Node*** 1001 1002 1003 1004

The command implies that all nodes get boundary condition *free* for all degrees of freedom, except for the four nodes 1001 1002 1003 1004, which are fully fixed.

This record may be repeated.

SWITCHES ***Keyword*** *SubKey* Value

Parameter	*Description*	*Default*
KeyWord SubKey	Definition of actual switch / parameter. Available keywords / Subkeys: *KeyWord* *Description* **General** **:** General Settings **Defaults** **:** Version defaults **WaveData** **:** Hydrodynamic options **StatusPrint** **:** Status.text file options **Write** : Misc. dump of FE-data **Iteration** : Iteration procedure **Nodedata** **:** Node definition options **Solution** **:** Solver options **StrainCalc** **:** Strain Calculation options **Results** **:** Result generation options **WindData** **:** Wind response options **Earthquake** **:** Earthquake options	

With this record the user may set specific analysis or result parameters.

Examples *:*

SWITCHES	WaveData	Timeinc	0.1	! Hydrodyn force calc every 0.1sec
SWITCHES	NodeData	DoublyDef	ON	! Accept repeated definition of same node
SWITCHES	StatusPrint	MaxElem	50	! Extend list in *res_status.text*
SWITCHES	Results	ShellComp	19	! Compute and store 19 shell results
SWITCHES	EarthQuake	Delay	10	! Delay the motion history with 10 sec

This record may be repeated.

See next page for complete list of SWITCHES options.

KeyWord	SubKey	Value	Description	Default
General	*IndefLimit*		Min / Max imperfection (in CINIDEF).	0.05 / 1%
Defaults	*Version*	ver	850: switch to version 8-5 defaults	870
WaveData	*TimeInc*	val	Time between each hydrodyn calc.	Every
	NoDoppler	-	Switches OFF Doppler effects.	ON
	NoStore		Switches OFF storing of wave data for visualize.	ON
	TidalLevel	Level	Specify Tidal Level	0
	Accuracy	val	Change accuracy. 0: old accur, 1: new accur	1
	SeaDim	X , Y-dim	Specify size of sea surface used in xact	2 λ
	StreamOrd	order	Stream Function order	10
	SecOrder	ON/OFF	Switch on Second order wave selected members	OFF
	KC_CdDepth	ON/OFF	Reference Z. ON: Elem location, OFF: MWL	ON
NodeData	*DoublyDef*	ON/OFF	ON: Accept doubly defined nodes with same coo	OFF
StatusPrint	*MaxElem*	val	Max element in status print	10
Iterations	*RLF_Calc*	-	Activate "Residual Load Factor" method	OFF
Write	*FE_Model*	IDAdd Case stp	Writes deformed FE model at given case stp	OFF
	LinDepAlt	-	Writes ZL-springs for each BLINDP2	Off
Solution	*FracRepeat*	MxRep	Max fracture repeat	10
	PlateEdge	ON/OFF	Avoiding I-girder to buckle about weak axis if the beam element is attached to a plate element	OFF
	Impact	*UnLoFact*	Load factor during unloading after boat impact	0.02
	PanCake	*ON/OFF*	Account for "pancake" failure	OFF
	HingeLim	*L/D*	Restrict hinges for short elements. Recommended	OFF
StrainCalc	*InclDent*	ON/OFF	OFF: not included. ON: included	ON
	Algorithm	Val	0: old. 2: new, incremental.	2
	Visualization	ON/OFF	Including Gradients. ON/OFF	ON
Results	*ShellComp*	Val	Number of shell results	5
	Overturn	Val	Specify X Y Z for overturn moment calculation	Estim.
WindData	*ReynDep*	ON/OFF	Switch to Reynolds-number dependent Cd	OFF
EarthQuake	*Delay*	Val	Delays earthquake with specified time	0
	Stretch	Val	Stretches the motion history with specified value	1

KeyWord	SubKey	Value	Description	Default
Joint	*ShortCan*	ON/OFF	Detect and account for short can effect	OFF
	EccUpdate	ON/OFF	"Repair" joint ecc to avoid short joint elements	OFF
	EyeLift	Val	Location of joint surface node. 1.0 is on leg surf.	1.2
FE_Model	*Hing2Elm*	ON/OFF	Replace BEAMHING with ZL-spring	OFF
	Hing2Elm	HingStf	Specify Stf of "fixed" dofs	Estim.
	Hing2Elm	ReleaseS	Specify Stf of released dofs.	0.0
	Hing2Elm	IDAdd	Specify number to be added to generated IDs	77E6
Soil	*DiscVisual*	Val	Specify PY and TZ relative weight factor for size	1 100
DentPlot	*Store*	ON/OFF	Stores dent depth to be visualized in xact	OFF
ShellOpt	*AxCorrect*	ON/OFF	Corrects under prediction of Axial resistance.	ON

SpriScal	KeyWord	LoadCase	MatIDs

Parameter	*Description*	*Default*
KeyWord	Increase : Introduce spring. (Increase from zero) Decrease : Remove spring. (Reduce to zero)	
LoadCase	Load case for introduction / removal of spring	
MatID	One or several Material id's separated by space. The materials are either of type MREF or HYPELAST.	

With this record the user may scale the properties of non-linear springs. It is typically used to simulate changing boundary conditions during the analysis.

The scaling starts at the specified load case. If the option "increase" is selected, the spring is "dead" up to the specified load case and gets the actual stiffness when the "wake-up" case is completed.

If "decrease" is selected, it is opposite: The spring has the specified stiffness from the beginning of the simulations and will "fade out" during the load case. When the load case is completed, the spring is "dead", (zero stiffness).

Example 1 *:*

```
‘              KeyWord       LoadCase   Material
SpriScal    Increase           2       1000

CUSFOS    10    100   1    1
'          lc      inc  max  n     min
            1      0.1   1  100   0.001  ! Basic Load
            2      0.1   1  100   0.001  ! Increase Spring (dummy load)
            1      0.1   2  100   0.001  ! Continue  Load
```

Non-linear springs referring to material 1000 will "wake-up" gradually (in 10 steps) from load case 2. When load case 1 continues, the spring is 100% present.

Example 2 *:*

```
‘              KeyWord       LoadCase   Material
SpriScal    decrease           2       1000

CUSFOS    10    100   1    1
'          lc      inc  max  n     min
            1      0.1   1  100   0.001  ! Basic Load
            2      0.1   1  100   0.001  ! decrease Spring (dummy load)
            1      0.1   2  100   0.001  ! Continue  Load
```

Non-linear springs referring to material 1000 will be present in the analysis (contribute to stiffness) from the start of the analysis. During load case 2, the springs will gradually reduce the stiffness. When load case 1 continues, the springs are no longer present.

This record may be repeated.

ILLEGAL KeyWord SubKey [Data]

Parameter	*Description*	*Default*
Keyword	Keyword describing the actual "Value":	
	BeamLength : Re-defining Minimum Beam Length to diameter ratio.	0.5
	Eccent : Re-defining Max Eccent to Length ratio.	0.5
	SoilThick : Re-defining Minimum soilthick to pile diameter ratio	0.5
	ShellAngle : Re-defining Minimum corner angle of shells.	10°
SubKey	*Accept* : "Data" defines new acceptance criterion	
	Element : "Data" contains a list of elements	
	Pile : "Data" contains a list of piles	
	UsersRisk : If "Data" is "ON", the user accepts all illegal elements without explicit specification.	

This record is used to bypass the element mesh checking if illegal shaped elements are detected by USFOS. The bad shaped elements are printed on the "out" file and in special "Label" files. Bad shaped elements could cause numerical problems and are not recommended. Bad-shaped, elastic elements with no significance for the structural ultimate strength or actual results could be accepted.

NOTE ! The user is 100% responsible for the element mesh used in the USFOS analysis!

Examples:

```
'          Key-1      Opt    Value
ILLEGAL BeamLength  Accept  0.1       ! Redefine Minimum Beam length/Diam ratio to 0.1
ILLEGAL BeamLength  Elem   1001 1002 ! Accept short elements 1001 and 1002

'          Key-1      Opt    Value
ILLEGAL BeamLength  Accept   0.1     ! Redefine Minimum Beam length/Diam ratio to 0.1
ILLEGAL BeamLength  UsersRisk ON     ! Accept all short elements

'          Key-1      Opt    Value
ILLEGAL SoilThick  Accept   0.4      ! Redefine Minimum SoilThick / Diam ratio to 0.4
ILLEGAL SoilThick  Pile   9001 9002  ! Accept thin soil for piles 9001 and 9002

'          Key-1      Opt    Value
ILLEGAL Eccent     Accept   2.0      ! Redefine Max Eccent / Beam length ratio to 2.0
ILLEGAL Eccent     Elem   1001 1002  ! Accept big eccents for elements 1001 and 1002

'          Key-1      Opt    Value
ILLEGAL ShellAngle Accept   7.0      ! Redefine Min corner angle to 7 degrees
ILLEGAL ShellAngle Elem     10 20    ! Accept small angles for shell elements 10 and 20
```

This record could be repeated.

6.3.16 Hydrodynamics

WAVEDATA	I_case	Type	Height	Period	Direct	Phase	Surflev	Depth	N_ini	
									X_1	f_1
									X_2	f_2
									...	...
									X_n	f_n

Parameter	*Description*	*Default*
l_case	Load case number. The wave is activated by using the LOADHIST command referring to this load case number + a TIMEHIST of type 3	
Type	Wave Type 1 : Airy, Extrapolated 1.1 : Airy, Stretched 2 : Stoke's 5'th (Skjelbreia, Hendrickson, 1961) 3 : User Defined 4 : Stream Function Theory (Dean, Dalrymple) Unit	
Height	Wave height [m]	
Period	Wave period [s]	
Direct	Direction of wave relative to global x-axis, counter clockwise [dg]	
Phase	Wave phase [dg]	
Surflev	Surface Level (Z-coordinate) expressed in global system [m]	
Depth	Water depth [m]	
N_ini	Number of initialization points defining wave 'envelope'	0
X_1	X-coordinate of first grid point (starting with largest negative x-coord.)	
f_1	Scaling factor of the wave height at first grid point. See Figure 6-1	

With this record, the user may specify a wave to be applied to the structure as hydrodynamic forces.
The wave is 'switched' ON according to the LOADHIST/TIMEHIST definition. TIMEHIST **type 3 must** be used.

Wave forces are applied on the structural members, who are **wet at the time of load calculation**, and relative velocity is accounted for if the record REL_VELO is specified in the control file.

All wave definitions with same load case number will be applied at the same time adding the wave kinematics before the forces are calculated, ('irregular' wave).

Current to be combined with the actual wave **must have same load case number!**
The current influences the travelling speed of the wave.

The time between calculations of wave forces is controlled by the referred TIMEHIST record, (dTime). The calculated wave forces are written to file if WAVCASE1 is specified in the control file.

In XACT the surface elevation is visualized. Applying a mesh on the surface (Verify/Show mesh) the waves become clearer, (Result/deformed model must be activated with displacement scaling factor=1.0). By pointing on the sea surface using the option Clip/Element, the surface will disappear.

NOTE! SI units must be used (N, m, kg) with **Z-axis pointing upwards!**
This record may be repeated

WAVEDATA Lcase Type Hs T Direct Seed Surflev Depth N_ini
X_1 f_1
X_n f_n
nFreq SpecType TMin TMax Grid (Opt) {Data}

Parameter	*Description*	*Default*
l_case	Load case number. The wave is activated by using the LOADHIST command referring to this load case number + a TIMEHIST of type 3	
Type	Wave Type = *Spect*	
	Unit	
Hs	Significant Wave height [m]	
T	Peak period of spectre, (***NOTE***: *This parameter is* ***Tz*** *if PM spectre*) [s]	
Direct	Direction of wave relative to global x-axis, counter clockwise [dg]	
Seed	Wave seed (input to random generator) [-]	
Surflev	Surface Level (Z-coordinate) expressed in global system [m]	
Depth	Water depth [m]	
N_ini	Number of initialization points defining wave 'envelope (see previous page and Figure 6-1).	0
nFreq	Number of frequencies	
SpecType	Spectre type. Jonswap : Jonswap Spectre PM : Pierson-Moscovitz User : User Defined Spectre	
TMin	Lowest wave period to be used in the wave representation	
TMax	Highest wave period to be used.	
Grid	Discretization type: 1 : Constant dω in the interval tmin-tmax 2 : Geometrical series from Tp 3 : Constant area for each S(ω) "bar"	2
"Opt"	Optional Data. If Jonswap : Gamma parameter If User Defined : Number of points in the ω – S(ω) curve Else : Dummy	
{Data}	If User Defined : The nPoint ω – S(ω) points defining S(ω) Else : Dummy	

With this record, the user may specify an irregular wave to be applied to the structure as hydrodynamic forces as described on the previous page.

Example:

```
        LCase  Typ    Hs    Tp    Dir  Seed SurfLev  Depth  nIni
WaveData  3   Spect  12.8  13.3  45   12      0      176     0
'          nFreq    SpecType  TMin  TMax  (Grid)
            30       Jonsw     4     20
```

See the example collection on www.usfos.com for more examples.

This record is given once.

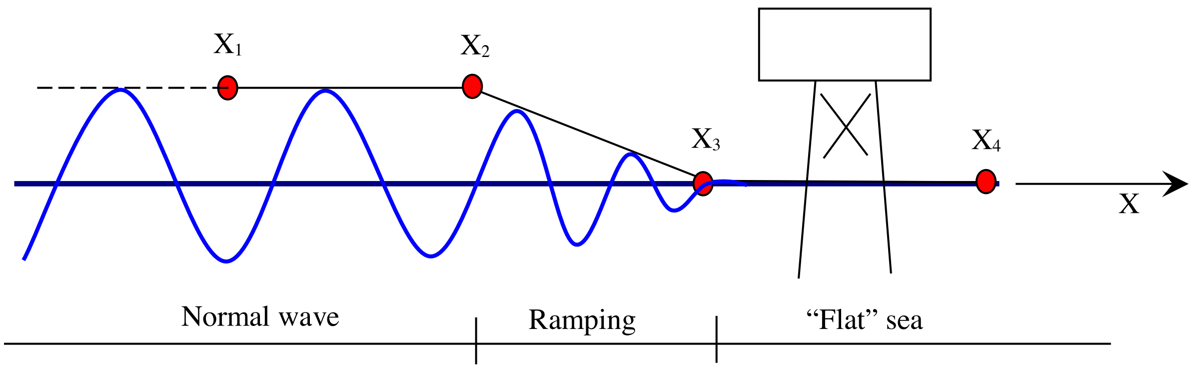


Figure 6-1 Initialisation of wave

CURRENT l_case Speed Direct Surflev Depth Z_1 f_1
... ...
Z_n f_n

Parameter	*Description*		*Default*
l_case	Load case number. The current is activated by using the LOADHIST command referring to this load case number + a TIMEHIST of type 3	Unit	
Speed	Current Speed to be multiplied with the factor f giving the speed at actual depth, (if profile is defined)	[m/s]	
Direct	Direction of wave relative to global x-axis, counter clockwise	[deg]	
Surflev	Surface Level (Z-coordinate) expressed in global system	[m]	
Depth	Water depth	[m]	
Z_1	Z-coordinate of first grid point (starting at Sea Surface)	[m]	
f_1	Scaling factor of the defined *speed* at first grid point.		
....	Similar for all points defining the **depth profile** of the current		

With this record, the user may specify a current to be applied to the structure as hydrodynamic forces.
The current is 'switched' ON according to the LOADHIST/TIMEHIST definition. TIMEHIST **type 3 must** be used. If the current should vary over time, the CURRHIST command is used.

Wave forces are applied on the structural members, who are **wet at the time of load calculation**, and relative velocity is accounted for if the record REL_VELO is specified in the control file.

Current to be combined with waves **must have same load case number!**

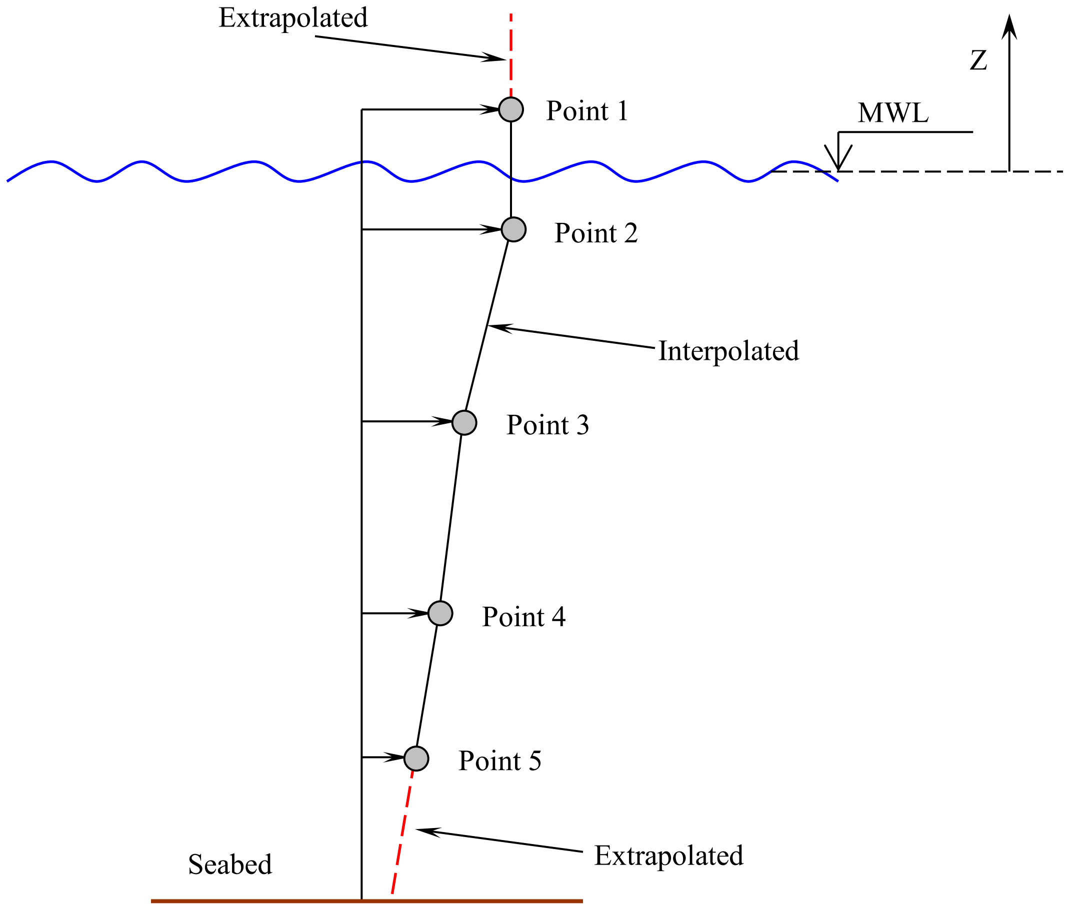
Time between calculations of wave forces is controlled by the referred TIMEHIST record, (dTime). The calculated wave forces are written to file if WAVCASE1 is specified in the control file.

In XACT the surface elevation is visualised. Applying a mesh on the surface (Verify/Show mesh) the waves become clearer, (Result/deformed model must be activated with displacement scaling factor=1.0). By pointing on the sea surface using the option Clip/Element, the surface will disappear.

NOTE! **SI units** must be used (N, m, kg) with **Z-axis pointing upwards!**

This record may be repeated

Defining current speed using depth profile. (Same definition also for other depth profiles, such as marine growth, hydrodynamic coefficients etc.).




```
'          lCase speed Dir SurfLev Depth  Z   Fac
 Current    3     1.5   45    95     95    5   1.0  ! Point 1 Above MWL
                                         -10   1.0  ! Point 2
                                         -30   0.7  ! Point 3
                                         -60   0.5  ! Point 4
                                         -80   0.2  ! Point 5
```

The *Local* Z-coordinate starts at MWL and point upwards.
=> positive Z means above MWL and negative Z means below MWL.

Points are given from top towards bottom.

NOTE
Beams located above point-1 or below point-5 use the *extrapolated* values indicated with red dotted line in this example. The current speed is *interpolated* between the discrete points.

Figure 6-2 - Depth Profile Definition.

CURRHIST	$Time_1$ $Time_2$ … $Time_n$	f_1 f_2 …. f_n

Parameter	*Description*	*Default*
$Time_1$ f_1	Time of first point [s] *Scaling factor* of the defined *speed* at first time. Similar for all points defining the **time history** of the current	

With this record, the user may specify a time history of the current. The current specified under CURRENT will be scaled according to the specified time history.

This record is given only once

REL_VELO nAvrg

If **REL_VELO** is specified, the relative velocity between the structure and the wave particles are accounted for in connection with the calculation of drag forces.

This is also the way to switch on hydro-dynamical damping.

Option 1:
If the relative velocity should be calculated on basis of an average structural velocity, the average velocity of the "nAvrg" last analysis steps is used. NAvrg = 0 is default, which means that the current structural velocity is used.

This record is given only once

WavCase1 l-case iTotal

Parameter	*Description*	*Default*
l-case	The generated loads are written to file with *l-case* as the first load case number. Each time new wave forces are calculated the forces are written, and the load case number is incremented with one.	1
iTotal	iTotal = 0 : Incremental forces are written iTotal = 1 : Total forces are written	0

The user may use this record to let USFOS write to file the forces generated in connection with the options WAVEDATA/CURRENT/WINDFIELD.

Together with the BeamLoad records, the LCASETIM record is printed connecting the load case to a physical time for the actual load.

NOTE ! This option should be used only if USFOS is used as a wave load pre-processor due to the extra time consumption and disc space requirements.

This record is given only once

M_GROWTH	Z_1	Add_T_1
	Z_2	Add_T_2
	...	...
	Z_n	Add_T_n

Parameter	*Description*	*Default*
Z_1	Z-coordinate of the first grid point defining the marine growth profile Z=0 defines the sea surface, and **all Z-coordinates are given relative to the sea surface**, Z-axis is pointing **upwards**. Z>0 means *above* the sea surface. Z_1 is the upper point (and Z_n is the lower point at seabed). Z_1 is the upper point (and Z_n is the lower point at seabed). See Image on page 6.3-120.	
Add_T_1	Additional thickness to be applied on members at elevation Z_1	
Z_2	Z-coordinate of the second grid point defining the marine growth profile	
Add_T_2	Additional thickness to be applied on members at elevation Z_2	

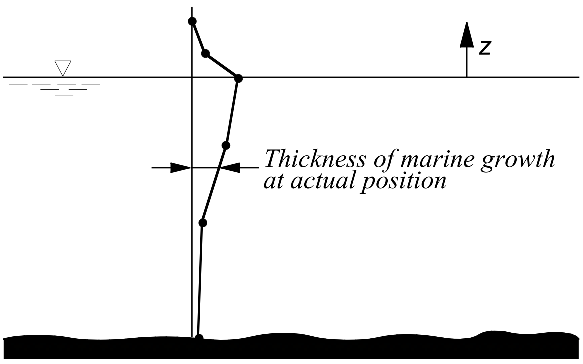
With this record the user may define a marine growth depth profile to be applied to the structural members.

Between the tabulated values, the additional thickness is interpolated, values outside the table are *extrapolated*.

The hydrodynamic diameter $D_{hyd} = D_{struc} + 2 * Add\_T$.

In the .out -file, the hydrodynamic diameter is listed for all *potential wet* beam elements.

Data should be specified above the sea surface according to the measured marine growth. Ensure that extrapolation gives correct Add_T, (dry elements become wet due to surface wave elevation).



NOTE! SI units must be used (N, m, kg) with **Z-axis pointing upwards!**

This record is given only once.

Hyd_CdCm Cd Cm elnox1 elnox2 .... elnoxn

Parameter	*Description*	*Default*
Cd	Drag Coefficient to be applied to the listed elements	0.7
Cm	Mass Coefficient to be applied to the listed elements	2.0
elnox1 elnox2 . elnoxn	External element number 1 to be defined with the actual Cd / Cm External element number 2 . External element number n to be defined with the actual Cd / Cm **NOTE !** If no elements are specified, all beam elements will be defined with the actual Cd and Cm.	All

This record is used to assign Drag- and Mass Hydrodynamic Coefficients to beam elements.

Example:
```
Hyd_CdCm   0.7    1.8
Hyd_CdCm   1.0    2.0    1010 1020 1030 2010 2020 2030
```

New defaults Cd / Cm are applied to all beam elements.
However, elements 1010 1020 1030 2010 2020 and 2030 are defined with different Cd and Cm coefficients (the latest Hyd_CdCm definition overrides previous definitions).

NOTE ! This record overrides Cd and Cm defined under the commands: Hydro_Cd and Hydro_Cm, (see next pages).

This record may be repeated.

Hydro_Cd	Z_1 Z_2 ... Z_n	Cd_1 Cd_2 ... Cd_n	

Parameter	*Description*	*Default*
Z_1	Z-coordinate of the first grid point defining the Cd profile (Z=0 defines the sea surface, and all Z-coordinates are given relative to the surface, Z-axis is pointing upwards. Z>0 means *above* the sea surface). Z_1 is the upper point (and Z_n is the lower point at seabed). See Image on page 6.3-120.	
Cd_1	Drag Coefficient to be used for elements at elevation Z_1	
Z_2	Z-coordinate of the second grid point defining the Cd profile	
Cd_2	Drag Coefficient to be used for elements at elevation Z_2	

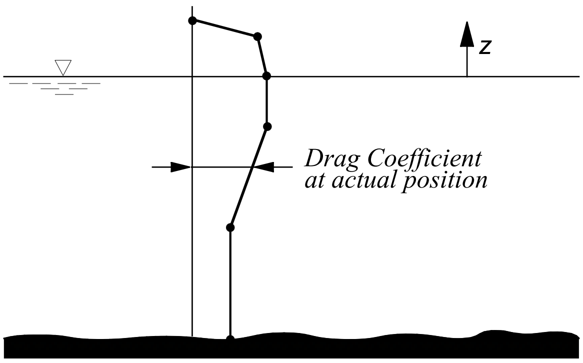
This record is used to define a **Drag Coefficient depth profile**.

Between the tabulated values, the Cd is interpolated. Values outside the table are *extrapolated*.

In the .out -file, the drag coefficient is listed for all *potential wet* beam elements.

Data should be specified above the sea surface. Ensure that extrapolation gives correct Cd, (dry elements become wet due to surface wave elevation).

The command HYD_CdCm overrides this command.



NOTE! **SI units** must be used (N, m, kg) with **Z-axis pointing upwards!**

This record is given only once.

Hydro_Cm	Z_1	Cm_1
	Z_2	Cm_2
	...	...
	Z_n	Cm_n

Parameter	*Description*	*Default*
Z_1	Z-coordinate of the first grid point defining the Cm profile (Z=0 defines the sea surface, and all Z-coordinates are given relative to the surface, Z-axis is pointing upwards. Z>0 means *above* the sea surface). Z_1 is the upper point (and Z_n is the lower point at seabed). See Image on page 6.3-120.	
Cm_1	Mass Coefficient to be used for elements at elevation Z_1	
Z_2	Z-coordinate of the second grid point defining the Cm profile	
Cm_2	Mass Coefficient to be used for elements at elevation Z_2	

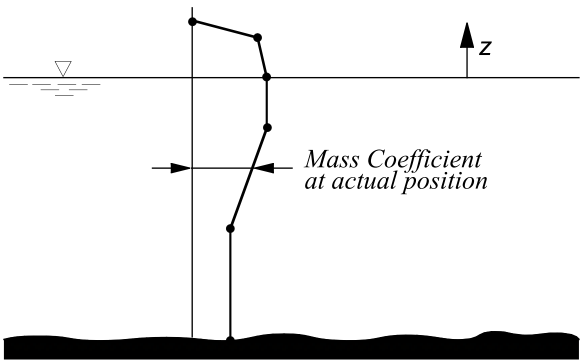
This record is used to define a **Mass Coefficient depth profile**.

Between the tabulated values, the Cm is interpolated. Values outside the table are *extrapolated*.

In the .out -file, the mass coefficient is listed for all *potential wet* beam elements.

Data should be specified above the sea surface. Ensure that extrapolation gives correct Cm, (dry elements become wet due to surface wave elevation).

The command HYD_CdCm overrides this command.



NOTE! **SI units** must be used (N, m, kg) with **Z-axis pointing upwards!**

This record is given only once.

Wave_KRF KRF		
Parameter	*Description*	*Default*
KRF	Wave Kinematics reduction Factor: Particle Velocities are multiplied by this factor.	1.0

The user may use this record the user may specify the reduction factor to be used in connection with calculation of wave forces (WAVEDATA).

2-D theory may over-estimate the kinematics of real 3-D ocean waves, and reduction of the particle velocities may give better correspondence with field measurements.
The corrected particle velocity is calculated as follows:

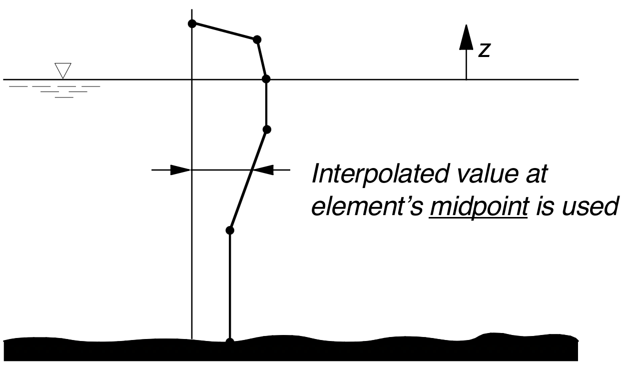
$$Vel_{corr} = Vel_{2\text{-}D} * KRF$$

This record is given only once

Wave_KRF *Profile* Z_1 KRF_1 ... ... Z_n KRF_n		
Parameter	*Description*	*Default*
Z_1	Z-coordinate of the first grid point defining the Integration Point profile (Z=0 defines the sea surface, and all Z-coordinates are given relative to the surface, Z-axis is pointing upwards. Z>0 means *above* the sea surface). Z_1 is the upper point (and Z_n is the lower point at seabed). See Image on page 6.3-120.	
KRF_1	Kinematics Reduction Factor to be used for elements at position Z_1	
Z_2	Z-coordinate of the second grid point.	
KRF_2	Kinematics Reduction Factor to be used for elements at elevation Z_2	

This record is used to define a **Kinematics Reduction Factor depth profile**, and is an extended version of the original *Wave_KRF* command. Between the tabulated values, the KRF is interpolated. Values outside the table are *extrapolated*. In the .out -file, the interpolated wave kinematics reduction factor used for each beam element is listed. Selected values are also visualized in XACT under Verify/Hydrodynamics.
Data should also be specified *above* the sea surface. Ensure that extrapolation gives correct KRF, (dry elements become wet due to surface wave elevation).

The command "`HYDROPAR WaveKRF`" overrides this command.



NOTE! SI units must be used (N, m, kg) with **Z-axis pointing upwards!**
This record is given only once.

Wave_KRF	3D_*Profile*	Z_1	KRF_X_1	KRF_Y_1	KRF_Z_1
		...	...		
		Z_n	KRF_X_n	KRF_Y_n	KRF_Z_n

Parameter	*Description*	*Default*
Z_1	Z-coordinate of the first grid point defining the Integration Point profile (Z=0 defines the sea surface, and all Z-coordinates are given relative to the surface, Z-axis is pointing upwards. Z>0 means *above* the sea surface). Z_1 is the upper point (and Z_n is the lower point at seabed). See Image on page 6.3-120.	
KRF_X_1	Kinematics Reduction Factor used for elements at position Z_1, X-dir	
KRF_Y_1	Kinematics Reduction Factor used for elements at position Z_1, Y-dir	
KRF_Z_1	Kinematics Reduction Factor used for elements at position Z_1, Z-dir	
Z_n	Z-coordinate of the n'th grid point, etc.	

This record is used to define a **Kinematics Reduction Factor depth profile**, and is an extended version of the original *Wave_KRF* command.

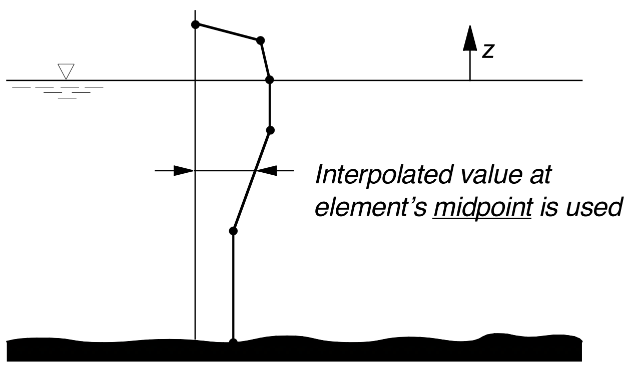
Between the tabulated values, the KRF is interpolated. Values outside the table are *extrapolated*. In the .out -file, the interpolated wave kinematics reduction factor used for each beam element is listed.

Different factors for the different global directions of the wave kinematics, X- Y and Z are possible. The selected values are printed in the "OUT" file.

NOTE! Only one KRF factor per element is visualized in in XACT. The visualized factor is the value for global X-direction.

Data should also be specified *above* the sea surface. Ensure that extrapolation gives correct KRF, (dry elements become wet due to surface wave elevation).

The command "`HYDROPAR WaveKRF`" overrides this command.



NOTE! SI units must be used (N, m, kg) with **Z-axis pointing upwards!**
This record is given only once.

BUOYANCY l_case write

Parameter	Description	Default
L_case	The buoyancy forces are *added* to the actual load case. If no load case is specified, the buoyancy forces are added to the WAVEDATA lcase	WAVEDATA lcase
Write	Write option: *Write* : buoyancy forces are written to a separate file *NoWrite* : buoyancy forces are *not* written to file.	*NoWrite*

If **BUOYANCY** is specified, buoyancy effects are accounted for. By default all elements are buoyant, but using the FLOODED command, it is possible to remove buoyancy from the internal volume for selected elements.

The buoyancy forces are updated every time wave loads are calculated, and the current position of the sea surface defines whether an element becomes buoyant or not at any time.

This record is given only once.

FLOODED elnox1 elnox2 .... elnoxn

Parameter	Description	Default
elnox1	External element number 1 to be defined flooded	
elnox2	External element number 2	
.	.	
elnoxn	External element number n to be defined non buoyant	

This record is used to define flooded elements, and this command has only meaning if BUOYANCY is specified. The elements are filled with seawater unless other is specified.
It is recommended to use the new input (see below).
This record may be repeated

FLOODED ListType {Id_List}

Parameter	Description	Default
ListType	Data type used to specify the elements: Element : The specified Id's are element numbers. Mat : The specified Id's are material numbers Geo : The specified Id's are geometry numbers.	
{Id_List}	Group : The specified Id's are group numbers. One or several id's separated by space	

This record is an extended version of the original FLOODED command specified above.

Example 1 *:*

Flooded *Group 1 5 6*

All elements in groups (sets) 1, 5 or 6 will become flooded.

NOTE: The last definitions will override previous definitions if same element is defined more than once

This record may be repeated.

MaxWave Criterion dT EndT Write

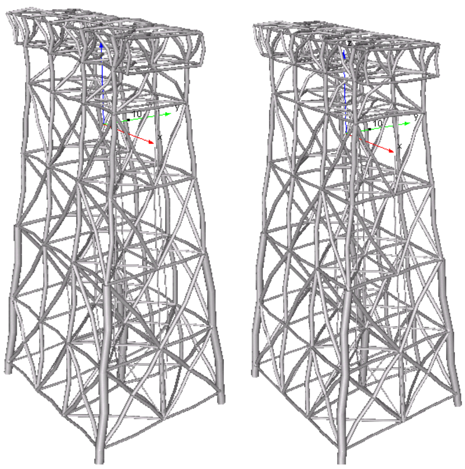
Parameter	*Description*	*Default*
Criterion	Criterion used to identify "worst wave phase" of the actual wave: BaseShear : Base Shear is used OverTurn : Overturning moment is used	
dT, EndT	The wave (defined by the WAVEDATA record) is stepped through with a time increment of dT up to time EndT. The worst phase detected in the specified interval is used.	
Write	Write option: Write : The wave forces are written to file noWrite : No writing	

With this record the user may us USFOS to identify the worst wave phase to be used in a static 'pushover' analysis. The load case no of the specified WAVEDATA record may be referred to from the CUSFOS or CICYFOS record. (NOTE: only ***one*** WAVEDATA load case is possible to specify using this option).

<u>*Example 1 :*</u>

```
'           Typ            dT     EndT     option
MaxWave   BaseShear     0.5     16     noWrite

'           l_case  Typ     H      Per  Dir  Phase   Surflev  Depth
WaveData     2     Stoke   25.0   16   45     0       0.0     100
'
CUSFOS    10  10  1.0  1.0
'          lcomb  lfact  mxld nstep minstp
             1     0.5   1.0    10   0.010     !  Dead Weight
             2     0.1   5.0   200   0.001     !  Max Wave forces found by USFOS
```

The Stoke wave will be 'stepped' through the platform with a time interval of 0.5 s up to time = 16s. The wave forces at the time (phase) giving the maximum base shear are assigned to load case no 2 (override read-in node and element loads with this loadcase no). LoadCase no 2 is referred to as usual in the CUSFOS record.



Hint:

If the automatic member imperfection is activated (CINIDEF) referring to the WAVEDATA load case, USFOS will use the selected hydrodynamic force pattern when the directions of the individual member imperfections are applied. To the left, the wave direction is 45°, and in the case to the right, the wave direction is 180° opposite (225°), and the member imperfection is adjusted automatically.

This record is given only once!

MaxWave KeyWord LoadCase(s)

Parameter	Description	Default
KeyWord	Keyword defining the actual parameter(s) *WaveLCase* : Specify Actual Wave Load Case to process. If only one WaveData record is defined, this command could be omitted. *AddLCase* : Specify additional loads to be added on top of the wave loads. Could be basic read-in loads or wind loads.	
LoadCase	Actual Load Case numbers	

This is an extension of the MaxWave command, where the user may specify additional information relevant for the wave load calculation.

Example 1 :

```
'             Typ       lCase
MaxWave     WaveLCase     5
```

The MaxWave will be performed using WaveData with load case 5. This means that the input may contain several WaveData definitions

Example 2 :

```
'             Typ      lCase
MaxWave     WaveLCase     5
MaxWave     AddLCase     10
```

The MaxWave will be performed using WaveData with load case 5. However, when the total load vector is computed, the forces from load case 10 will be added on top of the wave forces.

For example could load case 10 contain slamming loads (wave in deck) with time history information specified. The time when the sum of all loads gives the max peak, (base shear or overturn), will then be selected. NOTE: In the CUSFOS input, it should be referred to the Wave load case (in this example lc 5)

The additional load case could be either ordinary read-in loads (concentrated or distributed) **together with time variation information** (**TimeHist + LoadHist**), or it could be Wind Loads computed internally by USFOS.

This record may be repeated

WavMxScl Scale		
Parameter	*Description*	*Default*
Scale	Scaling factor used to scale the computed wave loads from MaxWave option before assigning into the actual load vector.	1.0

This record is used to scale the wave forces calculated by USFOS under the MaxWave option. The scaled forces are copied into the actual load vector *before* the analysis starts.

This option is required when the force unit used in the analysis is *not* Newton (e.g. kN or MN).

Example

WavMxScl 1.0E-6

Will multiply the generated wave forces (which are calculated in Newton) with 1.0E-6, and the wave forces are then stored in MN.

This record is given only once.

WET_ELEM All		
Parameter	*Description*	*Default*
All	All elements are assumed potential wet	off

This record is used to force USFOS to check *all* elements for hydrodynamic forces during the dynamic analysis. By default, only elements, which can be reached by the highest wave defined, are checked for hydrodynamic forces (this saves analysis time as f ex the whole topside structure is left out in cases where the highest wave never hit the topside).

In cases where the structure is dropped from a position above the sea surface, this switch should be set ON.

Wave_Int NIS Elem_1 Elem_2 Elem_3 ....

Parameter	*Description*	*Default*
NIS	Number of Integration sections used for wave force calculations.	2
Elem_1 Elem_2	Element 1D of first beam element using NIS integration sections Element ID of second element etc... NOTE ! If no elements are specified, all beam elements are using NIS integration sections.	

This record is used to define number of integration sections to be used in connection with wave load calculations.

Example 1

```
Wave_Int    4    1001    1002    1003
```

Means that beam elements 1001, 1002 and 1003 use 4 integration points. For the other beam elements 2 points are used (default).

Example 2

```
Wave_Int    3
Wave_Int   10    101    102
```

Means that default number of integration sections (points) are changed from 2 to 3. However, for elements 101 and 102, 10 points are used.

This record may be repeated.

Wave_Int	*Profile*	Z_1	$nInt_1$
		Z_2	$nInt_2$
		...	...
		Z_n	$nInt_n$

Parameter	*Description*	*Default*
Z_1	Z-coordinate of the first grid point defining the Integration Point profile (Z=0 defines the sea surface, and all Z-coordinates are given relative to the surface, Z-axis is pointing upwards. Z>0 means *above* the sea surface). Z_1 is the upper point, see Image on page 6.3-120.	
$nInt_1$	Number of Integration Points to be used for elements at position Z_1	
Z_2	Z-coordinate of the second grid point.	
$nInt_2$	Number of Integration Points to be used for elements at elevation Z_2	

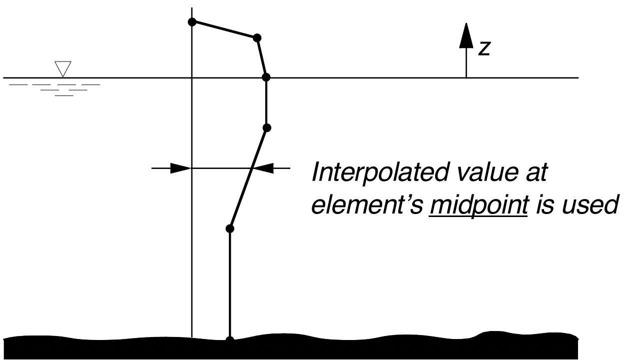
This record is used to define an **Integration Point depth profile**, and is an extended version of the original *Wave_Int* command (described on previous page).

Between the tabulated values, the *nInt* is interpolated. Values outside the table are *extrapolated*.

In the .out -file, the interpolated number of integration points used for each beam element is listed. Selected values are also visualized in XACT under Verify/Hydrodynamics.

Data should also be specified *above* the sea surface. Ensure that extrapolation gives correct *nInt*, (dry elements become wet due to surface wave elevation).

The command "`HYDROPAR   WaveInt`" overrides this command.



NOTE! SI units must be used (N, m, kg) with **Z-axis pointing upwards!**

This record is given only once.

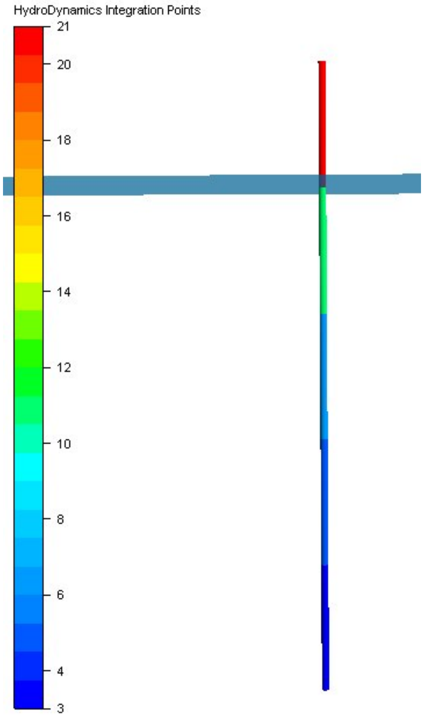
Wave_Int	*Mesh*	Z_1 Z_2 ... Z_n	$Dist_1$ $Dist_2$ ... $Dist_n$

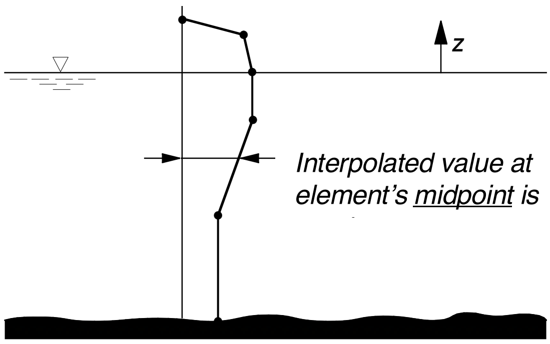
Parameter	*Description*	*Default*
Z_1	Z-coordinate of the first grid point defining the Integration Point profile (Z=0 defines the sea surface, and all Z-coordinates are given relative to the surface, Z-axis is pointing upwards. Z>0 means *above* the sea surface). Z_1 is the upper point. see Image on page 6.3-120.	
$Dist_1$	Proposed *distance between* integration points to be used for elements at position Z_1	
Z_2, $Dist_2$	Z-coordinate and distance of the second grid point.	

This record is used to define an **Integration Point depth profile**, and is an extended version of the original *Wave_Int* command (described on previous pages). Between the tabulated values, the ***Dist*** is interpolated. Values outside the table are *extrapolated*. Instead of giving the number of points, USFOS will find the number needed to give the specified distance between the integration points.

In the .out -file, the interpolated number of integration points used for each beam element is listed. Selected values are also visualized in XACT under Verify/Hydrodynamics (see image).

The command "`HYDROPAR` *`WaveInt`* " overrides this command.





Verification of Integration points in Xact

NOTE! Z-axis is pointing upwards!

This record is given only once.

CurrBloc Type [Data]		
Parameter	*Description*	*Default*
Type Data	Current Blockage Type (currently, only "User" is available) Required data for the different options: Type = ***User*** : "Data" is just a scaling factor (Fac) Type = ***API Taylor***: Blockage for the different members is computed according to API, Taylor's formulas.. Two different options exist: ❑ Common : Averaged factor are used for all elements ❑ Individual : Specific blockage for the different elements.	

The user may use this record to specify the reduction factor to be used in connection with calculation of current forces.

Vel_{corr} = Vel * Fac

Example 1:

CurrBloc User 0.85 :
All elements get a blockage factor of 0.85

Example 2:

CurrBlock API Taylor Common :
All elements get the computed (averaged) factor according to API (Taylor)

Example 3:

CurrBlock API Taylor Individual :
Computed factors will be assigned to the individual elements according to API Taylor.

This record is given only once

CurrBloc	*Profile*	Z_1	$Block_1$
		Z_2	$Block_2$
		...	...
		Z_n	$Block_n$

Parameter	*Description*	*Default*
Z_1	Z-coordinate of the first grid point defining the Integration Point profile (Z=0 defines the sea surface, and all Z-coordinates are given relative to the surface, Z-axis is pointing upwards. Z>0 means *above* the sea surface). Z_1 is the upper point. see Image on page 6.3-120.	
$Block_1$	Current Blockage to be used for elements at position Z_1	
Z_2	Z-coordinate of the second grid point.	
$Block_2$	Current Blockage to be used for elements at elevation Z_2	

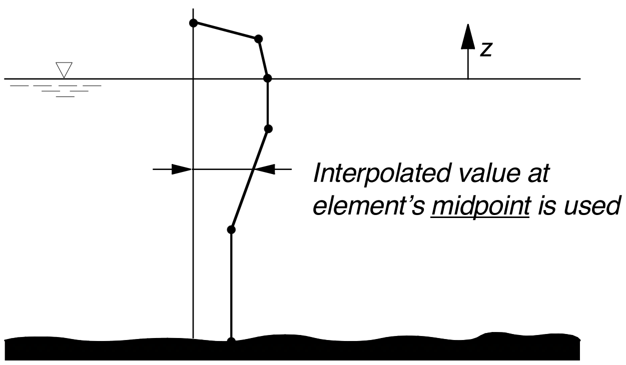
This record is used to define a **Current Blockage depth profile**, and is an extended version of the original *CurrBlock* command, (previous page).

Between the tabulated values, the Block value is interpolated. Values outside the table are *extrapolated*.

In the .out -file, the interpolated blockage factor used for each beam element is listed. Selected values are also visualized in XACT under Verify/Hydrodynamics.

Data should also be specified *above* the sea surface. Ensure that extrapolation gives correct Block, (dry elements become wet due to surface wave elevation).

The command "`HYDROPAR   CurrBlock` " overrides this command.



NOTE! **SI units** must be used (N, m, kg) with **Z-axis pointing upwards!**

This record is given only once.

BUOYHIST **Hist_ID** ListType {Id_List}

Parameter	*Description*	*Default*
Hist_ID	History ID to be used to scale the current buoyancy forces	
Type	Data type used to specify the elements: Element : The specified Id's are element numbers. Geo : The specified Id's are geometry numbers. Group : The specified Id's are group numbers.	
{Id_List}	One or several id's separated by space	

With this record the user may define time dependent buoyancy on the individual elements. The elements are specified either directly (element list) or in-directly (using material, geometry or group references).

Example 1 *:*

BuoyHist *10 Mat 1 5 6*

All elements referring to materials 1, 5 or 6 will have a time dependent buoyancy according to time history with Id 10

Example 2 *:*

BuoyHist *10 Mat 1 5 6*
BuoyHist *11 Elem 1001 1002 1003*

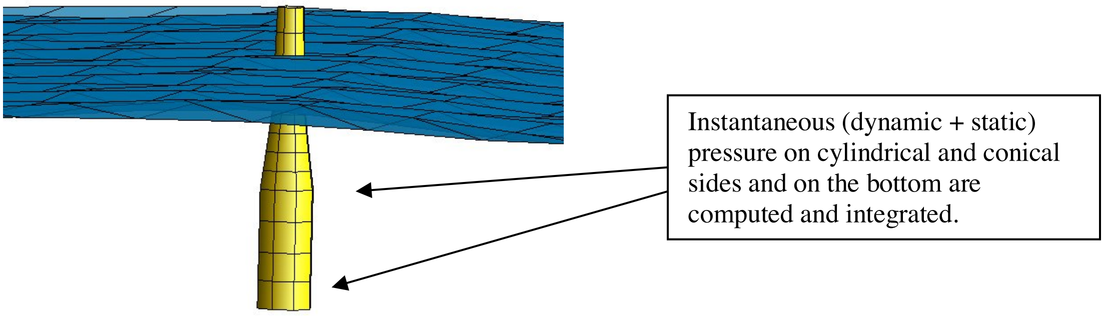
All elements referring to materials 1, 5 or 6 will have time dependent buoyancy according to time history with Id 10. Elements 1001, 1002 and 1003 will follow history no 11.

NOTE: The last definitions will override previous definitions if same element is defined more than once

This record may be repeated.

BUOYFORM	Form ListType {Id_List}	
Parameter	*Description*	*Default*
Form	Buoyancy Formulation: Arch : Simple "Archimedes" calculation Panel : Based on Pressure (dyn + static).	
ListType	Data type used to specify the elements: Element : The specified Id's are element numbers. Geo : The specified Id's are geometry numbers. Group : The specified Id's are group numbers.	
{Id_List}	One or several id's separated by space	

With this record the user may define special buoyancy formulation on selected elements. The elements are specified either directly (element list) or in-directly (using material, geometry or group references). The Panel model is rather time consuming, and it is recommended to us this formulation on special, selected elements only, (buoyancy tanks etc).



Example 1 *:*

```
BuoyForm   Panel  Mat  1  5  6
```

All elements referring to materials 1, 5 or 6 will have the buoyancy computed according to "panel" formulation. All other elements are using the simple, default Archimedes' formula, (buoyancy=weight of submerged water).

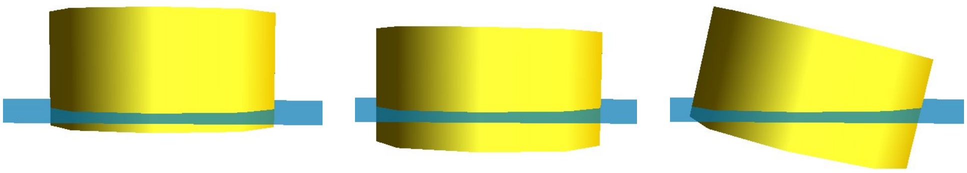
NOTE: The last definitions will override previous definitions if same element is defined more than once

This record may be repeated.

INTFLUID Density FillTyp Data ListType {Id_List}

Parameter	*Description*	*Default*
Density	Density of the internal fluid [kg/m³]	
FillTyp	Filling Type : TimeDep : The internal fluid level (fillratio) is defined by a TimeHist Drained : The fill ratio is computed as the element goes in and out of the water.	
Data	If "*TimeDep*" : History ID If "*Drained*" : Drain time	
ListType	Data type used to specify the elements: Element : The specified Id's are element numbers. Geo : The specified Id's are geometry numbers. Group : The specified Id's are group numbers.	
{Id_List}	One or several id's separated by space	

With this record the user may define the internal fluid of one or more cylindrical elements. The elements are specified either directly (element list) or in-directly (using material, geometry or group references). The fillratio (0-1) is specified using a time history, where the time history function value is directly the instantaneous fill ratio. The free surface calculation (no sloshing effects) detects instabilities and a new equilibrium position is found automatically. The model (which requires that "panel" buoyancy is defined for the element(s)), is rather time consuming, and it is recommended to us this option on special, selected elements only, (buoyancy tanks etc.).



Empty Cylinder

Cylinder is partly filled.

Becomes unstable (free surface) and finds a new equilibrium position.

Example 1 :

```
'          Dens   FillTyp  Hist_ID  ListType   MatID
IntFluid    900   TimeDep    11       Mat        1

'            ID    Type     T1   T2   Factor
TimeHist     11   S_Curve   20   30    0.2
```

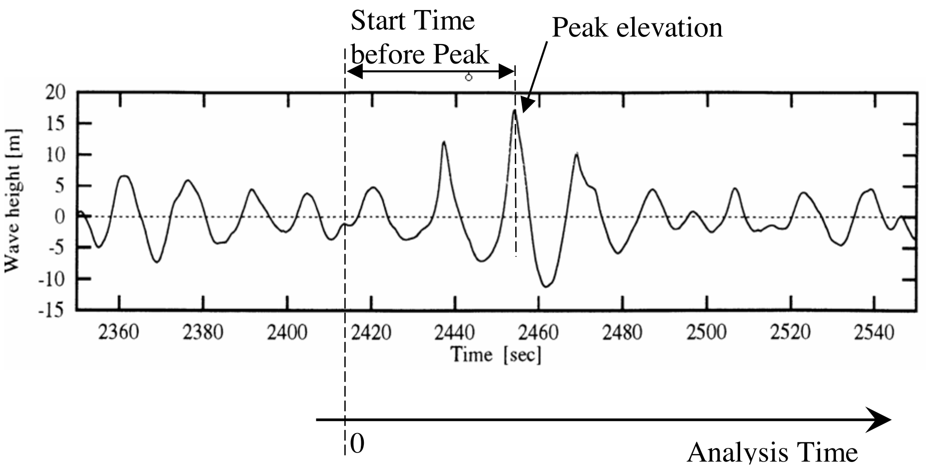
All elements referring to materials 1 will have an internal fluid with density 900 kg/m³. The cylinder is filled to 20% (0.2) between time t=20s to t=30s (See TimeHist 11). As the figures describe, the cylinder will first move vertical without tilting, but becomes unstable and tilts to a new equilibrium position.

NOTE: The free surface calculation requires the Panel buoyancy formulation on the actual elements. The "drained" option works for the conventional "Archimedes" buoyancy only.

This record may be repeated.

SpoolWave *Time Order dT StormLen Crit*		Unit
Time *Order* *dT* *StormLen* *Crit*	The analysis starts "*time*" seconds before the actual peak Select 1^{st} , 2^{nd} 3^{rd} etc. *highest* wave crest by specifying 1, 2, 3 etc. A negative number means the 1^{st} , 2^{nd} 3^{rd} etc. *lowest* wave through. Time increment during search for the peak waves Search for highest waves from time=0 to time=StormLen Criterion to use for selection of peak wave: Elev : Peak Wave Elevation is used Height : Max trough -to – crest is used	[s] [-] [s] [s] [-]

With this record, the user defines how to search for the highest waves in an irregular wave field. The analysis time=0 will be moved forward to the specified "Time before Peak" as shown in the figure.



Example 1:

```
'             Time     Order    dT    StormLen    Crit
SpoolWave     30         1      0.5    3600    elevation
```

Search for the highest wave elevation within a 1-hour storm (the WaveData command is specified separately). A delta T of 0.5 seconds is used during this check.

Example 2:

```
'             Time     Order    dT    StormLen    Crit
SpoolWave     30         3      0.5    3600    elevation
```

Search for the 3^{rd} highest wave elevation within a 1-hour storm. (*To ensure that the 2^{nd} , 3^{rd} etc peaks are not a part of any of the higher peaks, a minimum time between peaks are required. In the automatic search procedure, this minimum time between peaks is set equal to Tp.*)

NOTE! This option must be used only in connection with time domain simulations of irregular waves (WaveData *Spect* …)

This record is given once.

NONHYDRO	ListType	{Id_List}

Parameter	*Description*	*Default*
ListType	Data type used to specify the element(s): Element : The specified Id's are element numbers.	
{Id_List}	One or several id's separated by space. No ID's means all elements	

With this record the user may define non-hydro element, (elements which are bypassed in all hydrodynamic calculations). The elements are specified (element list).

Example 1 :

```
NonHydro Elem  1  5  6
```

Elements 1, 5 and 6 are bypassed during the hydrodynamic calculations

This record may be repeated.

HYDROPAR	KeyWord Value List Type {Id_List} ……	
Parameter	*Description*	*Default*
KeyWord	Keyword defining actual parameter to define: *KeyWord* — *Actual Definitions of "Value"* *HyDiam* : Hydrodynamic Diameter (override default) *Cd* : Drag Coefficient *Cm* : Mass Coefficient *Cl* : Lift Coefficient (not imp) *BuDiam* : Buoyancy Diameter (override default) *IntDiam* : Internal Diameter (override default) *WaveInt* : Number of integration points (override default) *CurrBlock* : Current Blockage factor *FluiDens* : Density of internal fluid *MgrThick* : Marine Growth Thickness *MgrDens* : Marine Growth Density *FloodSW* : Switch for flooded/no flooded (override default) *DirDepSW* : Switch for use of direction dependent Cd *FillRatio* : Fill ratio, (0-1) of member with internal fluid *WaveKRF* : Wave Kinematics Reduction Coeff *BuoyLevel* : Definition of complexity level of buoyancy calculations.	
Value	Actual Parameter value.	
ListTyp	Data type used to specify the element(s): *Element* : The specified Id's are element numbers. *Mat* : The specified Id's are material numbers *Geo* : The specified Id's are geometry numbers. *Group* : The specified Id's are group numbers.	
Id_List	One or several id's separated by space	

With this record, the user defines various hydrodynamic parameters for elements. Some of the parameters could be defined using alternative commands (F ex Hyd_CdCm etc.), but parameters defined under HYDROPAR will ***override all previous definitions***.

NOTE!
Since this command gives the user the possibility to define "any" combinations of values of the parameters, (with possible unwanted side effects), the user should use this option with care. Checking the hydrodynamic parameters graphically in xact and from the printed tables is recommended.

This record could be repeated

Below, the "HYDROPAR" keywords are described in detail:

HYDROPAR Keyword …….

Keyword	*Description*	*Default*
HyDiam	The hydrodynamic diameter is used in connection with drag- and mass forces according to Morrison's equation.	Struct Do
Cd / Cm	Drag- and Mass coefficients used in Morrison's equation.	0.7 / 2.0
Cl	Lift coefficients (normal to fluid flow). NOTE: Not implemented hydro.	0
BuDiam	Buoyancy calculations are based on this diameter.	Struct D
IntDiam	Internal diameter of the pipe. Relevant in connection with (completely) flooded members and members with special internal fluid.	Do-2T
WaveInt	Number of integration points per element	2
CurrBlock	Current blockage factor. Current is multiplied with this factor.	1.0
FluiDens	Density of *internal* fluid. Relevant for flooded members.	1024
MgrThick :	Thickness of marine growth specified in meter.	0.0
Marine Growth	Density of marine growth. Specified in [kg/m^3]	1024
Thickness	Switch (0/1) for flooded / non-flooded members. (internal use)	0
MgrDens *FloodSW*	Switch (0/1) for use of direction dependent drag coefficients. If switch is set to 1, special ElmCoeff data have to be defined for the element.	0
DirDepSW	Fill ratio of flooded member. By default is a flooded member 100% filled throughout the simulation. Fill ratio could be time dependent.	1
FillRatio	Wave kinematics reduction coefficient. Particle velocity used for actual elements is multiplied with this factor.	1.0
WaveKRF *BuoyLeve*	Specification of buoyancy calculation method. By default, the buoyancy of the (thin) steel wall is ignored for flooded members. If Level=1 is specified, a far more complex (and time consuming) calculation procedure is used. Flooded members on a floating structure going in and out of water should use Level=1 calculation.	0
SecOrder	Switch ON/OFF 2nd order wave theory on specified elements	OFF

HYDMASS Type Opt Value ListType {Id_List}		
Parameter	*Description*	*Default*
Type	Mass type. Only NODE is available	
Opt	Degree of freedom to assign a nodal added mass: XTransl Translation mass in Global X-direction YTransl Translation mass in Global Y-direction ZTransl Translation mass in Global Z-direction Coeff_X Mass coefficients in X-direction Coeff_Y Mass coefficients in Y-direction Coeff_Z Mass coefficients in Z-direction	
Value	Depends on "Opt": "XTransl" etc Added mass in [kg] in actual global direction "Coeff_X" etc. Mass coefficient in actual global direction.	
ListType	Data type used to specify the node(s). NODE : The specified Id's are Node IDs.	
{Id_List}	One or several id's separated by space.	

With this record the user may define concentrated hydrodynamic added mass to be assigned to nodes. The mass will contribute to the inertia only. If the "Coeff" option is specified ***in addition***, the wave acceleration will be computed and add forces to the nodes.

Example 1 *:*

```
         Typ.   Opt.    Mass  ListType   IDs ….
HydMass NODE. XTransl  1000   Node.      9. 10
```

Nodes 9 and 10 get an additional added mass of 1000kg.

If the following input is given ***in addition***, the wave acceleration forces are computed for the two nodes. The mass coefficient is here defined to 1.0 in X-direction.

```
         Typ.   Opt.     Cm   ListType   IDs ….
HydMass NODE. Coeff_X   1.0   Node.      9. 10
```

This record may be repeated.

HYDROXTR	Type Opt Val_X Val_Y Val_Z ListType {Id_List}	
Parameter	*Description*	*Default*
Type	Data type. Only NodeDrag is available	
Opt	Reference system: Only GLOB is available	
Val_X	CdA in X-direction. Unit is [m^2]	
Val_Y	CdA in Y-direction. Unit is [m^2]	
Val_Z	CdA in Z-direction. Unit is [m^2]	
ListType	Data type used to specify the node(s). NODE : The specified Id's are Node IDs.	
{Id_List}	One or several id's separated by space.	

With this record the user may define concentrated hydrodynamic drag forces to be assigned to nodes.

Example 1 :

```
              Typ.    Opt.    CdA_X   CdA_Y CdA_Z ListType   IDs ….
HydroXtr  NodeDrag  Glob    1000    1000    0      Node.      9. 10
```

Nodes 9 and 10 get an additional drag area of 1000m^2 in X- and Y-directions. No extra drag in Z-direction.

This record may be repeated.

6.3.17 Aerodynamics

**** The OLD input syntax *****

WINDFIELD l_case T_ini Z_bott U_{10} Power Rho WO_x WO_y WO_z
alpha beta1 gamma n_ini

Parameter	*Description*		*Default*
l_case	Load case number. The wind is activated by using the LOADHIST command referring to this load case number + a TIMEHIST of type 3		
T_ini	Time for applying the pre generated, fluctuating wind field	Unit	
Z_bott	Lower level of mean wind profile	[m]	
U_{10}	Mean wind 10 meter above Z_bott	[m/s]	
Power	power law coeff. defining the mean wind profile $U_z=U_{10}\ (z/10)^{power}$		
Rho	Air Density	[kg/m^3]	
WO_x	Wind Origin: x-coordinate in global system	[m]	
WO_y	Wind Origin: y-coordinate in global system	[m]	
WO_z	Wind Origin: z-coordinate in global system	[m]	
alpha	Rotation of wind axes about global x-axis	[deg]	
beta	Rotation of wind axes about global y-axis	[deg]	
gamma	Rotation of wind axes about global z-axis	[deg]	
n_ini	Number of points in the initialization curve for the fluctuating wind		0

With this record, the user may specify a wind field to be applied to the structure as aero dynamical forces.
The wind is 'switched' ON according to the LOADHIST/TIMEHIST definition. TIMEHIST **type 3 must** be used.

The time between calculations of wave forces is controlled by the referred TIMEHIST record, (dTime). The calculated wave forces are written to file if WAVCASE1 is specified in the control file.

NOTE! SI units must be used (N, m, kg) with **Z-axis pointing upwards!**

This record is given only once

NEW input syntax

WindField L_Case GWF_Type Ux Uy Uz Z_0 Z_{BOTT} Rho Power		Unit
LCase	Load Case ID for this wind definition	
GWF_Type	Global Wind Field Type: □ Uniform : Uniform wind in the entire space □ Z_Profile : The Wind speed increases for increasing Z □ 3D_Tab1 : The Global Wind is specified in Tables. Type 1. □ 3D_Tab2 : The Global Wind is specified in Tables. Type 2.	
Ux – Uz	X- Y- and Z component of the Global wind. Referred to Global Coords. The wind vector definition depend on Wind Field Type: □ Uniform : The uniform wind Vector □ Z_Profile : Wind Vector at elevation Z=10m above surface □ 3D_Tab1 : --- Dummy --. □ 3D_Tab2 : --- Dummy --.	[m/s]
Z_0 Z_{BOTT}	Z-coordinate of the Wind Coord System Origin Specified in **Glob** Coord Skip wind calculation for elements below Z_{BOTT} . Specified in Local Wind Coordinates. Default is 0.0.	[m]
Rho	Density of Air (If Omitted, ρ = 1.293 is used)	[kg/m^3]
Power	Power Law for Z-varying wind. $U(z) = U_{10}\left(\frac{Z}{10}\right)^{POWER}$ Where U_{10} is the Ux, Uy, and Uz specified at 10 m above ground.	

With this record, the user defines the Global wind Field to be used in the simulations..

Examples:

```
'           LCase    Type      Ux   Uy   Uz   Z0   Zbott   Rho   Power
 WindField    11     Uniform    10   10    0                               ! Min input
 WindField    12    Z_Profile   10   10    0    0      0   1.293   0.2   ! Max input
 WindField    13     3D_Tab1     0    0    0  150    150   1.293         ! Max input
```

This record could be repeated.

WindHist DOF Hist_ID		Unit
DOF	Actual Wind Vector Degree of Freedom: X (or 1) : X-Component of wind vector Y (or 2) : Y-Component of wind vector Z (or 3): Z-Component of wind vector	
Hist_ID	The Actual Wind Vector is Scaled according to the time history (command TimeHist) with the actual ID.	

With this record, the user defines the Global wind Filed to be used in the simulations..

Example:

```
'           LCase   Type     Ux  Uy  Uz  Z0   Zbott   Rho  Power
 WindField    11   Uniform   10  10   0                          ! Min input

'            Dof   HistID
 WindHist      X    101
 WindHist      Y    102


'          HistID    Type     T1     T2    fac
 TimeHist   101     S_Curve   0     10    1.0
 TimeHist   102     S_Curve  20     30    1.0
```

Means that the X-component of the wind will be initialized gradually from 0-10 m/s between time= 0 and 10 sec. Up to time=20s, the Y component is zero. From time=20, the Y-component gradually increases, and from time=30s, both X- and Y-components are 10m/s. (which gives a speed of 14.1m/s with 45° direction).

This record could be repeated..

WINDPAR	GeoID	Type	Height	Width	Np_T	Np_D	Np_L	Np_M	{Deg Coeff}$_1$ {Deg Coeff}$_2$ .... {Deg Coeff}$_N$

Parameter	*Description*	*Default*
GeoID	Cross section geometry ID	
Type	Input Type = 0	
Height	Profile Height	
Width	Profile Width	
Np_T Np_D Np_L Np_M	Number of points (deg/coeff) defining the Tangential Curve (=0, not imp) Number of points (deg/coeff) defining the Drag coefficient curve Number of points (deg/coeff) defining the Lift coefficient curve Number of points (deg/coeff) defining the Moment curve	
Deg	Cross section position (angle)	
Coeff	Corresponding coefficient NOTE: The four curves are given in sequence starting with the Tangential curve (Np_T grid points), then Np_D Drag coefficient points etc. Total number of coefficient points are then NpT+NpD+NpL+NpM.	

This record is used to define the aerodynamic coefficients for one cross section ID.
This record may be repeated.

Example 1, Pipe Cross section with diameter 0.55m:

	GeoID	Type	H	W	npt	npd	npl	npm	Deg	Coeff
WINDPAR	1001	0	0.55	0.55	0	1	0	0	0.0	0.7

Example 2, I Cross section:

	GeoID	Type	H	W	npt	npd	npl	npm	Deg	Coeff
WINDPAR	2001	0	0.8	0.6	0	5	3	1	-180	1.0
									-90	2.0
									0	1.0
									90	2.0
									180	1.0
									-180	0.0
									0	0.0
									180	0.0
									0	0.0

W_Coeffs ID Type {Data}		Unit
ID	Wind Coefficient ID, (a number) to be referred to.	
Type	Coefficient Type to define: □ Drag : Drag Coefficient Cd □ Lift : Lift Coefficient Cl □ Moment: Torsion Moment Cm □ Tang : Tangential (longitud) Ct	
Data	Actual pairs α - coefficient values.	

With this record, the user defines the Basic Aerodynamic Coefficients .

This record could be repeated..

W_Coeffs ID Combine ID_{DRAG} ID_{LIFT} ID_{MOM}		Unit
ID	Wind Coefficient ID, (a number) to be referred to.	
Combine	Keyword indicating that the command is used to combine basic drag- lift- and moment coefficient tables info one coefficient group to be assigned to the actual finite elements.	
ID_{DRAG}	ID of the Drag Coefficient table	
ID_{LIFT}	ID of the Lift Coefficient table, etc.	

With this record, the user defines the Combined Set of Aerodynamic coefficients, which should be referred to by the Finite elements.

Example:

```
`            ID      keyword  ID_Drag  ID_lift  ID_mom
 W_Coeffs   100   combine     101      102     103       ! Wing Profile
 W_Coeffs    10   combine      11                        ! Pipe Profile
```

Will create a coefficient set with ID=100 which combines the individual drag, lift and moment coefficient definitions. The next set uses only drag coefficient (for pipes).

This record could be repeated.

ElmCoeff ID List Type ID's		Unit
ID	Wind Coefficient ID, either a combined W_Coeffs or a complete coefficient package, (see above).	
ListType	Actual Cross section to assign the wind coefficients to: □ Pipe : Assign Coeffs to all elements with pipe cross section □ Elem : Assign Coeffs to all listed elements. □ Mat : Assign Coeffs to all listed materials □ Geo : Assign Coeffs to all listed geometries □ Group : Assign Coeffs to all listed groups.	
ID_s	ID of the actual cross section, elements, materials etc..	

With this record, the user assigns a complete set of Aerodynamic coefficients to a series of element, which are specified through their geometry (type and/or ID).

Example

```
'           Coeff_ID       ListType    ( IDs … )
 ElmCoeff       10          Pipe             ! Assign to all pipes
 ElmCoeff     1000          Wing             ! Assign to all Wing sections
 ElmCoeff     1200          Group   10 20 30 ! Assign to group 10 20 1nd 30
```

This record could be repeated..

MaxWind Criterion dT EndT Write

Parameter	*Description*	*Default*
Criterion	Criterion used to identify "worst wind state" of the actual wind field: Baseshear : Base Shear is used OverTurn : Overturning moment is used	
dT, EndT	The wind (defined by the WindField record) is stepped through with a time increment of dT up to time EndT. The worst time detected in the specified interval is used.	
Write	Write option: Write : The wind forces are written to file noWrite : No writing	

With this record the user may use USFOS to identify the worst wind state to be used in a static 'pushover' analysis. The load_case no of the specified WINDFIELD record may be referred to from the CUSFOS record.

Example 1 :

```
'            Typ      dT   EndT   option
MaxWind  BaseShear  0.5   16   noWrite
'            Typ      lCase
MaxWind  WindLCase  3     ! Use Wind Field load case 3

'          l_case  Type    ux  uy  uz    Z_0  z_bott Rho   Power
WindField    2   Z_Prof   20   0   0    100    0.0 1.225   0.2
WindField    3   Z_Prof    0  20   0    100    0.0 1.225   0.2
'
CUSFOS      10  10  1.0  1.0
       Lc    lfact  mxld nstep minstp
        1      0.5  1.0   10   0.010 ! DeadWeight
        3      0.1  5.0  200   0.001 ! Max Wind forces found by USFOS
```

The wind will be 'stepped' through the structure with a time interval of 0.5 s up to time = 16s. The wind forces at the time giving the maximum base shear are assigned to loadcase no 3, (override read-in node and element loads with this loadcase no). LoadCase no 3 is referred to as usual in the CUSFOS record.

This record is given only once!

6.3.18 Earthquake

NODEDISP I_case Node_ID dof_code [Values]

Parameter	*Description*	*Default*
I_case	Load Case number for the prescribed displacement, which is referred, to on the LOADHIST record.	
Node_ID	External node number	
dof_code	Integer number defining the prescribed degrees of freedom. The dof's are specified from 1 through 6, in global coordinates: 1 : Displacement in global X-direction 2 : Displacement in global Y-direction 3 : Displacement in global Z-direction 4 : Rotation about global X-axis 5 : Rotation about global Y-axis 6 : Rotation about global Z-axis	
Values	Reference values for the prescribed displacements. 0.0 (zero) defined fixed dof. **NOTE!** Values should only be specified for the dof's defined in the dof_code !	

This record defines prescribed displacements. The **dof_code** is formed by the dof numbers for all prescribed degrees of freedom at the node, **written consecutively, without blanks.**

Example 1:

```
NODEDISP   3   170  12   1.0      0.0
NODEDISP   3   180  12   1.0      0.0
```

Example 2:

```
NODEDISP   4   170  12   0.707    0.707
NODEDISP   4   180  12   0.707    0.707
```

Example 3:

```
NODEDISP   5   170  3    0.5
NODEDISP   5   180  3    0.5
```

In example 1 the nodes 170 and 180 are both given a prescribed displacement of 1.0 in global X-direction, global Y displacements are fixed.
In example 2 the same two nodes are both given a displacement value of 0.707 for both X- and Y-axis, (45 deg).
In example 3 the same two nodes are both given a prescribed displacement in global Z-axis with magnitude 0.5
Note that the values are ***reference values*** - if input is given in *time history format*, the total displacement will be a multiple of this value and the instantaneous value of the time history assigned through the LOADHIST record. If the displacement history defined in the referred TIMEHIST is the absolute displacements, the reference values should be set = 1.0.
For movement in directions different from the global axes, the reference values are used to define the actual directions, (see example 2).
If input is given in the conventional CUSFOS format, this value will be incremented in the same manner as external loads.

Note that the NODEDISP data are given *priority above the conventional boundary conditions*, (both free and fixed dofs may become prescribed)

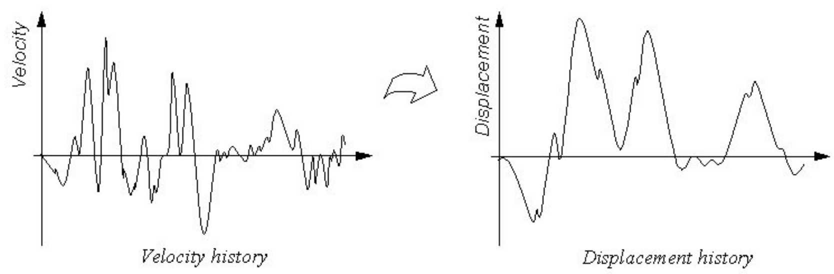
NODEVELO I_case Node_ID dof_code [Values]

Parameter	*Description*	*Default*
I_case	Load Case number for the prescribed velocity, which is referred, to on the LOADHIST record.	
Node_ID	External node number	
dof_code	Integer number defining the prescribed degrees of freedom. The dof's are specified from 1 through 6, in global coordinates: 1 : velocity component in global X-direction etc, see under NODEDISP	
Values	Reference values for the prescribed velocity components. 0.0 (zero) defined fixed dof. **NOTE!** Values should only be specified for the dof's defined in the dof_code !	

This record defines prescribed velocity. The **dof_code** is formed by the dof numbers for all prescribed degrees of freedom at the node, **written consecutively, without blanks.**

See NODEDISP for example of use.

Internally in USFOS the prescribed velocity history, (NODEVELO+TIMEHIST defines the history), is integrated to prescribed displacement history and applied as prescribed displacement.
The generated history is identified on the .out file with ID = -Original ID.



Note that the values are ***reference values***: the total velocity will be a multiple of this value and the instantaneous value of the time history assigned through the LOADHIST record. If the velocity history defined in the referred TIMEHIST is the absolute velocity, the reference values should be set = 1.0.
For movement in directions different from the global axes, the reference values are used to define the actual directions, (see example 2, NODEDISP).

Note that the NODEVELO data are given *priority above the conventional boundary conditions*, (both free and fixed dofs may become prescribed)

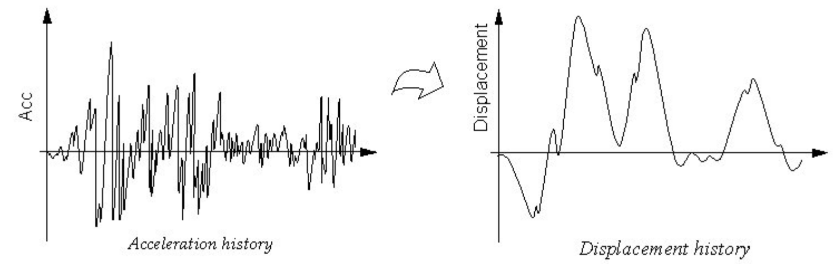
NODEACC I_case Node_ID dof_code [Values]

Parameter	*Description*	*Default*
I_case	Load Case number for the prescribed accelerations, which is referred, to on the LOADHIST record.	
Node_ID	External node number	
dof_code	Integer number defining the prescribed degrees of freedom. The dofs are specified from 1 through 6, in global coordinates: 1 : acceleration component in global X-direction etc., see under NODEDISP	
Values	Reference values for the prescribed acceleration components. 0.0 (zero) defined fixed dof. **NOTE!** Values should only be specified for the dofs defined in the dof_code !	

This record defines prescribed acceleration. The **dof_code** is formed by the dof numbers for all prescribed degrees of freedom at the node, **written consecutively, without blanks.**

See NODEDISP for example of use.

Internally in USFOS the prescribed acceleration history (NODEACC+TIMEHIST defines the history) is integrated to prescribed displacement history and applied as prescribed displacement.
The generated history is identified on the .out file with ID = -Original ID.



Note that the values are ***reference values***: the total acceleration will be a multiple of this value and the instantaneous value of the time history assigned through the LOADHIST record. If the velocity history defined in the referred TIMEHIST is the absolute acceleration, the reference values should be set = 1.0.
For movement in directions different from the global axes, the reference values are used to define the actual directions, (see example 2, NODEDISP).

Note that the NODEACC data are given *priority above the conventional boundary conditions*, (both free and fixed dofs may become prescribed).

This record could be repeated.

SOILACC l_case Type [data] dof_code [Values]

Parameter	*Description*	*Default*
l_case	Load Case number for the prescribed displacement, which is referred to on the LOADHIST record.	
Type	Definition type: Type = 1 : The soil is identified by Z_top Z_bott Type = 2 : The Soil is identified by the Pile ID	
data	Depends on Type=1 or Type=2. For Type = 1: Data : Z_Top Z_Bott. (Soil Coordinate System) For Type = 2: Data : Pile ID	
dof_code	Integer number defining the prescribed degrees of freedom. The dof's are specified from 1 through 6, in global coordinates: 1 : Displacement in global X-direction 2 : Displacement in global Y-direction 3 : Displacement in global Z-direction	
Values	Reference values for the prescribed displacements. 0.0 (zero) defined fixed dof. **NOTE!** Values should only be specified for the dof's defined in the dof_code !	

This record defines prescribed accelerations in the soil. The **dof_code** is formed by the dof numbers for all prescribed degrees of freedom at the soil nodes, **written consecutively, without blanks.**

Example 1:

```
   '          LC   Type  z_Top    z_Bott   DOF    Val1  Val2
  SOILACC     3     1    0.0     -20.0     12     1.0   0.5 ! Layer 1
  SOILACC     3     1  -20.0     -50.0     12     1.5   0.5 ! Layer 2
  SOILACC     3     1  -50.0    -100.0     12     2.5   1.0 ! Layer 3
```

See also examples on www.USFOS.com and release notes for USFOS version 8-5.

Note that the values are ***reference values*** - if input is given in *time history format*, the total acceleration will be a multiple of this value and the instantaneous value of the time history assigned through the LOADHIST record. If the displacement history defined in the referred TIMEHIST is the absolute displacements, the reference values should be set = 1.0.
For movement in directions different from the global axes, the reference values are used to define the actual directions, (see example 2).

This record could be repeated.

SOILDISP l_case Type [data] dof_code [Values]

Parameter	*Description*	*Default*
l_case	Load Case number for the prescribed displacement, which is referred to on the LOADHIST record.	
Type	Definition type: Type = 1 : The soil is identified by Z_top Z_bott Type = 2 : The Soil is identified by the Pile ID Type = 3 : The soil is identified by the Pile ID and Z_top Z_bott.	
data	Depends on Type=1, 2 or Type=3. For Type = 1: Data : Z_Top Z_Bott. (Soil Coordinate System) For Type = 2: Data : Pile ID For Type = 3: Data : Pile ID and Z_Top Z_Bott.	
dof_code	Integer number defining the prescribed degrees of freedom. The DOFs are specified from 1 through 6, in global coordinates: 1 : Displacement in global X-direction 2 : Displacement in global Y-direction 3 : Displacement in global Z-direction	
Values	Reference values for the prescribed displacements. 0.0 (zero) defines fixed dof. **NOTE!** Values should only be specified for the dofs defined in the dof_code !	

This record defines prescribed displacements in the soil. The **dof_code** is formed by the dof numbers for all prescribed degrees of freedom at the soil node(s), **written consecutively, without blanks.**

Example 1:

```
'           LC   Type   z_Top    z_Bott   DOF    Val1  Val2
SOILDISP    3     1     0.0     -20.0     12     1.0   0.5 ! Layer 1
SOILDISP    3     1   -20.0     -50.0     12     1.5   0.5 ! Layer 2
SOILDISP    3     1   -50.0    -100.0     12     2.5   1.0 ! Layer 3
```

Example 2:

```
'           LC   Type   Pile_ID   DOF    Val1  Val2
SOILDISP    3     2      1001     12     1.0   0.5 ! Subsidence of pile
```

Example 3:

```
'           LC   Type   PileID z_Top   z_Bott   DOF    Val1
SOILDISP    3     3     1001    0.0    -20.0     1     0.5 ! Layer 1
SOILDISP    3     3     1001  -20.0    -50.0     1     1.0 ! Layer 2
```

See also examples on www.USFOS.com and release notes for USFOS version 8-5 and 8-9.

Note that the values are ***reference values*** - if input is given in *time history format*, the total displacement will be a multiple of this value and the instantaneous value of the time history assigned through the LOADHIST record. If the displacement history defined in the referred TIMEHIST is the absolute displacements, the reference values should be set = 1.0.
For movement in directions different from the global axes, the reference values are used to define the actual directions, (see example 2).
If input is given in the conventional CUSFOS format, this value will be incremented in the same manner as external loads.

This record could be repeated.

SPRIDISP l_case Elem_ID dof_code [Values]

Parameter	*Description*	*Default*
l_case	Load Case number for the prescribed displacement, which is referred, to on the LOADHIST record.	
Elem_ID	Spring Element ID. NOTE! The spring material must be "MOVSPRI"	
dof_code	Integer number defining the prescribed degrees of freedom. The dof's are specified from 1 through 6, in global coordinates: 1 : Displacement in global X-direction 2 : Displacement in global Y-direction 3 : Displacement in global Z-direction 4 : Rotation about global X-axis 5 : Rotation about global Y-axis 6 : Rotation about global Z-axis	
Values	Reference values for the prescribed displacements. 0.0 (zero) defined fixed dof. **NOTE!** Values should only be specified for the dof's defined in the dof_code !	

This record defines prescribed displacements. The **dof_code** is formed by the dof numbers for all prescribed degrees of freedom at the spring node, **written consecutively, without blanks.**

Example 1:
```
SPRIDISP   3   170   12   1.0     0.0
SPRIDISP   3   180   12   1.0     0.0
```
Example 2:
```
SPRIDISP   4   170   12   0.707   0.707
SPRIDISP   4   180   12   0.707   0.707
```
Example 3:
```
SPRIDISP   5   170   3    0.5
SPRIDISP   5   180   3    0.5
```

In example 1 the springs 170 and 180 are both given a prescribed displacement of 1.0 in global X-direction, global Y displacements are fixed.
In example 2 the same two springs are both given a displacement value of 0.707 for both X- and Y-axis, (45 deg).
In example 3 the same two springs are both given a prescribed displacement in global Z-axis with magnitude 0.5
Note that the values are ***reference values*** - if input is given in *time history format*, the total displacement will be a multiple of this value and the instantaneous value of the time history assigned through the LOADHIST record. If the displacement history defined in the referred TIMEHIST is the absolute displacements, the reference values should be set = 1.0.
For movement in directions different from the global axes, the reference values are used to define the actual directions, (see example 2).
If input is given in the conventional CUSFOS format, this value will be incremented in the same manner as external loads.

Note that the SPRIDISP does not change the boundary conditions. The stiffness given under MATERIAL, type MOVSPRI is used.

6.4 SESAM STRUCTURAL FILE FORMAT

This section describes the input records of the FEM structural model and loads. The SESAM modules GeniE, PREFRAME, WAJAC and WALOCO may generate these records.

<table>
<tr><td colspan="3">RECORD IDENTIFICATOR   parameter no. 1   parameter no. 2   parameter no. 3</td></tr>
<tr><td>Parameter</td><td>Description</td><td>Default</td></tr>
<tr><td>parameter no. 1</td><td>Description of contents and default</td><td></td></tr>
<tr><td>parameter no. 2</td><td>Description of contents and default</td><td></td></tr>
<tr><td>parameter no. 3</td><td>Description of contents and default</td><td></td></tr>
<tr><td colspan="3">Optional box for comments, notes, exceptions etc.</td></tr>
</table>

The data items listed as *dummy* are set by the SESAM interface file format. They are skipped by USFOS. However, they must be present in the input data stream, or succeeding items of that record will be misinterpreted.

When input is given manually, there is no need to split input records over more lines. USFOS uses free format reading. One data record starts with the record identificator, and terminates when the next record identificator is encountered. Whether this is at the next line, or after several lines of data items is irrelevant to USFOS as long as the correct number of data items is given, and the specified order of the data items is satisfied.

6.4.1 Geometry Data

GNODE nodex nodeno		
Parameter	*Description*	*Default*
nodex	External node number specified by the user	
nodeno	Internal node number	
This record contains the correspondence between external, user defined, node numbering and internal node numbering in USFOS This record consists of one line, and has to be repeated once for every nodal point in the structure		

GCOORD nodeno xcoord ycoord zcoord		
Parameter	*Description*	*Default*
nodeno	Internal node number specified by the user	
xcoord	Cartesian x-, y-, and	
ycoord	z-coordinates of node *nodeno*	
zcoord		
This record contains coordinates of each node This record consists of one line, and has to be repeated once for every nodal point in the structure		

6.4.2 Connectivity Data

The **GELMNT1** record is compulsory, and contains the internal node numbers at the element ends. This record also contains the correspondence between external, user defined, element numbers and internal element numbers.

The **GELREF1** record is compulsory and contains reference to element data.

GELMNT1	elnox nodin1	elno (nodin2)(nodin3)(nodin4)	eltyp	dummy

Parameter	*Description*	*Default*
elnox	External element number specified by the user	
elno	Internal element number	
eltyp	Element type number 15 = Beam element (2 nodes) 18 = Spring to ground (1 node) 9 = Membrane element (4 nodes)	
nodin1	Internal node number of local node 1	
nodin2 . . .	Internal node number of local node 2	

This record defines the element topology, and contains the correspondence between external, user defined, element numbers and internal element numbering in USFOS

This record consists of two lines, and has to be repeated once for every element in the structure

GELREF1	elno	matno	dummy	dummy
	dummy	dummy	dummy	dummy
	geono	dummy	eccno	transno
	eccno(1)	eccno(2)	eccno(3)	eccno(4)

Parameter	*Description*	*Default*
elno	Internal element number	
matno	Material number	
geono	Cross sectional geometry number. Refers to record GBEAMG, GIORH, GBOX, GPIPE or GELTH	
eccno	Eccentricity vector reference number. Refers to GECCEN > 0: eccentricity reference number = 0: no eccentricity specified = -1: eccentricity reference numbers for all nodes are specified in line 4 of this record	
transno	Reference number of local element coordinate system (record GUNIVEC) or of local nodal coordinate system (BNTRCOS) in case of spring to ground or super element (ref AMATRIX)	
eccno(1)	Eccentricity vector reference number of node 1. Refers to record GECCEN	
eccno(2) . . .	Eccentricity vector reference number of node 2. Refers to record GECCEN	

This record contains references to element data

This record consists of three or four lines, depending on definition of nodal eccentricities

If the eccentricities at all element nodes are equal (ECCNO ≥ 0), then line 4 of this record is omitted for those elements

6.4.3 Cross Sectional Data

GBEAMG	geono	dummy area	xix	
	xiy	xiz	xiyx	wpix
	wpiy	wpiz	shary	sharz
	sceny	scenz		

Parameter	*Description*	*Default*
geono	Geometry number referred to in GELREF1	
area	Cross sectional area	
xix	Torsional moment of inertia about the shear centre	
xiy	Moment of inertia about y-axis	
xiz	Moment of inertia about z-axis	
xiyz	Product of inertia	
wpix	Plastic torsional section modulus about shear centre	
wpiy	Plastic sectional modulus about y-axis	
wpiz	Plastic sectional modulus about z-axis	
shary	Shear area in direction of y-axis	
sharz	Shear area in direction of z-axis	
sceny	Location of shear centre relative to y-axis	
scenz	Location of shear centre relative to z-axis	

This record contains the cross sectional parameters for a general cross section. GBEAMG input is skipped if GIORH, GBOX or GPIPE is given for the same cross section

This records consists of four lines

GIORH	geono	hzi	ty	bt
	tt	bb	tb	sfy
	sfz	hzj	bfy	bfz

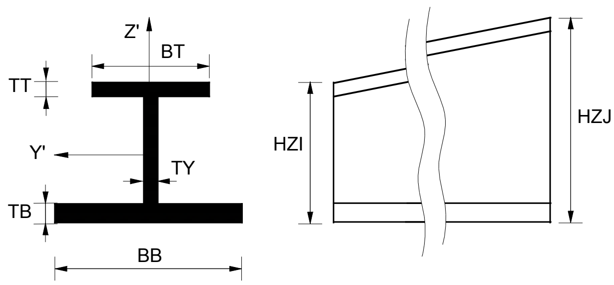
Parameter	*Description*	*Default*
geono	Geometry number referred to in GELREF1	
hzi	Height of beam	
ty	Thickness of beam web	
bt	Width of top flange	
tt	Thickness of top flange	
bb	Width of bottom flange	
tb	Thickness of bottom flange	
sfy	Shear area factor for y-axis direction. Shear area = *sfy* * shear area calculated by USFOS	1.0
sfz	Shear area factor for z-axis direction. Shear area = *sfz* * shear area calculated by USFOS	1.0
hzj	Height of beam at local node 2, **NOT** relevant for USFOS	
bfy	Shear buckling factor for y-axis direction	0.0
bfz	Shear buckling factor for z-axis direction	0.0

This record contains sectional geometry for an I or H type cross section
This record consists of three lines

Shear area calculated by USFOS for the local y- and z- directions:

A_s = I*t/S, where:

I = moment of inertia, y (z) axis
t = thickness
S = static moment y (z) axis



(Shear buckling effects are discussed in the USFOS Theory Manual)

GBOX	geono	hzi	ty	tb
	tt	by	sfy	sfz
	hzj			

Parameter	*Description*	*Default*
geono	Geometry number, referred to in GELREF1	
hzi	Height of beam et local node 1	
ty	Thickness of vertical walls (webs) of box section	
tb	Thickness of bottom flange	
tt	Thickness of top flange	
by	Width of beam	
sfy	Shear area factor of y-axis. Shear area = *sfy* * shear area calculated by USFOS	
sfz	Shear area factor of z-axis. Shear area = *sfz* * shear area calculated by USFOS	
hzj	Height of beam at local node 2, **not** relevant to USFOS	

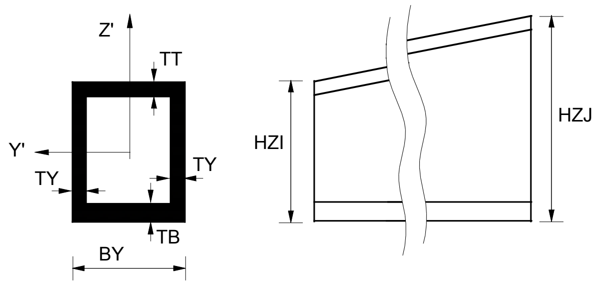
This record contains the cross sectional geometry of a box-type cross section

This record consists of three lines

Shear area calculated by USFOS for the local y- and z-directions:

A_s = I*t/S, where:

I = moment of inertia, y (z) axis
t = thickness
S = static moment, y (z) axis



GPIPE	geono	di	dy	t
	sfy	sfz		

Parameter	*Description*	*Default*
geono	Geometry number, referred to in GELREF1	
di	Inner diameter of pipe	
dy	Outer diameter of pipe	
t	Thickness of pipe	
sfy	Shear area factor of y-axis. Shear area = *sfy* · shear area calculated by USFOS	
sfz	Shear area factor of z-axis. Shear area = *sfz* · shear area calculated by USFOS	

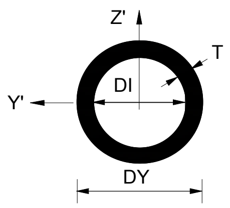
This record contains the cross sectional geometry of a pipe cross section

This record consists of two lines

Shear area calculated by USFOS for the local y- and z-directions:

$A_s = I \cdot t/S$, where:

I = moment of inertia, y (z) axis
t = thickness
S = static moment, y (z) axis



GELTH geono th dummy dummy		
Parameter	*Description*	*Default*
geono	Geometry number, referred to in GELREF1	
th	Thickness of the element	
This record contains of the plate thickness of the deck plate element This record consists of one line		

6.4.4 Element Data

GECCEN eccno ex ey ez

Parameter	*Description*	*Default*
eccno	Eccentricity number referred to in record GELREF1	
ex ey ez	Eccentricity vector in global coordinates. The vector points from the global node towards the local element node.	

This record contains the eccentricity vector of each nodal eccentricity. The eccentricity vector is given in global coordinates and points from the global node towards the local element node (ref Figure 3.3)

This record consists of one line

GUNIVEC transno unix uniy uniz

Parameter	*Description*	*Default*
transno	Unit vector number referred to in record GELREF1	
unix uniy uniz	Unit in global coordinates. The vector specifies the direction along local element z-axis	

This record consists of one line

6.4.5 Material Data

MGSPRNG	matno	ndof	k_{11}	k_{21}
	k_{31}	k_{41}	k_{51}	k_{61}
	k_{22}	k_{32}	k_{42}	k_{52}
	k_{62}	k_{33}	k_{43}	k_{53}
	k_{63}	k_{44}	k_{54}	k_{64}
	k_{55}	k_{65}	k_{66}	

Parameter	*Description*	*Default*
matno	Material number	
ndof	Number of nodal degrees of freedom (*ndof* = 6)	
k_{11}	Elements of the spring stiffness matrix	
k_{21} : : k_{ij} . . k_{66}	$k = \begin{bmatrix} k_{11} & & & & & \\ k_{21} k_{22} & & & & & \\ k_{31} k_{32} & k_{33} & & & & \\ k_{41} k_{42} & k_{43} & k_{45} & & & \\ k_{51} k_{52} & k_{53} & k_{54} & k_{55} & & \\ k_{61} k_{62} & k_{63} & k_{64} & k_{65} & k_{66} & \end{bmatrix}$	

This record defines the stiffness characteristics of linear springs to ground

Only the elements below the main diagonal are given, i.e. a symmetric stiffness matrix is assumed

The elements are referred to the global coordinate system, or to a local nodal coordinate system, if one is defined by record BNTRCOS and referenced by record GELREF1

MISOPL	matno	matyp	dummy	dummy
	dummy	dummy	dummy	dummy
	np	sig1	eps1	sig2
	eps2	....	sign(np)	eps(np)

Parameter	*Description*	*Default*
matno	Material number	
matyp	Material type: matyp = 4: Elastoplastic with kinematic hardening matyp = 2: Nonlinear hyper elastic matyp = 0: Nonlinear hyper elastic	
dummy	Parameters generated by SESAM pre-processor with no relevance for the present material model	
np	Number of points to represent the uniaxial stress-strain (force-displacement) curve for increased loading. Note NP < 25	
sig1	Stress/force at the first point representing the stress-strain curve (force-displacement curve)	
eps1	Corresponding strain/displacement at the first point representing the stress-strain curve (force-displacement curve)	
sig2	Stress at the second point representing the curve	
eps2	Corresponding strain at the second point representing the curve	
.		
.		
sig(np)	Stress/force at the last point representing the curve	
eps(np)	Corresponding strain/displacement at the last point representing the curve (see figure)	

With this record, the force-displacement (moment-rotation) curves for nonlinear spring element are defined

6.4.6 Boundary Conditions

BNBCD	nodeno	ndof	fix1	fix2
	fix3	fix4	fix5	fix6

Parameter	*Description*	*Default*
nodeno	Internal node number	
ndof	Number of degrees of freedom. *ndof* = 6	
fix1 fix2 fix3 fix4 fix5 fix6	Boundary condition of relevant degrees of freedom in global degrees of freedom fix i = 0: free = 1: fixed = 4: retained degree of freedom fix1: translation in x-direction fix2: translation in y-direction fix3: translation in z-direction fix4: rotation about x-axis fix5: rotation about y-axis fix6: rotation about z-axis	

This record specified the boundary condition of each relevant degree of freedom

Nodes with fix = 4 are called super nodes and may only be specified in a substructure analysis, refer input record SUBSTRU

This record consists of two lines

BNTRCOS	transno	t_{11}	t_{21}	t_{31}
	t_{12}	t_{22}	t_{32}	t_{13}
	t_{23}	t_{33}		

Parameter	*Description*	*Default*
transno	Transformation number referred to in GELREF1	
t_{11}	Nine direction cosines of the rotation transformation matrix	
t_{21}		
:		
:		
t_{32}		
t_{33}		

The transformation matrix **T** describes the transformation defined by

$$\mathbf{r}_{Loc} = \mathbf{T}\,\mathbf{r}_{Glob}$$

where $\mathbf{r}_{Loc}$ refers to the local coordinate system and $\mathbf{r}_{Glob}$ refers to the global coordinate system

$$T = \begin{bmatrix} \cos(x_L, X_G) & \cos(x_L, Y_G) & \cos(x_L, Z_G) \\ \cos(y_L, X_G) & \cos(y_L, Y_G) & \cos(y_L, Z_G) \\ \cos(z_L, X_G) & \cos(z_L, Y_G) & \cos(z_L, Z_G) \end{bmatrix}$$

This record consists of three lines

BELFIX	fixno	opt	trano	dummy
	fix1	fix2	fix3	fix4
	fix5	fix6		

Parameter	*Description*	*Default*
fixno	Fixation number, FIXNO is referred to from GELREF1	
opt	opt = 1 is mandatory, (either fully fixed or fully released).	
trano	trano = 0 is mandatory, (The fixation is then defined in **local** element coordinate system).	
fix1	Fixation Code of local X-direction. 0 : Fully Released. 1: Fully Fixed.	
fix2	Fixation Code of local Y-direction. 0 : Fully Released. 1: Fully Fixed.	
.	.	
.	.	
.	.	

This record specified internal hinge definition to be referred to from the GELREF1 record.

This record consists of three lines.

6.4.7 Load and Temperature Increments

The BNLOAD record defines nodal loads. The BELOAD1 record defines element line loads. The BGRAV record defines gravitational loads.

BNLOAD	llc	dummy	dummy	dummy
	nodeno	ndof	rload1	rload2
	rload3	rload4	rload5	rload6

Parameter	*Description*	*Default*
llc	Load case number	
nodeno	Internal node number	
ndof	Number of degrees of freedom. *ndof* = 6	
rload1 rload2 rload3 rload4 rload5 rload6	Load in direction of the relevant degree of freedom in the global coordinate system rload1: load in x-direction rload2: load in y-direction rload3: load in z-direction rload4: moment about x-axis rload5: moment about y-axis rload6: moment about z-axis	

This record defines nodal loads

This record consists of three lines

Note! The load is handled as conservative loading by USFOS

BELOAD1	llc	dummy	dummy	OPT
	elno	l1	l2	edof
	dummy	rint1(1)	rint2(1)	rint3(1)
	rint1(2)	rint2(2)	rint2(3)	

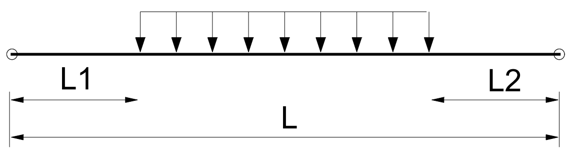
Parameter	*Description*	*Default*
llc	Load case number	
OPT	Option for reference l1/l2. OPT=0 : L1/L2 refers to dist from the ends of the flex part of the element OPT=1 : L1/L2 refers to distance from the nodes (disregarding eccs.).	
elno	Internal element number	
l1	Distance along the element from node 1 to the position where the line load starts acting	
l2	Distance along the element from node 2 to the position where the line load ends	
edof	Number of specified intensities (*edof* = 6)	
rint1(1) rint2(1) rint3(1)	Load intensities at local node 1 in global x-, y- and z-directions	
rint1(2) rint2(2) rint2(3)	Load intensities at local node 2 in global x-, y- and z-directions	

This record defines element line loads

The record may be repeated for each element, with different values for *l1* and *l2*

This record consists of four lines

Note! The load is handled as conservative loading by USFOS



BGRAV	llc	dummy dummy dummy	
	gx	gy gz	

Parameter	*Description*	*Default*
llc	Load case number	
gx gy gz	Component of gravity vector global x-, y- and z-directions	

This record defines gravitational loads

This record consists of two lines

Note! The load is handled as conservative loading by USFOS

BNWALO	llc	nodeno dummy ndof		
	rload1	rload2	rload3	rload4
	rload5	rload6		

Parameter	*Description*	*Default*
llc	Load case number	
nodeno	Internal node number	
ndof	Number of degrees of freedom. *ndof* = 6	
rload1 rload2 rload3 rload4 rload5 rload6	Load in direction of the relevant degree of freedom in the global coordinate system rload1: load in x-direction rload2: load in y-direction rload3: load in z-direction rload4: moment about x-axis rload5: moment about y-axis rload6: moment about z-axis	

This record defines nodal loads from wave load program

This record consists of three lines

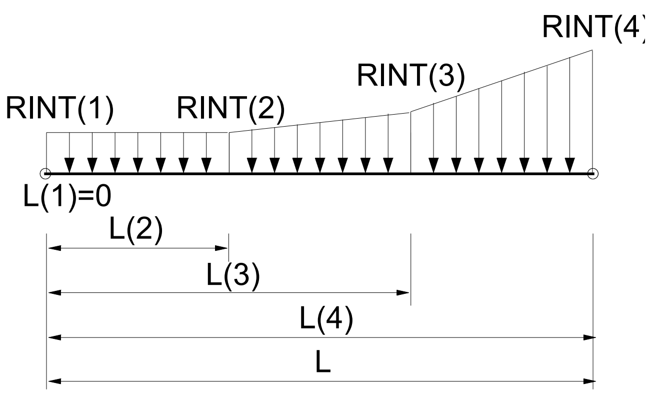
Note! The load is handled as conservative loading by USFOS

BEWALO1	llc	elno	dummy n	
	n	l(1)	l(2)	...
	l(n)	rint1(1)	rint2(1)	rint3(1)
	rint1(2)	rint2(2)	rint3(2)	...
	...	...	rint1(n)	rint2(n)
	rint3(n)			

Parameter	*Description*	*Default*
llc	Load case number	
elno	Internal element number	
n	Number of load points along the element	
l	Total element length	
l(1) l(2) : l(n)	Distance from local node 1 to load point no. 1 Distance from local node 1 to load point no. 2 : Distance from local node 1 to load point no. n	
rint1(1) rint2(1) rint3(1)	Load intensities at load point 1 in global x-, y- and z-directions	
rint1(2) rint2(2) rint3(2)	Load intensities at load point 2 in global x-, y- and z-directions	
:		
rint1(n) rint2(n) rint3(n)	Load intensities at load point n in global x-, y- and z-directions	

This record defines line loads from wave load program

Note! The load is handled as conservative loading by USFOS



BEISTE	llc	dummy	dummy	top
	elno	nnod	dummy	t1
	t2	dummy	dummy	dummy

Parameter	*Description*	*Default*
llc	Load case number	
top	Temperature specification option = 1: Mean element temperature specified = 2: Different temperature specified at each node	
elno	Internal element number	
nnod	Number of element nodes (*nnod* = 2)	
t1	Mean element temperature (*top* = 1) or temperature at local node 1 (*top* = 2)	
t2	Temperature at local node 2 (*top* = 2)	

This record contains specification of temperature increments without gradients over the element cross section

This record consists of two or three lines

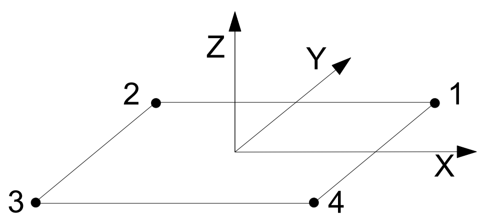
BEUSLO	llc	lotyp	dummy	dummy
	elno	nnod	dummy	dummy
	rload1	rload2	rload3	rload4

Parameter	*Description*	*Default*
llc	Load case number	
lotyp	Load type (only load type -1, normal pressure, non-conservative load is available)	
elno	Internal element number	
nnod	Number of nodes of the element	
rload1	Intensity of pressure at local node 1. Positive pressure is directed along local element z-axis	
rload2 . . .	Intensity of pressure at local node 2. Positive pressure is directed along local element z-axis	

This record defines element surface loads

The record may be repeated for each element

This record consists of 3 lines



6.4.8 Hydrodynamic Added Mass

BEMASS1	dummy	elno	dummy	np
	c(1)	c(2)	....	
	c(np)	am(1,1)	am(2,1)	am(3,1)
	am(1,2)	....	am(3,np)	

Parameter	*Description*	*Default*
elno	Internal element number	
np	Number of points along beam length for which data are given	
c(i)	Distance along the element from node 1 to the position no <i>/*ltot*	
am(j,i)	Added mass intensity for the j'th DOF in the beam element local coordinate system at point no. i	

This record defines added mass data for the beam element, generated by the wave load module WAJAC

This record consists of four lines

NOTE! This record has been changed in connection with the 7-7 release of USFOS in order to match the new, official SESAM format.

6.4.9 Nodes with point masses

BNMASS	nodeno	ndof	mass1	mass2	
	mass3	mass4	mass5	mass6	

Parameter	*Description*	*Default*
nodeno	Internal node number	
ndof	Number of degrees of freedom. *ndof* = 6	
mass1	Mass with respect to first degree of freedom	
mass2	Mass with respect to second degree of freedom	
mass3	:	
mass4	:	
mass5	:	
mass6	:	

This record specifies the nodal point masses for each relevant degree of freedom

This record consists of two lines

6.4.10 Nodes with initial conditions

BNINCO	dummy	dtype	dummy	dummy
	nodeno	ndof	velo1	velo2
	velo3	velo4	velo5	velo6

Parameter	*Description*	*Default*
dtype	Type of condition. Only legal value: 2 = velocity	
nodeno	Internal node number	
ndof	Number of degrees of freedom. *ndof* = 6	
velo1	Initial velocity in first degree of freedom	
velo2	Initial velocity in second degree of freedom	
velo3	:	
velo4	:	
velo5	:	
velo6	:	

This record specifies the node initial velocities for use in dynamic analysis

This record consists of three lines

6.4.11 Super-element definition

AMATRIX	nfield	matno	dummy nnod	
	nsub	dummy dummy	dummy	
	matrtyp	matrref	matrform	dummy
	....	....	....	....
	matrtyp	matrref	matrform	dummy

Parameter	*Description*	*Default*
nfield	Number of data fields on this record (including this and embedded bland fields)	
matno	Material number (reference number) for this AMATRIX record, specified on the GELREF1 record	
nnod	Number of nodes on this element (must correspond to specification on GELMNT1 record)	
nsub	Number of data fields in each sub-record (= 4)	
matrtyp	Matrix/vector type indicator: 1 = stiffness matrix 4 = load vector	
matrref	Matrix reference no. pointing to the reference *matrref* on the corresponding ADMSTIFF or AMDLOAD records	
matrform	= 0 Element vectors are stored (load) = 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	

This record specifies the stiffness properties and loads for a "premade" reduced super-element. The element may have "any" number of nodes, but the stiffness matrix must be symmetric

This record consists of minimum three lines

AMDLOAD	nfield	matrref	snodi	llc
	ndof	rload1	rload2	rload3
	rload4	rload5	rload6	

Parameter	*Description*	*Default*
nfield	Number of data fields on this record (including this)	
matrref	Reference number for this load vector record	
snodi	Supernode sequence number of super-element or internal node number of basic element	
llc	Load case number	
ndof	Number of degrees of freedom, *ndof* = 6	
rload1 rload2 rload3 rload4 rload5 rload6	Load in direction of the relevant degree of freedom rload1: load in x-direction rload2: load in y-direction rload3: load in z-direction rload4: moment about x-axis rload5: moment about y-axis rload6: moment about z-axis	

This record defines nodal loads for a basic element or reduced super-element. Each record consist of three lines and contains the load terms of one node for one load case

The loads are refereed to the global coordinate system, or to a local system if one is defined by record BNTRCOS and referred to in the GELREF1 record

Note! The load is handled as conservative loading by USFOS

AMDSTIFF	nfields	matrref	snodi	snodj
	coddof	k_{11}	k_{21}	k_{31}
	k_{41}	k_{51}	k_{61}	k_{12}
	k_{22}	...	...	k_{66}

Parameter	*Description*	*Default*
nfields	Number of data fields on this record (including this)	
matrref	Reference number for this stiffness matrix record	
snodi/ snodj	Supernode sequence number of super-element or internal node number of basic element	
coddof	Coded form of sub matrix dimension (1000*idof + jdof = 6006)	
k_{11} : : k_{66}	Sub matrix stiffness terms	

This record contains stiffness terms for a nodal sub matrix of an element stiffness matrix. It may be reduced super-element or basic element stiffness matrix. Each record contains the stiffness terms connecting one node *snodi* to another *snodj* or itself. The matrices are stored as sparse block data, which means that sub matrices with all terms equal to zero are not stored

The stiffness is referred to the global coordinate system, or to a local system if one is defined by record BNTRCOS and refereed to in the GELREF1 record

6.5 UFO STRUCTURAL FILE FORMAT

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 (IDs). These files may be written with a text editor, or generated by pre-processors.

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.

6.5.1 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.

6.5.2 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 loads.

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
M_x, ..	Concentrated Mass in X-, Y- and Z-direction.
M_Rx,..	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 $\mathbf{T_{11}}$ $\mathbf{T_{12}}$ $\mathbf{T_{13}}$
$\mathbf{T_{21}}$ $\mathbf{T_{22}}$ $\mathbf{T_{23}}$
$\mathbf{T_{31}}$ $\mathbf{T_{32}}$ $\mathbf{T_{33}}$

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.

LocNMass *RefType* NodeID ElemID		Unit
RefType	Reference Type = "Node"	
NodeID	Actual node (with a concentrated mass).	
ElemID	The Concentrated mass' properties are referred to the Element's local coordinate system	

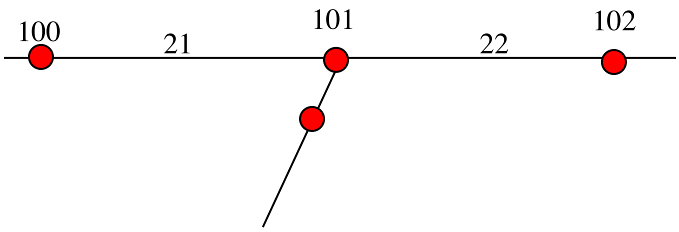
With this record, the user defines that the concentrated (nodal) masses are referred to a given element's local coordinate system.

Example:

```
`          Type    NodeID   ElemID
 LocNMass  Node     101       22

`         NodeID   X  Y  Z  rX   rY  rZ
 NodeMass  101     0  0  0  1E4   0   0
```

The rotation mass with value 10.000 kgm^2 will refer to element 22's local system, i.e., the mass becomes a local torsion mass of element 22.

This option is typically used in connection with modelling of rotating structures, where the properties should follow the structure's large displacements and rotations.



This record could be repeated.

WEIGHT	**Package** PackID KeyWord Value ListType (Id_List)
Parameter	*Description*
Package PackID	Indicates definition of a weight package: Actual Package ID
KeyWord	Actual keywords: *Concentrated* : Specification of concentrated mass. *Distributed* : Specification of distributed mass. *History* : Specification of time history to control mass Activate : Activation of actual weight package.
Value	If "*Concentrated*" : Value is actual mass [kg] If *"Distributed* : Value is actual mass per length [kg/m] If "*History*" : Value is actual time history ID controlling mass If "*Activate"* : No Value is given
ListType	Concentrated masses: How to define actual nodes. Only "Node" is available. Distributed masses How to define actual elements: Elem : Element List Mat : Material List Geo : Geometry List Group : Group List
Id_List	Actual ID(s) to be given actual mass

With this record, the user defines a concentrated/distributed mass, assigned to one or more nodes/elements, which could vary as a function of time.

Example:

```
'               PackID  KeyWord        Value    ListType  IDs
 Weight  Package   1     Concentrated   5000     Node      1
 Weight  Package   2     Concentrated   5000     Node      3

'
'               PackID  KeyWord     Hist_ID
 Weight  Package   1     History       2
 Weight  Package   2     History       3

'               PackID  KeyWord
 Weight  Package   1     Activate
 Weight  Package   2     Activate
```

..defines a concentrated mass of 5000 kg in nodes 1 and 3. The masses are controlled in time by individual time histories: Node 1 mass follows history no 2, while node-3 mass is changed according to time history 3. Both packages are activated.

See examples on www.usfos.com and release notes for USFOS 8-5.

This record may be repeated.

SURFIMP KeyWord { data }	
Parameter	*Description*
KeyWord	Actual keywords: *LoadCase* : Surface impact is a separate load case. *Attach* : Surface impact comes from an attached contact spring
data	Actual input parameters. See description below.

With this record, the user defines a concentrated force acting on the surface of a beam-element. Local denting of the tube wall is accounted for. The concentrated force is either an external load or caused by an attached spring.

NOTE! Relevant for PIPE cross sections only.

This record may be repeated.

SURFIMP "LoadCase" LCase Type ID Extent Fx Fy Fz

Parameter	*Description*
KeyWord	Actual keyword = LoadCase
LCase	Load Case number
Type	Load Type: *Node* : The surface load is specified for a node *Elem* : The surface load is specified for an element
ID	*Node* : Node ID (one number) *Elem* : Elem ID and position (two numbers)
Extent	Extent of the impact zone. See BIMPACT for description
Fx, Fy, Fz	Concentrated force components referring to Global System.

With this record, the user defines a concentrated force acting on the surface on a beam-element, pipe section.

Example 1:

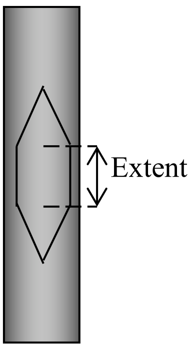
```
'          key        LCase     Type    NodeID    Extent     Fx     Fy     Fz
 SurfImp  LoadCase    2         Node       2        0.5     100E3    0      0
```

This defines a concentrated force of 100 kN acting in X-direction in node 2. The extent of the surface contact is 0.5m. USFOS will detect the pipe dimensions and compute the force-indentation curve. The denting of the tube wall is included for the beam elements attached to node 2. (Similar to BIMPACT).

Example 2:

```
'           Key      LCase  Type    ElemID    End  Extent     Fx     Fy     Fz
 SurfImp  LoadCase   3      Elem       1       3     0.1      1E6    0      0
```

This defines a concentrated force of 1 MN acting in X-direction at **midspan** of element 1. The extent of the surface contact is 0.1m. USFOS will detect the pipe dimensions and compute the force-indentation curve. The denting of the tube wall is included for the actual beam element. (Similar to BIMPACT). Since the load attacks at midspan, USFOS will divide element 1 in two and let the concentrated force act on the middle-node.



See examples on www.usfos.com and release notes for USFOS 8-8.

This record may be repeated.

SURFIMP	"Attach"	Elem ID	End	Extent

Parameter	*Description*
KeyWord	Actual keyword = Attach
Elem ID	Elem ID
End	Element End
Extent	Extent of the impact zone. See BIMPACT for description

With this record, the user defines a concentrated force acting on the surface on a beam-element, pipe section.

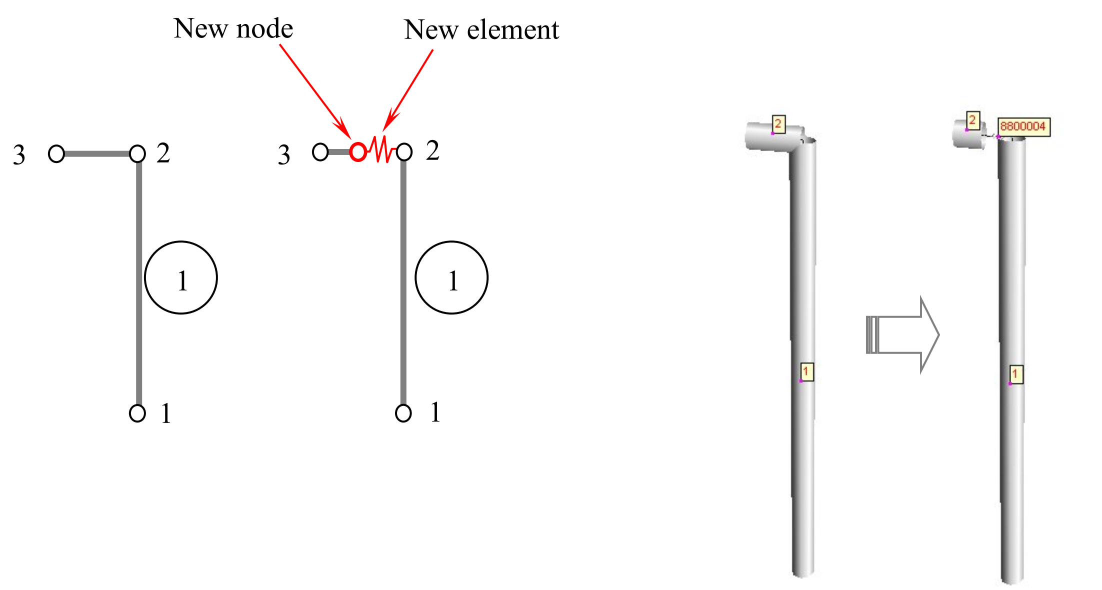
Example 1:

```
'          key     Elem ID    End    Extent
 SurfImp  Attach      2         2     0.5
```

This defines an attachment to element-2 in end-node 2. The extent of the surface contact is 0.5m. USFOS will detect the other element attached to end-node 2 and insert a contact spring between user's element and element-1. The surface contact spring characteristics is based on the force-indentation of element 1.

One new node and two new elements are included, (one to represent the denting and one to represent pipe-wall connection for large dents).

This option could be used to model for example a boat impact scenario.



See examples on www.usfos.com and release notes for USFOS 8-8.

This record may be repeated.

6.5.3 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 Ecc1. Node 2 of the beam has an eccentricity defined by 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 an 8-node hexahedron solid element to be used in the finite element analysis.

NOTE: Not implemented in USFOS

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 loads.

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 –3000, (loads are accumulated).

This record may be repeated.

SHELPRES	**Load Case** **LoTyp** **Pressure** **<List Type>** [Data]
Parameter	*Description*
Load Case	Load Case Number
LoTyp	LoadType (=Def).
Pressure	Actual Pressure (Positive value acts in the direction of local Z-axis of the elem)
<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 *Local* coordinate system.

Example 1:

SHELPRES 5 Def -1000.0 *Element* 1020 2020 3030

Defines a pressure of -1000 [N/m^2] in local Z-direction for elements 1020, 2020 and 3020.

Example 2:

SHELPRES 3 Def -2200.0 *Material* 245 355

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

Example 3:

SHELPRES 4 Def -2000.0 *Element*
SHELPRES 4 Def -1000.0 *Element* 1020 2020 3030

Defines a pressure (load case number 4) with intensity –2000 [N/m^2] in local Z-direction for all shell elements in the model, except for the elements 1020, 2020 and 3020, which get an intensity of –3000, (loads are accumulated).

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 for example 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 for example 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 for example 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.

X_ELMASS mass ListType ElemID ….

Parameter	*Description*
mass	Mass per unit length to be added on beam element
ListType	Elem : Specification element-by-element
ElemID	Element IDs, (beam elements only)

With this record, the user defines an extra distributed beam mass.

Example 1:

```
 '         mass [kg/m]   ListType  Elem1 Elem2 ..
X_ELMASS    2000         Elem      101   102   103
```

A translation mass of 2000 kg/m will be added to elements 101, 102 and 103. The option has impact on the following:

- Structural mass (inertia)
- Reaction forces (if gravity is specified)

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 1 If **no** elements are specified, all beam elements are specified with the actual added mass.
	NOTE 2 This option has impact on the inertia only. (Does not result in increased reaction forces)

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!

BEAMTYPE Type ListType { List }		
Parameter	*Description*	*Default*
Type	*Jacket* : The default "*USFOS ultimate strength*" element. (required for ultimate strength assessment of frames) *Riser* : Special element suited for slender systems (for example mooring / riser analysis)	*Jacket*
ListType	Specification of actual element(s): *Element* : Element ID list *Material* : Material ID list *Geometry*: Geometry ID list *All* : All beam elements become "riser", no further input	
{ List }	List of actual IDs	

This record is used to specify elements, which should use a special element formulation.

Example:

```
BeamType   Riser   Mat  100  200
```

All beam elements referring to one of the material ID's: 100 or 200 will use the special "riser" element formulation.

This record may be repeated

SPRITYPE Type ListType { IDList }		
Parameter	*Description*	*Default*
Type	*LumpSoil*: The spring is interpreted as a "Lumped Soil" element	
ListType	Specification of actual element(s): *Element* : Element ID list *Material* : Material ID list	
{ IDList }	List of actual ID's	

This record is used to specify springs, which should use a special element formulation.

Example:

```
'             ID      Node   Mat
 Sprng2Gr   1001       1   1000         ! One-Node Spring to ground
'
'              Model   ListType  ID
 SpriType    LumpSoil   Elem    1001   ! Use the Lumped Soil formulation
'
'                  P-Y   P-Y   T-Z    rX      rY     rZ
 MREF    1000    1001   1001  1003   0       0      0
'
'                                        - Curve for P-Y
'             MatID      P        d
 ElPlCurve    1001 -2001      -1.050
                   -2000      -0.050
                   -1000      -0.010
                    1000       0.010
                    2000       0.050
                    2001       1.050
'
'                                        - Curve for T-Z
'             MatID      P        d
 ElPlCurve    1003  -200E3    -1.000
                    -100E3    -0.010
                     100E3     0.010
                     200E3     1.000
```

A special foundation formulation is used for spring 1001. The same material has to be referred to for DOFs, 1 and 2, and this curve is used for the lateral (P-Y) response. DOF-3 refers to the vertical (T-Z) response. The P-Y capacity becomes the same for all directions in the horizontal plane.

This record may be repeated

6.5.4 Cross Section Data

PIPE Geom ID D_o T ShearY ShearZ D_2

Parameter	*Description*
Geom ID	User defined (external) geometry number
D_o	Outer Diameter of the pipe.
T	Wall thickness of the pipe.
ShearY ShearZ	Shear area factor of Y-axis. Shear area = ShearY * Calculated shear area. Shear area factor of Z-axis. Shear area = ShearZ * Calculated shear area If ShearY and ShearZ are omitted or equal to zero, ShearZ and ShearZ are both set equal to 1.0.
D_2	Outer Diameter at end 2 of element. If Omitted, end 2 gets same diameter as end 1.

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).

Example 2:

```
‘               ID          Do        T     ShY       ShZ        D2
        PIPE   45025       0.450  0.025    1          1        0.250
```

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). End 2 of all elements referring to this cross section get diameter =0.250.

This record may be repeated.

BOX **Geom ID** **H** **T_side** **T_bott** **T_top** **Width** ShearY ShearZ	
Parameter	*Description*
Geom ID	User defined (external) geometry number
H	Height of the profile.
T_side	Thickness of the sidewalls 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.
ShearY ShearZ	Shear area factor of Y-axis. Shear area = ShearY * Calculated shear area. Shear area factor of Z-axis. Shear area = ShearZ * Calculated shear area If ShearY and ShearZ are omitted or equal to zero, ShearZ and ShearZ 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** ShearY ShearZ	
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
ShearY ShearZ	Shear area factor of Y-axis. Shear area = ShearY * Calculated shear area. Shear area factor of Z-axis. Shear area = ShearZ * Calculated shear area If ShearY and ShearZ are omitted or equal to zero, ShearZ and ShearZ are both set equal to 1.0.
With this record, the user defines an 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 T_side	Height of the profile. Thickness of the vertical part.
Width T_bott	Width of the profile 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.	

SURFPIPE Geom ID D_o T

Parameter	*Description*
Geom ID	User defined (external) geometry number
D_o	Outer Diameter of the surface (pipe shape).
T	Wall thickness of the pipe (used to compute the internal area). .

Bu default, the surface geometry is identical to the cross section geometry.

With this record, the user re-defines the surface geometry to be used in the finite element analysis. This command does not define the cross section parameters, and therefore a conventional cross section has to be defined for this Geom-ID.

The purpose of this optional command is to re-define the section shape, for example when a buoyancy element should be defined or when the GENBEAM section is used.

The SURFPIPE data are used in connection with the following operations:

- Visualization in Xact
- Hydrodynamic calculations (Drag, and Mass forces. Buoyancy)
- Aerodynamic calculations (Drag forces)
- Radiation (FAHTS) (Basis element for the meshing)

Example 1:

```
PIPE        45020   0.200  0.020    ! Structural Properties
SURFPIPE    45020   0.450  0.020    ! Buoyancy element
```

Defines a pipe identified by the ID: 45020 with a cross section diameter = 0.20 and wall thickness = 0.020. The diameter used in the hydrodynamic calculations is 0.450m

Example 2:

```
GENBEAM    10   1.20E-1 4.4E-4 2.20E-6 2.20E-6 2.2E-5  2.2E-5 2.20E-5 1.2E-1 1.2E-1
SURFPIPE   10   0.300  0.020    ! Riser shape
```

Defines the diameter to be 0.300m in the hydrodynamic calculations of a riser element. The element gets its structural properties from the GENBEAM definition.

This record may be repeated.

6.5.5 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 P_2 δ_2 P_n δ_n	Definition of the first point of the force/displacement curve Definition of the second point of the force/displacement curve 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 hyper elastic 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 hyper elastic 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 P_2 δ_2 P_n δ_n	Definition of the first point of the force/displacement curve Definition of the second point of the force/displacement curve 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.

MATERIAL	**Mat ID** **Type** **{ Data }**
Parameter	*Description*
Mat ID	User defined (external) material number
Type	Actual Material Type: *Elastic : Elastic Isotropic* *Plastic : Elasto-Plastic* *CompSpri: Compression Spring* *TensSpri : Tension Spring* *Grout : Grout (inside pipes)* *Fric : Friction spring (see separate page)* *General : Just general curves* *Curve : Non-Linear spring with One NL DOF.* *Bearing : Linear Bearing* *LoadElem: Load element* *Equipment: Equipment* *MovSpri : Moving Spring (SpriDisp)*
Data	Data required by the actual material type, see below. **NOTE !** Zeros at the end of the record may be omitted.

With this record, the user defines different types of material.

Elastic & Plastic:

MATERIAL	MatID	Elastic	E-Mod	Poiss	Dummy	Density	ThermX
MATERIAL	MatID	Plastic	E-Mod	Poiss	Yield	Density	ThermX

E-Mod : Modulus of elasticity
Poiss : Poisson's ratio
Dummy : Dummy parameter (Yield is dummy for elastic material)
Yield : Yield stress (Relevant for plastic material)
Density : Material Density
ThermX : Thermal Expansion Coefficient

CompSpri & TensSpri:
(This material could be used together with *dynamic* simulations for a special beam formulation, which his activated using the command BeamType 5 {ListType} {IDList})

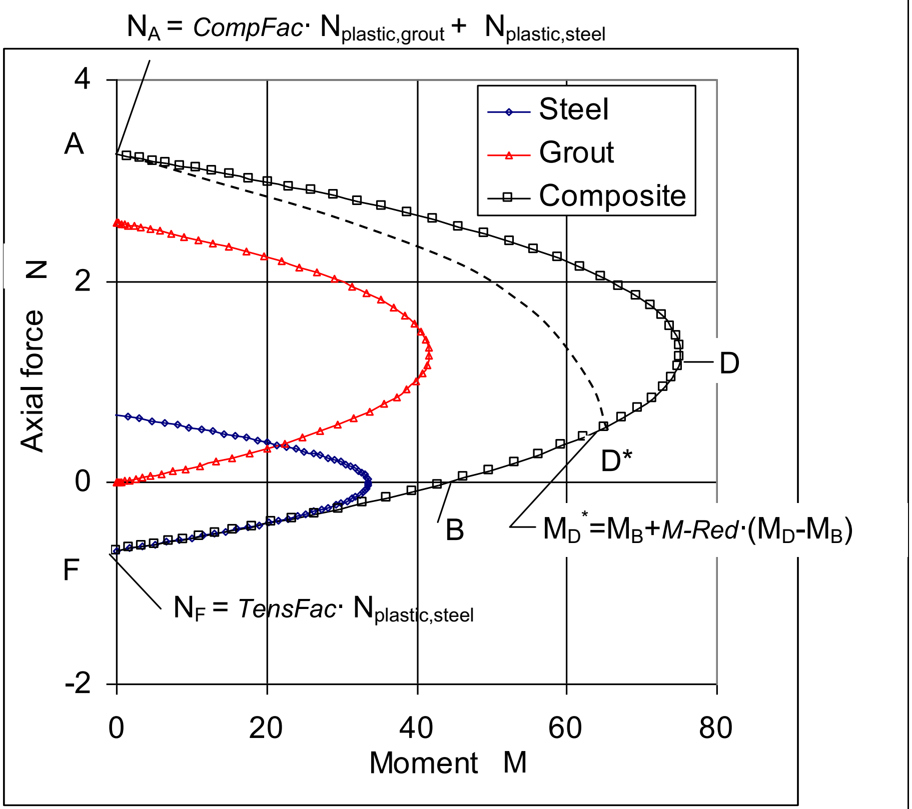
MATERIAL	MatID	CompSpri	Typ	K	Gap	Density	History
MATERIAL	MatID	TensSpri	Typ	K	Gap	Density	History

Typ : Spring Behaviour Typ = 1 :
Typ = 2 :
K : Spring Stiffness
Gap : Gap/Slack before spring is activated
Density : Material Density
History : Spring length follows actual time history.

This record may be repeated.

MATERIAL Mat ID ***Grout*** E Yield Dens TensFac CompFac M-Red I-Red ShearFac

Parameter	*Description*	*Default*
Mat ID	User defined material ID to be referred to from the commend *GROUTED*	
Grout	Grout material keyword	
E	Elastic modulus for grout	
Yield	Yield stress for grout	
Dens	Grout density or equivalent density (specific mass). See note below.	
TensFac	Tension capacity factor (N_F = TensFac· $N_{plastic,steel}$)	1.12
CompFac	Compression capacity factor (N_A = CompFac· $N_{plasticgrout}$ + $N_{plastic,steel}$)	1.0
M-Red	Moment reduction factor (M_D^*=M_B+M-Red·(M_D-M_B) 0 < M-Red < 1)	1.0
I-Red	Moment of inertia reduction factor (EI =$(EI)_{steel}$+(I-Red·EI$)_{grout}$)	1.0
ShearFac	Shear capacity factor ($Q_{plastic}$ = ShearFac· $(A\tau_Y)_{steel}$)	1.0
	See Figure below for definitions	



Plastic axial force-bending moment interaction for grouted tube

Density:
The mass from the grout is computed based on the grout density and the internal area of the pipe.

Grout is an *attribute* to elements; i.e. the elements are “filled with grout”. (no change in the normal input) Commenting out the **Grouted** command gives the normal, steel beam with no grout.

For tubular members with internal grout the material is referred to using the command:

Grouted <MatID> Elem Elem1 Elem2 …
Grouted <MatID> Mat Mat1 Mat2 …
Grouted <MatID> Geo Geo1 Geo2 …

NOTE:
It is recommended to model grouted members with two beam elements (use for example the REFINE command). Modelling with *one beam element* may give a discontinuous force-bending moment interaction history at member mid span when a plastic hinge is introduced**.**

This record may be repeated

MATERIAL ***Mat ID*** **Fric** ***SpriType*** ***Coeff*** ***Stiff*** ***FricDof(s)*** ***ForceDof***

Parameter	*Description*
Mat ID	Material ID
Fric	Material Type
SpriType	Actual Spring Types: *Comp* : *Compression spring* *Tens* : *Tension spring* *Symm* : *Symmetric spring (tension & compression)*
Coeff	Friction coefficient
FricDof(s)	Specification of the friction degrees of freedom, (one coded number)
ForceDof	Specification of the degree of freedom which controls the friction force

With this record, the user defines different friction springs.

The special spring, (1- or 2-node), uses the same, specified elastic stiffness in the actual degrees of freedom ("friction" and "force"). However, depending of the spring type and force, the stiffness is set equal to zero:

- Tension spring : Zero stiffness when spring goes into compression
- Compression spring : Zero stiffness when spring goes into tension
- Symmetric spring : Stiffness is kept always for the "force" degree of freedom

The maximum friction force = Coeff x ForceDofForce.

Example 1:

```
'          MatID   Typ  SpriType   Coeff   Stf  FricDofs  ForceDof
Material   100    Fric  Comp       0.5   1E6     12         3
'
```

The spring has following properties:

- Stiffness = 1E6 [N/m]
- Friction coefficient = 0.5
- Degrees of freedom 1 and 2 (X and Y) are friction dependent
- The friction capacity depends on the force in Z-direction.
- The spring takes compression only.

This spring could for example be a vertical support, which will slide in the horizontal plane when the horizontal force exceeds the actual friction capacity (the normal force x friction coefficient)

This record may be repeated.

MATERIAL ***Mat ID*** **General** **CurveType** **{ Data }**

Parameter	*Description*
Mat ID	Material ID
General	Material Type
CurveType	Actual Curve Type: *S_Curve : S-shaped curve (see TimeHist)* *Points : Discrete points (see TimeHist)* *TwoLevel: Two-level S_Curve (see TimeHist)*
Data	Data defining the actual curve. See TimeHist definitions.

With this record, the user defines a general curve.

Example:

```
'        MatID   Typ   CurveType  Work1  Work2 Fac
Material  102  General  S_Curve    0.1    0.5  0.9 ! cyclic damage
'
```

This record may be repeated.

MATERIAL	***Mat ID***	**Curve**	**DOF**	**kElast**	**P**	**delta**
					P	**delta**
					P	**delta**

Parameter	*Description*
Mat ID	Material ID
Curve	Material Type : Curve
DOF	Actual degree of freedom with non-linear curve
kElast	Elastic Stiffness to be used for all the remaining 5 degrees of freedom.
P delta	Pairs of Force displacement defining the non-linear curve. First point is 0.0 0.0.

With this record, the user defines a non-linear curve, which creates a non-linear spring.

Example:

```
'          ID    n1    n2     mat     geo
 Beam       1     1     2      2       1

'          MatID   Type       Dof   kElast      P      Delta
 Material  2       Curve       1    1E6         0        0.0
                                               2E6       0.1
                                               3E6       1.1
                                             3.1E6     100.0
```

The "Beam" will be converted to a 2-node Non-Linear spring with the axial degree of freedom defined by the P delta curve. The other 5 degrees of freedom of the spring get stiffness 1E6.

This record may be repeated.

MATERIAL ***Mat ID*** **Bearing** **Type** **s11 s22 s33 s44 s55 s66**

Parameter	*Description*
Mat ID	Material ID
Bearing	Material Type : Bearing
Type	Bearing type. (only *lin* is available)
s11-s66	Elastic Stiffness for the 6 degrees of freedom.

With this record, the user defines the linear stiffness of a bearing spring element.

Example:
```
'         ID    n1    n2     mat    geo
 Beam      1     1     2       2      1

'         MatID  M-Type   Type   s11   s22   s33   s44   s55   s66
 Material  2    Bearing   Lin   1E6   1E6   1E6   1E6    0     0
```

The "Beam" will be converted to a 2-node Linear bearing spring with no stiffness for DOFs 5 and 6.
The other 4 degrees of freedom of the spring get stiffness 1E6.
This material is typically used for Zero-Length springs, which create internal hinges.

This record may be repeated.

MATERIAL *Mat ID* **LoadElem**

Parameter	*Description*
Mat ID	Material ID
LoadElem	Material Type : LoadElem

This is a dummy material used by the Load elements (ref LOAFIELD). The 4 node shell element referred to this material becomes a non-structural load element.

This record may be repeated.

MATERIAL *Mat ID* **Equipment** **Mass**

Parameter	*Description*
Mat ID	Material ID
Equipment	Material Type : Equipment
Mass	Equipment Mass

This is a material used by the Equipment elements to define the equipment mass

This record may be repeated.

MATERIAL ***Mat ID*** **MovSpri Opt s11 s22 s33 s44 s55 s66**

Parameter	*Description*
Mat ID	Material ID
MovSpri	Material Type : Moving Spring
Opt	Spring type. (only *lin* is available)
s11-s66	Elastic Stiffness for the 6 degrees of freedom.

With this record, the user defines the linear stiffness of a bearing spring element.

Example:

```
'            ID    n1     mat    geo
 Sprng2Gr     1     1      2      1

'        MatID  M-Type   Type   s11   s22   s33   s44   s55   s66
 Material  2    MovSpri   Lin    1E6   1E6   1E6   1E6   0     0
```

The 1-node "Sprng2Gr" will be converted to a 1-node Linear moving spring with no stiffness for DOFs 5 and 6. The other 4 degrees of freedom of the spring get stiffness 1E6.

This material is used for springs, which are using the load of type SPRIDISP. The stress-free conditions are changed during the analysis, and these changes will set up internal forces in the spring. Iterations are required when this option is used.

This record may be repeated.

6.5.6 Misc. Data

GRAVITY **Load Case** **Ax** **Ay** **Az**	
Parameter	*Description*
Load Case	Load Case Number
Ax Ay Az	Acceleration field in global X-direction Acceleration field in global Y-direction 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 Translation Acceleration Field.

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.

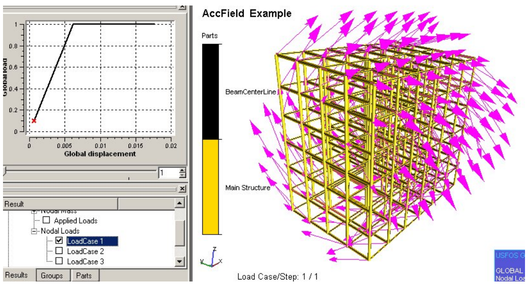
ACCFIELD Load Case Type Opt aRx aRy aRz Xc Yc Zc

Parameter	*Description*
Load Case	Load Case Number
Type	Actual Field Type. *Rot* : Actual Field is a Rotation Field
Opt	Actual Option: *Stat* : The Acc field is a static field in a fixed coordinate system
aRx aRy aRz	Rotating Acceleration field around global X-axis Rotating Acceleration field around global Y-axis Rotating Acceleration field around global Z-axis
XYZc	Coordinate of the Rotation Centre specified in Global coordinate system

With this record, the user defines Rotation Acceleration Field.

Example:

```
'                                                     < Centre >
'          LoadCase  Type   Opt     aRx   aRy  aRz   X    Y    Z
 AccField     1       Rot   Stat     1     0    0   30    0   30
```



defines an acceleration field of magnitude 1.0 [rad/s²] about Global X-axis. The rotation centre has coordinates (30, 0, 30). All data refer to Global coordinate System.

This record may be repeated.

LOAFIELD **Load Case** **Operation** **[Data]**	
Parameter	*Description*
Load Case	Load Case Number
Operation	*Define* : Define the load element pressure Assign : Assign the pressure to structural elements either by distance or referring to elem, mat or geo *Remove* : Individual removal of pressure on specific elements

With this record, the user defines a Load Field, which will be mapped to the shell element within the area (and distance) of the actual load element.

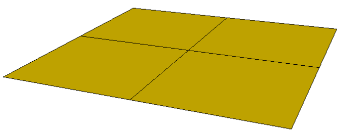
Examples:

A load element is a "QuadShell" with special Material- and Geometry.

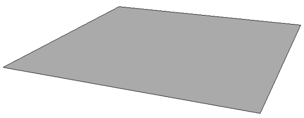
```
`            Elem n1 n2 n3 n4   Mat  Geo
 QuadShell   1000  1  2  3  4   100  100

 Material  100    Loadelem
 Geometry  100    Loadelem
```

The load element properties are defined using the LoaField command with different options.

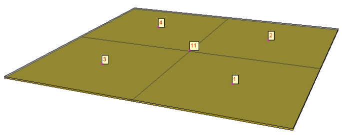
```
'
'           LoadCase   Operation   Option-1    Option-2  Value   LoadElmId
 LoaField     100        Define    LoadElem    Pressure  -10E3     1000
 LoaField     100        Assign    Distance     0.020
 LoaField     100        Remove    Elem         4
'

 LoaField     200        Define    LoadElem    Pressure  -20E3     2000
 LoaField     200        Assign      Mat         1
```



4 shell elements



1 load panel



Shells and Panel

The shell elements within a distance of 20mm get the actual pressure from the load element.

This record may be repeated.

EQUIP_1N	**Elem ID nod1 mat geo**
Parameter	*Description*
Elem ID	User defined (external) element number
Nod 1	The equipment is attached to the actual Node.
mat	Material Reference for the Equipment
geo	Geometry reference for the Equipment

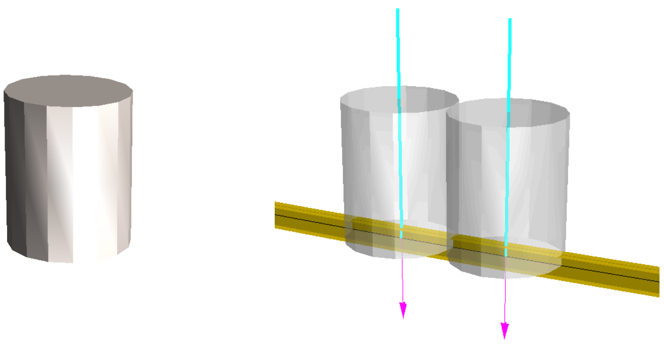
With this record, the user defines a 1-node equipment mass element to be used in the finite element analysis.

Example 1:

```
`           Elem    n1   Mat     Geo
 Equip_1N   1001    13   10      10

 '           ID      Type      ShapeNo
 Geometry    10    TankVert      1

'            ID     Typ        Mass
 Material    10    Equip       10E3
```

..defines a 1-node equipment element with ID = 1001, connected to the node with ID = 13. The mass is 10 mT (10,000 kg), and the node will be assigned a mass of 10,000 kg.

In xact, the mass-element will be visualized as shown in the images. The blue bars indicate the location of the node masses from the two mass-elements. The red arrows show the applied load if gravity is activated.



This record may be repeated.

EQUIP_2N   Elem ID   nod1 nod2   mat geo

Parameter	*Description*
Elem ID	User defined (external) element number
Nod 1-2	The equipment is attached to the actual 2 Nodes.
mat	Material Reference for the Equipment
geo	Geometry reference for the Equipment

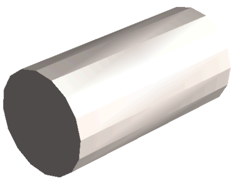
With this record, the user defines a 2-node equipment mass element to be used in the finite element analysis.

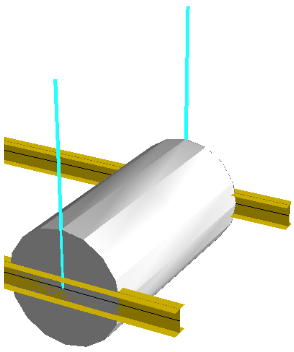
Example 1:

```
`           Elem    n1   n2   Mat     Geo
 Equip_2N   2001    12   22   10      10

 '           ID      Type      ShapeNo
 Geometry    10    TankHoriz      1

'            ID     Typ         Mass
 Material    10    Equip        20E3
```

..defines a 4 node equipment element with ID = 2001, connected to the two nodes with ID = 12 and 22. The total mass is 20 mT (20,000 kg), and each of the nodes will be assigned a mass of 10,000 kg.

In xact, the mass-element will be visualized as shown in the images. The blue bars indicate the node-masses.





This record may be repeated.

EQUIP_4N	**Elem ID nod1 nod2 nod3 nod4 mat geo**
Parameter	*Description*
Elem ID	User defined (external) element number
Nod 1-4	The equipment is attached to the actual 4 Nodes.
mat	Material Reference for the Equipment
geo	Geometry reference for the Equipment

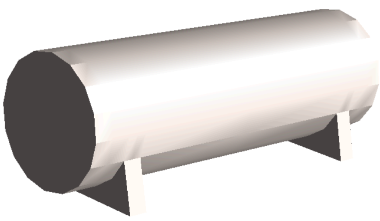
With this record, the user defines a 4-node equipment mass element to be used in the finite element analysis.

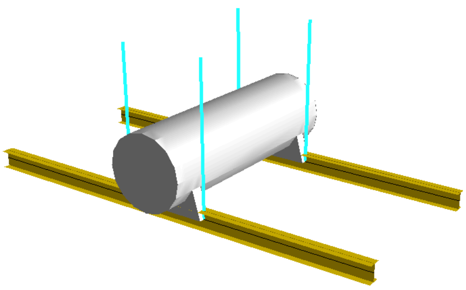
Example 1:

```
`           Elem    n1  n2  n3  n4    Mat     Geo
 Equip_4N   1234     1   2   3   4    10      10

 '            ID      Type      ShapeNo
 Geometry    10    TankHoriz       1

'             ID     Typ         Mass
 Material    10    Equip        20E3
```

..defines a 4 node equipment element with ID = 1234, connected to the four nodes with ID = 1, 2, 3 and 4. The total mass is 20 mT (20,000 kg), and each of the nodes will be assigned a mass of 5,000 kg.

In xact, the mass-element will be visualized as shown in the images. The blue bars indicate the node-masses.





This record may be repeated.

6.5.7 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.

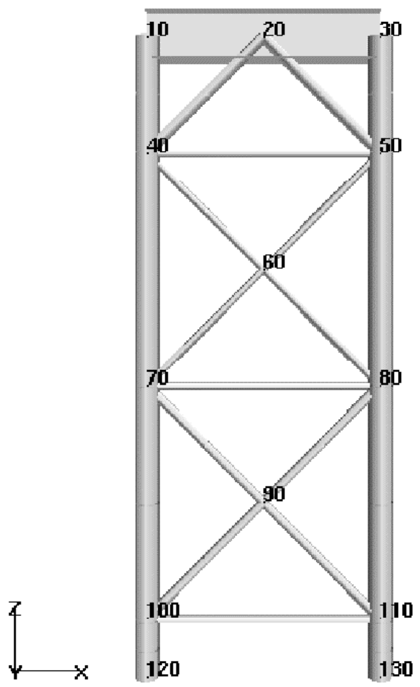


Figure 6-3 Node Numbers

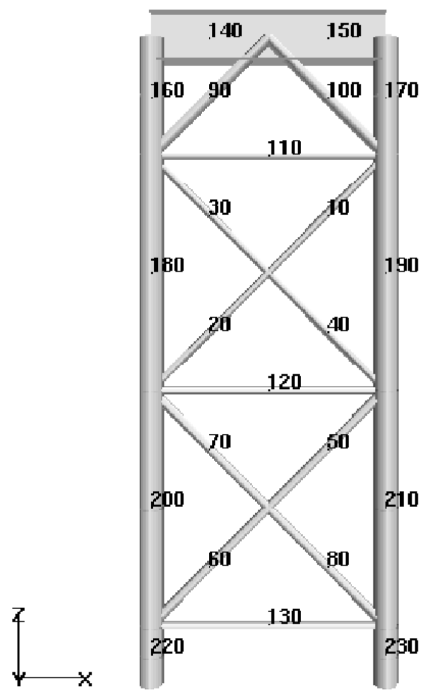


Figure 6-4 Element Numbers

UFO input file describing the Zayas 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 Shy ShZ
 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
```