SINTEF

**SINTEF** Structural Engineering

Address: N-7034 Trondheim, NORWAY.

Tel +47 7359 5611
Fax: +47 7359 2660

# MEMO

| MEMO CONCERNS | FOR YOUR ATTENTION | COMMENTS ARE INVITED | FOR YOUR INFORMATION | AS AGREED |
|---|---|---|---|---|
| **Release Notes**<br>**USFOS Version 7-6** | | | | |
| DISTRIBUTION | | | | |
| Members of USFOS user group | | | x | |

ONE COPY TO RECORDS OFFICE

| FILE CODE | CLASSIFICATION |
|---|---|
| | Open |
| ELECTRONIC FILE CODE | |

| PROJECT NO. | DATE | PERSON RESPONSIBLE/AUTHOR | NUMBER OF PAGES |
|---|---|---|---|
| 22L050 | 1999-04-20 | Tore Holmås | 18 |

# *Release notes*
# *USFOS 7-6 1999*

## Contents:

# 1. Introduction

The current version of USFOS (version 7-6, 99-04-20) is the final release of the 97-98 user group development period.

The 7-6 version is the USFOS version, which will be used when the next millennium is entered. As USFOS does *not* use date as input to the calculations (print of time for analysis initiation only), the change from year 1999 to 2000 is assumed to cause no problems.

By artificially changing the date to the year 2000 one customer has tested the USFOS package on their own computers with following conclusion: *"We have successfully tested USFOS , XFOS and POSTFOS thru 2-3 crucial date changes and has worked in all cases."*

The current release with date 1999-04-20 contains following:

- CD-ROM
- Updates of User's Manual
- Release Notes (this MEMO)

# 2. Contents of CD-ROM

## 2.1. Overview

The CD contains documentation, examples and new versions of the program codes, and the organisation is described in Figure 2.1-1. Both UNIX and NT solutions are collected in the same CD.

Contents of 'usfos7-6'

| Name | Size | Type | Modified |
|---|---|---|---|
| Document | | File Folder | 15.06.99 13:41 |
| Examples_PC | | File Folder | 15.06.99 13:41 |
| Examples_UNIX | | File Folder | 15.06.99 13:45 |
| Usfos_for_DecAlpha | | File Folder | 15.06.99 13:44 |
| Usfos_for_HP | | File Folder | 15.06.99 13:43 |
| Usfos_for_SGI | | File Folder | 15.06.99 13:44 |
| Usfos_for_Windows_NT4.0 | | File Folder | 15.06.99 13:44 |

**Figure 2.1-1 Contents of CD-ROM**

## 2.2. New versions of the program codes

Under each file folder (f ex "USFOS_for_Windows_NT4.0"), two folders, (bin and etc) are located. The "*bin*" folder contains the program code, while the "*etc*" folder contains set up files.

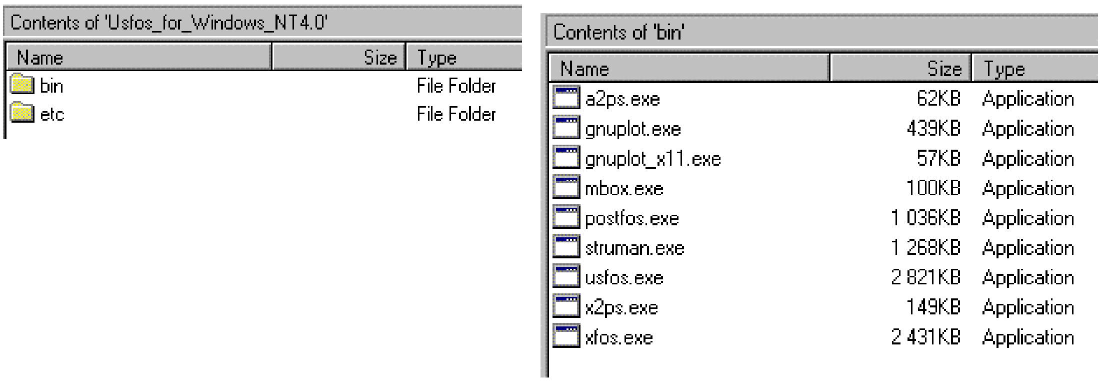



**Figure 2.2-1 Program Code located in "bin" folder**


**Figure 2.2-2 Files in "etc" folder. NT (to the left) and UNIX (to the right)**

### *Installation on UNIX:*

- Create a root directory for USFOS, (the new "*USFOS_HOME* " directory)
- Copy the actual "bin" and "etc" directories to *USFOS_HOME*
- Copy the "Examples_UNIX" and "Document" directories to *USFOS_HOME*.
- Define the USFOS_HOME variable in the USFOS.cshrc/USFOS.kshrc files

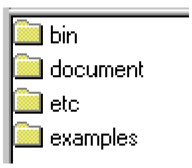



**Figure 2.2-3 Contents of "$USFOS_HOME" folder after installation**

***Installation on Windows NT 4.0***

- Copy the new ".exe" files located in the "bin" folder to the existing "USFOS_HOME/bin" folder
- Copy the new "postfos.inca" file located in the "etc" folder to the existing "USFOS_HOME/etc" folder
- Copy the "Examples_PC" and "Document" folders to the existing USFOS_HOME.

***NOTE ! :*** *If USFOS has never been installed on NT before, please contact SINTEF.*

***For all systems:***

- Copy the file: "USFOS.key" (delivered on a separate diskette) to the actual "USFOS_HOME/etc" directory.

## 2.3. Manual

The User's manual is updated, and (paper) copies of the actual pages are delivered. In addition, the most important part of the manual, the "Input Description" is available for "on-line" reading using f ex. Adobe Acrobat Reader or any other "PDF readers".

Contents of 'Document'

| Name | Size | Type |
|---|---|---|
| UM_6_input_description.pdf | 1 875KB | Portable Document Format |

A ***free*** "PDF-reader" is available on www.adobe.com .

## 2.4. Examples

Approx. 40 examples are given under the "Examples" directories. The contents of the UNIX and PC examples are identical, (the only reason for having two folders is due to computer compatibility, UNIX and PC represent the files differently).

The input files are located in separate folders, one example per folder, see Figure 2.4-1. In each folder, following files are found:

Head.fem : USFOS control parameters

Stru.fem : Structure and load description in either SESAM or UFO file format. In some cases *both* SESAM and UFO formats are given for the same example, and then the "stru-file" has a postfix, u for UFO and s for SESAM. Any of the two variants (stru_u.fem or stru_s.fem) should produce the same results. The USFOS control parameters are unaffected by the file format used to describe the structure and loads. (See also Chapter 3).

SINTEF

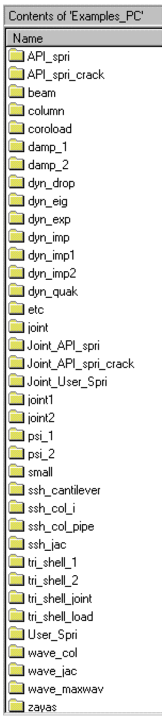



**Figure 2.4-1 Example folders available for UNIX and NT( PC)**

# 3. Input File Formats

In the current version of the User's manual, one chapter describing the UFO file format is added. The UFO file format is used to describe the same type of information, which normally is described in SESAM file format, and has been used since 1994 by non-SESAM users. The type of information is: Nodal ID's, Coordinates and Boundary conditions, Element ID's, connectivity and properties etc. USFOS recognises the file format automatically, and the results are unaffected by the structural/load file format used. **However, *mixing* commands from the two input formats are not possible.**

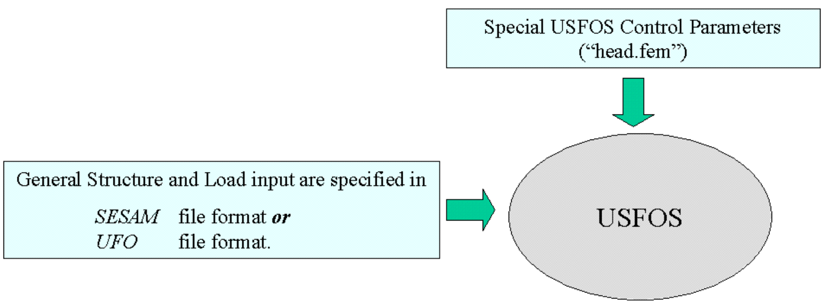



**Figure 2.4-1 Input files to USFOS**

In the USFOS User's manuals, following sections are found:

6.3 USFOS Control Parameters
6.4 SESAM Structure and Load
6.5 UFO Structure and Load

Following "style guide" is recommended see Figure 2.4-1:

- Use the "USFOS control file" for the USFOS control parameters.
- Use the "Structural file" for the structure and load input (described in either SESAM or UFO). Sometimes it's convenient to spread the structure/load input in two files ("Structure file" and "Load file").

# 4. New Features

## 4.1. Shell Element

From version 7-6, a non linear triangular shell element is available. The element is specified through general SESAM input format, element type 25, or using the TRISHELL command (UFO input). The thickness is specified similar to the existing membrane element. The non linear material parameters are given in the "usual" MISOIEP record. Both concentrated load, conservative distributed load and pressure load are available. In Table 4.1-1, the necessary input records are given for both file formats.

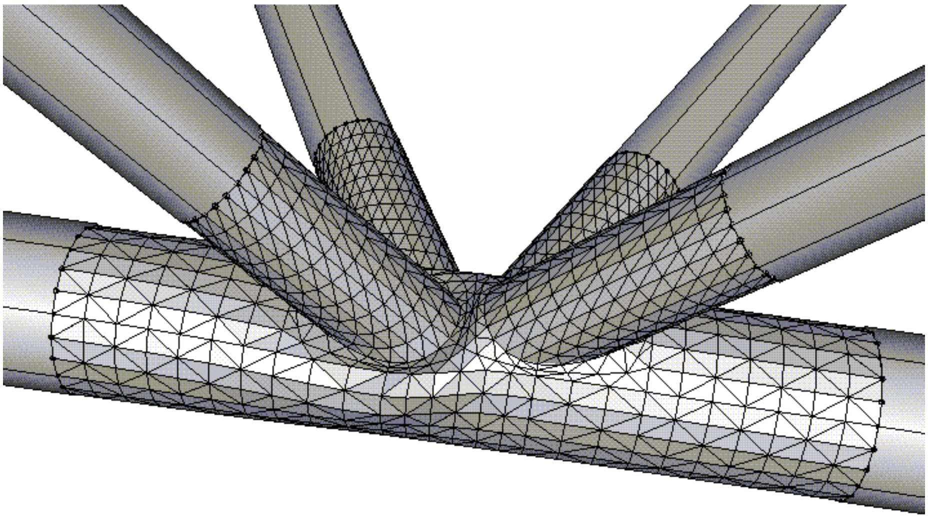


**Figure 4.1-1 Non linear shell element in USFOS.** (Example tri_shell_joint)

| Item | SESAM file format | UFO file format |
|---|---|---|
| Element definition | GELMNT1/GELREF1 | TRISHELL |
| Plate thickness | GELTH | PLTHICK |
| Material properties | MISOIEP | MISOIEP |
| Concentrated (nodal) Load | BNLOAD | NODELOAD |
| Pressure load (non-conservative) | BEUSLO | PRESSURE |
| Distributed (conservative) load | | SHELLOAD |

**Table 4.1-1 Input records for triangular shell element**

For more detailed description, see User's manual Ch. 6. See also following example folders:

- *tri_shell_1*
- *tri_shell_2*
- *tri_shell_joint*
- *tri_shell_load*

***Result presentation:***

The results for the shell element is presented in XFOS and available element results are plastic strain and plastic utilisation. These result types are new, and are accessed through the new "button":

GENERAL

ELEMENT

In Figure 4.1-2, the dialogue box used for shell element selection is shown.

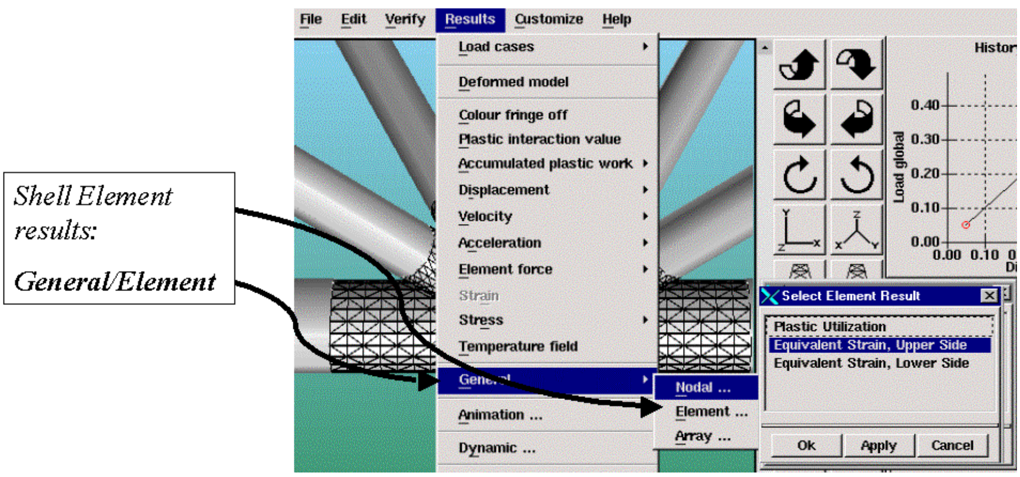



**Figure 4.1-2 Selecting Shell Element Result**

By default, no element mesh is shown on the model image, but using the *Verify/Show Mesh* option as shown in Figure 4.1-3, the user may switch on the mesh. The same button is used to switch off the mesh visualization.

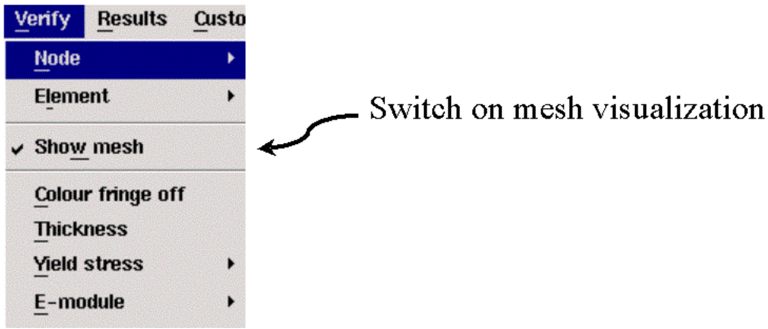



**Figure 4.1-3 Switching ON/OFF mesh visualization**

## 4.2. Shell formulation for beams

In addition to having the triangular, shell element available directly "one by one" as an ordinary element, the shell element is possible to access through the shell sub structure option. An ordinary beam element (pipe, box etc) is then represented by shell elements (in stead of the normal beam formulation, see Figure 4.2-1). As the physical member is represented by shell elements, effects like local buckling, torsion buckling, etc is predicted with high accuracy. The necessary commands (subshell and meshpipe) used to define one "shell-beam" element are described in figure Table 4.2-1overleaf.

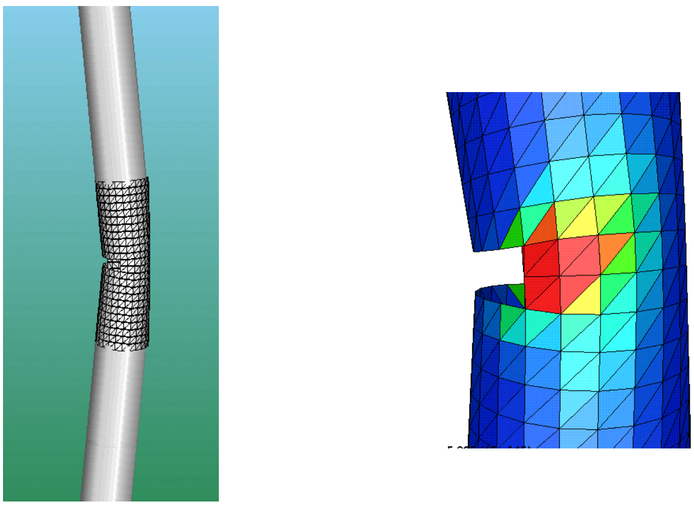


**Figure 4.2-1 Shell formulation on selected beam element**

Several simple examples are given on the CD-ROM:

- ssh_cantilever
- ssh_col_I
- ssh_col_pipe
- ssh_jac

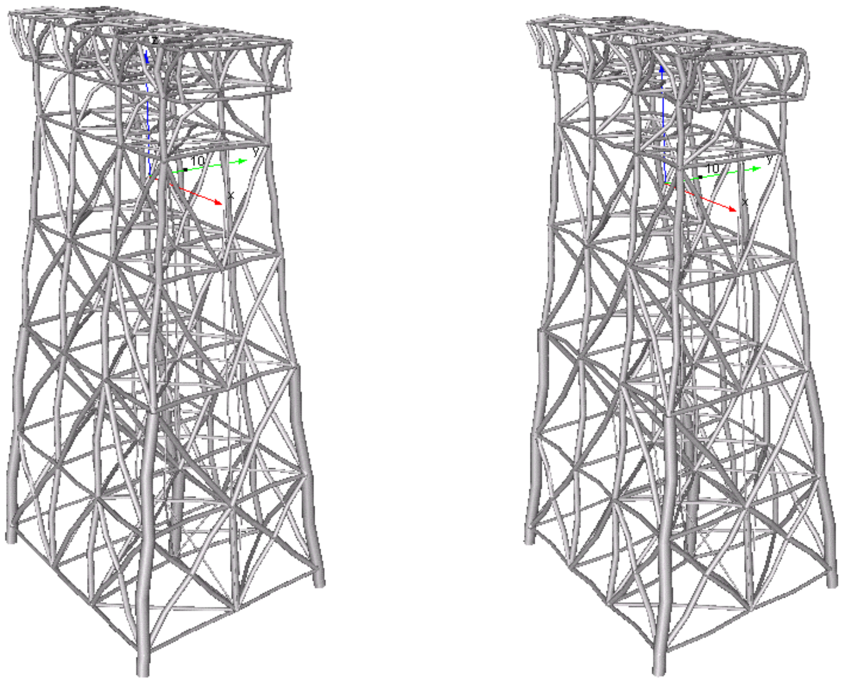
```
'                                                      - Use Shell_Beam for elem 12
'          Elem_Id
SUBSHELL     12
'                                                      - Define mesh density
'         n_Length    n_Circ   Elem_Id
MESHPIPE      6          12        12
```

**Table 4.2-1 Input commands defining shell formulation on beam elements**

## 4.3. Extreme Wave calculation/Automatic member imperfections

Modules for calculation of hydrodynamic forces are included in USFOS. This means that using separate wave load pre processor is not needed. Using the USFOS hydrodynamic in connection with *static* "push over" analysis will typically contain following:

- Specify the actual wave (type, height, period, direction…)
- Specify the corresponding current (if any)
- Switch on buoyancy (optional)
- Specify criterion to be used for selecting worst wave position (max base shear or max overturning moment)



Direction of wave ←

Direction of Wave →

**Figure 4.3-1 Automatic member imperfection according to wave force direction**

USFOS will then step through the actual wave and identify the worst wave position (the position causing the highest base shear or overturning moment). The hydrodynamic forces from this wave phase (position) are saved (in memory) to be used in the pushover analysis. The calculated *buoyancy* forces are possible to separate from the other hydrodynamic forces, and the user may specify how to use the buoyancy forces, (add to an existing deadweight loadcase etc.).

SINTEF

Applying member imperfections, one by one, is a time consuming task, but by using the new option **CINIDEF**, the correct member imperfection is applied automatically for all beam elements. The most common buckling curves are available defining the size of the imperfection, (see User's manual Ch. 6). The *direction* of the imperfections follow the direction of the loads for a specified load case.

In Figure 4.3-1, the jacket to the right is exposed to waves with direction 45°, while the jacket to the left is exposed to a wave with opposite direction (225°). It is seen that the direction of the imperfections are opposite in the two cases (size is scaled).

All necessary input is shown in Table 4.3-1, and it should be noted that these few commands replace 1000's of input lines and use of separate wave load pre-processor / load files.

***Comments to the input given in*** Table 4.1-1 *(see also example folder wave_maxwav):*

- Load case 1 is used for "dead weight" and calculated buoyancy
- Load case 2 is used for the extreme wave

Load case 1 is not scaled beyond factor 1.0 (that’s why the calculated buoyancy forces is separated from the other hydro. forces and added to this load case). Load case 2 forces are scaled to platform collapse.

- The direction of the member imperfections (CINIDEF par. no 2 and 3) follows the direction of the member forces defined by load case 2 (which is the calculated wave forces).
- The size of the imperfection (CINIDEF par. no 1) is calculated according to "Chen column curve".

- A Stoke 5'th wave with height 25m, period 16s, 45° direction is applied. The sea surface is located for global Z-coordinate=0.0. Water depth is 100m.
- A current profile with peak value 2 m/s is defined with same direction as the wave. From depth 20m (Z=-20m relative to the sea surface), the current is reduces linearly.

- The actual wave is 'stepped through' the structure with time increment 1 s. The wave position giving the highest *base shear* in the interval Time = 0 -20s is used in the "push over" analysis.

NOTE As all hydrodynamic calculations are using SI units, the forces are calculated in N (Newton). If f ex. MN is used as force unit, the wave forces must be scaled before they are used in the "pushover" analysis. The command **WAVMXSCL** <factor> is used, (see also User's manual, Ch 6). In the current example, the wave forces are scaled with a factor 1.3 (just for demo purpose).

- For both the buoyancy forces and the wave forces, it is possible to print the calculated forces to separate files, but in the example, printing is switched off (nowrite).

```
' ----------------------------------------------------------------------
'    Lcomb 1 is gravity loads and static deck loads+calculated buoyancy,
'    Lcomb 2 is Stoke Wave 45 deg diretion
' ----------------------------------------------------------------------
'         nloads   npostp   mxpstp   mxpdis
CUSFOS    10        15       1.00     0.05
'         lcomb    lfact    mxld     nstep     minstp
           1       1.0      1.0      10        0.05    ! Dead + Buoyancy
           2       0.5      3.0      50        0.001   ! Wave
           2       0.1      6.0     100        0.001   ! Wave
'
' ----------------------------------------------------------------------
'         Apply automatic out of straightness.
'         Use loads from Waves (lcase 2)
' ----------------------------------------------------------------------

'          Size    Pat    LoadCase
cInidef    70      1        2
'
' ----------------------------------------------------------------------
'         Separate Bouyancy from wave forces.
'         Add Buoyancy to load case 1
' ----------------------------------------------------------------------
'
'            lCase      Option
BUOYANCY      1       noWrite
'
'                                              - Define Wave:
'
'           Ildcs  <type>   H    Period   Direction Phase  Surf_Lev Depth
WAVEDATA      2   Stoke  25.0    16.0       45      0.0     0.0     100
'
'           Ildcs  Speed Direction Surf_Lev Depth    [Profile]
CURRENT       2     2      45        0.0    100      0.0  1.0
                                                   -20.0  1.0
                                                  -100.0  0.0
                                                  -110.0  0.0
'
' ----------------------------------------------------------------------
'     Identify Worst Phase (Max Base Shear) and do not create a loadfile
' ----------------------------------------------------------------------
'           Criterion   dT     EndT   Write
MaxWave      Baseshear  1.0    20.0  noWrite
'
' ----------------------------------------------------------------------
'    Scale the Wave load. This option is required when Force Unit is not N.
'   (generated wave loads are always using Newton).
'    In this demo case, scale by 1.3 :
' ----------------------------------------------------------------------
WavMxScl    1.3
```

**Table 4.3-1 Input for automatic wave calculations and automatic member imperfections**

SINTEF

## 4.4. Pile – Soil / Automatic generation of piles and soil capacity

The automatic generation of piles and corresponding soil capacities is a powerful option, which requires a few input lines only. The user's structure ends at "mud line", and all elements below mud line are generated automatically by USFOS, see Figure 4.4-1. In Table 4.4-1 overleaf, the necessary commands used to produce the foundation model shown in the figure are given.

See also the example in folder PSI_2.

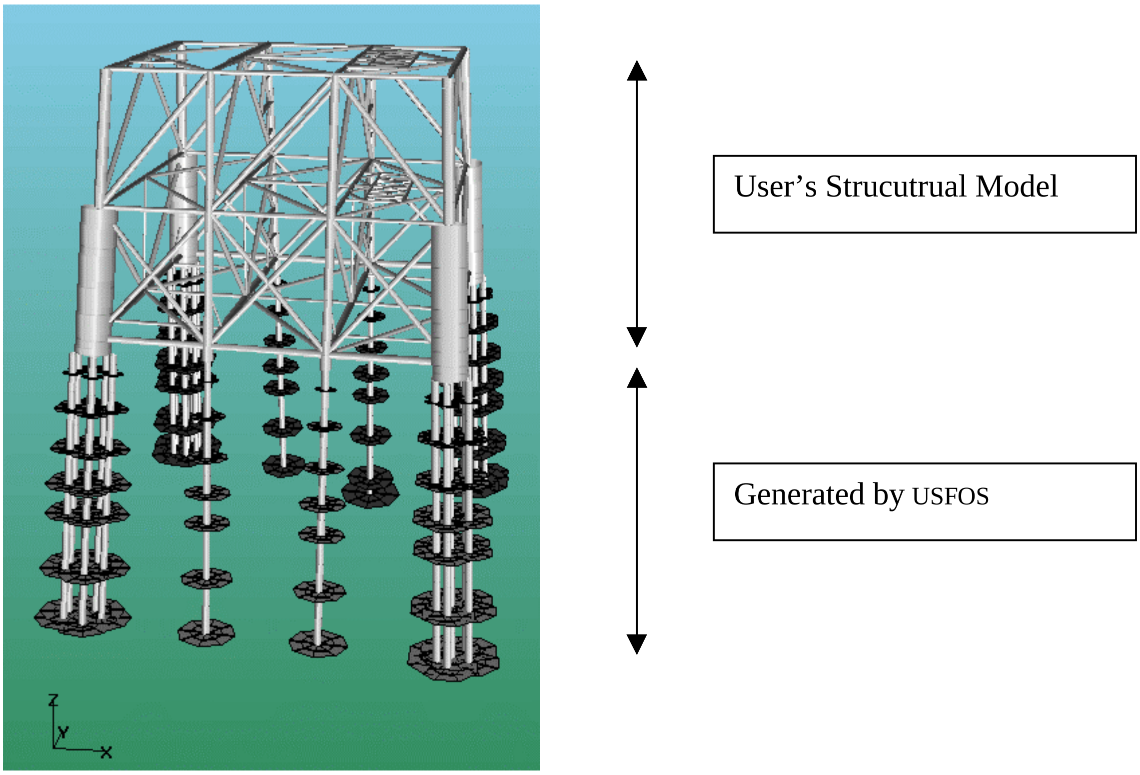



**Figure 4.4-1 Automatic generation of piles and soil capacity**

***Comments to the input in Table 4.4-1:***

- The foundation consists of 4 pile clusters, each with 7 piles, and 4 single piles.
- This foundation is defined as 8 **PILE** elements, which refer to one of the two **PILEGEO** records.
- PILEGEO number 1 consists of 7 pipes with diameter 1.22m. The individual positions are specified through local Y- and Z-co ordinates referring to the PILE local axis.
- The PILE local x-axis goes (downwards) from the pile head towards the pile tip.
- PILEGEO number 2 is a single pile, here defined as a group with only one pipe in the centre of the pile element axis. (The single pile option could also been used, see UM Ch 6).

- For all the 8 piles, the same soil exists (refer all to the same **SOILCHAR** record)
- The SOILCHAR is specified with 3 clay layers and 3 sand layers. However, in order to obtain a reasonable element density in the rather thick sand layer no. 2 (-24.1 to -48.8m), the same soil property (no. 501) is referred to three times. (The soil spring is inserted in the middle of the layers defined under SOILCHAR.
- The soil strength is calculated according to API 1993 by specifying the geotechnical data in the command **API_SOIL.**

```
'            Elem ID   np1    np2  Soil ID   Pile_mat  Pile_geo lcoor  Imper
 PILE            1     1    1001    10         99        1    0
 PILE            2     2    1002    10         99        2    0
 PILE            3     3    1003    10         99        2    0
 PILE            4     4    1004    10         99        1    0
 PILE            5     5    1005    10         99        1    0
 PILE            6     6    1006    10         99        2    0
 PILE            7     7    1007    10         99        2    0
 PILE            8     8    1008    10         99        1    0
'
''           ID      Type    Do     T     Npile      Y_loc  Z_loc
 PILEGEO      1        2    1.22  0.05      7          0.0   0.0
                                                       2.1   2.1
                                                       3.0  -1.4
                                                       0.6  -2.5
                                                     -1.65 -1.65
                                                      -2.5   0.6
                                                      -1.5   2.7
'            ID      Type    Do     T     Npile      Y_loc  Z_loc
 PILEGEO      2        2    1.22  0.05      1          0.0   0.0

'           ID  Type  Z_Mud D_ref Ffac  Lfac   Z1     Z2    API_Soil ID
 SOILCHAR   10  API -93.725  1.0  1.0    1.0   -1.0  -5.2     101    ! Clay
                                                -5.2 -12.5     201    ! Clay
                                               -12.5 -18.3     301    ! Clay
                                               -18.3 -24.1     401    ! Sand
                                               -24.1 -28.3     501    ! Sand
                                               -28.3 -42.7     501    ! Sand
                                               -42.7 -48.8     501    ! Sand
                                               -48.8 -67.0     601    ! Sand
'
'         ID   Type    load    Gam   Plug  Su    eps50  APIJ   Tresf QPLim  iDyn
 API_SOIL 101 SoftClay Static  9500   1   50E3  0.013  0.25   0.74  0.2E6   0
 API_SOIL 201 StifClay Static  9500   1  120E3  0.012  0.25   0.72  1.2E6   0
 API_SOIL 301 StifClay Static  9500   1  150E3  0.010  0.25   0.73  1.0E6   0
 API_SOIL 401 StifClay Static  9500   1  190E3  0.019  0.25   0.75  1.9E6   0

'         ID  typ      load    Gam  Plug Phi   Delta  rNq           QPlim  iDyn
 API_SOIL 501 Sand    Static  8000   0   33      22    22           1.4E7   0
 API_SOIL 601 Sand    Static  8000   0   37      26    23           1.1E7   0
```

**Table 4.4-1  Input for automatic calculation of piles and soil capacities**

## 4.5. Dynamic Analysis results. Time Series

A dynamic analysis may involve a large number of analysis steps (1000 - 100 000), and saving of analysis results is then a challenge. It is then necessary to select a few results, which could be saved every analysis step, while the rest of the results could be saved more seldom. In this way, the user obtain following:

- High density on the time series of the selected (most important) results
- Acceptable density on the results presented in XFOS for inspection of the global behaviour of the structure (f ex generation of animation etc).

The few, selected result quantities are stored in a separate file with extension *.dyn* in addition to the usual *.raf* file. The dynamic results are accessed from XFOS through the result/dynamic_result dialogue box, see Figure 4.5-1.

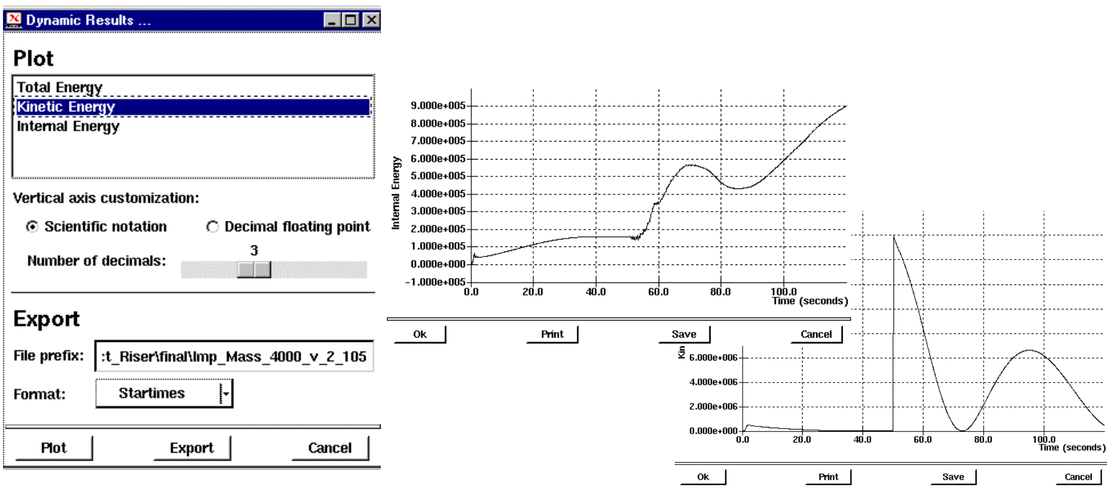



**Figure 4.5-1 Selecting Dynamic Results from XFOS**

Following results are

- NODAL
  - Displacement
  - Velocity
  - Acceleration
  - Relative displacement (between two nodes)

- ELEMENT
  - Displacement
  - Force

- GENERAL
  - Internal Energy
  - Plastic Energy
  - Kinetic Energy
  - Total Energy

See Table 4.5-1 for example of use:

```
'              Type   Node_ID   Dof
 DYNRES_Node   Dis      10       1
 DYNRES_Node   Dis     130       1
 DYNRES_Node   Vel     130       1
 DYNRES_Node   Acc     130       1
'
'              Type   Node_ID   Dof  Node_ID Dof
 DYNRES_Node  RelDis    10       1    130     1
'
'
'              Type   Elem_ID   End   Dof
 DYNRES_Elem   Disp     20       2     1
 DYNRES_Elem   Force    20       1     1

                  Type
 DYNRES_General  Wint
 DYNRES_General  Wplast
 DYNRES_General  Wkin
 DYNRES_General  Wtot
```

**Table 4.5-1 Input for "Dynamic result" saving**

See also in the example folders:

- dyn_drop
- dyn_imp
- dyn_imp2
- dyn_quak

## 4.6. Impact Analysis including "dash-pot" dampers

As an alternative to the standard impact options (BIMPACT, DYNIMPCT), it is sometimes necessary to model *both* the structure *and* the impacting object. The impacting object is defined as a separate structure and is assigned the appropriate properties (mass, initial velocity etc). In order to determine the contact between the two structures, a non linear spring is used. In Figure 4.6-1 this spring is seen between the impacting structure (the pipe) and the slender frame. The spring properties (P_d curve) is shown in the figure, and the curve is specified in the example file described in Table 4.6-1. See also example folder ***damp_2***.

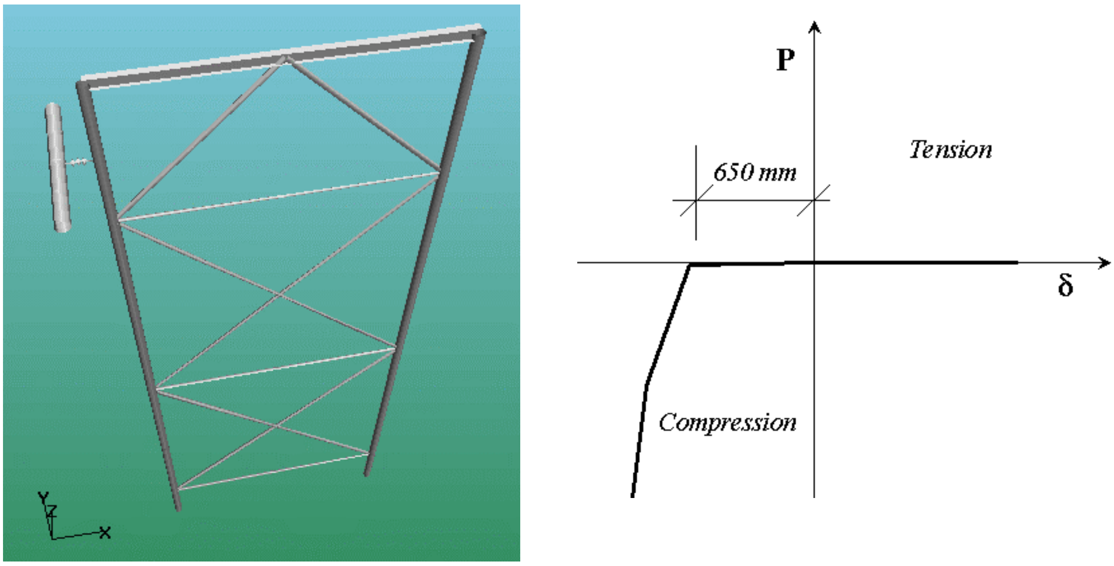



**Figure 4.6-1 Slender frame impacted by a separate structure with contact spring**

The presence of physical dampers (like the ones in your car) will reduce the damage on the structure, and boat fenders is often equipped with dampers mounted in parallel with springs. USFOS 7-6 is extended to cover this type of 'suspension details', and the 'dash pot' damper characteristics (*C*) for a given non linear spring is specified in the input file.

***Comments to the input in*** Table 4.6-1***:***

- The impacting object (just a pipe) consists of 3 nodes and 3 beam elements.
- One beam element (the contact spring) refers to **MREF** material, and is then automatically transferred to a 2 node non linear spring
- The MREF material refers further to P_d curves, one per degree of freedom. In this example, only axial stiffness is included, and the other references are set equal to zero (means no stiffness in theses degree of freedoms)

SINTEF

- The Axial stiffness is defined as a hyper elastic material with a curve as shown in the figure above. The spring bust be compressed 0.650 m before any force is activated, and the stiffness increases after a deformation of 0.100 m. (The hyper elastic material has no elastic unloading: the forces follow the input curve during loading, as well as unloading).

- The non linear spring (with element ID = 1000) is given an Axial damping characteristics of 20 000 N/(m/s) using the **SPRIDAMP** command. The damper forces will be activated once the relative speed between the two element ends are different from Zero, and the direction of the force is always opposite to the velocity (like hydrodynamic drag damping).

- The impacting body is given an initial velocity of 2 m/s in positive X-direction using the **INI_VELO** command and *material* specification. All elements with the specified material ID (here no. 10) will be given the specified initial velocity.

```
' ======================================================================
'     Impacting Object with mass : 10000 kg
' ======================================================================
'
 NODE     1001        -1.000     -2.000     22.860
 NODE     1002        -1.000       .000     22.860   0 1 1 1 1 1
 NODE     1003        -1.000      2.000     22.860
'
'        Elem ID     np1      np2   material geom  lcoor   ecc1   ecc2
 BEAM     1001      1001      1002       10    10
 BEAM     1002      1002      1003       10    10
 BEAM     1000      1002        45     1000     0    ! Spring with damper
'
 PIPE       10       0.4         0.020
'
 MREF   1000    1001  0  0  0  0  0
'
'           ID        P       d
 HypElast  1001    -10.0E6  -1.000
                   -10.0E5  -0.750
                     0.0    -0.650
                     0.0    -0.100
                     0.0     0.100
                     0.0     1.000
'           Dof     C [ N/(m/s) ]    Elem_1   Elem_2 .....
 SpriDamp  Axial       2.0E4          1000
'                                            - Initial Velocity applied

'                                              to material 10

'

'           Type  Time    Vx    Vy  Vz  Vrx Vry Vrz    Mat_ID
 INI_VELO    Mat   0.0    2.0    0  0    0   0   0       10
```

**Table 4.6-1 Input for defining an Impact Object with nonlinear damper/spring**

See User's Manual, Ch 6 for further details. See also in the example folders:

- Damp_1
- Damp_2

# 5. New/modified input identifiers

Since last main release (7.4), following input identifiers are added/extended:

| | | |
|---|---|---|
| ***TRISHELL*** | : | Specification of triangular non linear shell element. |
| ***SHELLOAD*** | : | Specification of distributed (conservative) load for shell element. |
| ***SUBSHELL*** | : | Switch ON shell formulation for specified beam element. |
| ***INI_VELO*** | : | Initial velocity of specified node(s) or bodies (materials). |
| ***DampRatio*** | : | Structural damping given in terms of damping ratios (and associated frequencies). Time dependent (optional). |
| ***DynRes_N*** | : | Dynamic Result, Nodal data |
| ***DynRes_E*** | : | Dynamic Result, Element data |
| ***DynRes_G*** | : | Dynamic Result, Global data |
| ***CINIDEF*** | : | Analysis Calibration to column buckling curves |
| ***API_SOIL*** | : | Automatic calculation of P-Y, T-Z and Q-Z according to API 1993 |
| ***MAXWAVE*** | : | Automatic selection of the "worst" wave load phase to be applied in a 'pushover' analysis (used together with *Wavadata/Current*). |
| ***WAVMXSCL*** | : | Scaling (du to units) the wave forces found using the MaxWave option. |
| ***WAVE_INT*** | : | User control of the number of integration points to be used along the different beam elements when calculating wave loads. |
| ***BUOYANCY*** | | Calculate and add buoyancy forces to specified loadcomb. |
| ***COROLOAD*** | : | Specification of distributed element loads in local coordinate system. |
| ***INVISIBLE*** | : | Making non-linear springs invisible in xfos, (f ex contact springs). |
| ***WET_ELEM*** | : | Check all elements for hydrodynamic forces |
| ***USERFRAC*** | : | User defined fracture. "Old" identifier, but extended options (Loadcomb/Loadlevel, Time, Utilization, Strain) |
| ***ACTIVELM*** | : | Specification of elements to be "waked up" at a given loadcomb. "Old" identifier, but extended to Dynamic Analysis. |