USFOS

Reality Engineering

USFOS AS

Phone: +47 905 05 717

www.USFOS.com

Enterprise No.: NO 986 827 374 MVA

MEMO

MEMO CONCERNS	FOR YOUR ATTENTION	COMMENTS ARE INVITED	FOR YOUR INFORMATION	AS AGREED
Release Notes **USFOS Version 8-9**				
DISTRIBUTION Members of USFOS user group				X

FILE CODE	CLASSIFICATION
	Confidential
REFERENCE NO.	

PROJECT NO.	DATE	PERSON RESPONSIBLE / AUTHOR	NUMBER OF PAGES
	2019-11-01	Tore Holmas	42

Release Notes

USFOS 8-9, Nov 2019

This memo contains project information and preliminary results as a basis for final report(s).
USFOS AS accepts no responsibility of this memo and no part of it may be copied.

1 Introduction

The current official version of USFOS is version 8-9 with release date 2019-11-01. The release contains the following:

- Release Notes (this MEMO)
- Updated software www.usfos.com
- Extended examples library www.usfos.com
- Updated manuals www.usfos.com

Except for this MEMO, no written information will be distributed in connection with this release. All information is stored on the WEB.

2 Changes in version 8-9

Comparison of 8-9 vs. older USFOS versions could give somewhat different results due to modifications of the algorithms and default parameters.

3 News in USFOS version 8-9 - 2019.

3.1 Introduction

Some of the new features are described by examples in this memo, in the examples collection on the web and in the updated manuals.

USFOS 8-9 is built on the usual platforms: Win32, Win64, LINUX-and MacOSX. The utility software is available on all platforms.

3.2 How to install/ upgrade your USFOS version

3.2.1 Windows (64bit)

USFOS could be upgraded in different ways (as usual):

- Alt 1: Download the new "*setup.exe*" and u-install/install USFOS, (same as for release 8-7). This operation requires administrator rights on the PC.

- Alt 2: Download module by module and copy into the application folder, (typical "*C:\Program Files\USFOS\bin*". This operation requires write access on **C:**, but no administrator rights are required since this is just file copy).

Alternative 1 updates all modules and the on-line manuals.

Alternative 2 requires following download and operations:

❑ USFOS 64bit module, unzip and copy into	C:\Program Files\USFOS\bin
❑ xact (complete 64bit package), unzip and copy into	C:\Program Files\USFOS\bin
❑ USFOS manual. Copy into	C:\Program Files\USFOS\bin

Similar procedure is used for other modules, (for example STRUMAN, FAHTS).

3.2.2 Windows (32bit)

No set-up script is made for USFOS 8-9 32bit windows. However, version 8-9 becomes available by downloading the central modules (similar to Alternative-2 above):

o USFOS 32bit module, unzip and copy into	C:\Program Files\USFOS\bin
o xact 32bit, (complete package), unzip and copy into	C:\Program Files\USFOS\bin
o USFOS manual. Copy into	C:\Program Files\USFOS\bin

3.2.3 LINUX

Updated versions of USFOS, xact and utility tools are downloaded module-by-module as usual.

The simulation engine is available in two versions, USFOS and USFOS_gF. The last version is compiled using gFortran and requires that the library "libgfortran.so.3" exists on the computer.

On Ubuntu, the library is installed using the command:

```
sudo apt-get install libgfortran3
```

3.2.4 MACOSX

Updated versions of USFOS, and utility tools are downloaded module-by-module as usual.

The simulation engine is available in two versions, USFOS and USFOS_gF. The last version is compiled using gFortran and requires that the library "libgfortran.so.3" exists on the computer.

3.3 Enhanced Graphical User Interface

The graphical user interface (*xact*) has been enhanced since last year's release. The GUI version released together with USFOS 8-9 is "3.1.8" for the Win-64bit and LINUX versions. The 32bit version of XACT.exe is not changed since v8-7, but the updated 32bit "dll" gives extended functionality.

3.3.1 Export to CeetronCloud.

The manual found on our web under USFOS user's gives a complete description:
https://www.usfos.com/manuals/usfos/users/documents/Xact_Cloud_Sharing.pdf

Brief summary:

Ceetron's "*one click cloud sharing*" makes it possible to share USFOS analysis results (animations) with "one click". The current result and settings in XACT (such as displacement scaling, fringe range etc.) are sent to the Cloud. The "Send to CeetronCloud" button is found under "File".

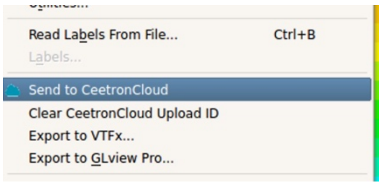


Figure 3-1 – The "Send to CeetronCloud" button in XACT is located under File

When "things" are established, *this is the only click*, and the link to the Cloud could be sent to "anybody" for visualization in a web browser, (PC/iPad/iPhone). Remember to set sharing level.

Just press the link to see the example described on the next pages:

`https://cloud.ceetron.com/v/WIvz-H1DTmWqwrvmf-7x3w`

Two things to establish once-for-all:

1. Create an account: Go to *cloud.ceetron.com* and press *Signup,* (a limited space is free)
2. Fetch your "Upload ID" to specify in the "Send to CeetronCloud" dialogue first time.

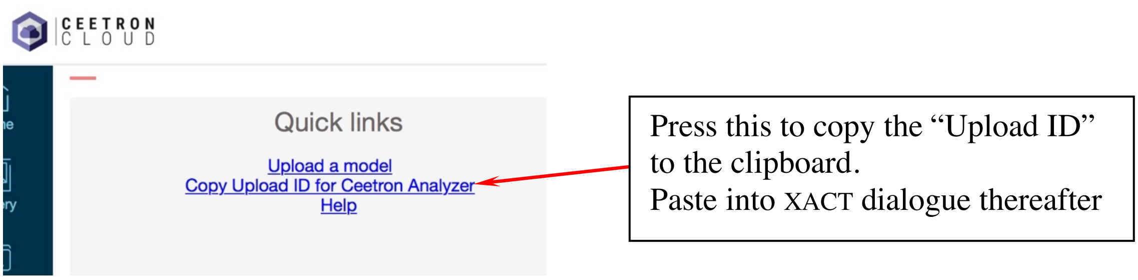


Figure 3-2 - Fetching the "Upload ID" to be given to XACT the first time.

Example: The "north sea jacket"

Figure 3-3 shows the usual XACT display after the RAF file is opened. The model is set to the wanted view, and the displacements are scaled with a factor 5 to easier see the response. The fringes are updated min/max –0.2 to 1 (to avoid the dark blue colour, just "Hollywood").

These settings will be kept after upload to CeetronCloud

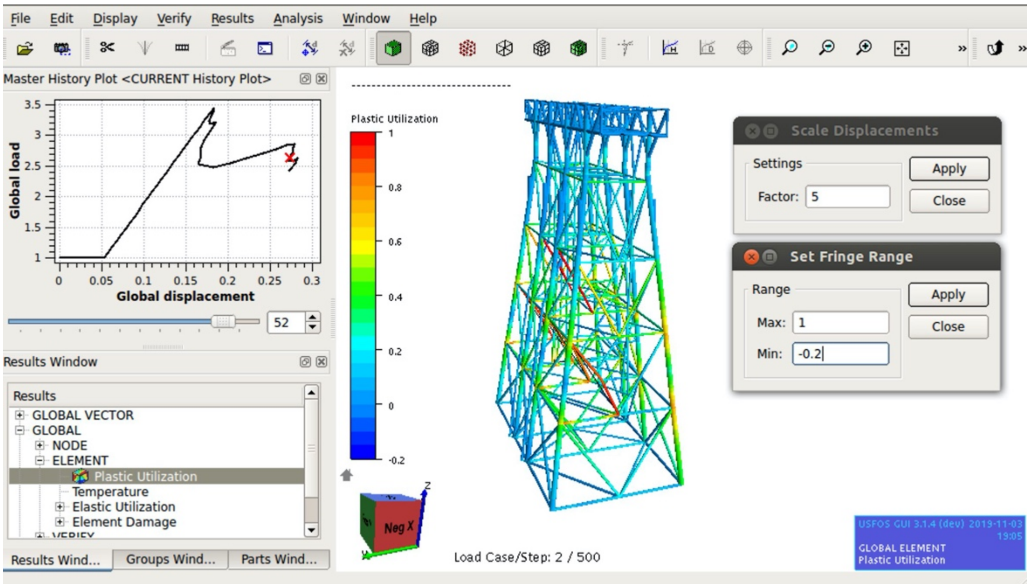


Figure 3-3 - xact showing plastic utilization. Scaled displacements and fringe-range is set.

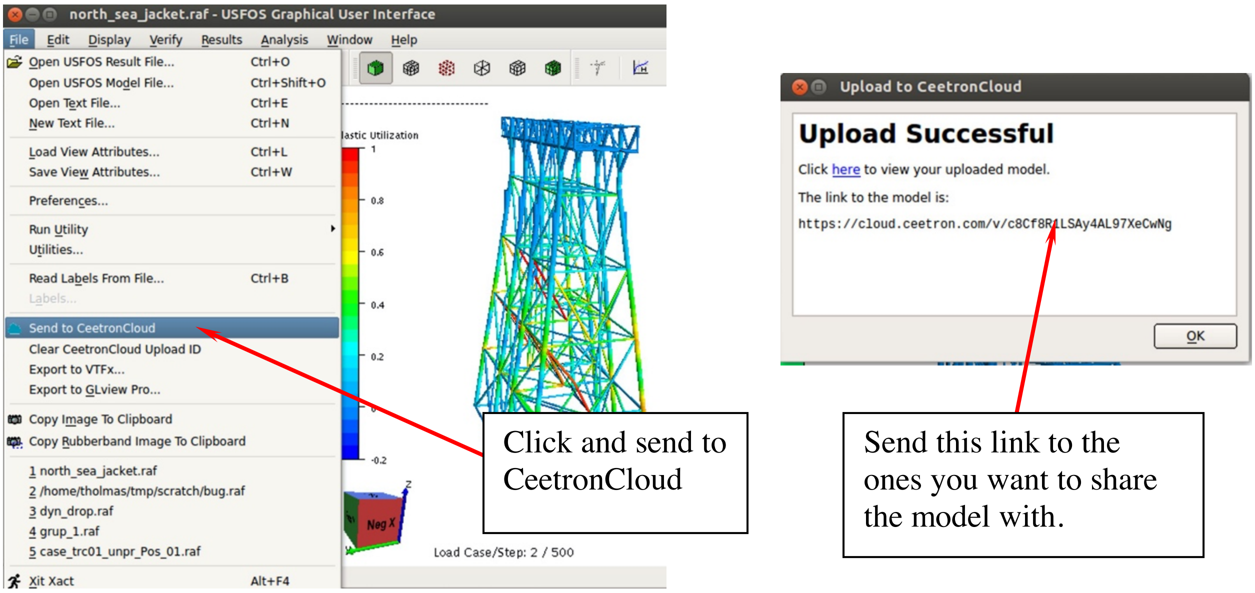


Figure 3-4 – Press the "Send to CeetronCloud" button.

By clicking on the link the model stored on Ceetron Cloud will appear:

https://cloud.ceetron.com/v/WIvz-H1DTmWqwrvmf-7x3w

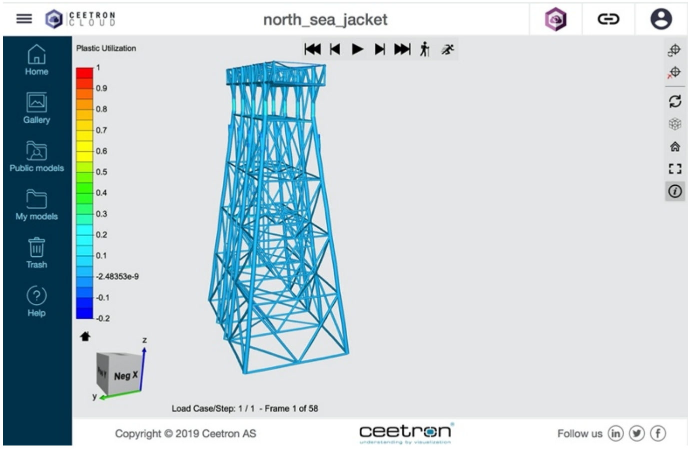


Figure 3-5 - "Opening Scene". View as defined in XACT

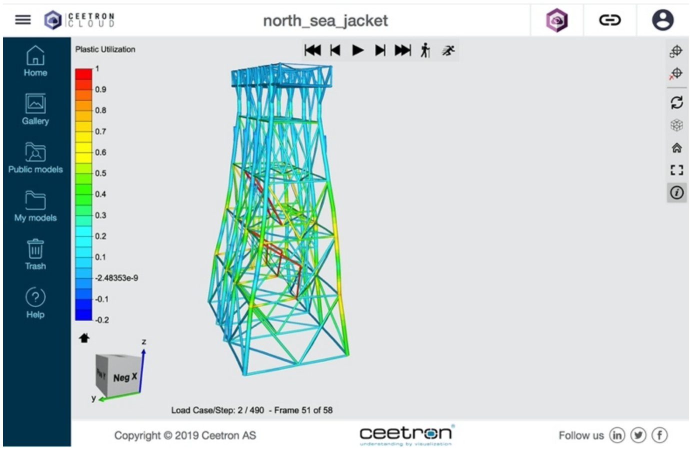


Figure 3-6 – Playing animation or stepping through the analysis

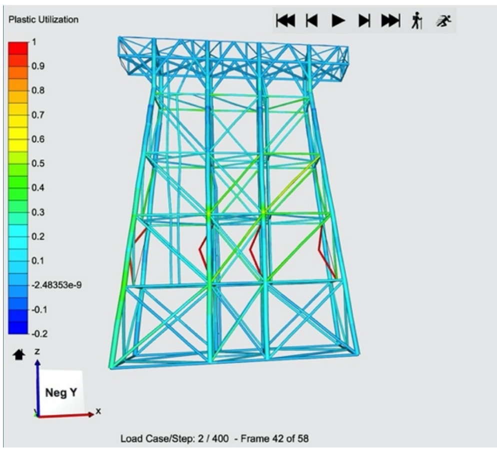


Figure 3-7 – Rotate, zoom pan (similar to XACT).

3.3.2 Visualization of Node Set

Sets of nodes could be visualized in XACT similar to "element groups". The menu is found in the window in the left lower corner.

The marked set is visualized similar to Element Groups. Clear selection removes all node set labels.

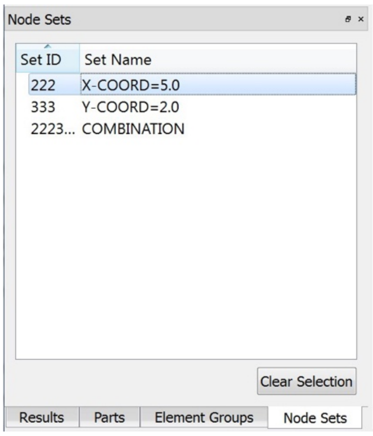


Figure 3-8 – Menu for Visualization of Node Set.

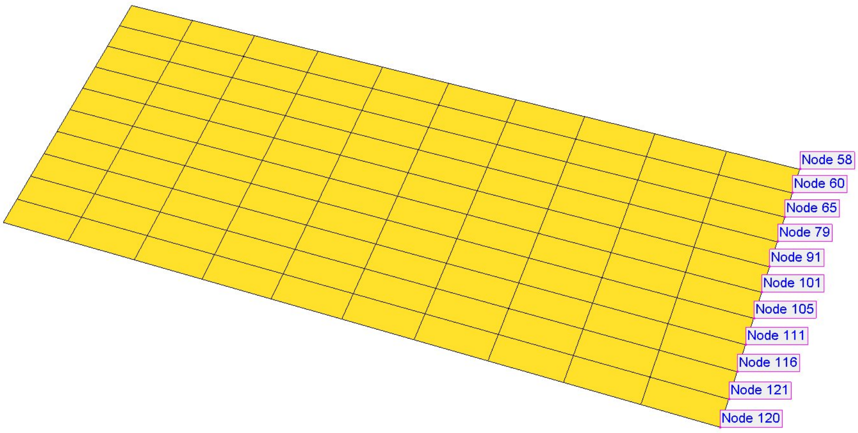


Figure 3-9 - Visualization of Node Set "222, X-Coord=5.0"

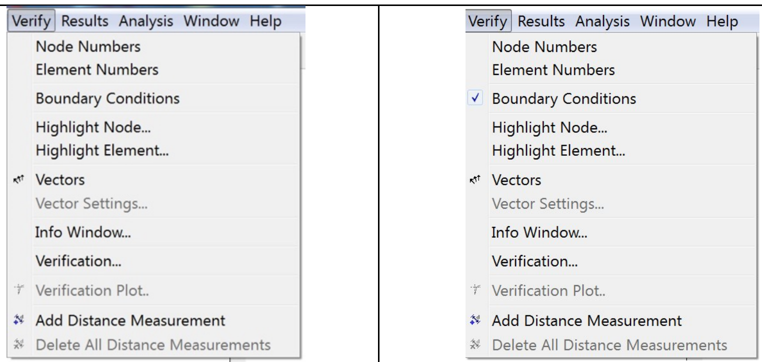
3.3.3 Visualization of Boundary conditions

The boundary conditions used in the actual simulated could be visualized. Changes made using "Chg_Boun" are included. Nodes with some boundary conditions (i.e. not 100% free) get a label with the node ID and 6 boundary codes. The boundary conditions refer to the Global System.

NOTE-1:
Nodes with no elements attached are automatically set fully fixed by USFOS, (see node 999 in Figure 3-10, image to the left). These nodes are also visualized.

NOTE-2:
The clip function removes visualization of nodes not attached to the visible elements. After reset, the nodes not attached to any element remains invisible.

Switching on/off visualization of boundary conditions is done under Verify / Boundary Conditions.



Menu for Verify Boundary Conditions	Boundary Conditions visualization is ON

Figure 3-10 - Menu for switch on/off visualization of boundary conditions

The boundary codes shown in the labels are:

- 0: DOF is free
- 1: DOF is fully fixed
- 2: DOF is prescribed (see Figure 3-12)
- 5: DOF has a spring to ground.

The “surplus” node no 999 is shown in the image to the left. It is recommended to remove surplus nodes (this could be done using STRUMAN).

After the clipping operation, the surplus nodes are not visualized.

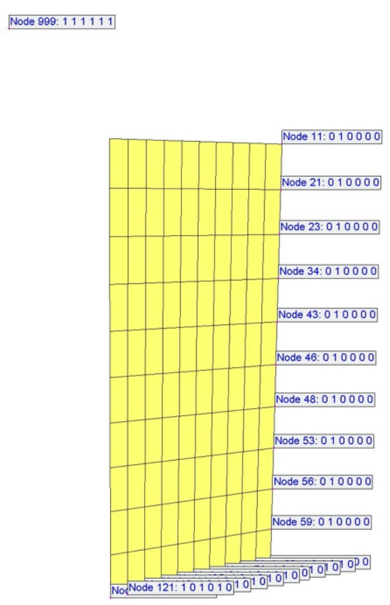
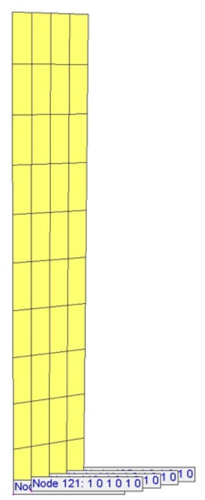
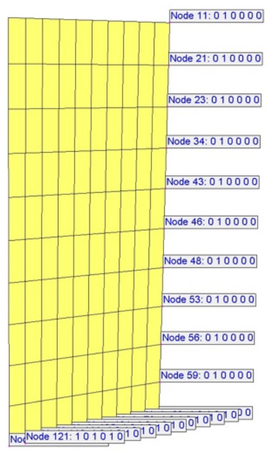
		
Initial visualization. Nodes with no elements attached are visualized	Clipping model shows boundary on visible nodes	Resetting after clipping removes all nodes with no elements attached (node 999 is removed)

Figure 3-11 - Visualization of boundary conditions.

Figure 3-12 show a case with DOF-X prescribed, rotation about Y-axis free, and the other fixed.

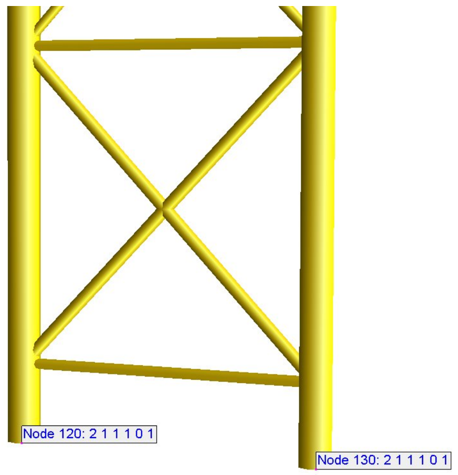


Figure 3-12 - Visualization of Prescribed displacement in X-direction.

3.4 Foundation

3.4.1 Mono Pile

If e.g. a wind tower foundation is modelled, the “monopile” is recommended. (One single “jacket pile under-estimates the resistance for short “stocky” piles).

The “monopile” is using a series of standard piles along the circumferential of the monopile and will therefore describe the bending resistance more accurately.

See more detailed description in the theory manual found under:
www.USFOS.com manual / USFOS / theory.

The monopile is defined as usual with one PILE element. The PILEGEO defines the monopile diameter and thickness. By default, 8 integration points along the circumferential is used, but the user may increase this to, say 16 as shown in the example.

```
'
''
'        Pile_id    Nodex1    Nodex2   SoilId  PileMat  PileGeo    Lcoor
 PILE      1001        2       2000     1000      2      1000       0
'
`
'          ID      Type       Do     T      nPoints
 PILEGEO   1000   MonoPile    10   0.050     16   ! Define monopile with 16 points
'
```

Figure 3-13 - Definition of Monopile

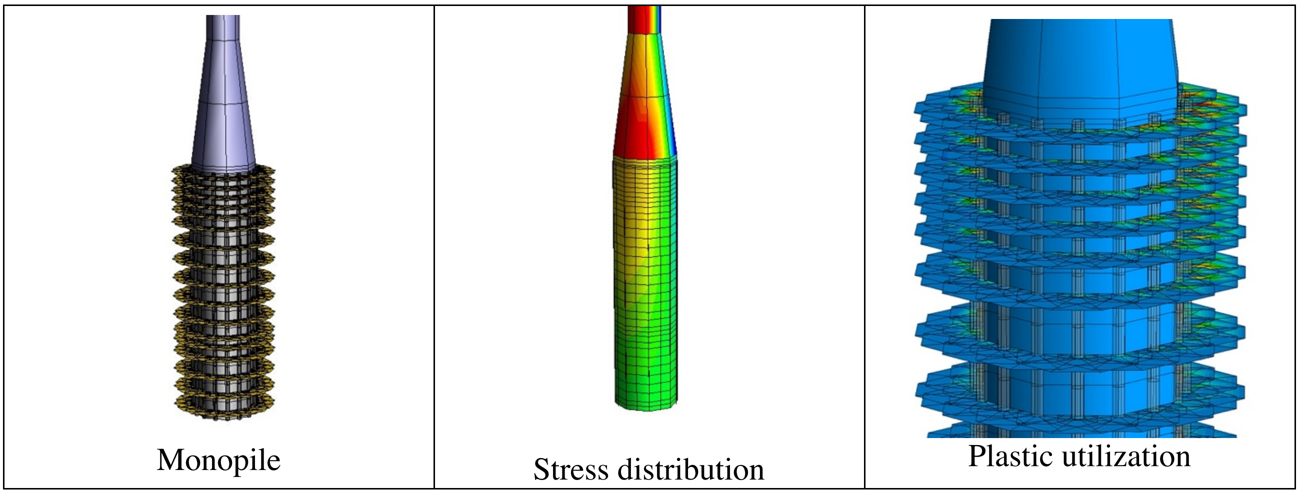


Figure 3-14 – Monopile visualization. Stresses in the pile and soil utilization.

3.4.2 Displacement Rate Dependency

A soil may behave differently depending on the loading rate (how rapid the displacements change). The soil will typically have higher resistance for higher displacement rates ("strain rate effect").

The "enhancement factor" is given as function of displacement speed.

This option could be useful in connection with earthquake analysis.

```
'
'        Pile_id    Nodex1   Nodex2  Soil_id Pile_mat  Pile_geo Lcoor
 PILE       9100       2        3      762       248   762032      0
'
'
'
'           KeyWord    ID   Key       Speed   Fac
 PileOpt    DispRate  200   T-Z        0       0
                                      0.1     0.2
                                      1.0     2.0
                                      2.0    10.0 ! "Stronger" at high speed
'
 PileOpt    DispRate  200   P-Y        0       0
                                      0.1     0.2
                                      1.0     2.0
                                      2.0    10.0
'
'           KeyWord    ID   Key       PileID
 PileOpt    DispRate  200   Assign     9100
'

'           KeyWord    ID   Key       nCyc    Fac
 PileOpt    CyclDegr  100   T-Z        0       1
                                       1      0.9
                                       5      0.5
                                      10      0.5
'
'           KeyWord    ID   Key       nCyc    Fac
 PileOpt    CyclDegr  100   P-Y        0       1
                                       1      0.8
                                       5      0.4
                                      10      0.4
''
'           KeyWord    ID   Key       PileID
 PileOpt    CyclDegr  100   Assign     9100
```

Figure 3-15 - Definition of Displacement Rate Dependency

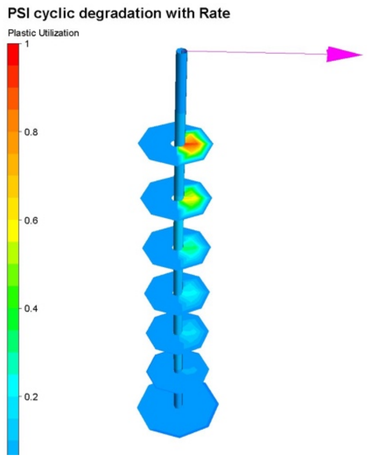
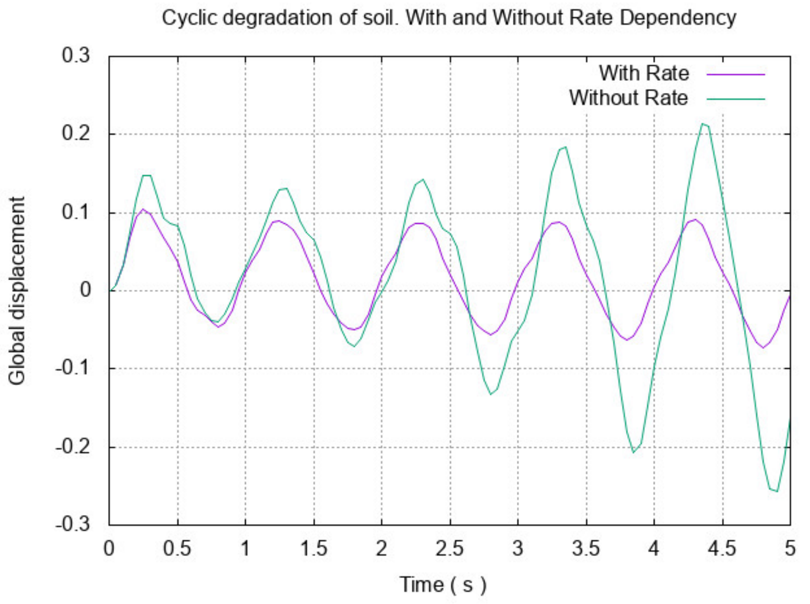
 Pile with horizontal load, P-Y	 Comparison with/without displacement rate dependency

Figure 3-16 – Sideways loaded pile with displacement rate dependency. .

3.4.3 SOILDISP. Assigned on individual Piles (type 3)

In cases were the “SOILDISP” is not the same for all piles, the “type-3” could be used. Here, the SOILDISP is assigned to piles, one by one.

The example shows a case where two piles are pushed together.

```
' --------------------------------------------------
'     Push the soil  Between  -19 and -40m
'     Different at diff levels. Same Load Case
'
'     Positive disp for Pile 1000
'     Negative disp for Pile 2000
'
' --------------------------------------------------
'
'          lCase  Type PileID   z_Top   z_Bot  DofCode  Xdisp
 SoilDisp    2      3   1000    -19     -21      1       0.2
 SoilDisp    2      3   1000    -21     -30      1       0.4
 SoilDisp    2      3   1000    -30     -40      1       0.1

 SoilDisp    2      3   2000    -19     -21      1      -0.2
 SoilDisp    2      3   2000    -21     -30      1      -0.4
 SoilDisp    2      3   2000    -30     -40      1      -0.1
```

Figure 3-17 - Definition of SOILDISP, type 3. Pile-by-Pile

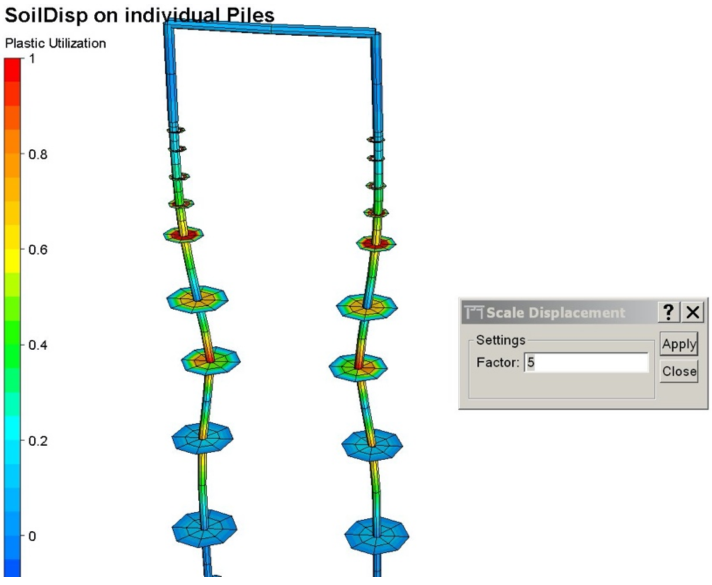


Figure 3-18 - The piles are pushed in opposite directions. Disp depends on Z.

3.4.4 Inner Pile. Longer than main pile

It is possible to add an inner pile to the main pile. The inner pile could be longer than the main pile as shown in the illustration. The “Zlmat” refers to the material used by the ZeroLength spring, which connects the inner pile to the main pile.

```
'               ID       Nod1   Nod2  SoilID      PileMat    PileGeo  LCoo
 Pile        6001         3      4       1        210345     11001     0
'
'               ID       Nod1   Nod2   MainPile   PileMat    PileGeo   ZLMat
 InnerPile  7001       103    104    6001         300       300       400
```

Figure 3-19 – Defining a pile inside pile 6001 going from node 103 to 104

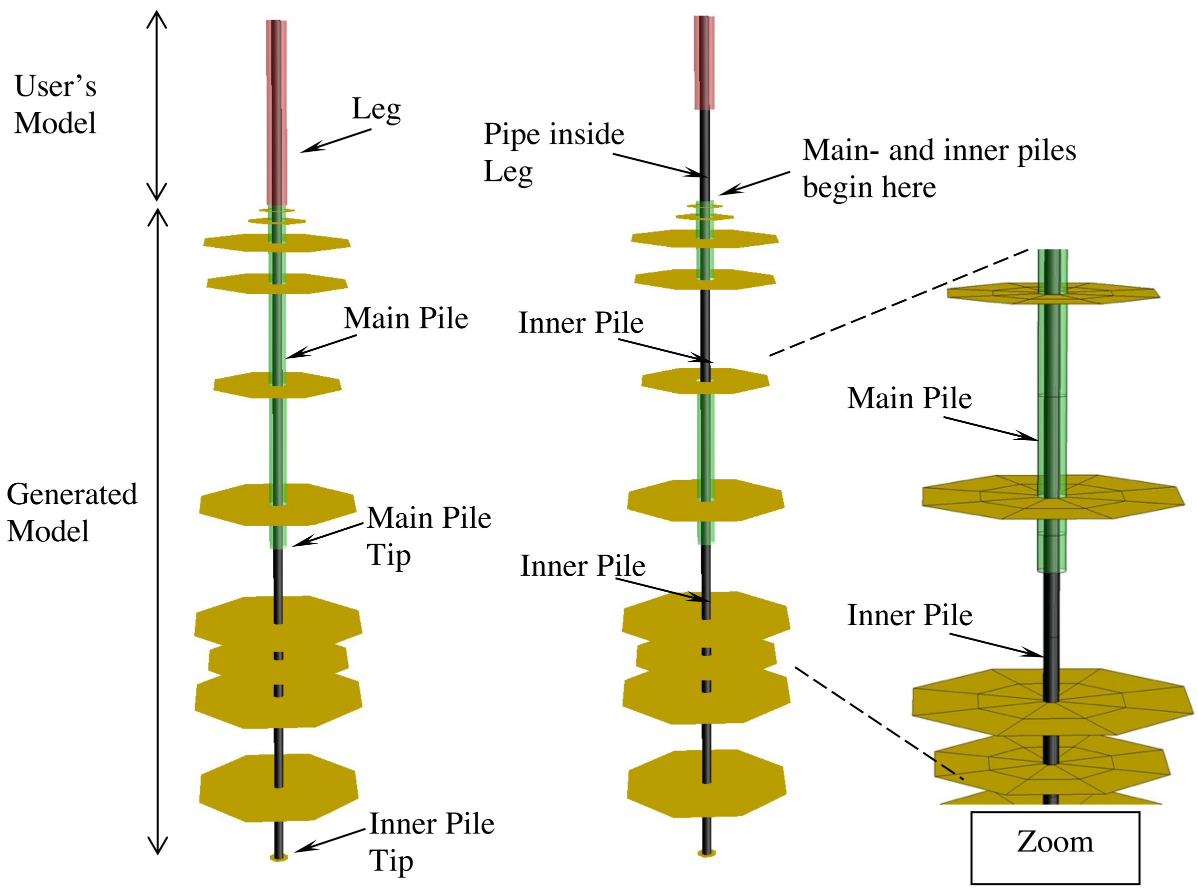


Figure 3-20 - Leg with inside pipe and a Pile with “Inner-Pile”

- o Mat : Referring to misoiep defining the inner pile material prop.
- o Geo : Referring to Pipe ID defining the inner pile cross section
- o ZLMat : Material for ZL-spring between inner & main pile.

```
'' ---------------------------------------------------------------------
'     One  Pile. Goes from Platform towards tip.
' ---------------------------------------------------------------------
'
'        ID     Nod1  Nod2 Soil_ID  Pile_Mat Pile_Geo Local_coord Imper
 Pile   6001      3     4     1      210345    11001    0
'
' ---------------------------------------------------------------------
'     Define inner pile between node 103 and 104.
'     Contact to Main pile uses material no 400 (Bearing)
' ---------------------------------------------------------------------
'
'            ID      Nod1  Nod2  MainPile    Mat     Geo     ZLMat
 InnerPile 7001      103    104   6001       300     300      400
'
'
'                    End   ListType    MatID
 ElmTrans   Loc    1      Mat       200       !  Use same local coosys as pipe i
'
'         ID       Do     T
 Pipe      1       1.2    0.05    ! Leg
 Pipe    100       0.6    0.02    ! Pipe inside leg
 Pipe    200       0.1    0.01    ! Dummy geo for ZL spring
 Pipe    300       0.6    0.02    ! Inner Pile
'
' ------------------------------------------------------------
'     Material
' ------------------------------------------------------------
'
'         matno    E-mod    poiss  yield   density
 MISOIEP    1      2.1E11     0.3   400E+06  7850   0 ! Outer Leg
 MISOIEP  100      2.1E11     0.3   300E+06  7850   0 ! Pipe inside Leg
 MISOIEP  300      2.1E11     0.3   500E+06  7850   0 ! Inner Generated
 MISOIEP  210345   2.1E11     0.3   345E+06  7850   0 ! Main Pile
'
'
'         ID   Typ    opt  s11  s22  s33 s44 s55 s66
 Material  400  Bear  Lin   1   2e7  3e7  0   0   0  ! Connect. between inn/out
'
'          MatId     X    Y     Z
 MREF       200      0   201   201    0  0   0       ! Connect betw. leg and inn
'
 HypElast   201   -1e7    -1.070
                   1e7     1.070
```

Figure 3-21 – Misc. command defining the model with Inner Pile

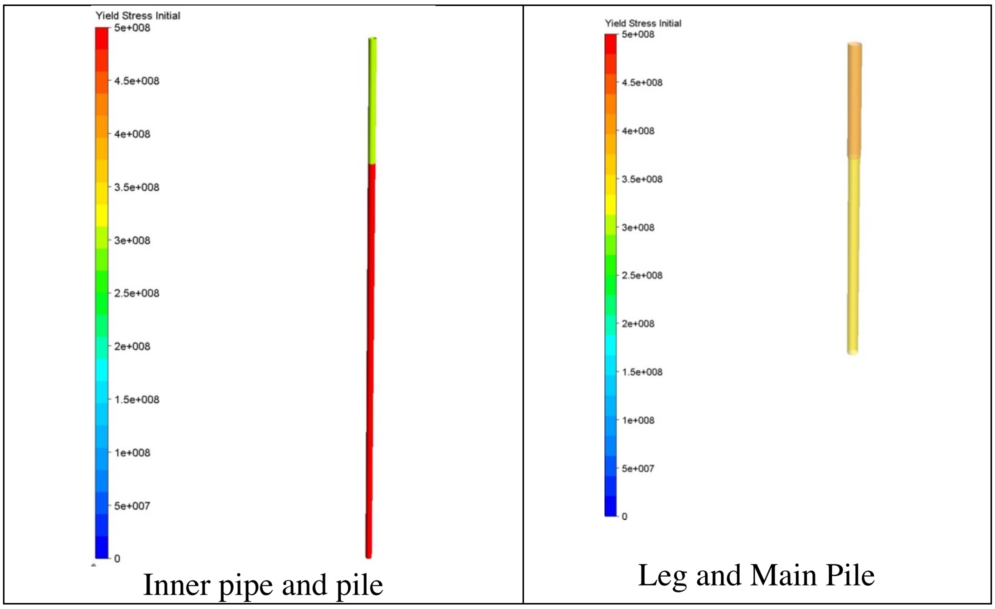
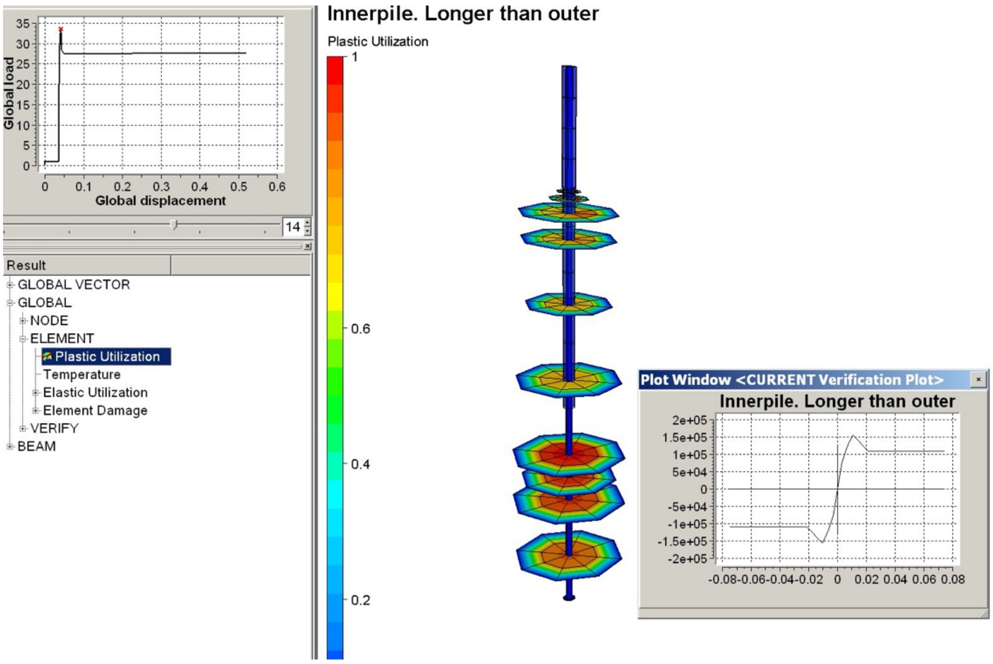


Figure 3-22 - Yield strength is different for the different components

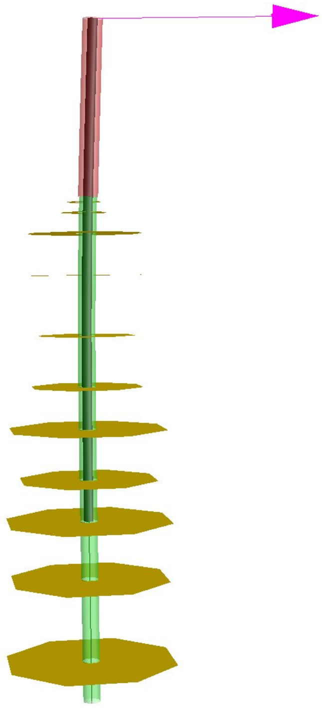


3.4.5 Inner Pile. Shorter than main pile

The inner pile could also be shorter than the main pile. The input is the same as for the long inner pile, but the inner pile tip is located higher up in the soil.

```
'            ID       Nod1   Nod2  SoilID      PileMat   PileGeo  LCoo
 Pile       6001        3      4      1        210345    11001     0
'
'            ID       Nod1   Nod2  MainPile   PileMat   PileGeo   ZLMat
 InnerPile  7001      103    104   6001         300      300       400
```

Figure 3-23 – Defining a pile inside pile 6001 going from node 103 to 104



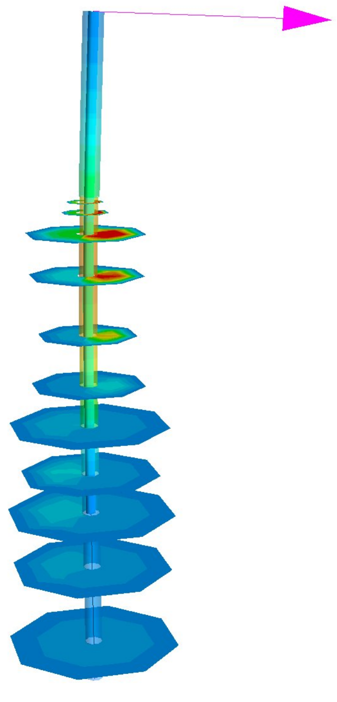
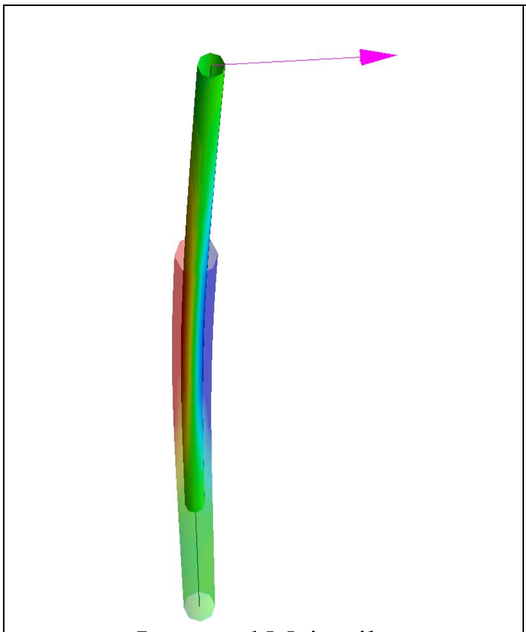


Figure 3-24 – Complete pile-soil system **Plastic Utilization**


Inner and Main pile

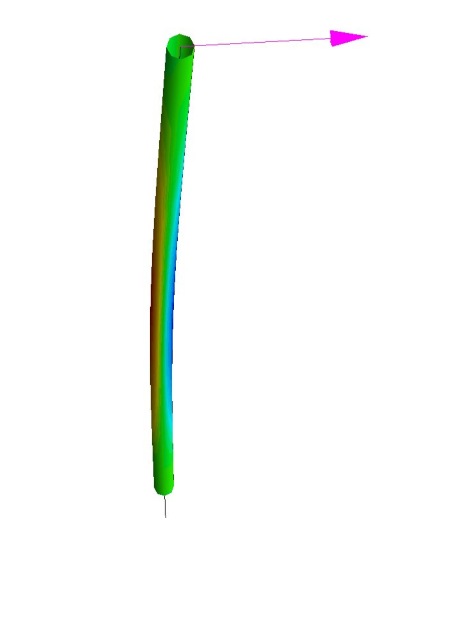

Inner pile

Figure 3-25 – Stresses in piles

3.5 Hydrodynamics

3.5.1 Wave Integration. Mesh

Defining distance between integration points using depth profile. Wants 1m between points in the upper layers. Length of each element is 20m.

Gives 21 points on the upper element (nInt = (Length / Dist) + 1).

```
'
'                              Z   Distance
 Wave_Int         Mesh       20.0     1
                              0.0     1
                            -10.0     2
                            -20.0     3
                            -50.0     5
                            -80.0    10
```

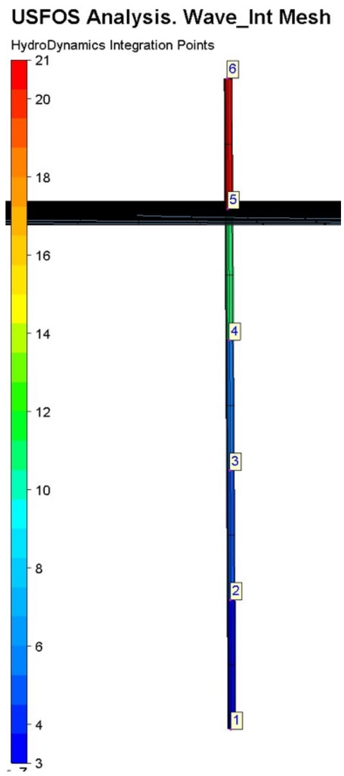


Figure 3-26 – Defining wave integration density.

3.5.2 KC-Table

It is possible to let USFOS assign drag- and mass coefficients based on specified tables and the (depth) location of the element. The depth is used to estimate the peak particle velocity need to compute the KC-number. Elements with marine growth use the data for "Rough", see image on next page.

```
'
'          Type       KC        Rough   Smooth
 Hydro_Cd  KC_Table  0.01       2.04    1.16
                     0.75       0.49    0.19
                     2.00       0.49    0.19
                     3.00       0.59    0.26
                     12.00      1.54    0.845
                     21.00      1.31    0.815
                     42.00      1.16    0.725
                     63.00      1.05    0.65

'
'          Type       KC       Rough    Smooth
 Hydro_Cm  KC_Table   0          2.0     2.0
                      1          2.0     2.0
                      2          2.0     2.0
                      3          2.0     2.0
                     12          1.6     1.6
                     20          1.2     1.6
                     60          1.2     1.6
```

Figure 3-27 – Defining wave integration density.

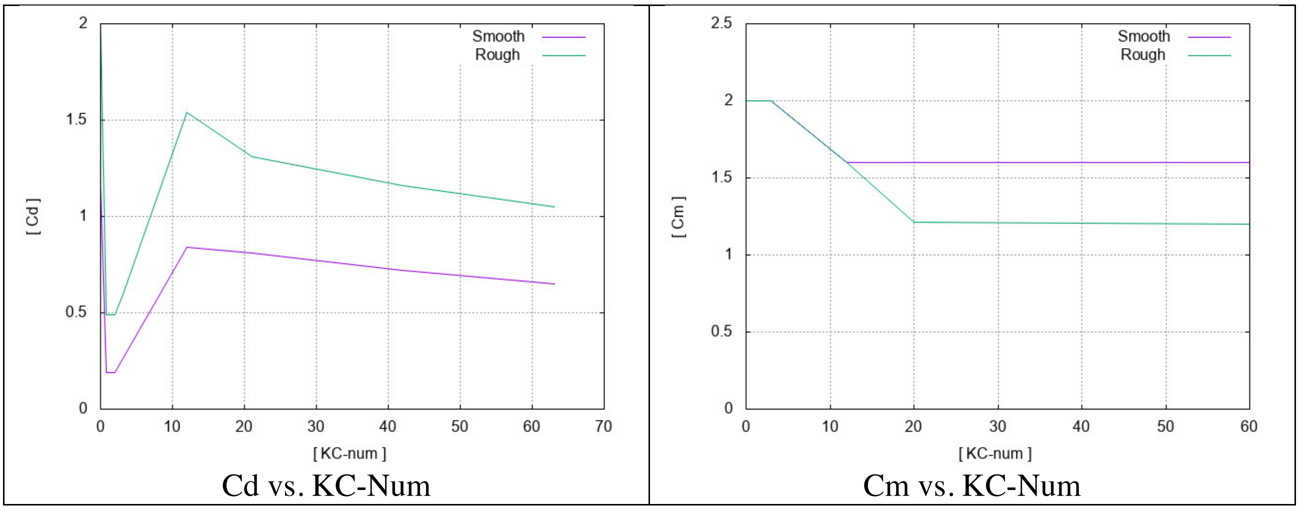


Figure 3-28 – Cd and Cm vs. KC-Num. Smooth and Rough cylinder

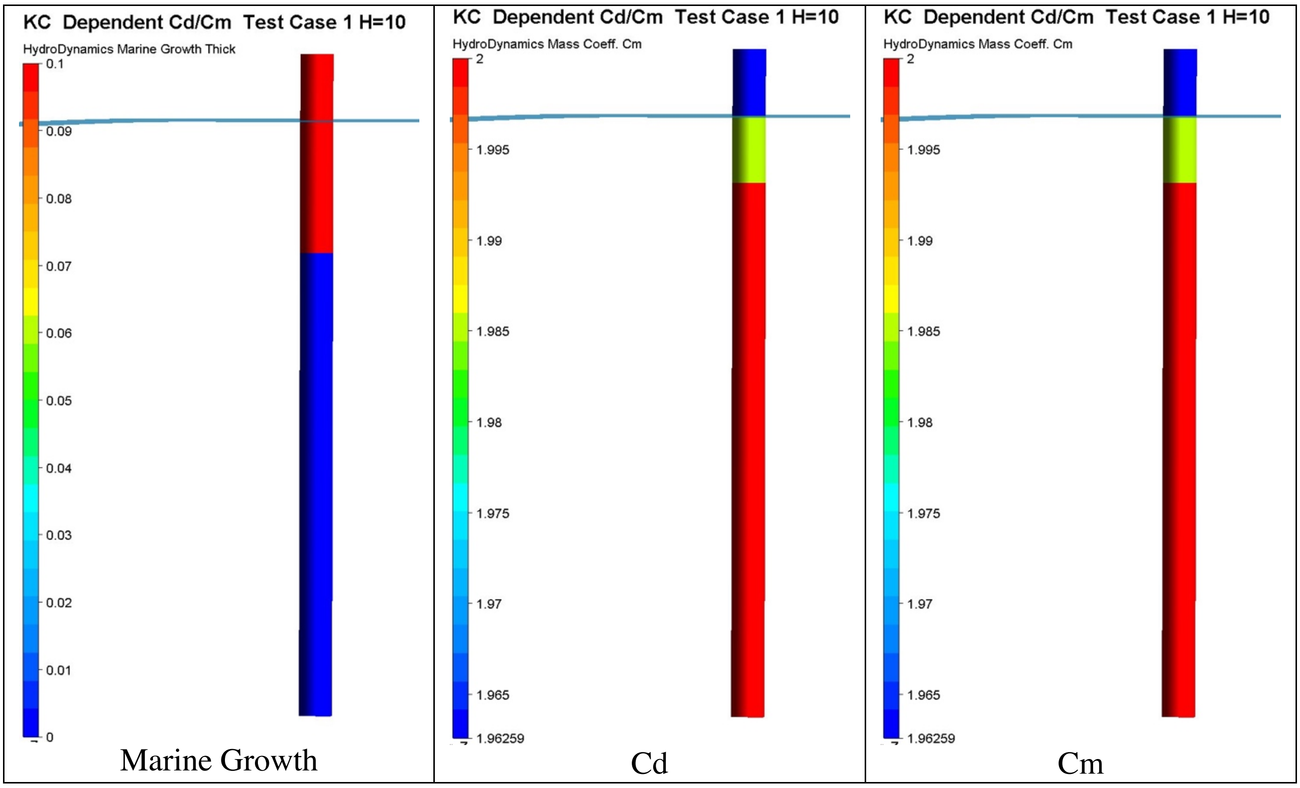


Figure 3-29 – Computed Cd and Cm. Rough is used for elements with marine growth.

3.5.3 2nd order wave

It is possible to switch ON 2nd order wave theory on selected members.

- The “Switches” command is a global switch needed to activate the 2nd order wave module.
- The HydroPar command defines actual elements to use the 2nd order theory.

2nd order wave calculation is very time consuming and should be used only when needed (this is why the Hydropar specification is introduced).
(See also “*spec2grid*” on page 38)

```
'         LCase  LoTyp Hs   Tp   Dir   Seed   Surf   Depth  nIni
 WaveData   2   Spect  15   15   000   033     0.0   100     4
'
     -1000   1
      -200   1
         0   0
       100   0
'
'      nfreq Type      T_min  T_max   grid   Gamma
        50   Jonsw       3    30.0      3     3.3

 Switches  WaveData  SecOrder ON
'
 HydroPar  SecOrder  ON   All
'
```

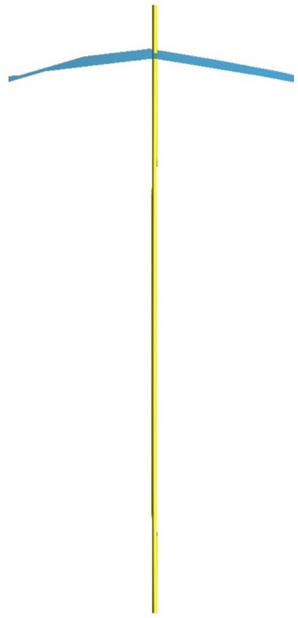
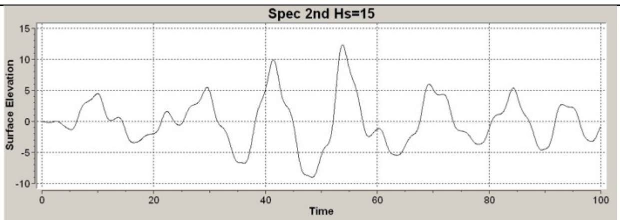
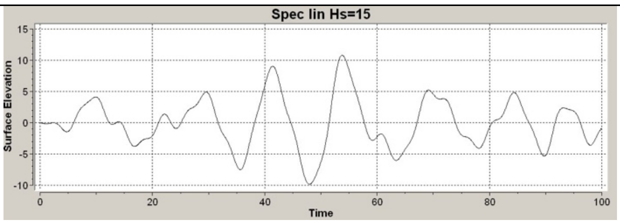


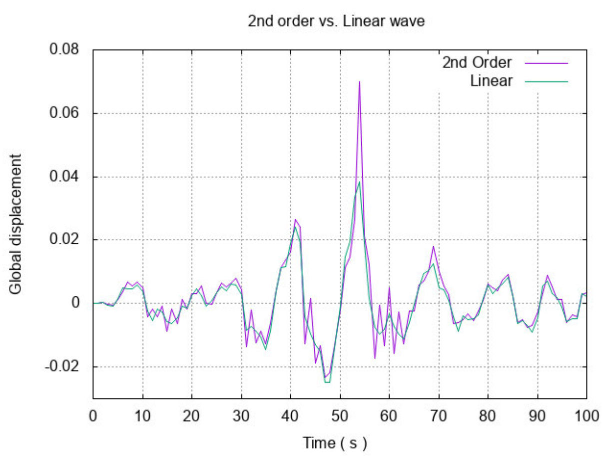
Figure 3-30 – Defining 2nd order wave for all elements along the slender cylinder.



2nd order wave



Linear Wave



Response. Violet curve is 2nd order.

Figure 3-31 – Wave elevation (left) and response (right)

3.5.4 Print of Hydro Masses

The generated hydrodynamic masses, both the “added mass” and the ingernal fluid mass are printed in the ”out file.

If the print switch (CPRINT) is set to 1, only the summary is printed. For higher print levels, the hydrodynamic masses are printed element-by-element in addition.

The masses are printed both referring to the elements’ local axes and referring to the global coordinate system.

```
        ---------------------------------------------------------
        ----          H Y D R O  -  M A S S E S             ----
        ---------------------------------------------------------

        ----    T O T A L    H Y D R O  -  M A S S E S     ----

     Mass item                  GlbX         GlbY         GlbZ
                               [ kg ]       [ kg ]       [ kg ]
     HydroDyn Added Mass :   9.467E+06    9.466E+06    2.614E+06
     Internal Fluid Mass :   0.000E+00    0.000E+00    0.000E+00
     Marine Growth  Mass :   2.003E+06    2.003E+06    2.003E+06

     Total               :   1.147E+07    1.147E+07    4.618E+06
```

Figure 3-32 - Summary of all Hydrodynamic masses

```
     ----      H y d r o D y n.     A d d e d  M a s s     ---

    Elem    LocX       LocY       LocZ       GlbX       GlbY       GlbZ
          [kg/m]     [kg/m]     [kg/m]     [ kg ]     [ kg ]     [ kg ]
       1  0.00E+00  1.27E+03  1.27E+03  1.81E+04  2.63E+04  2.12E+04
       2  0.00E+00  1.27E+03  1.27E+03  2.22E+04  3.22E+04  2.60E+04
       3  0.00E+00  1.27E+03  1.27E+03  2.83E+04  1.86E+04  2.35E+04
       4  0.00E+00  1.27E+03  1.27E+03  3.36E+04  2.21E+04  2.80E+04
       5  0.00E+00  1.27E+03  1.27E+03  2.22E+04  3.22E+04  2.60E+04



     ----    I n t e r n a l  and  M a r i n e  G r o w t h  --

    Elem      Internal Fluid           Marine Growth
            Distributed    Total      Distributed   Total
             [kg/m]        [ kg ]      [kg/m]       [ kg ]
       1     0.000E+00    0.000E+00    2.335E+02    4.986E+03
       2     0.000E+00    0.000E+00    2.335E+02    6.116E+03
       3     0.000E+00    0.000E+00    2.335E+02    5.366E+03
       4     0.000E+00    0.000E+00    2.335E+02    6.384E+03
```

Figure 3-33 – Element-by-element print of Hydrodynamic masses

3.5.5 Hydrodynamic Node Masses

The new command HydMass NODE is introduced for specification of concentrated hydrodynamic masses. These masses contribute only to the inertia, (and not influenced by the Gravity).

The masses refer to the Global Coordinate System.

The mass could be given either for all directions in once or one-by-one direction.

The example shows two ways to assign 10,000kg to node 11, all three directions.

Repeated mass definitions for a given node and DOF will be accumulated (see Figure 3-36).

```
'
'         type         dof         HydMass[kg]   ListType    IDs
 HydMass   NODE         AllTransl     10000         Node       11
'
'
```

Figure 3-34 – Define hydrodynamic mass of 10,000kg for all three global directions.

```
'
'         type          dof      HydMass[kg]   ListType    IDs
 HydMass   NODE         XTransl    10000         Node       11
 HydMass   NODE         YTransl    10000         Node       11
 HydMass   NODE         ZTransl    10000         Node       11
'
```

Figure 3-35 – Define hydrodynamic mass of 10,000kg, one-by-one direction.

```
'
'         type         dof         HydMass[kg]   ListType    IDs
 HydMass   NODE         AllTransl    10000/2       Node        11
 HydMass   NODE         AllTransl    10000/2       Node        11
'
```

Figure 3-36 – Repeated definitions are accumulated.

3.6 Node Set

Pre-defined sets of nodes could be defined for reference.

3.6.1 Change Boundary Conditions

The plate has no boundary conditions. The boundary conditions are specified (Chg_Boun) in e.g. the USFOS control file. The nodes are specified by use of node sets, which are defined using the NODESET command, (node-by-node or referring to coordinates).

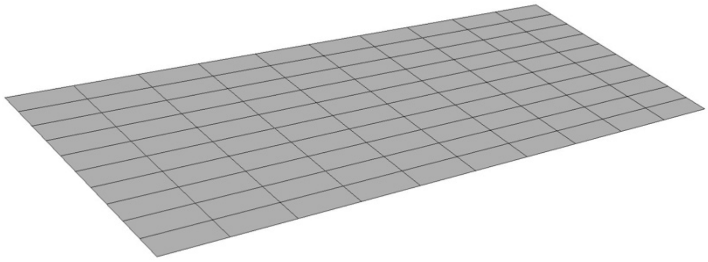


Figure 3-37 - Plate without boundary conditions

```
'
 Chg_Boun  1 1 1   1 1 1    nSet    111
 Chg_Boun  1 0 1   0 1 0    nSet    222
 Chg_Boun  0 1 0   0 0 0    nSet    333
'
 NodeSet   Define   111   Node            1  5   9
 NodeSet   Define   222   X_Coordinate    5.0   0.001
 NodeSet   Define   333   Y_Coordinate    2.0   0.001
'
'
 Name   NodeSet    111     node-by-Node_specification
 Name   NodeSet    222     X-coord=5.0
 Name   NodeSet    333     Y-coord=2.0
'
-
```

Figure 3-38 – Defining three node sets and change boundary conditions.

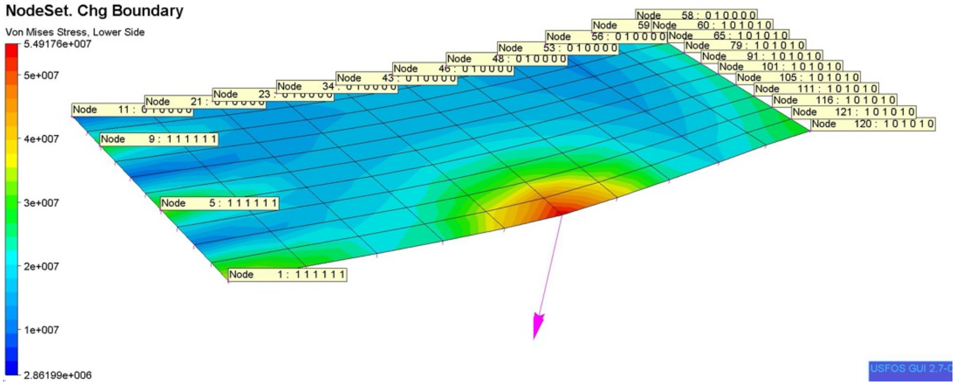


Figure 3-39 – USFOS analysis model with the boundary conditions applied.

3.7 Load Control

3.7.1 Termination

It is possible to set criteria for when a simulation should terminate: Either completely or just terminate the current cusfos "line" and continue on the next.

At present, two criteria are available:

- Utiliz : Plastic utilization
- Strain : Plastic strain.

As soon as one element exceeds the specified level, the analysis will terminate (or just jump to next line).

```
'
'              What    Crit      Level
 Terminate     All     Utiliz    0.90
'
'          nloads   npostp   mxpstp   mxpdis
 CUSFOS     10        50        1       1
'          lcomb    lfact    mxld     nstep     minstp
              1       0.1       8       200       0.001  ! Load
              1      -0.1     0.01      200       0.001  ! Offload
'
```

Figure 3-40 – Terminate simulation when (if) utilization exceeds 0.90.

```
'
'              What    Crit      Level
 Terminate     All     Utiliz    0.90
'
'          nloads   npostp   mxpstp   mxpdis
 CUSFOS     10        50        1       1
'          lcomb    lfact    mxld     nstep     minstp
              1       0.1       8       200       0.001  ! Load
              1      -0.1     0.01      200       0.001  ! Offload
```

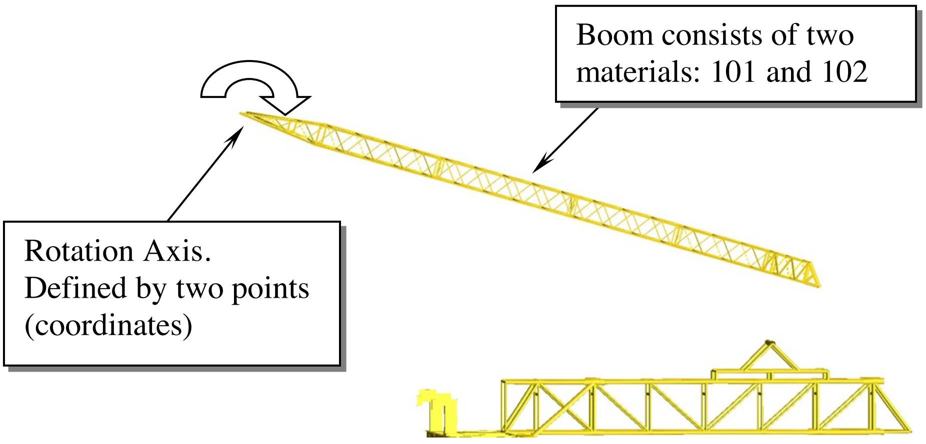
Figure 3-41 – Terminate current "line" and continue with next (offload) if utiliz>0.9.

3.8 Initial Velocity

3.8.1 Body Rotation

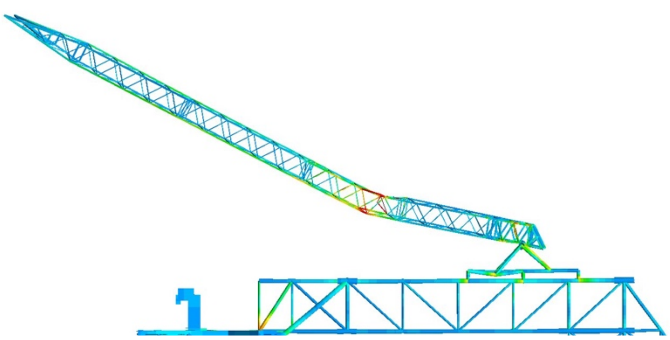
By assigning a given initial body rotation to e.g. a crane boom, it is possible to start the dynamic simulation immediately with the specified hit speed. The coordinates to two points define the rotation axis.

NOTE! Unique Material ID's are required for the actual object with body rotation. These Material-ID's are specified in the "BodyRotMat" input. The rotation speed is given in radians per second.




```
'' ---------------------------------------------------------------------
'     Initial Rotation about Crane Axis. Start object immediately
' ---------------------------------------------------------------------
'            Type       Time RotSpeed    x     y    z
 INI_VELO  BodyRotMat    0.0    0.5     0.0   -10   20 ! Point-1 on RotAxis
                                        0.0    10   20 ! Point-2 on RotAxis
'
'                                     Mat IDs
                                        101      102
```

Figure 3-42 – Define body rotation of materials 101 and 102. Specify rotation axis.



3.9 Joints

3.9.1 Stiffness Scale

The elastic stiffness of a tubular joint has little (no) impact on the ultimate strength level of the structure.

However, in cases with dynamic simulations the elastic stiffness may impact the eigen-periods of the structure. In case the eigen-periods need to be tuned it is possible to specify scaling factors to be used on the different joint degrees of freedom's elastic stiffness.

```
'
'              key        Factor      Opt     ID
 JntOption    StfScAxT     2.0       Joint    100
 JntOption    StfScAxC     2.0       Joint    110
 JntOption    StfScAxT     2.0       Joint     70
 JntOption    StfScAxT     2.0       Joint     80
 JntOption    StfScAxT     2.0       Joint     40
 JntOption    StfScAxT     2.0       Joint     50
'
 JntOption    StfScAxT     2.0       Joint     90
 JntOption    StfScAxT     2.0       Joint     60
'
 JntOption    StfScIPB     4.0       Joint     60
 JntOption    StfScOPB     4.0       Joint     60
```

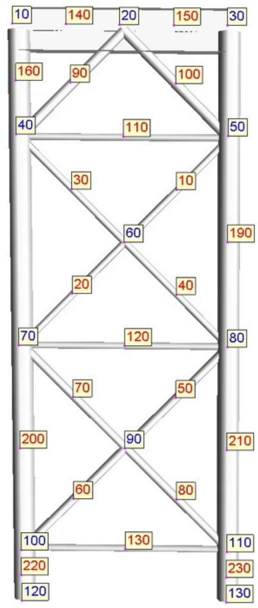


Figure 3-43 – Defining Double Elastic Stiffness of specified joints

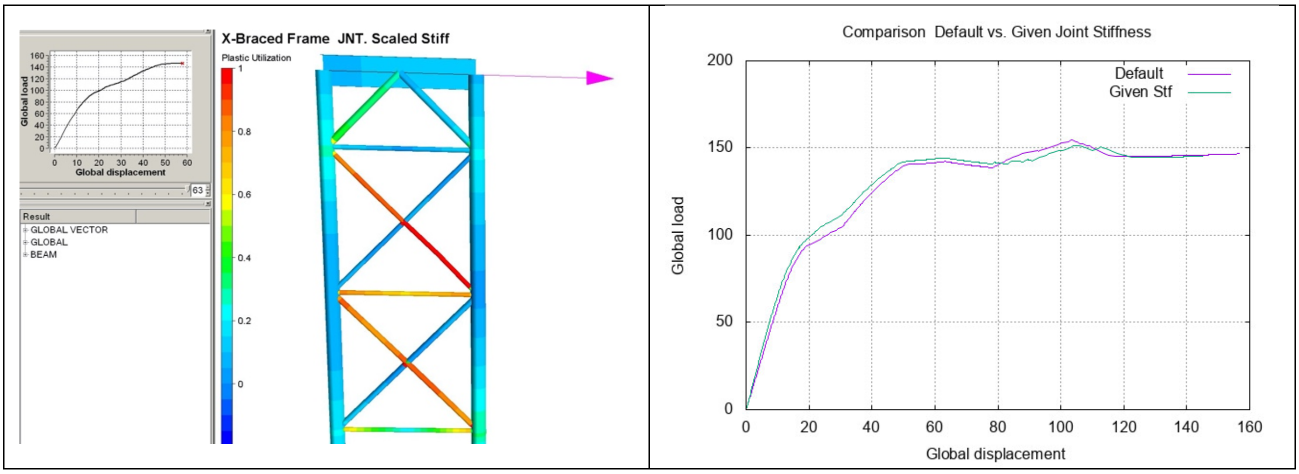


Figure 3-44 – Comparison of response. Default (pink) and Specified (green) stiffness.

3.9.2 Skip Check

If, of some reasons, one or more braces should be excluded in the joint capacity checking, the "SkipCheck" will insert an element with higher resistance than the incoming brace.

Both pipes and non-pipes could be excluded.

```
'
'            key        Opt       NodeID   BraceID
 JntOption SkipCheck  Connection    2        25     !
'
```

Figure 3-45 - Skip checking of braces

```
---------------------------------------------------
     -----     J O I N T   C A P A C I T I E S     -----
     ---------------------------------------------------

     -----------------------------
     Joint  :          2 , Rule: Norsok N-004, Yield: 4.000E+08
     Chord-1:         12
     Chord-2:         23

     Connect   Brace    Geom      Angle        Gap
      no        ID     (Shape)    (deg)
       1        25      NoChk      90          N/A
       2        24        K        45         Autom
```

Figure 3-46 – Print of Skipped braces

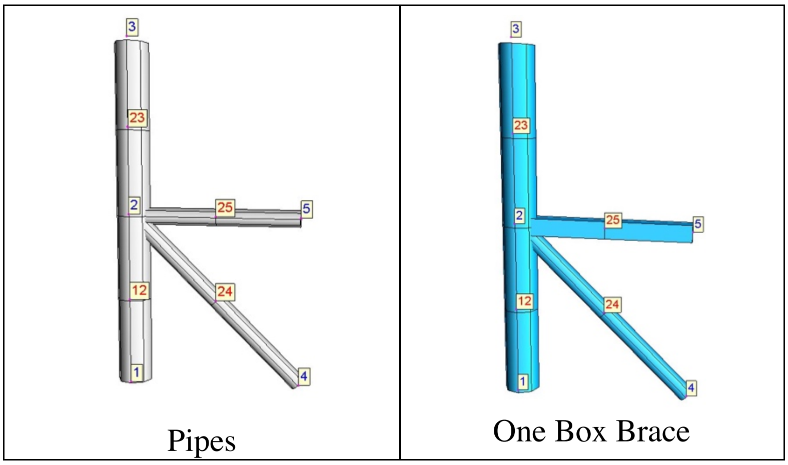


Figure 3-47 – Joint with brace 25 as either pipe (left) or box (right)

3.9.3 Pile-In-Leg using ZL-spring

When piles are modelled inside the leg it is convenient to use 2-node Zero Length non-linear springs to connect the (inner) pile with the (outer) leg.

It is now possible to use the CHJOINT option also when a ZL-spring is a “brace” to this joint. (All elements except the two specified chord members are treated as “brace).

The ZL-spring could be either a linear “bearing”, see Figure 3-49 or a “MREF” see Figure 3-50 on next page.

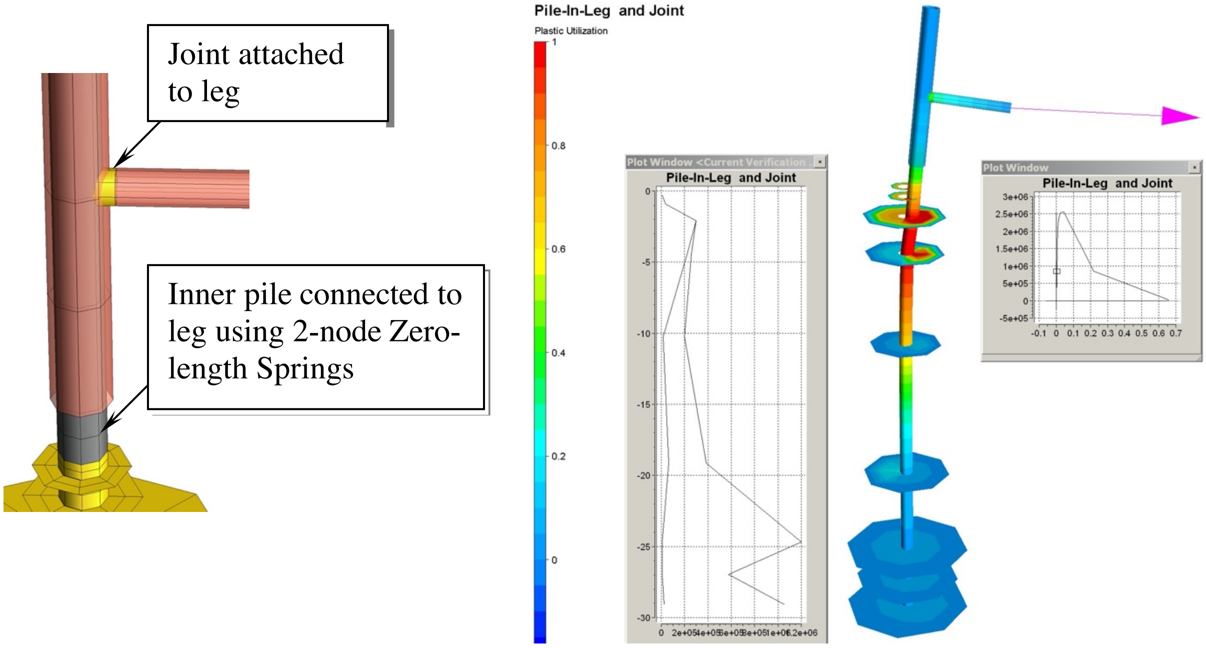


Figure 3-48 - The pile inside the leg is connected using ZL-spri. Also at Joint.

```
'          Node    Ch1   Ch2
 ChJoint    2       1     2

'      elm      n1     n2    mat   geo
 Beam   1001     1     11   1000    4
 Beam   1002     2     12   1000    4 ! Spring attached to "ChJoint Node
 Beam   1003     3     13   1000    4

 Beam    3      2     4      2     2
'
'          ID    Typ          s11   s22   s33   s44   s55   s66
 Material 1000  Bear   Lin      0    1e7   1e7     0     0     0
'
'           Type   End    ListTyp     ID ....
 ElmTrans    Loc    2      Mat        1000
```

Figure 3-49 - Definition of springs, which connects pile and leg. Bearing element.

```
' '
'          ID          s11  s22  s33  s44  s55  s66
 MREF     1000          0  1001  1001  0    0    0
'
'          ID        P      d
 HypElast 1001    -1e7   -1.000
                    0    -0.050 ! 50mm gap
                    0     0.050 ! 50mm gap
'                  1e7    1.000
'
'           Type  End   ListTyp    ID ....
 ElmTrans    Loc   2     Mat       1000
```

Figure 3-50 - Definition of spring material with gap

NOTE-1:
Make sure that the ZL-spring does no contribute significantly in the joint load transfer, (the purpose of the spring is to stabilize the inner pile).

=> Use a relatively soft ZL-spring (bearing) or a spring with gap.

NOTE-2:
The element forces normally refer to the element's local coordinate system, which for the ZL-springs is the same as the attached main beam, (e.g.: X is along the leg axis).
However, a ZL-spring with *bearing* material is an exception. The element forces for bearing elements refer to the Global Coordinate System.

3.9.4 Joint Fracture Handling

If the joint element exceeds the ultimate peak in the force-displacement or moment rotation curves the joint failure begins.

If the joint element fails to find equilibrium in such cases, the analysis often stopped with an error message.

In version 8-9, such cases are handled differently. A softening process starts, and the element will be degraded gradually and, in practice, be removed after some steps.

3.10 Temperature

3.10.1 Pushdown Time

FAHTS generates a "lcasetim" command for every new temperature load case. This command connects the load case to the fire time.

In addition to specifying the fire load case under Pushdown, it is possible to refer to this fire time.

NOTE! The exact time has to be found. USFOS will ***not*** search for nearest time,

```
'
'           KeyWord   Time
 PushDown    AtTime    30
'
```

Figure 3-51 - Perform pushdown for temperature state at time 30min

```
'
'           KeyWord   Time
 PushDown  UpToTime    30
'
```

Figure 3-52 - Perform pushdown for "all-time-high" temperature up to time 30min

3.11 Miscellaneous

3.11.1 DentPlot

The current dent dept could be stored for visualization in XACT using the switches command.

```
'
 Switches  Dent     Store
 XfosFull
'
'          key        LCase     Type    NodeID    Extent     Fx     Fy     Fz
 SurfImp  LoadCase    9999     Node     2997        2      -1E6     0      0
 SurfImp  LoadCase    9999     Node     3067        2      -1E6     0      0
'
```

Figure 3-53 – Define surface impact at two locations simultaneously. Store Dent info

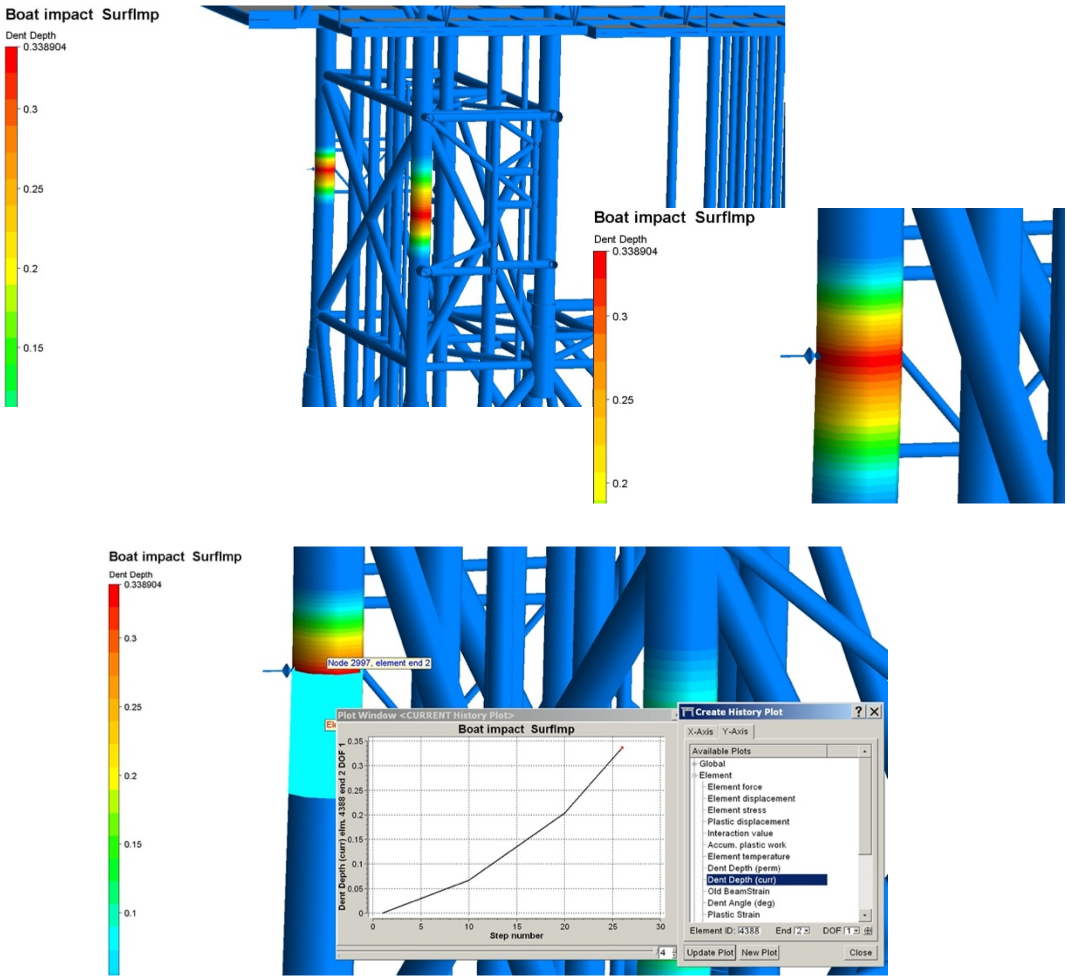


Figure 3-54 - Visualization of dent depth and plot of dent for a given element

3.11.2 Status Print, ("res_status.text")

The status print containing key results is also produced when the simulation stops with an error. The termination status, either "*Normal stop*" or "*Error exit*" is printed.

In case of error exit, the status print could include the step causing the error, but in most cases the last stored result step is the step prior to the error.

3.11.3 Plastic strain

The original "Beam Plastic Strain" is renamed to "Old BeamStrain".

A new strain "Plastic Strain" is introduced. This strain estimate has a more fixed extent of the plastic zone and is not dependent on the element forces. The strain estimate is based on the plastic rotations/displacements and the accumulated plastic work.

NOTE! The UserFrac uses the new plastic strain as the default strain criterion (keyword "Strain"). If the old beam plastic strain is wanted, use the keyword "OldStrain".

3.11.4 Input checking

The input has more checking, and input, which was accepted by previous versions, could stop.

On the other hand, USFOS 8-9 accepts Zero-Length springs to be attached to joints with CHJOINT option, (see Joint section 3.9.3 on page 32).

3.11.5 Shell Elements. Default Hardening set to 0.2%

The default hardening is set to 0.2% (up from 10^{-5}). This is the same hardening parameter as used on beam elements always.

The ultimate stress σ_U is estimated for the ultimate strain $\varepsilon_U = 0.1$, and this means that σ_U becomes approx. 40MPa higher than the yield stress. The change to a more "normal" hardening could make the simulations more stable. For strain exceeding ε_U the stress decreases, (softening).

3.11.6 Riser element / eccentricities

The eccentricities defined on riser elements are no longer removed.

3.11.7 STRUMAN possibilities

STRUMAN version 500 is extended, e.g. for converting file formats, (from SESAM, SACS and STAAD).

It is recommended to always let STRUMAN check the input.

NOTE! STRUMAN processes only the basic structural model data, such as nodes, elements, properties, loads, etc. Special USFOS "analysis control" kind of data is ***not*** processed.

The *not processed* commands are printed.

Model processing:

In addition to a pure file format converting, STRUMAN could also be used for simple model modification options, e.g.: creating groups, change element properties, change boundary conditions, remove elements, clip parts of an element etc. Just send us e-mail if you have some potential STRUMAN operations.

Example: Create group and assign new plate thickness:

```
 UFO
'
 Name Group 123   BOX_01
'           ID                Type   X     Y      Z
 GroupDef   123               Box   2.00  0.00   -0.1    ! Box-1
                                    2.50  1.8     0.1
'
                                    4.00  0.8    -0.1    ! Box-2
                                    4.50  1.2     0.1
'
'         Geo    T
 PlThick  888   0.050
'
'         Geo    ListType   GroupID
 Chg_Geo  888     Group      123
'
```

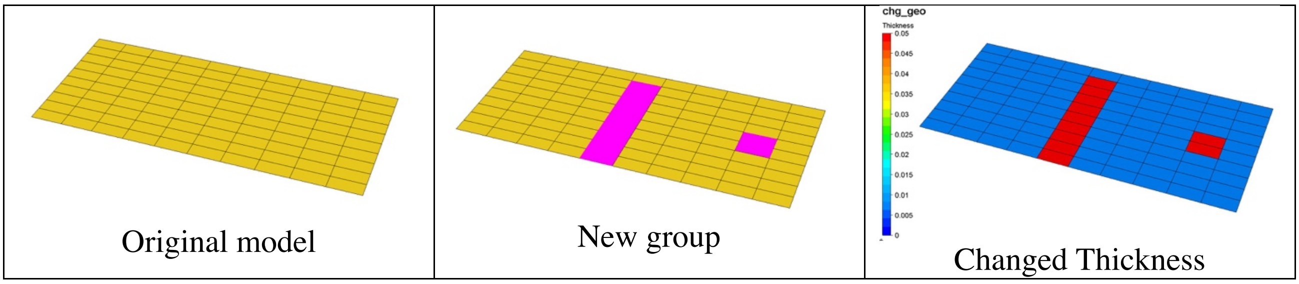


Figure 3-55 – Define a new group and assign new plate thickness to this group

3.11.8 "Spec2grid". Defining a 3D wave kinematics grid

Instead of let USFOS compute the wave kinematics for every element, it is possible to let "spec2grid" generate the wave kinematics in a 3D grid on a separate file prior to the USFOS simulations.

USFOS will read this "grid wave" file and interpolate the kinematics to the element's wave integration points.

If 2nd order wave theory is used, this is a practical way to save analysis time. The same wave field could be used in all actual directions (WAVEDATA "gridwave" has a transformation input).

For more information, contact support@USFOS.com

```
-----------------------------------------------
-----------------------------------------------
-----------------------------------------------
-----        S p e c  2  G r i d         -----
-----   Creates wave kinematics for      -----
-----   irregular waves. Stores in GWF   -----
-----                                    -----
-----   Both linear and 2nd order waves  -----
-----   Direction spreading available.   -----
-----                                    -----
-----   Version 120 / Mar  21, 2018      -----
-----              Usfos AS              -----
-----------------------------------------------
-----------------------------------------------
```

3.11.9 Shear Plasticity Spring. Earthquake.

The Spring Type “ShearPL” could be used to model a special shear connection between two structural parts. The Zero Length spring is modelled as usual using ElmTrans to define the local coordinate system.

The material properties are defined using the conventional MREF, which refers to elasto plastic curves.

The 2-node Zero-length spring has no coupling between the vertical force and bending. If the global overturn moment caused by the sliding ZL-spring is significant, the “SpriOTM 3” parameter given under “Switches Solution” could be used.

```
SpriType   ShearPL   Elem   1001

Switches   Solution   SpriOTM    3
```

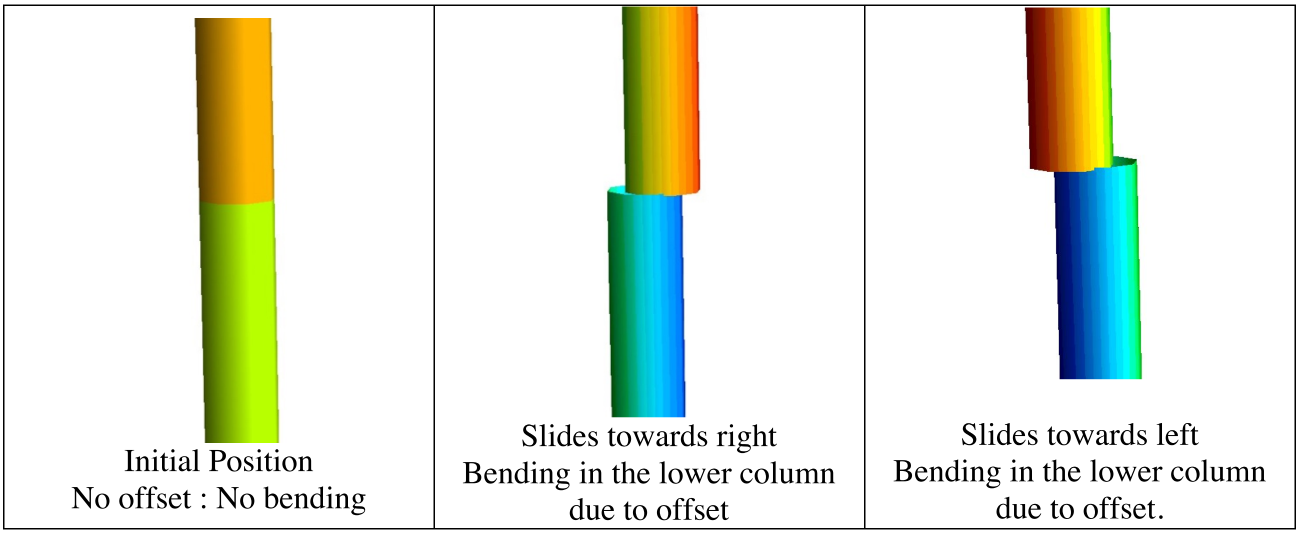


Figure 3-56 – 2-Node Zero Length spring. Shear Plasticity model

3.12 SWITCHES, (Special Options).

The command “SWITCHES” was introduced in 8-5 to switch on special options and is extended in version 8-9. Following “Switches” commands are available, (sub keys in bold are new):

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	890
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
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

3.13 Updates Usfos and Utility Tools

News, corrections and updates are described on the web, and it is recommended to check the following link:

https://www.usfos.com/news/index.html

3.14 New/modified input commands

Since last main release (8-8), following input identifiers are added/extended:

PILEGEO :	Extended command	*: MonoPile*
PILEOPT :	Extended command	*: DispRate*
SOILDISP :	Extended command	*: Type 3 (individual pile)*
INNERPILE :	New command	*: Inner pile (shorter or longer than main pile)*
WAVE_INT :	Extended command	*: "Mesh"(user defined dist between points)*
HYDRO_CD :	Extended command	*: KC number dependent*
HYDRO_CM :	Extended command	*: KC number dependent*
WAVEDATA :	Extended command	*: 2nd order wave*
HYDROPAR :	Extended command	*: Specify 2nd order for element(s)*
HYDMASS :	New command	*: Defining concentrated hydrodynamic masses*
NODESET :	New command	*: Defining a node set to refer to*
CHG_BOUN :	Extended command	*: Referring to nSet (node set)*
TERMINATE:	New command	*: Terminate simulation e.g. for a given strain.*
INI_VELO :	Extended command	*: Initial Rigid Body rotation*
JNTOPTION :	Extended command	*: Scaling of stiffness (e.g. in a dynamic simulation)*
JNTOPTION :	Extended command	*: Skip specified braces*
PUSHDOWN :	Extended command	*: Specifying "Time" (alternative to "loadcase")*
SPRITYPE :	New command	*: Defining Shear Plasticity spring.*
SWITCHES :	Extended command	*: See above.*

3.15 Documentation

The following documents, (updated or new), are available on the web:

- User's manual : Updated document (user's)
- Monopile Theory : New document (theory)
- XACT Cloud : New document (user's)