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 8-8**				
DISTRIBUTION Members of USFOS user group				X

FILE CODE	CLASSIFICATION
	Confidential

REFERENCE NO.

PROJECT NO.	DATE	PERSON RESPONSIBLE / AUTHOR	NUMBER OF PAGES
	2015-11-01	Tore Holmas	24

Release Notes

USFOS 8-8, Nov 2015

1 Introduction

The current official version of USFOS is version 8-8 with release date 2015-09-01. 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 8-8

Comparison of 8-8 vs. older USFOS versions could give somewhat different results due to:

- Mix of hinges and eccentricities. Hinges are removed if conflict (see also Hin2Elem)

- Different T-Z capacities in tension and compression are accounted for.

3 News in USFOS version 8-8 - 2015.

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 8-8 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 8-8 32bit windows. However, version 8-8 becomes available by downloading the central modules (similar to Alternative-2 above):

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

3.2.4 MAC-OSX

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

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 8-8 is “3.0” for the Win-64bit version. The functionality is the same on win32 and win64, but the win64 version has access to more memory and uses QT-4 library.

3.3.1 Updated Preferences. NOTE! Remembers Fringe Range.

The “Preferences” options are updated with following important changes:

1. Current Fringe Range is by default **kept** after opening a new RAF file if the “keep setting on new files” is ON. This is useful if the user wants a certain min/max range for all states. The preferences dialogue has an option to switch off this setting.
2. The viewpoint and zoom are kept.
3. The plot size could be customized (remembers last used size. The size could be set manually, for example width x height = 500 x 300)

3.3.2 Visualization of D-T ratio for Pipes

Diameter to thickness ratio is visualized for pipes. All other sections become grey.

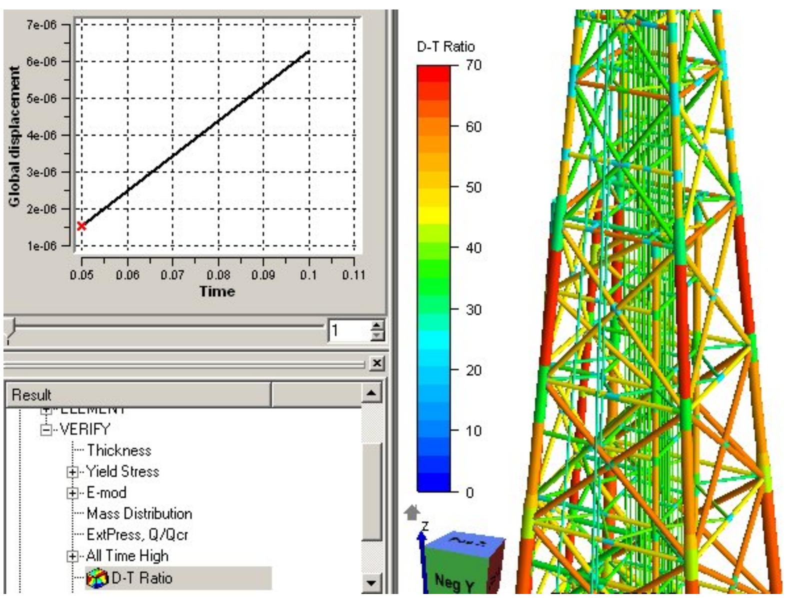


Figure 3-1 - Visualization of D-T ratio of pipes

3.3.3 Verify Slenderness of I-Profiles

USFOS has a function, which shows, graphically, the slenderness (i.e. the opposite of compactness) of the I-cross sections.

The function is based on the AISC-Standard: Specification for Structural Steel Buildings /3/. and performs code checking of the capacity of I-profiles with respect to strong - and weak axis bending, shear loading, compression buckling and lateral torsional buckling. The following colour convention is used to visualize the slenderness/compactness:

- Yellow to Red (> 0.67) : Slender section.

  The cross-sectional behaviour does not conform to the capacity formulations used by USFOS. If slender I-profiles are used in secondary or tertiary structural components, the utilization of the cross-section MUST be checked by means of the code-checking module in USFOS.

- Yellow to Light Blue (0.67 – 0.33) : Semi compact.

  Failure may occur earlier than predicted by USFOS , and the utilization should be checked by means of the code-checking module in USFOS. In order to ensure a high level of robustness, such cross-sections should preferably not be used for important main steel in compression

- Light Blue to Dark Blue (0.33 – 0.0) : Compact.

  The cross-sectional behaviour conforms to USFOS capacity formulations for all loading conditions. The use of compact sections for primary load-carrying members is recommended

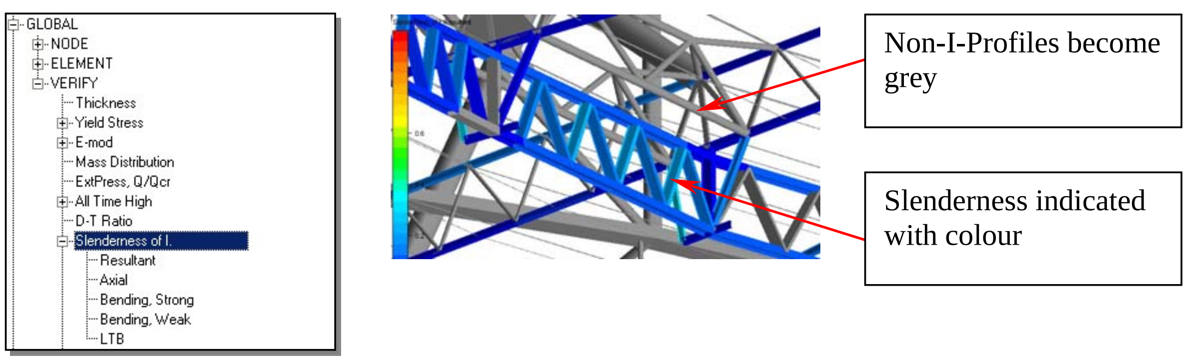
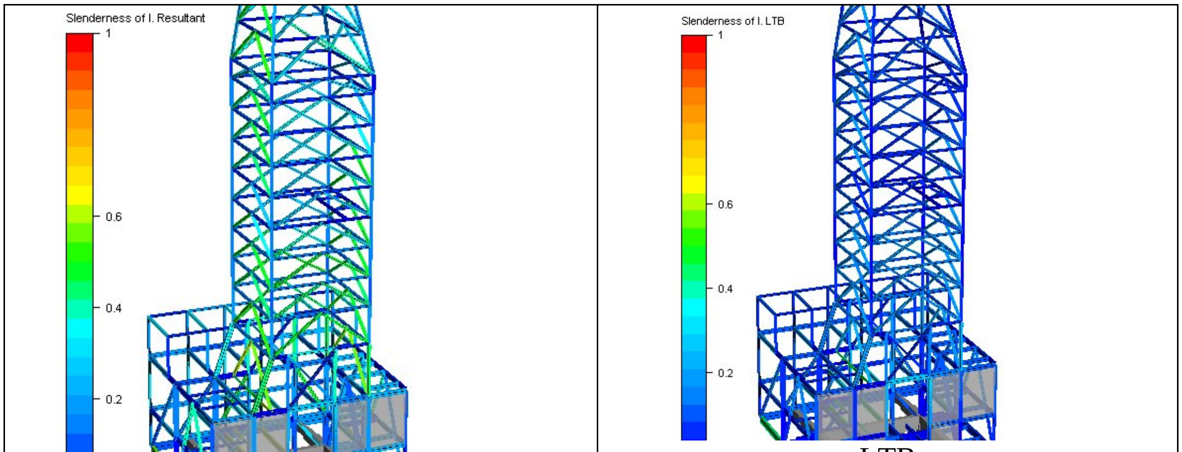


Figure 3-2 - Global - Verify - Slenderness of I



Resultant LTB

Figure 3-3 – Different "Slenderness"

3.3.4 Visualization of NonStru and Fracture elements

Non-structural elements are easier to identify when the “Nonstru visible” is selected. Earlier, the nonstru elements became blue if plastic interaction was selected. Now these elements become grey.

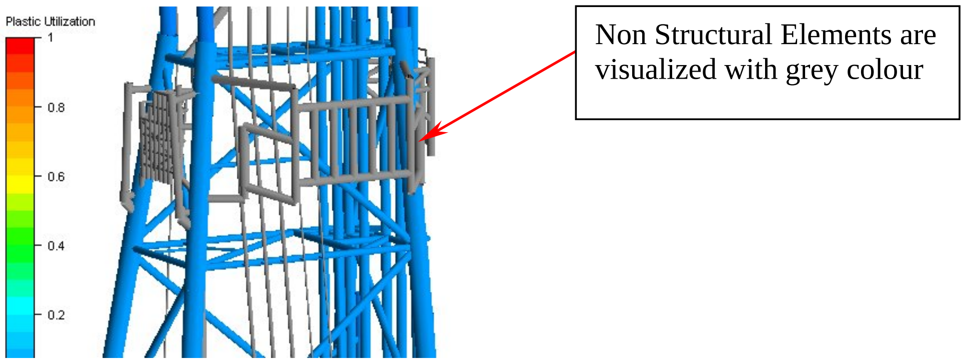


Figure 3-4 – Visualization of Plastic Utilization / NonStru elements.

When an element fractures, the element-forces are removed (sent into the end-nodes), and the element is visualized with grey when plastic interaction is selected.

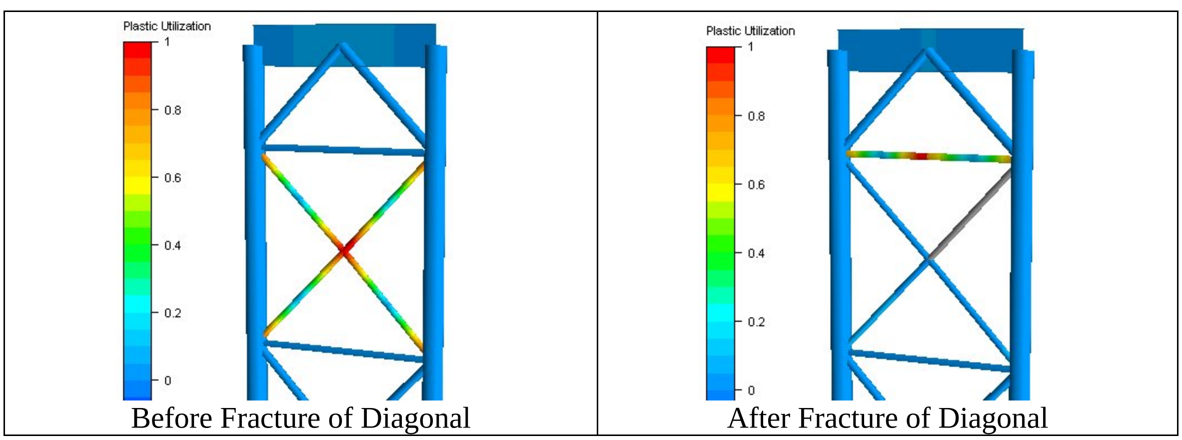


Figure 3-5 – Fracture elements become grey.

3.3.5 Visualization of Soil Strength

By default, the sizes of the soil discs are based on the relative strength, where the T-Z capacity is weighted 100 and P-Y is weighed 1.

The user may change this default using the “SWTCHES” command as follows:

```
'                                   P-Y   T-Z
 Switches   Soil    DiscVisual      50     50
```

```
'                                   P-Y   T-Z
 Switches   Soil    DiscVisual      100     1
```

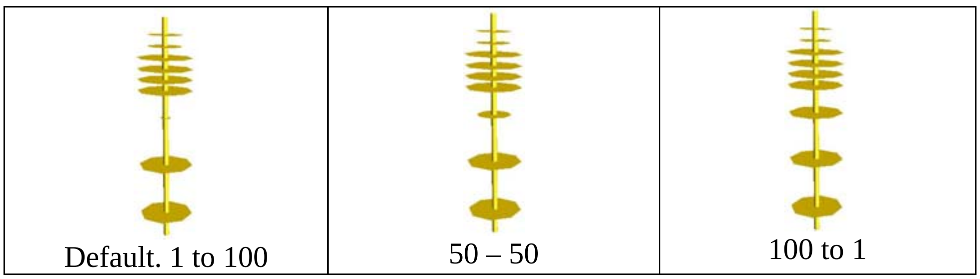


Figure 3-6 - Disc Size for three different weights between P-Y and T-Z.

3.3.6 Visualization of Absolute Displacements

Visualization of displacement ranges from lowest negative (blue) to highest positive (red) if NODE “Displacement” is selected.

If the user wants the largest deflection to become red, the Abs(Displacement) will visualize the absolute value of the displacement as shown in the figure.

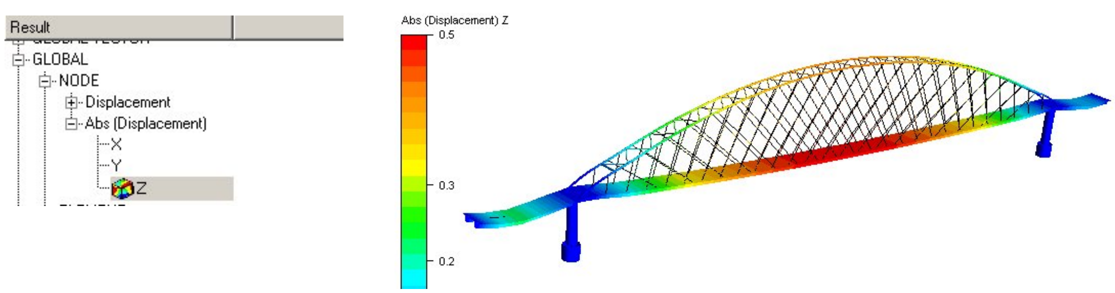


Figure 3-7 – Visualization of Absolute Z-displacement

3.4 Pile material.

Different pile material along a pile is defined using the command PILEMAT as shown below.

```
'
'            Mat  ID      E-mod  Poiss    Yield   Density
 MISOIEP       1000   2.100E+11   0.3  300E6      7850
 MISOIEP       1001   2.100E+11   0.3  600E6      7850
 MISOIEP       1002   2.100E+11   0.3  500E6      7850
 MISOIEP       1003   2.100E+11   0.3  400E6      7850
'
'
'        Pile_id     Nodex1    Nodex2   Soil_id Pile_mat   Pile_geo Lcoor Imper
 PILE       9100        2         3       762      1000     762032    0

'           Pile    Ztop   ZBotm  Material
 PileMat    9100       0     -3      1001    ! Use mat 1001 from  0 to  -3
                      -3     -6      1002    ! Use mat 1002 from -3 to  -6
                      -6    -10      1003    ! Use mat 1003 from -6 to -10
'
```

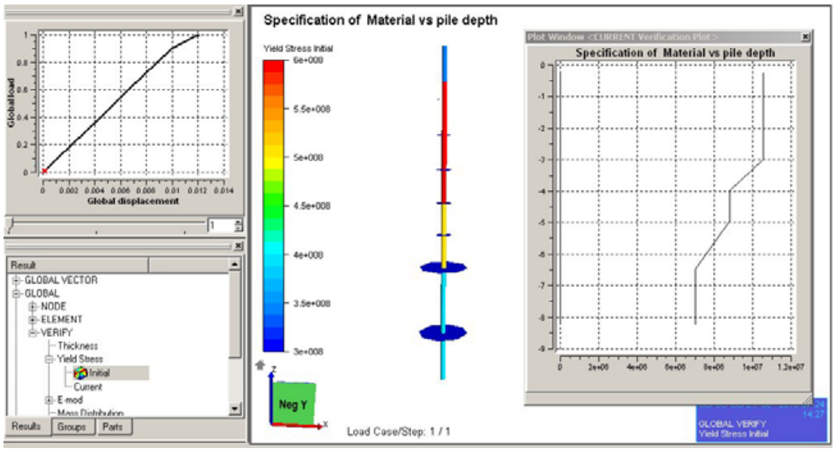


Figure 3-8 - Varying Yield stress along the pipe.

PileMat ALL (instead of PileMat ID) means that all piles get the actual material vs. depth.

This example is found on the web under “foundation”.

3.5 Pile Cross Sections.

A pile is normally a pipe cross section, and it has been possible to specify different diameter/thick along the pile using the command Pile_D-T.

A new option is available in version 8-8 where different pile cross sections (not limited to pipe section) along a pile is defined using the command PILEGEO **ChgCross** as shown below.

In this simple example, only pipes are used, but in principle, other section types could be assigned.

```
' '
'          Opt      Pile    Ztop   ZBotm  Geometry
 PileGeo ChgCross  9100      0     -3      2001   ! Use geo 2001 from  0 to  -3
                            -3     -6      2002   ! Use geo 2002 from -3 to  -6
                            -6    -10      2003   ! Use geo 2003 from -6 to -10
'
 Pipe  2001     0.150   0.050
 Pipe  2002     0.150   0.040
 Pipe  2003     0.150   0.030

'         Pile_id     Nodex1    Nodex2  Soil_id Pile_mat  Pile_geo Lcoor Imper
 PILE       9100        2         3       762      1000     762032       0
'
```

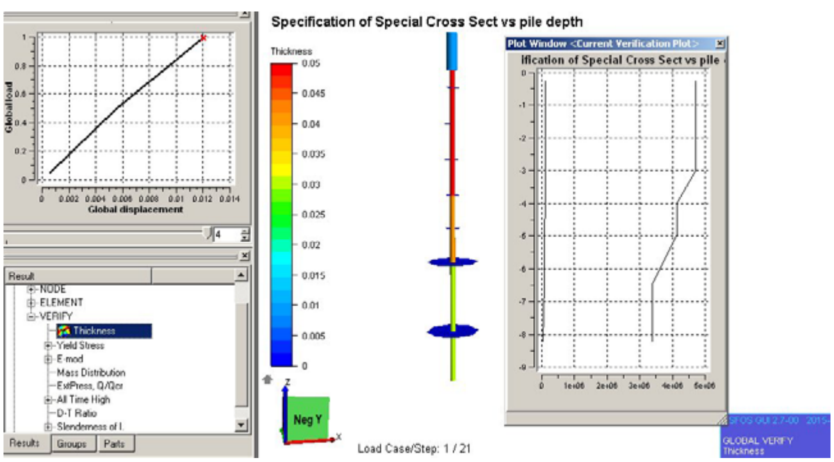


Figure 3-9 - Varying Pile Cross Section along the pipe.

This example is found on the web under “foundation”.

3.6 "Lumped" Soil.

If the resultant properties of the foundation are known, a "lumped soil" model could be used. The element is a 1-node spring to ground with non-linear properties (using MREF & ElPlCurve).

The soil curves are defined as follows:

- DOF-1 : P-Y curve
- DOF-2 : P-Y curve (same curve as for DOF-1)
- DOF-2 : T-Z curve

The command *SpriType Lumpsoil* is used to change the 1-node spring to a special lump-soil element.

```
'
 SpriType    LumpSoil   Elem  1001
'
'            ID      Node   Mat
 Sprng2Gr   1001       1   1000
'
'                  1      2     3    rX      rY     rZ
'                P-Y    P-Y   T-Z
 MREF    1000    1001   1001  1003   0       0      0
'
'
'            MatID      P        d
 ElPlCurve    1001  -1001      -1.050
                    -1000      -0.050
                     -900      -0.010
                      900       0.010
                     1000       0.050
                     1001       1.010
'
'            MatID      P        d
 ElPlCurve    1003   -200E3    -1.000
                     -100E3    -0.010
                      100E3     0.010
                      200E3     1.000
```

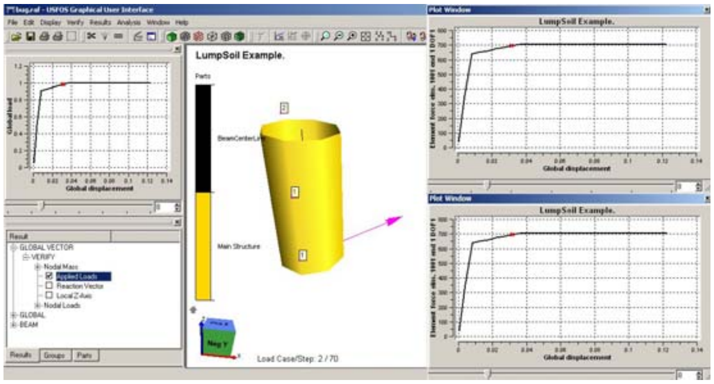


Figure 3-10 – Pipe supported by a "Lump Soil" element.

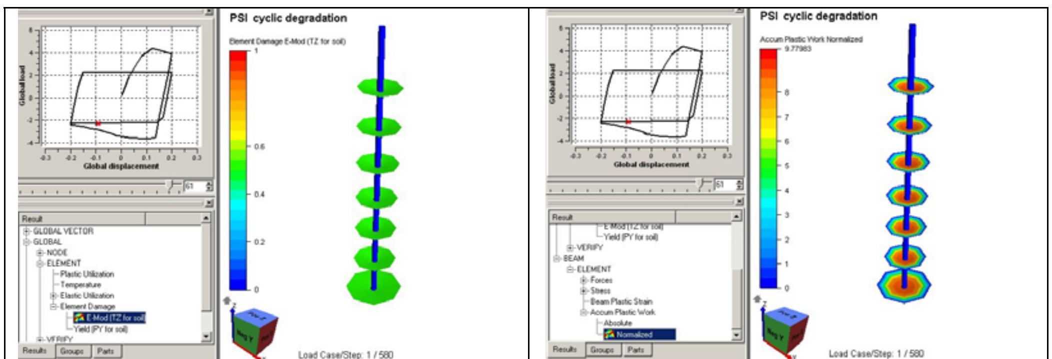
This example is found on the web under "foundation".

3.7 Soil Damage (cyclic degradation).

With the new pile option “CyclDegr” the user may define cyclic degradation of the soil with individual degradation of P-Y and T-Z. Factor 1.0 means the initial soil strength, and linear interpolation is used for the degradation vs. number of cycles. The cycles are derived from the accumulated plastic work, where one ½ cycle is defined as shown in Figure 3-12.

```
'          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
```



T-Z degradation (damage) | Accumulated plastic work

Figure 3-11 – Soil degradation as a function of accumulated plastic work.

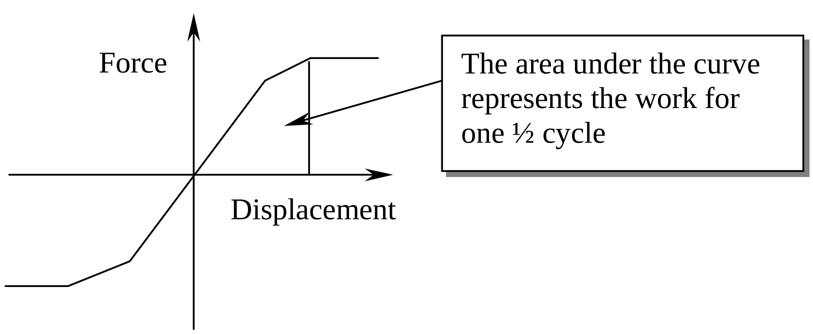


Figure 3-12 - Definition of work vs. cycle

This example is found on the web under “foundation”.

3.8 User Defined Soil Damping.

The user may define "dashpot" dampers for the different soil layer as shown below. T-Z and P-Y damping are defined to a certain ID (in the example = 100) and then assigned to the actual pile(s).

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

'            KeyWord      ID      Type       Z   Fac
 PileOpt     SoilDamp    100      T-Z        0   1E4/100
                                            -1   1E4/100
                                            -2   1E4/100
                                           -80   1E4/100

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

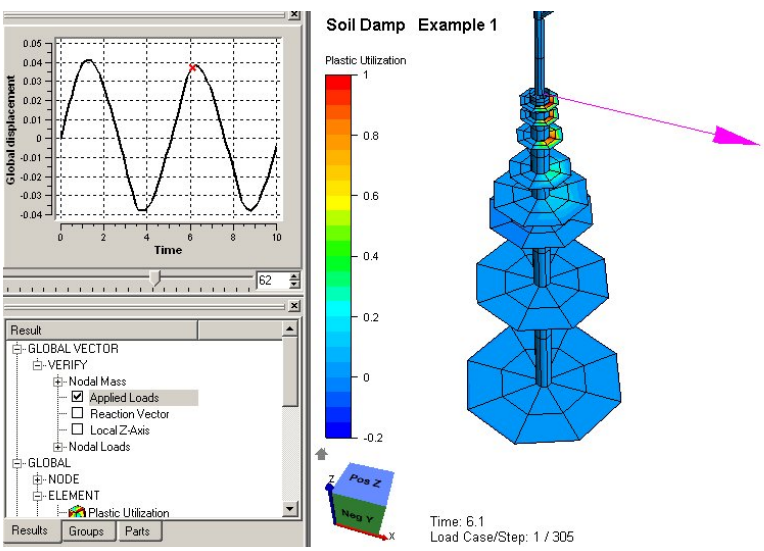


Figure 3-13 – User defined soil damping.

This example is found on the web under "foundation".

3.9 Surface Load on Pipe Sections

A conventional NODELOAD is applied on the Node. If the user wants to account for the denting of the tube wall, the new **SurfImp** load could be used.

In the example, element 1 (which goes from node 1 to 2) gets a surface impact load of 1MN in X-direction at mid-span (end-3). The extent of the impact zone is 0.1m.

```
'
'          Key      LCase  Type   ElemID   End  Extent    Fx    Fy    Fz
 SurfImp  LoadCase  3      Elem      1      3    0.1      1E6    0     0
```

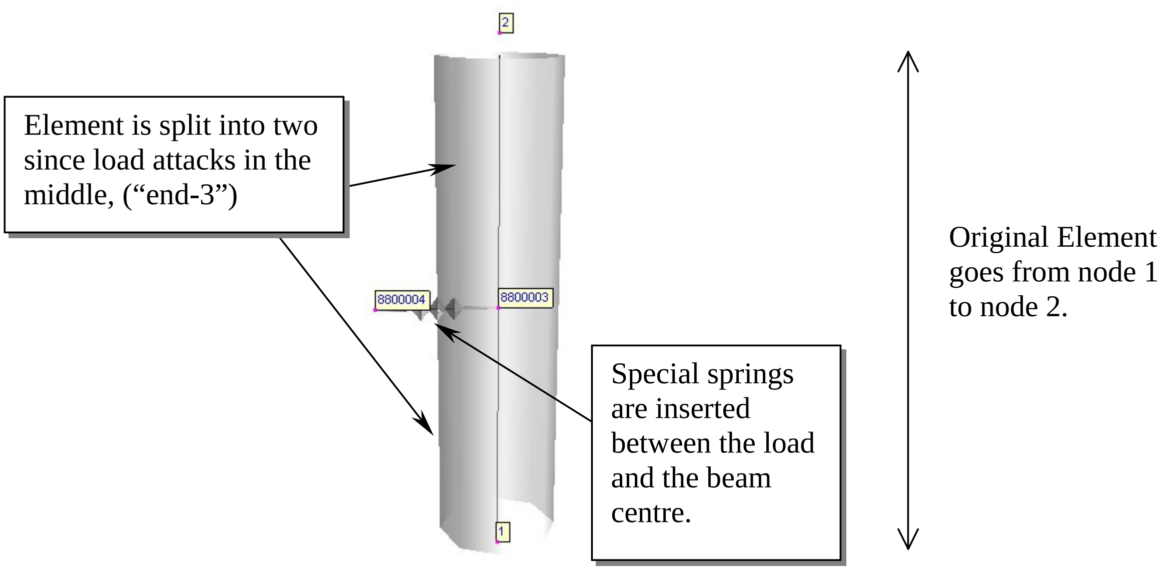


Figure 3-14 - Modified model. Extra elements are inserted automatically.

A special "attach" option makes it possible to create surface impact between different structures.

This example is found on the web under "basic loads".

3.10 Joint Options

3.10.1 Short Can Reduction

If the can is shorter than a certain length, the strength of the can is reduced. The user may either use the automatic option, where USFOS derives the parameters from the FE model, or speficy the parameters explicitly.

```
' '
 Switches   Joint  ShortCan  ON    ! Automatic ShortCan detection
'
```

Figure 3-15 – Automatic detection of Short Can Reduction parameters

```
' --------------------------------------------------------
'     Define Chord Geometries
' --------------------------------------------------------
'
'              KeyWord     Value    ListType    JointID
 JntOption     CanLength   0.400     Joint        100  110
 JntOption     CanLength   0.200     Joint         90   60
'
'              KeyWord     Value    ListType    JointID
 JntOption     CanThick    0.010     Joint        100  110
 JntOption     CanThick    0.005     Joint         60  90
'
 JntOption     CanDiam     0.500     Joint        100  110
 JntOption     CanDiam     0.150     Joint         60
'
'              KeyWord     Value    ListType    JointID    BraceID
 JntOption     CanLength   0.350    Connection     100        130
```

Figure 3-16 - Manual definition of Short Can Reduction parameters

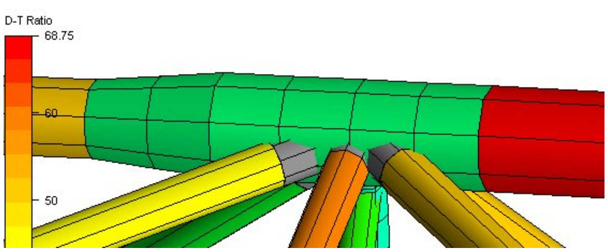


Figure 3-17 - Joint with Can

3.10.2 "Repair" eccentricities.

If the FE model has defined eccentricities in an "unfavourable" way (brace flushes the chord surface), this has negative side effects on the special joint element, which is inserted between the chord centre and the brace. The new "Switches Joint" command "EccUpd ON" will update the eccentricities are shown in Figure 3-18. The special element will go to the chord surface, where it meets the brace.

```
' '
 Switches   Joint  EccUpd  ON    ! Automatic eccentricity Repair
'
```

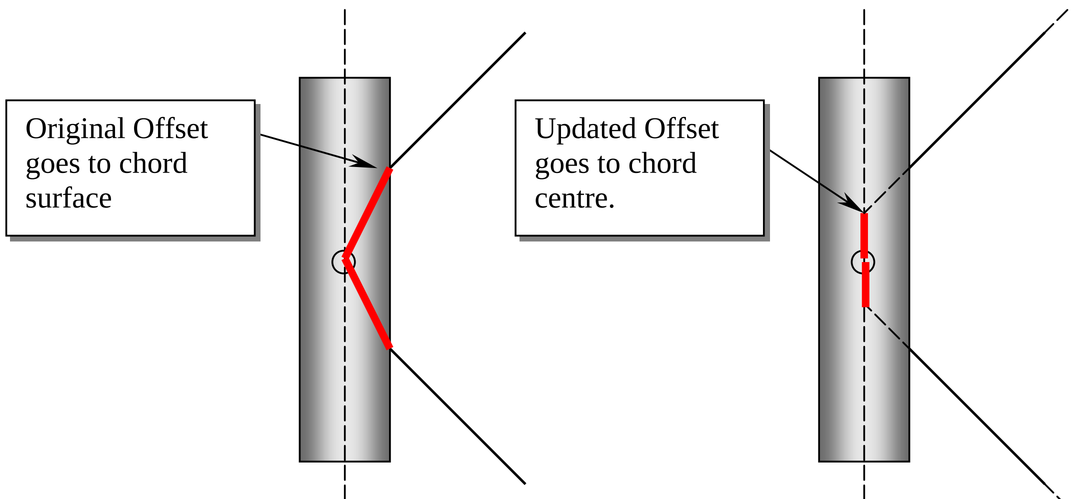


Figure 3-18 - Original and updated eccentricity

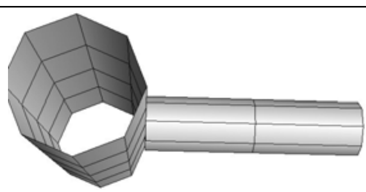
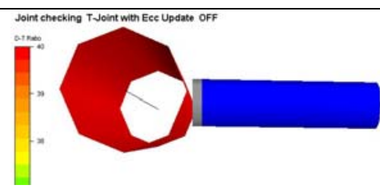
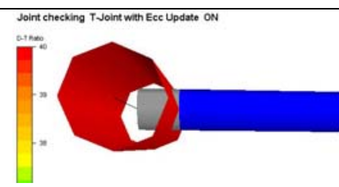
 Users Model. Brace Flushes leg surface	 Default model. Eccentricity is kept, but special element becomes short.	 New model. Eccentricity is moved into chord axis. Special element gets sufficient length and original brace flushes the chord surface

Figure 3-19 – T-joint with joint model. Default and new handling of offsets.

This example is found on the web under Joints.

3.10.3 Local Shell model (SubShell).

If the user wants to represent a beam with shells, the new “DumpFEM” option will generate a shell model for the selected element. The shell model contains the followings:

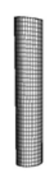
- Shell elements and properties derived from the original beam element
- Transition from shell to beam axis
- Original beam is set “NonStru”.

Such analyses have two steps:

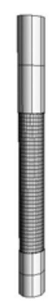
1. Generate the local shell model using the “SubShell” command
2. Include the generated shell model (for example using the “opt” input file)

```
'              ID   KeyWord
 SUBSHELL       2   DumpFEM
'
'
'            nLeng ncirc
 MESHPIPE     36    36 '
'
```

Users Model.



Generated shell model.



Assembled model

Figure 3-20 – Local Shell model.

3.10.4 Element degradation ("damage")

The user may define different ways to degrade the strength of a beam element. The Damage command has several options:

- After a certain load case (static)
- According to a time history (dynamic)
- As a function of accumulated plastic work (normalized).

The example shows the input to the "PlastWork" option, where two general material curves are used to define the degradation for E-mod and Yield. For Plastic work less than W1, no damage is applied, and is kept constant for work > W2.

```
'
'          Type      DamE  DamY  ListTyp  Ids
 Damage  PlastWork  101   102    Mat      1
'
'          MatID   Type    Curv     W1  W2    Fac
 Material 101   General  S_Curv  0.1 0.5  0.10  ! E-Mod
 Material 102   General  S_Curv  0.1 0.5  0.90  ! Yield
```

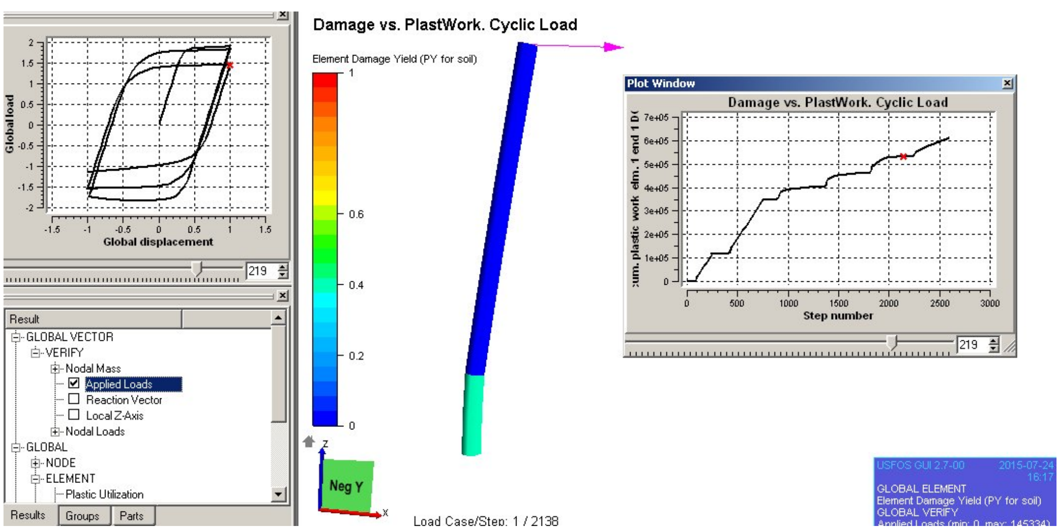


Figure 3-21 - Degradation of Yield strength as a function of plastic work.

3.11 Beamhing -> Linear Bearing

Beam hinges are by default handled using “static condensing” of the internal forces. Alternatively, the hinge could be represented by one extra “bearing” element and one extra node per beam end with hinge. These extra nodes and elements are created automatically if the Switches command shown below is defined.

It is also possible to give the released degrees of freedom some elastic stiffness (the “release” option). The default is zero stiffness for the hinge degrees of freedom.

The “fixed” (non-hinged) degrees of freedom are given a high stiffens (derived from the actual beam element’s stiffness), but the user may specify this stiffness (the HingStiff option).

```
'          <  end 1   >   <  end 2   >  ElemID
 BeamHing  1 1 1  1 0 0   1 1 1  1 0 0    1

'              key1       Key2        opt
 Switches   FE_Model     Hing2Elm    ON
 Switches   FE_Model     Hing2Elm    HingStiff   1E9
 Switches   FE_Model     Hing2Elm    Release     1E3
 Switches   FE_Model     Hing2Elm    IdAdd       7700000
```

In the example, one extra node and one extra element are inserted in both ends of beam element 1.

The user may control the node- and element IDs, (the “IdAdd” option). By default, the number 7700000 is added to the generated nodes and elements.

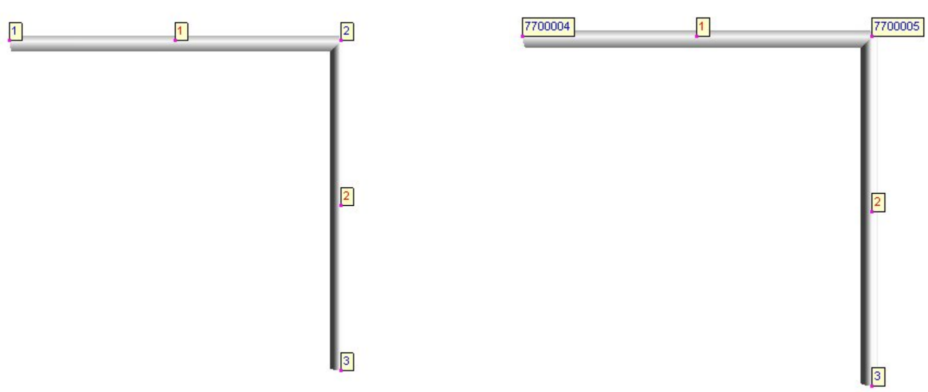


Figure 3-22 - Original model (left) and modified (right).

3.12 SWITCHES, (Special Options).

The command "SWITCHES" was introduced in 8-5 to switch on special options and is extended in version 8-8. 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	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
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

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:

http://www.usfos.no/news/index.html

3.14 New/modified input commands

Since last main release (8-7), following input identifiers are added/extended:

DAMAGE	:	New command	: Defines reduced capacity / gradual fracture
PILEMAT	:	New command	: Defining different pile material along pile
SPRITYPE	:	New command	: LumpSoil
SURFIMP	:	New command	: Load attacking surface of a pipe
DYNIMPCT	:	Extended command	*: Material Curve directly for ship.*
JNTOPTION	:	Extended command	*: Short Can reduction.*
PILEOPT	:	Extended command	*: Cyclic Degradation, Soil Damping*
PILEGEO	:	Extended command	*: Change cross section type for pile*
SUBSHELL	:	Extended command	*: Dump of FE-mesh.*
SWITCHES	:	Extended command	*: See above.*

3.15 Documentation

The following documentation, (updated or new), is available on the web:

- User's manual : Updated document
- Examples : New examples on the web