

 USFOS AS Phone: +47 905 05 717 www.USFOS.com Enterprise No.: NO 986 827 374 MVA		MEMO										
		Release Notes USFOS Version 8-8			FOR YOUR ATTENTION	COMMENTS ARE INVITED	FOR YOUR INFORMATION	AS AGREED				
					DISTRIBUTION Members of USFOS user group							X
FILE CODE	CLASSIFICATION Confidential											
REFERENCE NO.												
PROJECT NO.	DATE 2015-11-01	PERSON RESPONSIBLE / AUTHOR Tore Holmas			NUMBER OF PAGES 24							

Release Notes

USFOS 8-8, Nov 2015

1	INTRODUCTION	3
2	CHANGES IN VERSION 8-8	3
3	NEWS IN USFOS VERSION 8-8 - 2015.	4
3.1	INTRODUCTION	4
3.2	HOW TO INSTALL/ UPGRADE YOUR USFOS VERSION	4
3.2.1	Windows (64bit)	4
3.2.2	Windows (32bit)	4
3.2.3	LINUX	5
3.2.4	MAC-OSX	5
3.3	ENHANCED GRAPHICAL USER INTERFACE	6
3.3.1	Updated Preferences. <i>NOTE! Remembers Fringe Range.</i>	6
3.3.2	Visualization of D-T ratio for Pipes	6
3.3.3	Verify Slenderness of I-Profiles	7
3.3.4	Visualization of NonStru and Fracture elements	9
3.3.5	Visualization of Soil Strength	10
3.3.6	Visualization of Absolute Displacements	10
3.4	PILE MATERIAL	11
3.5	PILE CROSS SECTIONS.	12
3.6	“LUMPED” SOIL.	13
3.7	SOIL DAMAGE (CYCLIC DEGRADATION).	14
3.8	USER DEFINED SOIL DAMPING.	15
3.9	SURFACE LOAD ON PIPE SECTIONS	16
3.10	JOINT OPTIONS	17
3.10.1	Short Can Reduction	17
3.10.2	“Repair” eccentricities.	18
3.10.3	Local Shell model (SubShell)	19
3.10.4	Element degradation (“damage”)	20
3.11	BEAMING -> LINEAR BEARING	21
3.12	SWITCHES, (SPECIAL OPTIONS).	22
3.13	UPDATES USFOS AND UTILITY TOOLS	24
3.14	NEW/MODIFIED INPUT COMMANDS	24
3.15	DOCUMENTATION	24

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:

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

- o 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.
- o 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):

- | | |
|---|----------------------------|
| 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.

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.

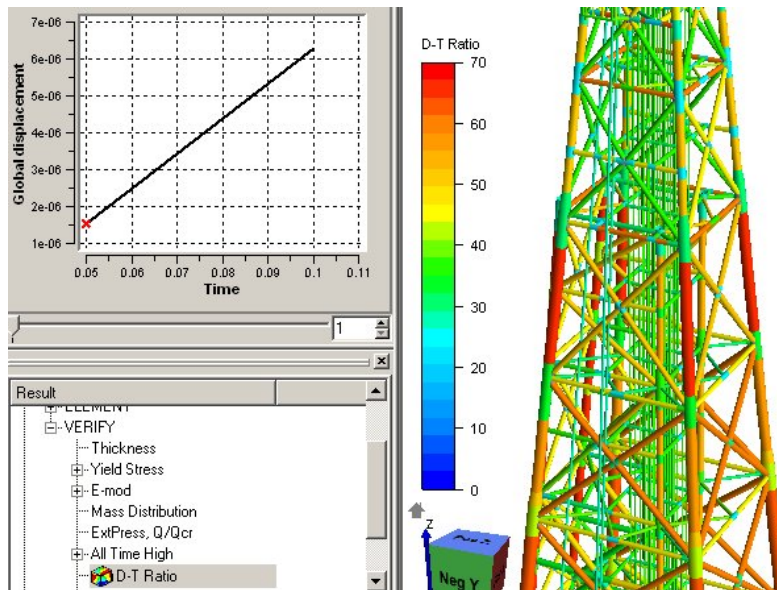


Figure 3-1 - Visualization of D-T ratio of pipes

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:

- o Yellow to Red ($\lambda > 0.67$) : 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.

- o 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

- o 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

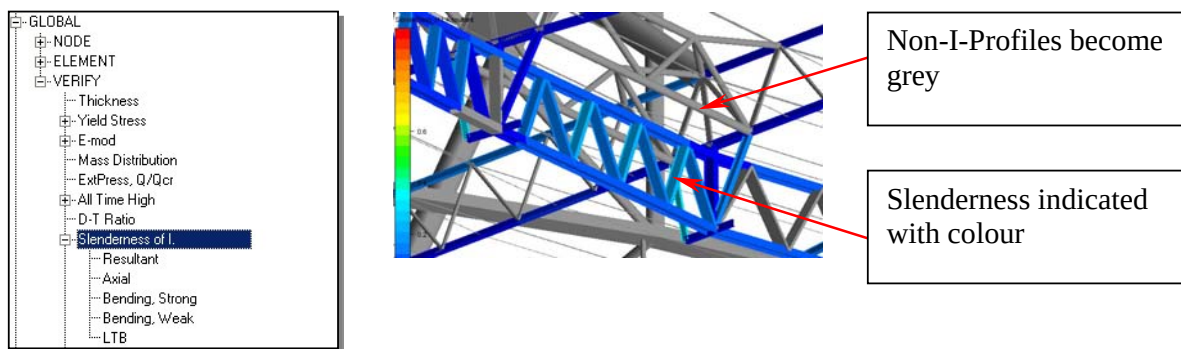


Figure 3-2 - Global - Verify - Slenderness of I

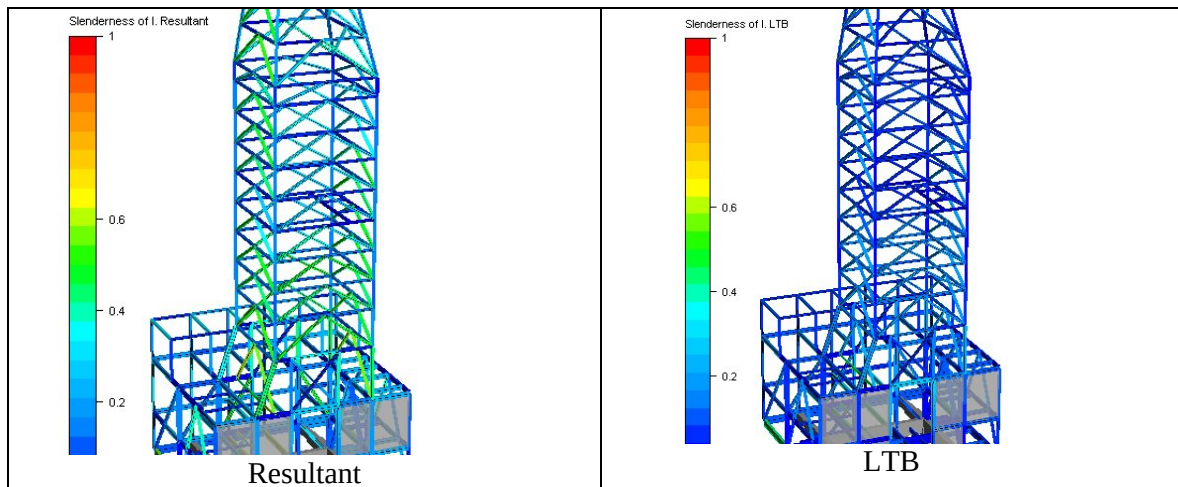


Figure 3-3 – Different “Slenderness”

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.

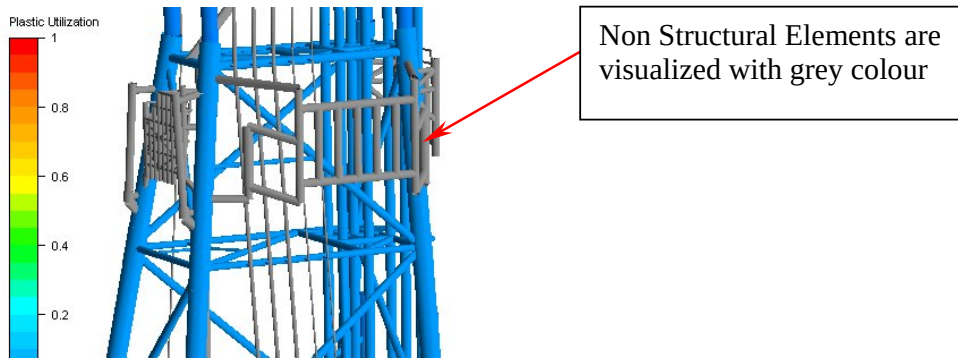


Figure 3-4 – Visualization of Plastic Utilization / NonStru elements.

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.

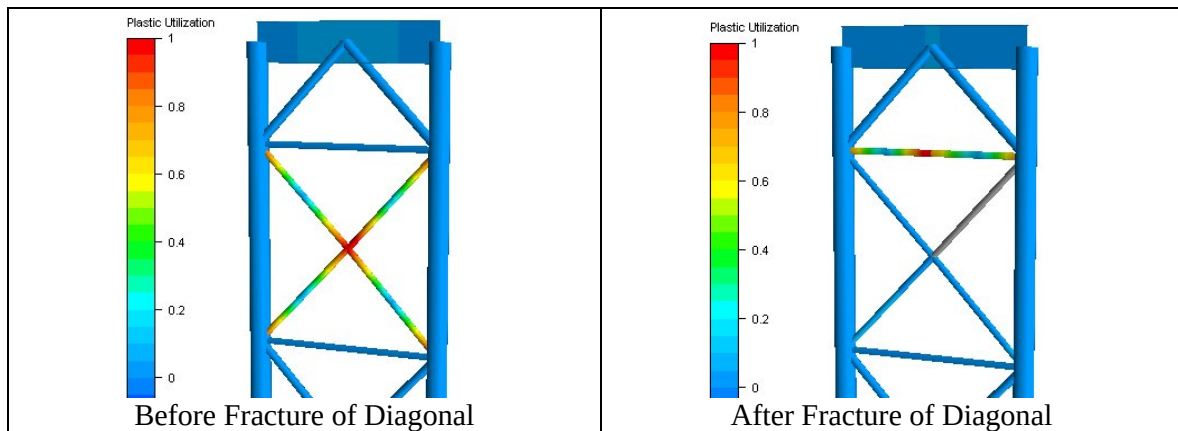


Figure 3-5 – Fracture elements become grey.

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 weighed 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

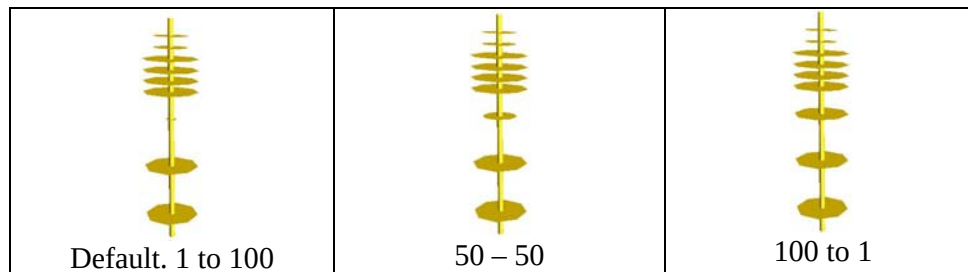


Figure 3-6 - Disc Size for three different weights between P-Y and T-Z.

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.

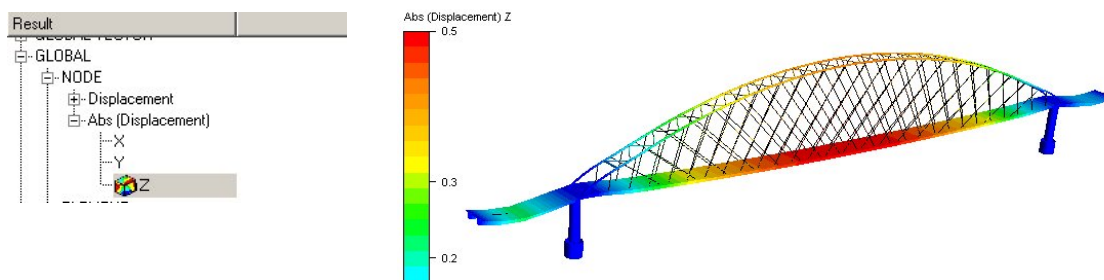


Figure 3-7 – Visualization of Absolute Z-displacement

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
FileMat	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

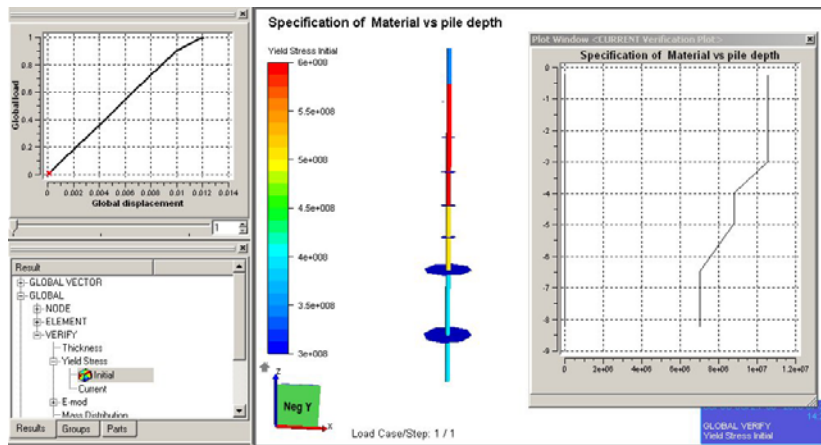


Figure 3-8 - Varying Yield stress along the pipe.

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	

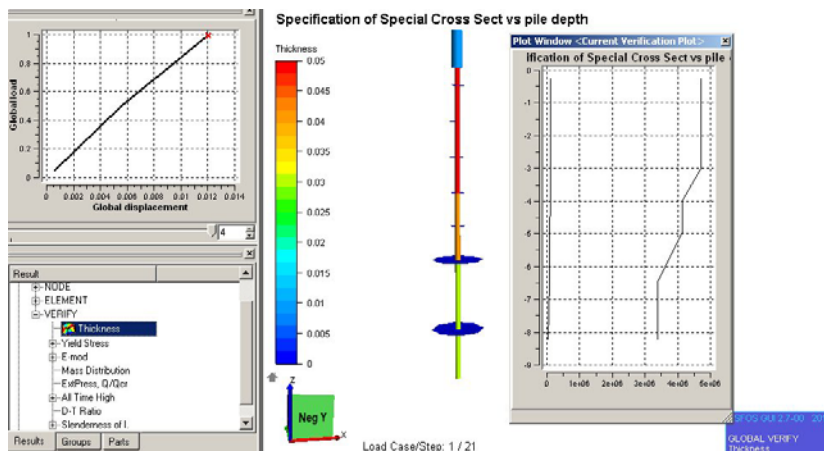


Figure 3-9 - Varying Pile Cross Section along the pile.

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:

- o DOF-1 : P-Y curve
- o DOF-2 : P-Y curve (same curve as for DOF-1)
- o DOF-3 : T-Z curve

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

```

SprType    LumpSoil    Elem 1001

Sprng2Gr   ID      Node  Mat
1001      1001      1    1000

MREF       1000      1    2    3    rX    rY    rZ
          P-Y      P-Y  T-Z
          1001    1001  1003  0      0      0

ElPlCurve   MatID    P      d
1001      1001    -1001   -1.050
          -1000   -0.050
          -900    -0.010
          900     0.010
          1000    0.050
          1001    1.010

ElPlCurve   MatID    P      d
1003      1003   -200E3   -1.000
          -100E3   -0.010
          100E3    0.010
          200E3    1.000
  
```

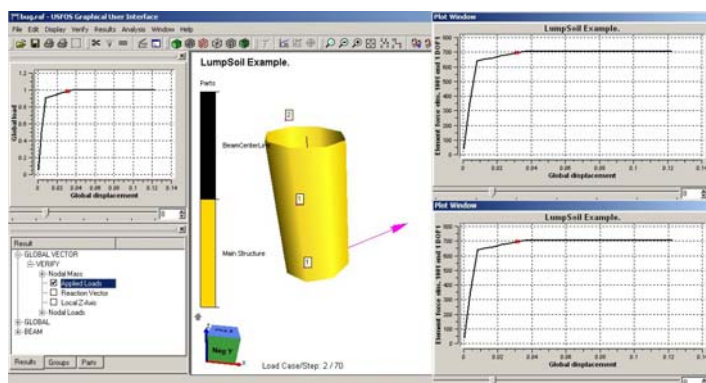


Figure 3-10 – Pipe supported by a “Lump Soil” element.

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	

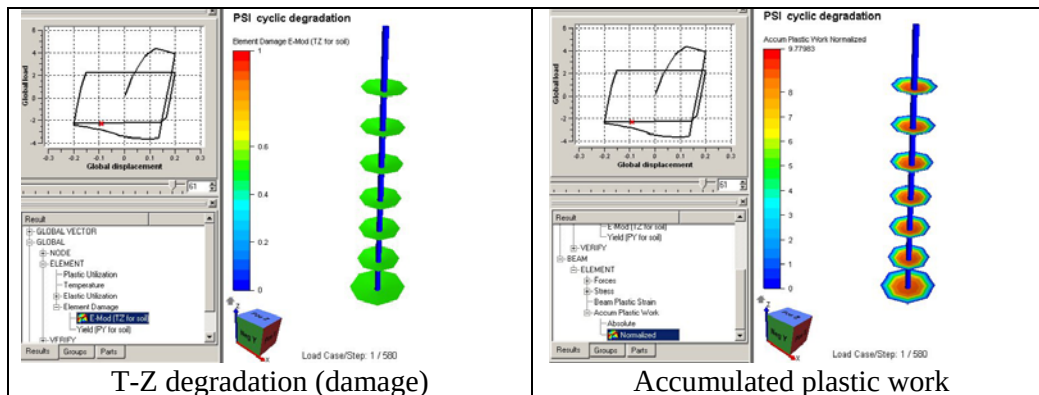


Figure 3-11 – Soil degradation as a function of accumulated plastic work.

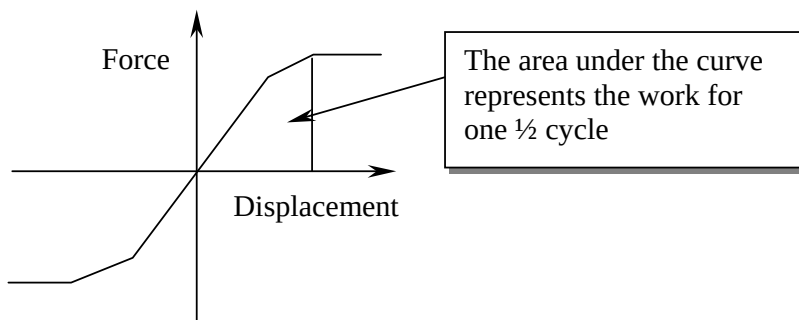


Figure 3-12 - Definition of work vs. cycle

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					
PileOpt	Keyword	ID	Type	Z	Fac
	SoilDamp	100	P-Y	0	1E4/100
				-1	1E4/100
				-2	1E4/100
				-80	1E4/100
PileOpt	Keyword	ID	Type	Z	Fac
	SoilDamp	100	T-Z	0	1E4/100
				-1	1E4/100
				-2	1E4/100
				-80	1E4/100
PileOpt	Keyword	ID	Type	PileID	
	SoilDamp	100	Assign	1001	

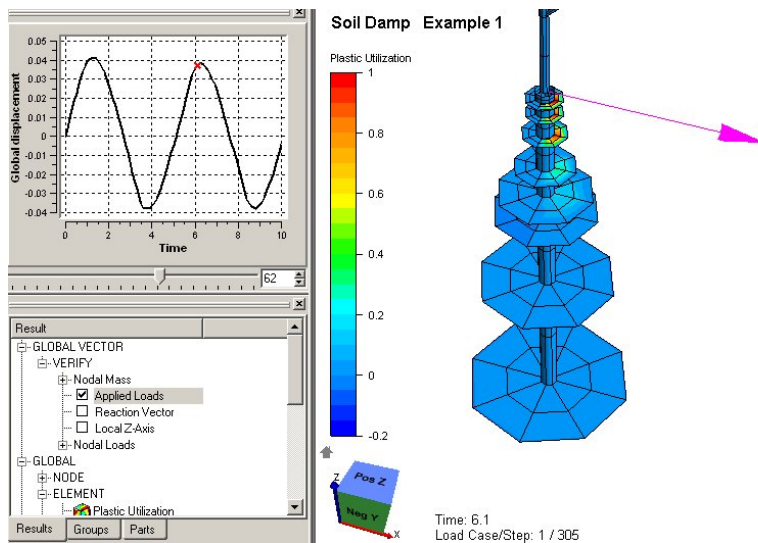


Figure 3-13 – User defined soil damping.

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

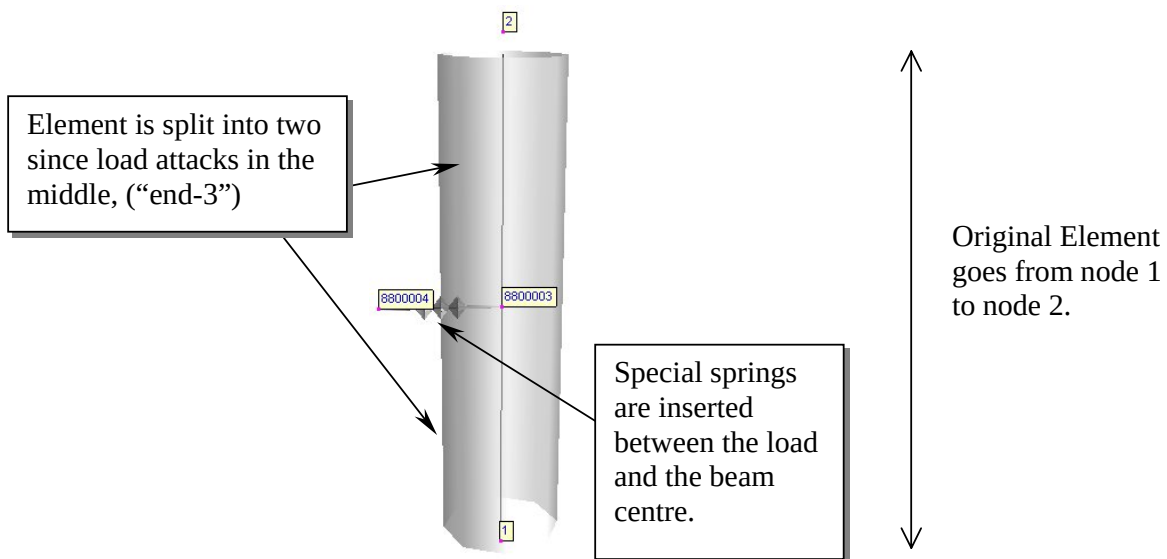


Figure 3-14 - Modified model. Extra elements are inserted automatically.

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 specify the parameters explicitly.

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

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

```
'
-----
Define Chord Geometries
-----
'
JntOption  Keyword  Value  ListType  JointID
JntOption  CanLength  0.400  Joint     100 110
JntOption  CanLength  0.200  Joint     90 60
'
JntOption  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
'
JntOption  Keyword  Value  ListType  JointID  BraceID
JntOption  CanLength  0.350  Connection 100      130
'
```

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

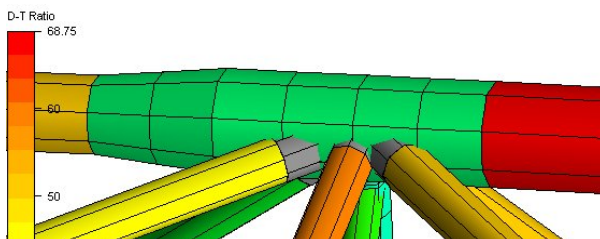


Figure 3-17 - Joint with Can

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.

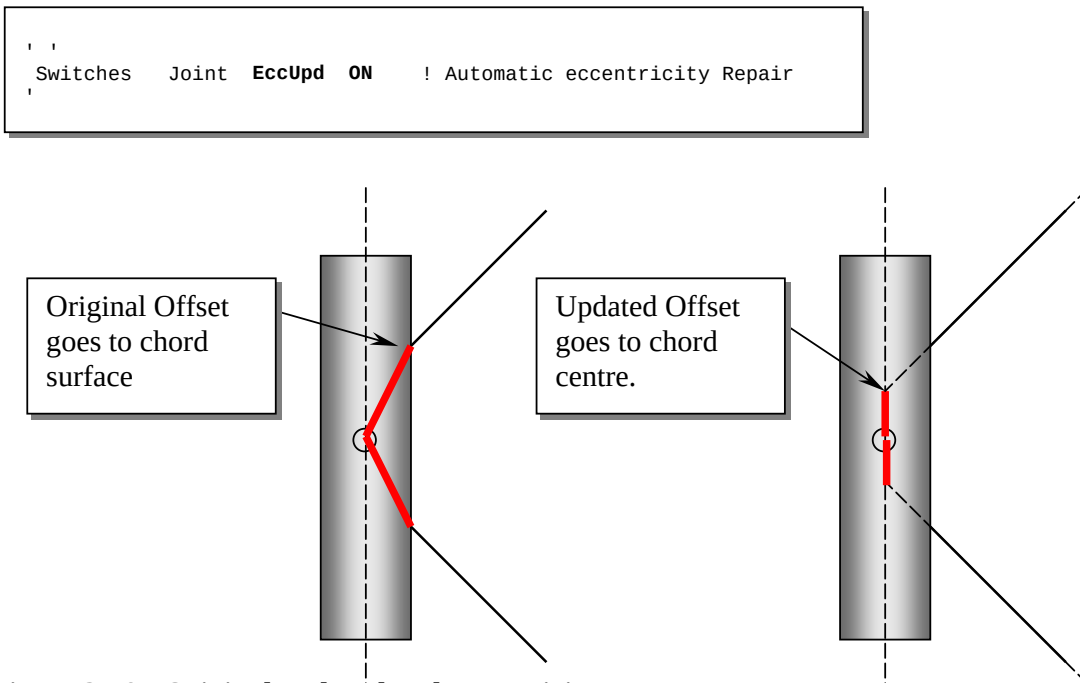


Figure 3-18 - Original and updated eccentricity

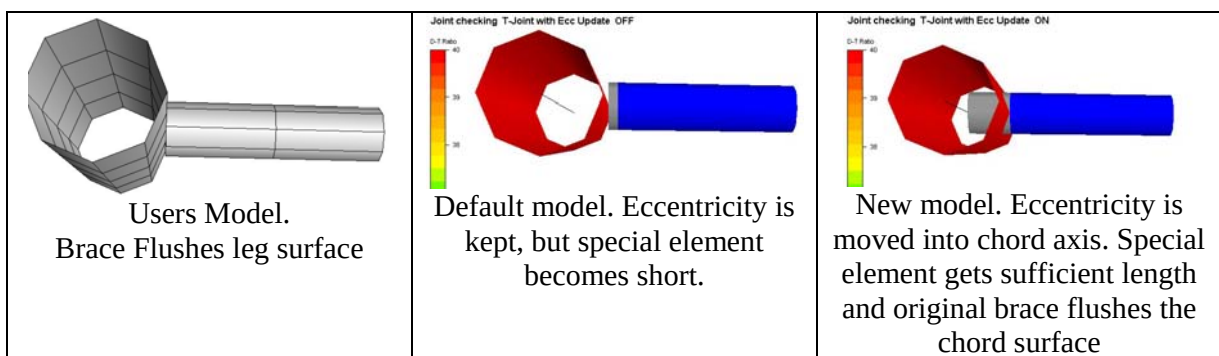


Figure 3-19 – T-joint with joint model. Default and new handling of offsets.

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:

- o Shell elements and properties derived from the original beam element
- o Transition from shell to beam axis
- o 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 '
  
```

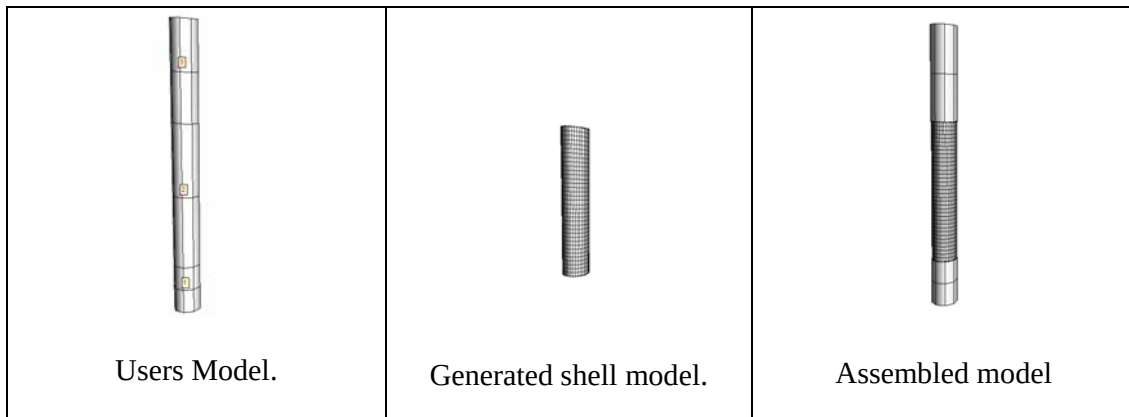


Figure 3-20 – Local Shell model.

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:

- o After a certain load case (static)
- o According to a time history (dynamic)
- o 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

```

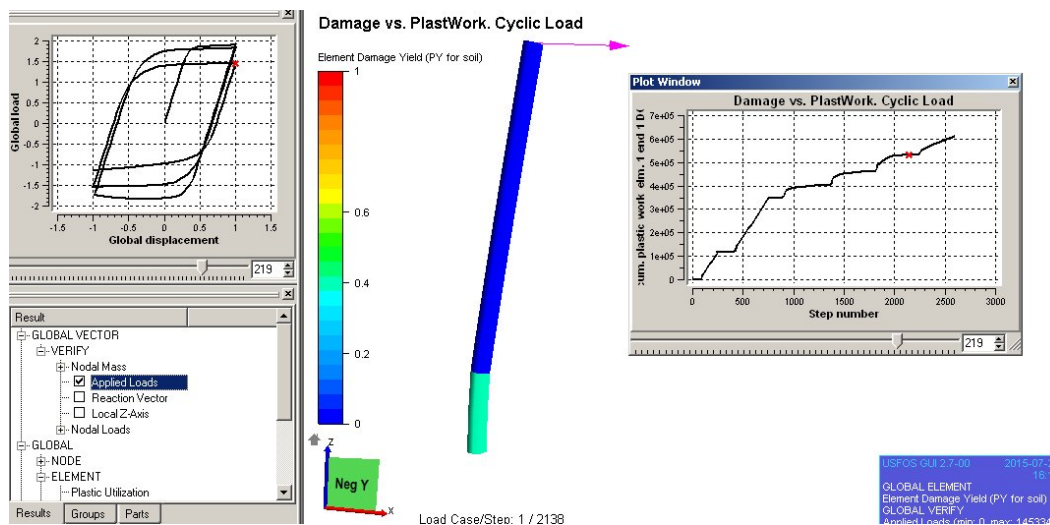


Figure 3-21 - Degradation of Yield strength as a function of plastic work.

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 >				
BeamHing					ElemID				
1	1	1	1	0	0	1	1	1	1
'									
Switches		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.

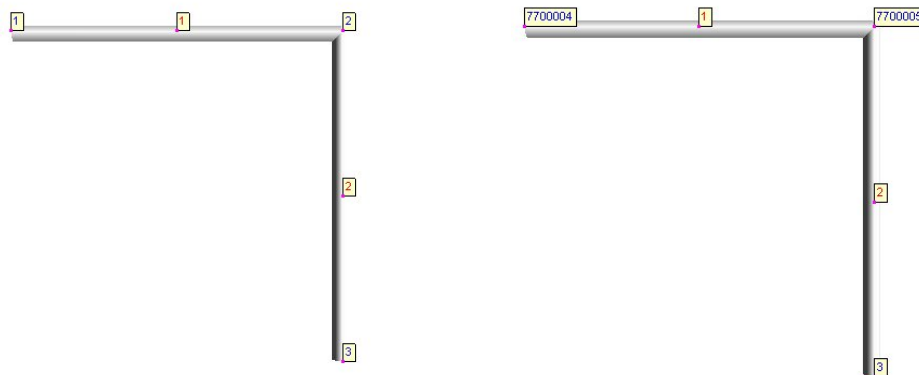


Figure 3-22 - Original model (left) and modified (right).

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	<i>IndefLimit</i>		Min / Max imperfection (in CINIDEF).	0.05 / 1%
Defaults	<i>Version</i>	ver	850: switch to version 8-5 defaults	870
WaveData	<i>TimeInc</i>	val	Time between each hydrodyn calc.	Every
	<i>NoDoppler</i>	-	Switches OFF Doppler effects.	ON
	<i>NoStore</i>		Switches OFF storing of wave data for visualize.	ON
	<i>TidalLevel</i>	Level	Specify Tidal Level	0
	<i>Accuracy</i>	val	Change accuracy. 0: old accur, 1: new accur	1
	<i>SeaDim</i>	X , Y-dim	Specify size of sea surface used in xact	2 λ
	StreamOrd	order	Stream Function order	10
NodeData	<i>DoublyDef</i>	ON/OFF	ON: Accept doubly defined nodes with same coo	OFF
StatusPrint	<i>MaxElem</i>	val	Max element in status print	10
Iterations	<i>RLF_Calc</i>	-	Activate “Residual Load Factor” method	OFF
Write	<i>FE_Model</i>	IDAdd Case stp	Writes deformed FE model at given case stp	OFF
	<i>LinDepAlt</i>	-	Writes ZL-springs for each BLINDP2	Off
Solution	<i>FracRepeat</i>	MxRep	Max fracture repeat	10
	<i>PlateEdge</i>	ON/OFF	Avoiding I-girder to buckle about weak axis if the beam element is attached to a plate element	OFF
	<i>Impact</i>	<i>UnLoFact</i>	Load factor during unloading after boat impact	0.02
StrainCalc	<i>InclDent</i>	ON/OFF	OFF: not included. ON: included	ON
	<i>Algorithm</i>	Val	0: old. 2: new, incremental.	2
	<i>Visualization</i>	ON/OFF	Including Gradients. ON/OFF	ON
Results	<i>ShellComp</i>	Val	Number of shell results	5
	Overturn	Val	Specify X Y Z for overturn moment calculation	Estim.
WindData	<i>ReynDep</i>	ON/OFF	Switch to Reynolds-number dependent Cd	OFF
EarthQuake	<i>Delay</i>	Val	Delays earthquake with specified time	0
	<i>Stretch</i>	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	: <i>Material Curve directly for ship.</i>
JNTOPTION	:	Extended command	: <i>Short Can reduction.</i>
PILEOPT	:	Extended command	: <i>Cyclic Degradation, Soil Damping</i>
PILEGEO	:	Extended command	: <i>Change cross section type for pile</i>
SUBSHELL	:	Extended command	: <i>Dump of FE-mesh.</i>
SWITCHES	:	Extended command	: <i>See above.</i>

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