<table>
<tr><td colspan="2" rowspan="2">USFOS
Reality Engineering

USFOS AS

Phone: +47 905 05 717
www.USFOS.com
Enterprise No.: NO 986 827 374 MVA</td><td colspan="5">MEMO</td></tr>
<tr><td>MEMO CONCERNS
Release Notes
USFOS Version 8-6</td><td>FOR YOUR ATTENTION</td><td>COMMENTS ARE INVITED</td><td>FOR YOUR INFORMATION</td><td>AS AGREED</td></tr>
<tr><td>FILE CODE</td><td>CLASSIFICATION
Confidential</td><td rowspan="2">DISTRIBUTION
Members of USFOS user group</td><td rowspan="2"></td><td rowspan="2"></td><td rowspan="2"></td><td rowspan="2">X</td></tr>
<tr><td colspan="2">REFERENCE NO.</td></tr>
<tr><td>PROJECT NO.</td><td>DATE
2012-01-01</td><td>PERSON RESPONSIBLE / AUTHOR
Tore Holmas</td><td colspan="4">NUMBER OF PAGES
32</td></tr>
</table>

Release Notes

USFOS 8-6, Jan 2012

1 Introduction

The current official version of USFOS is version 8-6 with release date 2012-01-01. The release contains the following:

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

NOTE, USFOS 8-6 comes with some important changes. Please read the section:

Important Changes in version 8-6

2 Important Changes in version 8-6

In order to improve the quality of the analysis results, several important changes are made. The changes could be split into two main groups:

1. More strict checking of the finite element model
2. New defaults and accuracy parameters

2.1 Finite Element model

2.1.1 Restrictions on element length to diameter ratio

The USFOS beam element is designed for being able to describe the behaviour of the physical member with only one element. Using very short element could cause unnecessary numerical problems and should be avoided. Element lengths as shown in Figure 2-1 is not recommended.

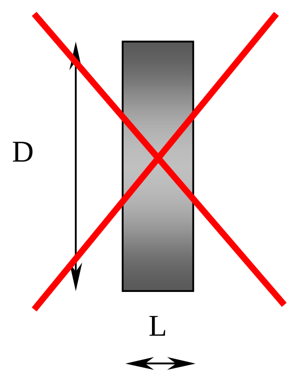

Figure 2-1 Illegal small Length-to-diameter ratio. Standard Beam element

Extreme **small** element-length –to – diameter ratios is checked for.

The ratio: **L / D** (*D is set to height of I- and Box sections*)

is computed for all standard beam elements (not special). If the ratio is less than 0.5, USFOS will report this as an illegal short element and stop with an error message.

Following information are given:

- The elements are printed in the "out" file (see Figure 2-5)
- The USFOS label file: *"prefix"_illegal_short_elements.usl*" could be opened in xact and the illegal elements are shown. (see Figure 2-6)

The best option is always to modify the structural model and remove unnecessary sub-divisions of members, (or even better: create a good model from the beginning). However, if the user decides to keep the short elements the "**`ILLEGAL`**" command could be used to bypass the check.

Figure 2-2 describes the recommended "bypassing": Specification one-by-one. This option means that the user has an overview over important and less important elements, (which f ex remain elastic and therefore are not so vulnerable for short length).

The option works as follows:

- A new minimum L/D ration is defined by the user
- The element to accept are listed

The minimum L/D could be re-defined several times. The commands are executed in the specified sequence.

```
'         Key-1      Opt      Value
Illegal BeamLength  Accept   0.3      ! Redefine Min L/D

'         Key-1      Opt   IDs
Illegal BeamLength  Elem   1001  1002 ! Accept elem 1001 and 1002
                           2001  2002 ! Accept elem 2001 and 2002
                           3001  3002 ! Accept elem 3001 and 3002


'         Key-1      Opt      Value
Illegal BeamLength  Accept   0.1      ! Redefine Min L/D

'         Key-1      Opt   IDs
Illegal BeamLength  Elem   7001  7002 ! Accept elem 7001 and 7002
```

Figure 2-2 Accepting short elements. Specification element-by-element

The "lazy" (not recommended) version is to accept all short elements without any specification. A warning will be printed in the output file, (see Figure 2-4).

```
'         Key-1      Opt      Value
Illegal BeamLength  Accept   0.001   ! Redefine Min L/D

'         Key-1      Opt
Illegal BeamLength   UsersRisk  ON   ! Accepting everything unchecked
```

Figure 2-3 Accepting short elements without element specification

```
---------------------------------------------------
--     * * *    W A R N I N G   * * *          --
--      Unconditionally Acceptance of          --
--     Very Short Beams on Users own Risk     --
---------------------------------------------------
```

Figure 2-4 The warning is printed in the output file.

In the output file (.out), the results from element checking are printed. If illegal short elements are found, the L/D is printed. The shortest element is printed, which in this example has a L/D ratio of less than 2%.

```
----    C H E C K I N G   B E A M - E L E M E N T   L E N G T H -----

        Minimum Length/Diam  for struct elem   :     0.500
        Minimum Length/Diam  for pile   elem   :     1.000
        Maximum Ecc/Length ratio               :     0.500
        Model to use for short elements        :         0


        ----------------------------------
        .... Checking Structural Elements
        ----------------------------------
        Illegal short struct element           :    605163 , L/D =   0.20
        Illegal short struct element           :    108100 , L/D =   0.17
        Illegal short struct element           :    118100 , L/D =   0.02
        Illegal short struct element           :    403458 , L/D =   0.30

         ......
         ......

        Illegal short struct element           :    403642 , L/D =   0.30
        Illegal short struct element           :    503200 , L/D =   0.40
        Illegal short struct element           :    503295 , L/D =   0.40
        Illegal short struct element           :    503456 , L/D =   0.40
        Illegal short struct element           :    503636 , L/D =   0.40

        Checking of struct elements completed.
        Smallest L/D found                     :     0.01706
```

Figure 2-5 The Errors and list of illegal elements are printed in the output file.

The “Label file” describes where the illegal elements are located, and the Illegal Length to Diameter ratio is printed for each element.

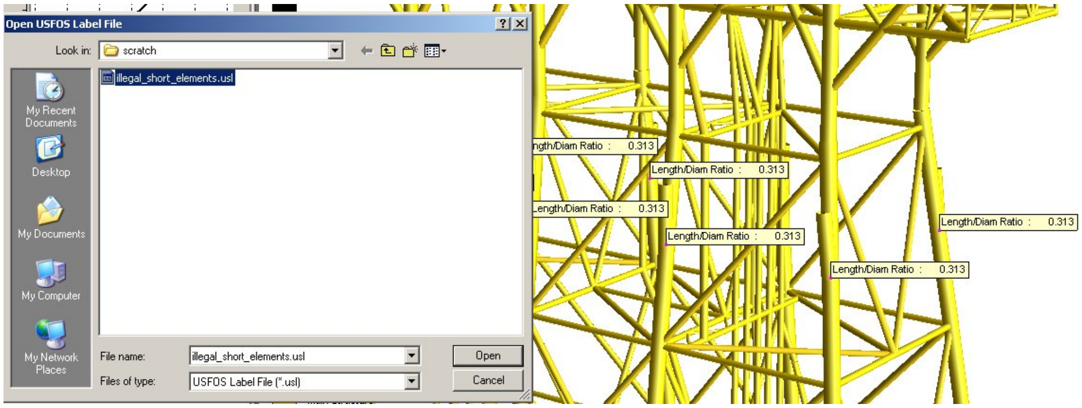

Figure 2-6 The short elements are documented in the Label File.

2.1.2 Restrictions on eccentricity to element length ratio

Another source for numerical inaccuracies/problems is using extreme eccentricities. Figure 2-7 describes an element, where the eccentricities are very big compared with the length of the element (the flexible part).

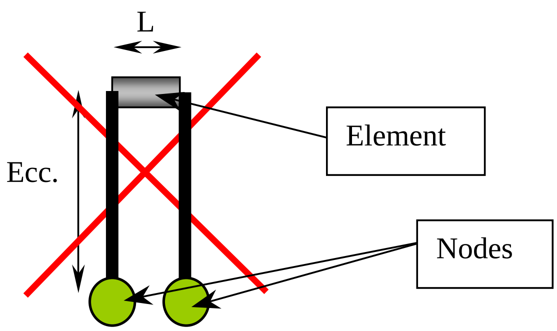

Figure 2-7 Illegal big Eccentricity-to-BeamLength ratio

Extreme **big** eccentricity –to – element lengths are checked for.

The ratio: **(e1+e2) / L** (*e1+e2 is the sum of eccentricities*)

is computed for all standard beam elements. If the ratio exceeds **0.5**, USFOS will report this as an illegal big eccentricity and stop with an error message.

Following information are given:

- The elements are printed in the “out” file, (see Figure 2-11)
- The USFOS label file: *“prefix”_illegal_big_eccentricities.usl*” could be opened in xact and the illegal elements are shown. (see Figure 2-12)

```
'          Key-1      Opt      Value
Illegal Eccentricity  Accept   3      ! Redefine Max e/L

'          Key-1      Opt   IDs
Illegal Eccentricity Elem  1001  1002 ! Accept elem 1001 and 1002
                           2001  2002 ! Accept elem 2001 and 2002
                           3001  3002 ! Accept elem 3001 and 3002


'          Key-1      Opt      Value
Illegal Eccentricity Accept   10      ! Redefine Max e/L

'          Key-1      Opt   IDs
Illegal Eccentricity Elem  7001  7002 ! Accept elem 7001 and 7002
```

Figure 2-8 Accepting big eccentricities elements. Specification element-by-element

The “lazy” (not recommended) version is to accept elements with big eccentricities without any specification. A warning will be printed in the output file, (see Figure 2-10).

```
'          Key-1      Opt      Value
Illegal Eccentricities Accept  10   ! Redefine Max e/L

'          Key-1      Opt
Illegal Eccentricities UsersRisk ON ! Accepting everything unchecked
```

Figure 2-9 Accepting big eccentricities without element specification

```
        ------------------------------------------------
        --    * * *   W A R N I N G   * * *         --
        --      Unconditionally Acceptance of         --
        --  Very  big  E/L ratios on Users own Risk   --
        ------------------------------------------------
```

Figure 2-10 The warning is printed in the output file.

```
----------------------------------
        .... Checking Eccentricities
        ----------------------------------
        Illegal big E/L   ratio for element    :    501110 , E/L =   0.75
        Illegal big E/L   ratio for element    :    505160 , E/L =   1.22
        Illegal big E/L   ratio for element    :    503190 , E/L =   0.78
        Illegal big E/L   ratio for element    :    425822 , E/L =   4.61

        .......
        .......

        Illegal big E/L   ratio for element    :    435822 , E/L =   4.61
        Illegal big E/L   ratio for element    :    445822 , E/L =   4.61
        Illegal big E/L   ratio for element    :    515822 , E/L =   4.61

        Checking of eccentricities completed.
        Biggest  E/L found                     :    14.82887
```

Figure 2-11 The Errors and list of illegal elements are printed in the output file.

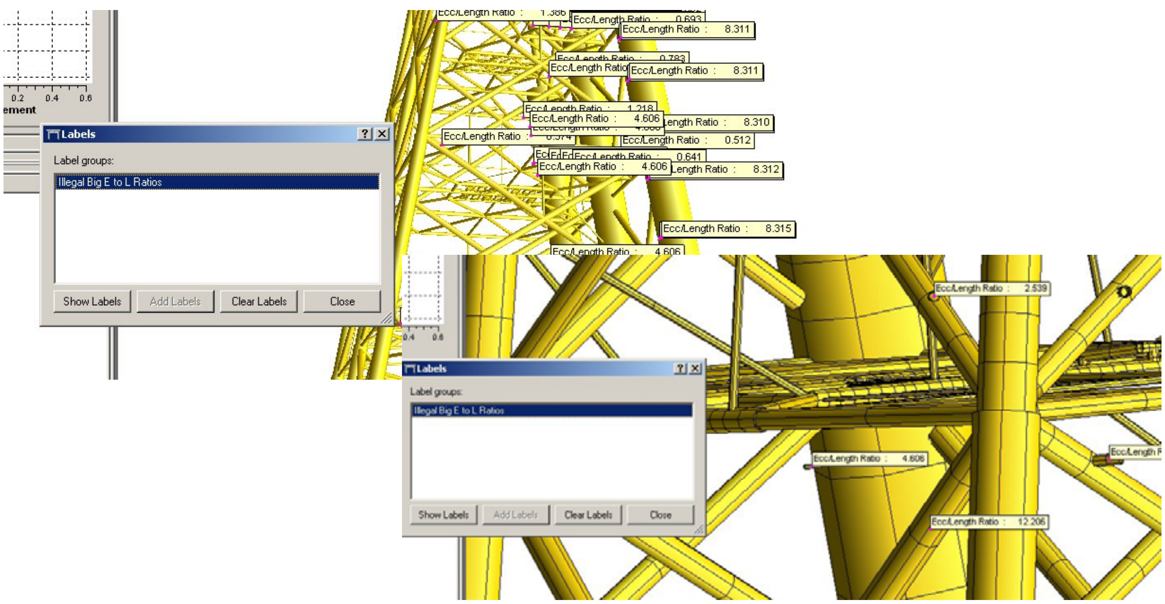

Figure 2-12 Elements with illegal big eccentricities are shown

2.1.3 Restrictions on Soil thickness to Pile diameter ratio

In order to ensure that the finite element model does not contain very short elements, the pile-soil model is checked for “too-thin-soil-layers”.

In a normal pile-soil model, the soil thickness is typically 3-10 times the pile diameter.

If the ratio is less than 1.0, USFOS will report this as an illegal thin soil layer and stop with an error message. The piles are printed in the “out” file.

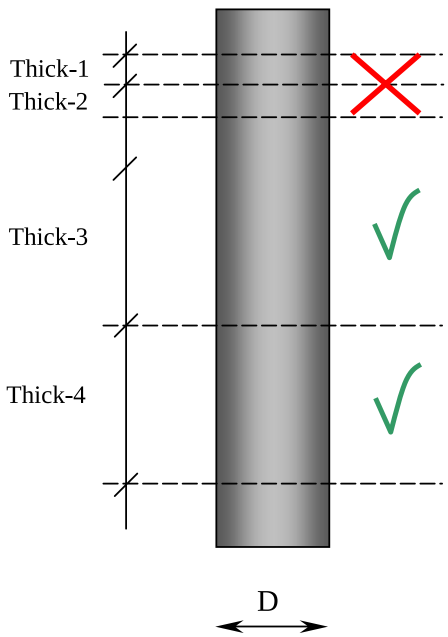

Figure 2-13 Illegal small Soil thickness to pile diameter ratio

The minimum thick/diameter ratio could be re-defined several times. The commands are executed in the specified sequence.

```
'            Key-1      Opt       Value
Illegal SoilThick  Accept   0.3       ! Redefine Min t/D

'            Key-1      Opt    IDs
Illegal Soilthick   Pile    10   20  ! Accept pile 10,20,30 40
                            30   40
```

Figure 2-14 Accepting thin soil layers. Specification pile-by-pile

The “lazy” (not recommended) version is to accept all short elements without any specification. A warning will be printed in the output file, (see Figure 2-17).

```
'          Key-1      Opt      Value
Illegal SoilThick  Accept   0.1   ! Redefine Min t/D

'          Key-1      Opt
Illegal Soilthick   UsersRisk  ON   ! Accepting everything unchecked
```

Figure 2-15 Accepting thin soil layers without specification

```
--------------------------------
.... Checking Pile Elements
--------------------------------
Illegal short element on pile        :        1 , L/D =   0.12
Illegal short element on pile        :        1 , L/D =   0.12
Illegal short element on pile        :        1 , L/D =   0.12
………….
…………
Illegal short element on pile        :        8 , L/D =   0.95
Illegal short element on pile        :        8 , L/D =   0.95
Illegal short element on pile        :        8 , L/D =   0.95

Checking of pile elements completed.
Smallest L/D found                   :     0.12432
```

Figure 2-16 The Errors and list of illegal elements are printed in the output file.

```
------------------------------------------------
--    * * *    W A R N I N G   * * *          --
--      UnConditionally Acceptance of         --
--  Very Thin Soil Layers on Users own Risk   --
------------------------------------------------
```

Figure 2-17 The warning is printed in the output file.

2.2 New defaults

Version 8-6 has several important changes, which means that the results may differ from previous versions of USFOS.

2.2.1 Default Joint Capacity.

The default joint capacity is “NORSOK”, and this also means following defaults:

- Element formulation “plasticity” is ON (same as the command Jnt_Form 3)
- Classification every step (same as the command JntClass 1)
- One equilibrium iteration (same as the command Liter)

2.2.2 Default Soil model.

The default soil model is set to the special “plasticity” model (same as “Spri_mod 1”). This formulation requires iterations, and iteration is switched on, (same as “Liter”)

2.2.3 Default Hydro accuracy.

The default calculation procedure for buoyancy is set to accuracy level 1. This means that the following are included by default:

- The buoyancy of the wall of a submerged pipe will be included, (minor impact on steel pipes).
- Weight of marine growth.
  Substantial change in vertical force when the MG goes in and out of water.
- Weight of internal fluid.
  Substantial change in vertical force if a flooded member goes in and out of water.

2.2.4 Default numerical procedure.

Iterations are switched on automatically if the model contains either joint capacity, (chjoint) or pile-soil model.

In addition is the model for strain estimation, (which are derived from the plastic hinge displacements and rotations), changed to an incremental solution. The incremental solution is more numerical stable, (f ex during unloading after a boat impact).

3 News in USFOS version 8-6 - 2012.

3.1 Introduction

The new features are described by examples in this memo and in the updated manuals.

3.2 How to upgrade your USFOS version

From release 8-5, USFOS could be upgrades in two different ways:

- Alt 1: Download the new "*setup.exe*" and u-install/install USFOS, (same as for release 8-5). 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 no installation operations are performed, (just file copy).

With alternative 1, all modules and the on-line manuals are updated. For alternative 2, following should be done:

- Download USFOS module , unzip and copy into C:\Program Files\USFOS\bin
- Download xact, (complete package) , unzip and copy into C:\Program Files\USFOS\bin
- Download USFOS and xact user's manuals. Copy into C:\Program Files\USFOS\bin

Alternative 2 means that the existing files located on the Application folder will be over-written, (take a backup copy of the actual files if you want to keep your existing USFOS modules).

Similar procedure for other USFOS modules (for example STRUMAN).

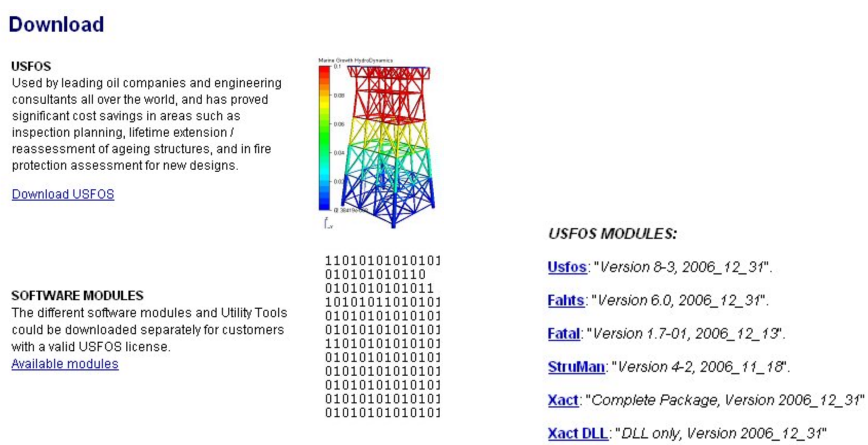

Figure 3-1 Download complete USFOS installation setup or the modules one by one

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 8-6 is "2.6". Check under help/about to ensure that the latest version is installed.

3.3.1 External Pressure

External pressure will reduce the section capacity for non-flooded members. In order to verify the reduction (0 means no reduction), the option shown in Figure 3-2 could be used. The result type is found under:

- Verify
    - ExtPress Q/Qcr

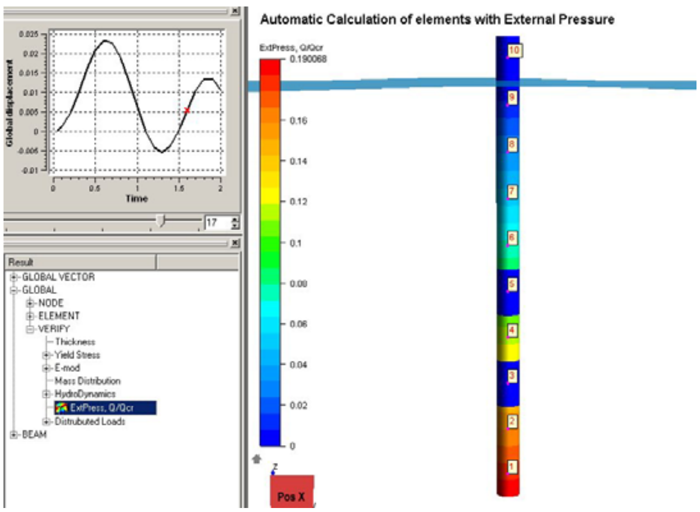

Figure 3-2 Verify External Pressure Effects

Also note that the input is simplified substantially:

```
ExtPres    Auto
```

USFOS will then automatically compute the external pressure effect relative to the surface defined under **WAVEDATA**. Flooded members get no external presssure effect.

NOTE! This automatic option is only valid when the USFOS hydrodynamic module is defined and activated.

3.3.2 Plot Size / Plot Scaling.

If a certain plot size should be kept for all new plot windows, the “Keep Last Plot Size” should be activated, (found under File / Preferences / Plot Settings, see Figure 3-3).

All plots will then keep the actual size.

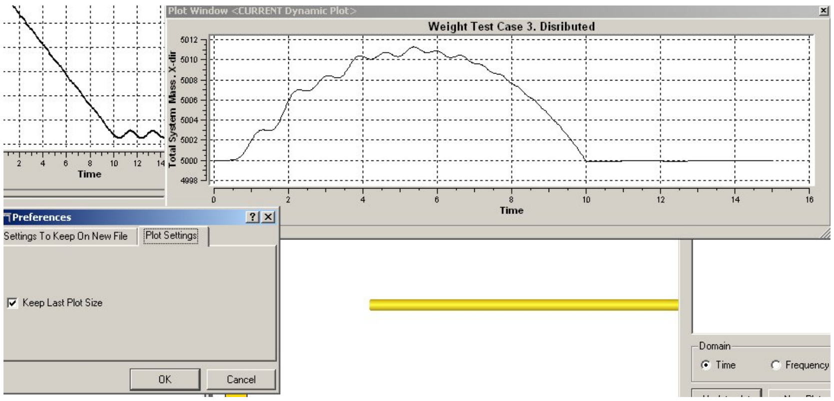

Figure 3-3 Switch ON "Keep Last Plot Size"

In order to increase the readability of plots, the same plot scale (min/max) should be used. By default, the “auto scale” option will scale both X- and Y axes every time the plot is updated. To keep the same scale, the following should be done:

- Switch off “auto scale” and define the wanted Min / Max
- Select “Update Plot” when a new curve is plotted

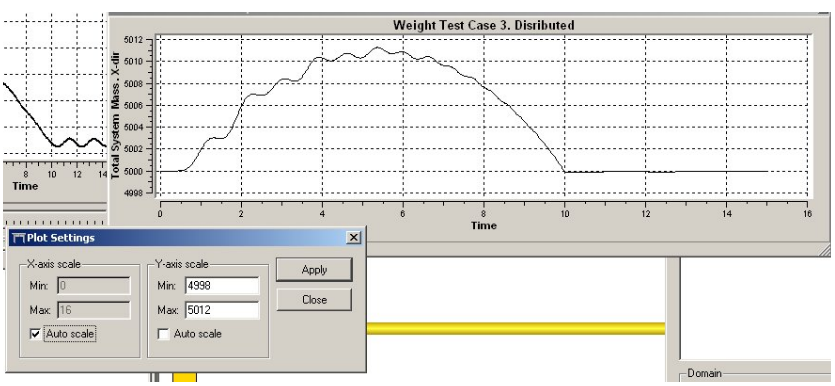

Figure 3-4 Select “Auto Scale” or “Fixed Scale”

3.4 Hydrodynamics

The hydrodynamic module has been improved and extended in release 8-6. The extensions covers the following:

3.4.1 Buoyancy

The default buoyancy accuracy is set to “1”, which means that the vertical (downward) forces from the structural steel, the marine growth and the internal fluid are computed always. The masses are multiplied with the instantaneous level of the gravity (NOTE, the correct gravity constant should be specified).

The vertical (upward) forces from the buoyancy will depend on the elements position in the water. This means that a flooded member with marine growth will become substantial heavier when the member becomes “dry”.

See the document, (which is found on the web):

> *“USFOS Buoyancy and Hydrodynamic Mass”,*

3.4.2 Tidal Level

The tidal level could be specified by the command:

```
‘          Key        Opt       Value
Switches WaveData  TidalLevel   +3
```

In practice this means that the surface elevation, (and the water depth), are increased with the actual level. The adjusted parameters are printed in the out file.

3.5 Tubular Joint Capacity Models

3.5.1 Geometry Modelling

When the joint option is used, each joint will be re-modelled by USFOS automatically. For example will a K-joint get two extra nodes (on the chord surface) and two extra special elements (with behaviour according to selected capacity curve). USFOS will compute the gap automatically based on the actual geometry input. If eccentricity is needed in order to get correct joint shape, the solution shown in Figure 3-6 is recommended. This model ensures that the special element gets a reasonable length, which again ensures that the local coordinate system of the element could be established and updated correctly.

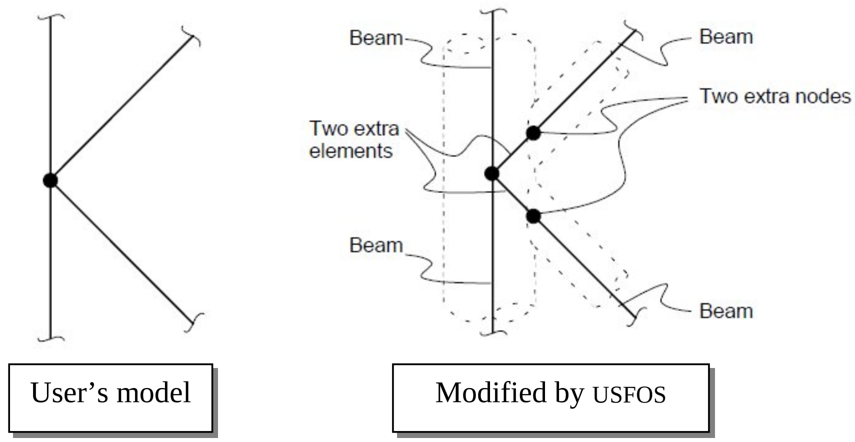

Figure 3-5 Automatic generation of extra nodes and elements of joints *without* eccentricity

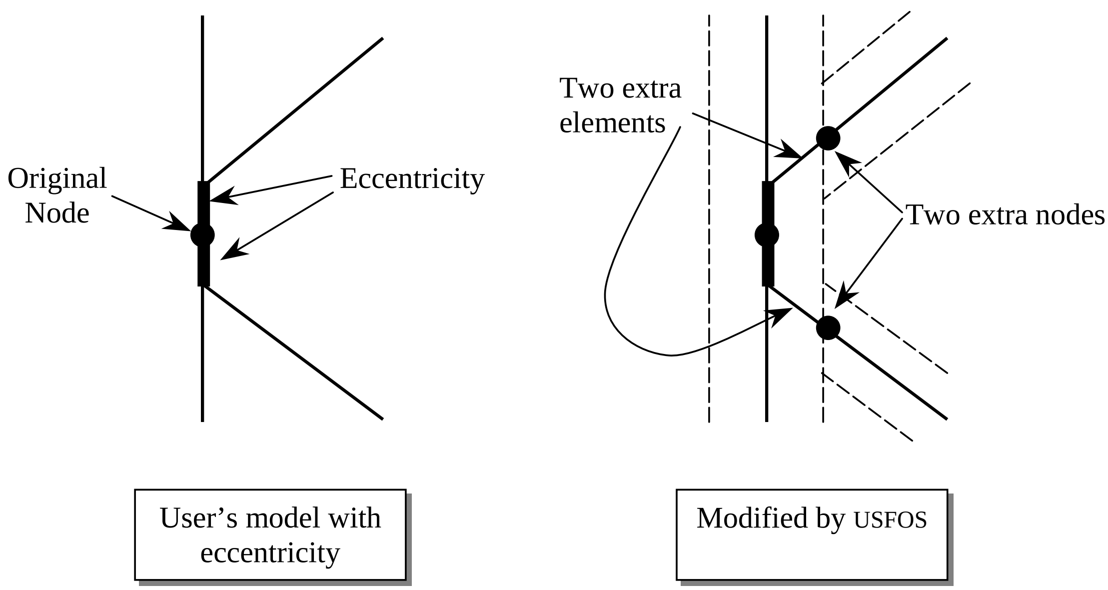

Figure 3-6 Automatic generation of extra nodes and elements of joints *with* eccentricity

The solution shown in Figure 3-7 is ***not*** recommended. The “flushing option” in the mesh generator will insert an infinite connection between the centre node of the chord (leg) and the chord surface. The special element inserted by USFOS will therefore become extremely short and may introduce following problems:

- Local coordinate system may rotate much for a small element deformation
- Element axis could rotate 180° if the special element yields in compression and gets substantial plastic deformations, (element axis is based on the flexible part of the element)

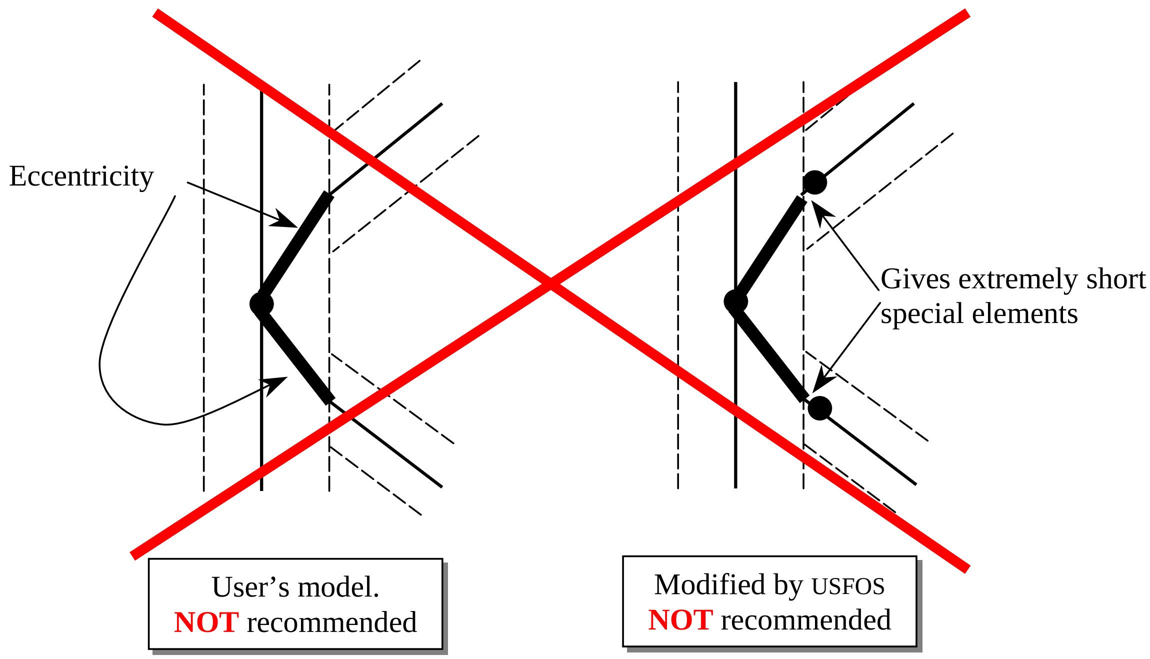

Figure 3-7 Not recommended User’s model. “Flushing” gives extremely short elements

3.5.2 New Joint Curve default

The default joint curve is changed to NORSOK. (Previous default is now accessed using the keyword oldAPI).

```
'         Node    Ch1   Ch2
 ChJoint   101     10    20
```

Figure 3-8 If no joint curve is specified (minimum input), the NORSOK is used.

3.5.3 API RP2A-WSD

A new capacity curve is added to the joint option: API RP2A-WSD. This option is activated for the following ChJoint command:

```
‘         Node   Ch1  Ch2  Geo   Rule
 ChJoint  101     10   20   0   APIWSD
```

Figure 3-9 Activating API-WSD joint capacity

3.5.4 ISO 19902

A new capacity curve is added to the joint option: ISO 19902. This option is activated for the following ChJoint command:

```
‘         Node   Ch1  Ch2  Geo   Rule
 ChJoint  101     10   20   0     ISO
```

Figure 3-10 Activating ISO 19902 joint capacity

3.5.5 Utility tool JntRes

A new tool, “**jntres**” is available on the web. The purpose of the tool is to give a summary of the joint behaviour in a simple text file and give an overview of the tables produced by the tool.

The “overview” (Table 3-1) sorts the connections in three groups based on the status at the end of the analysis:

- Connections exceeding peak axial
- Connections exceeding first yield (but have not reached peak)
- Connections, which have not reached first peak (remain elastic)

The “detailed” print (Table 3-2), gives a status for one connection from the first to the final analysis step. Following information are printed:

- Utilization : Plastics utilization of the connection element
- Status : Connection status:
    - Elastic, tension or compression
    - Plastic, tension or compression
    - Failure (post-peak), tension or compression
- Ductility limit : Based on the Axial degree of freedom
- Peak Axial : Current capacity (based on the instantaneous classification)
- Axial : Actual axial force in the connection

```
    ------------------------------------------------
    ------------------------------------------------
    ----                                      ------
    ----      J o i n t   R e s u l t s       ------
    ----                                      ------
    ----         S t a t u s                  ------
    ----                                      ------
    ------------------------------------------------
    ------------------------------------------------

    ------------------------------------------------
    ----     Connections Exceeding Peak Axial ------
    ------------------------------------------------

Connection  Node      Brace LoadCase StepNo LoadLevel Utiliz  Status     PeakAx
      3       11         1      1     75    6.020    0.998  CompFail    1.922E
      1       10         1      1     93    6.051    1.000  TensFail    2.489E

    ------------------------------------------------
    ----     Connections exceeding First Yield -----
    ------------------------------------------------

Connection  Node      Brace LoadCase StepNo LoadLevel Utiliz  Status     PeakAx
      9        8         2      1     42    4.188    0.566  TensPlast   2.859E
      5        7         1      1     57    5.168    0.578  TensPlast   2.339E
      6        7         2      1     57    5.168    0.589  CompPlast   2.891E
     14        5         1      1     57    5.168    0.596  TensPlast   2.267E
      8        8         1      1     63    5.368    0.596  CompPlast   2.205E
     15        5         2      1    102    6.044    0.470  CompPlast   3.495E
     10        8         3      1    123    5.975    0.257  CompPlast   1.733E

    ------------------------------------------------
    ----  Max Utiliz for Elastic Connections   -----
    ------------------------------------------------

Connection  Node      Brace LoadCase StepNo LoadLevel Utiliz  Status     PeakAx
      2       10         2      1    195    5.974    0.332  CompElast   1.659E
      4       11         2      1    189    5.974    0.412  CompElast   1.056E
      7        7         3      1    200    5.953    0.281  CompElast   1.665E
     13        4         3      1    200    5.953    0.132  CompElast   1.548E
     16        5         3      1    200    5.953    0.132  CompElast   1.570E
```

Table 3-1 jntres overview

```
    ------------------------------------------------
    ------------------------------------------------
    ----                                      ------
    ----      J o i n t   R e s u l t s       ------
    ----                                      ------
    ----    D e t a i l e d   P r i n t       ------
    ----                                      ------
    ------------------------------------------------
    ------------------------------------------------

    =================================
          Connection :          1
          Joint      :         10
          Brace      :          1
    =================================

LoadCase StepNo LoadLevel Utiliz  Status     PeakAxial   DuctAxial    Axial
     1      3    0.300    0.028  TensElast   2.748E+05     0.014   1.156E+04
     1      6    0.600    0.071  TensElast   2.764E+05     0.011   2.342E+04
     1      9    0.900    0.114  TensElast   2.767E+05     0.011   3.532E+04

     1     42    4.188    0.616  TensPlast   2.685E+05     0.011   1.673E+05
     1     45    4.481    0.665  TensPlast   2.662E+05     0.011   1.793E+05
     1     48    4.773    0.714  TensPlast   2.636E+05     0.011   1.910E+05


     1     87    6.064    0.999  TensPlast   2.484E+05     0.010   2.482E+05
     1     90    6.058    1.000  TensPlast   2.486E+05     0.010   2.486E+05
     1     93    6.051    1.000  TensFail    2.489E+05     0.010   2.488E+05
     1     96    6.047    1.000  TensFail    2.491E+05     0.010   2.490E+05


     1    195    5.974    1.000  TensFail    2.510E+05     0.011   2.031E+05
     1    198    5.952    1.000  TensFail    2.510E+05     0.011   2.013E+05
     1    200    5.953    1.000  TensFail    2.509E+05     0.011   2.005E+05
```

Table 3-2 Detailed print of one connection

3.6 Pile - Soil

3.6.1 Pile Option

The properties of a given “SOILCHAR” could be scaled, pile-by-pile using the new **PILEOPT** command. A certain depth-profile could be defined for each component, P-Y, T-Z and Q-Z, and this scaling could be assigned to one or more piles. The example shows three piles, all referring to same “SOILCHAR”, but with different scaling, (0.5, 0.3 and 0.1). This option could be used if for example the standard soil curve over/under estimates the strength.

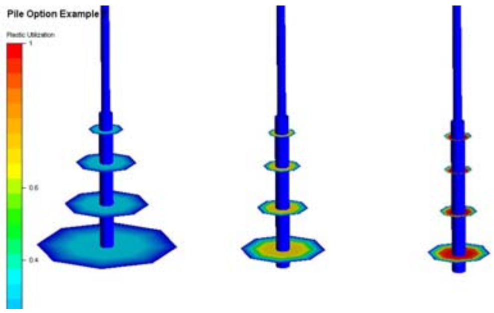

Figure 3-11 Three piles with same soil, but scaled differently

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

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

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

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

Table 3-3 Scaling all properties of pile 1001 with factor 0.5

```
' ------------------------------------------------------------
'    Define Pile Options (ID=200)  and Assign to Pile 1002
' ------------------------------------------------------------
'             KeyWord      ID      Type       Z     Fac
 PileOpt      SoilScal     200     P-Y        0     0.1
                                             -2     0.1
                                            -80     0.1

'             KeyWord      ID      Type       Z     Fac
 PileOpt      SoilScal     200     T-Z        0     0.2
                                             -1     0.2
                                            -80     0.2

'             KeyWord      ID      Type       Z     Fac
 PileOpt      SoilScal     200     Q-Z        0     0.3
                                            -80     0.3

'             KeyWord      ID      Type      PileID ....
 PileOpt      SoilScal     200     Assign      1002
```

Table 3-4 Scaling all properties of pile 1002 with factor 0.3

3.6.2 Soil Curve visualization

Under “Verify” the verification plot is found. In addition to the depth profiles of pile- and soil forces for a given pile, “Soil Curves” is found in the menu, (see figures).

The actual soil layer is selected by pointing on the actual soil disc.

The choice of “DOF X-axis” gives the following plots:

- Dof = 1 : T-Z curve
- Dof = 2 : P-Y curve

The soil curves are given per unit length of the pile. The Peak values in the text box are also capacity per unit length of the pile, (changed in version 8-6).

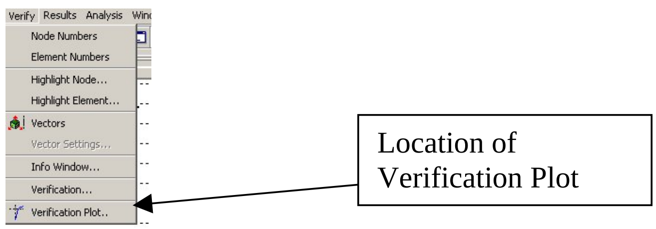

Figure 3-12 Location of Verification Plot Menu

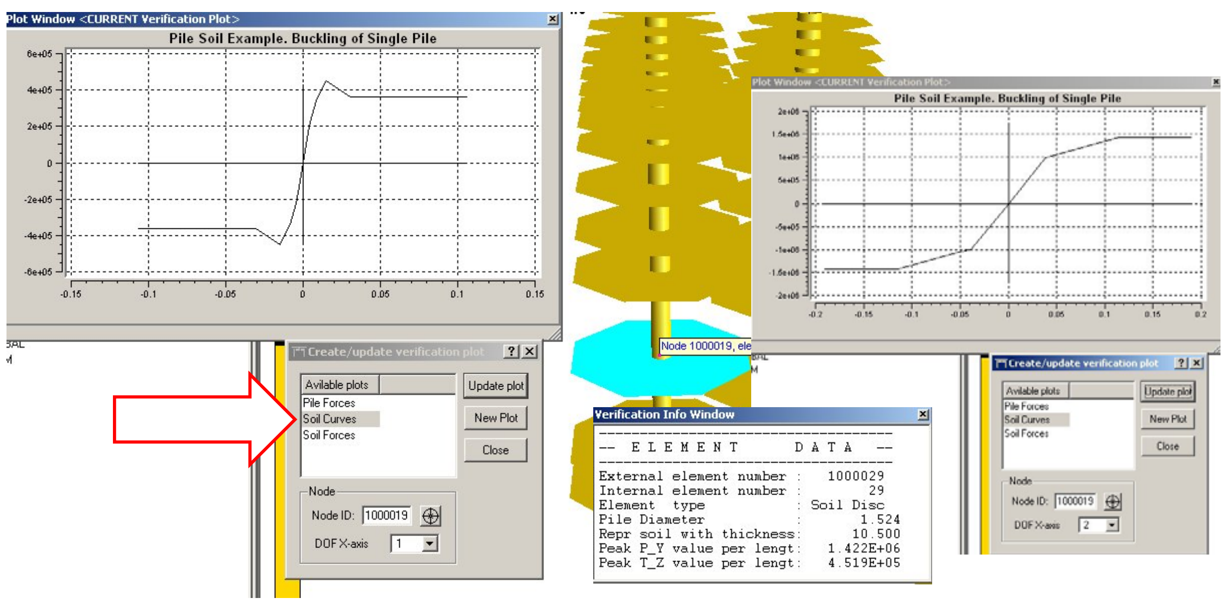

Figure 3-13 Visualization of T-Z and P-Y curves for a given soil layer.

3.6.3 Pile Upper Bound Capacity

The sum of all TZ peaks is computed for all piles and printed in the "out" file, (see Table 3-5). The compression capacity includes the peak QZ. This peak is an "upper bound" capacity since sum of peaks means that all peaks are activated simultaneously. For a real pile, some soil layer may have moved into the post-peak range before other layers are fully mobilized.

However, this is a useful reference, and the axial forces in the piles relative to this "upper bound" capacity are printed under the "reaction force", (percent of max theoretical). See Table 3-6.

```
---------- U P P E R   B O U N D   P I L E   C A P A C I T Y ------------
              ( Based on sum of soil-peaks for all layers )

     Pile      Max Tension   Max Compression      Pile
      ID        Capacity        Capacity         Length
         1      1.1E+08         1.3E+08           43.5
         2      1.9E+07         2.2E+07           47.4
         3      2.1E+07         2.4E+07           50.3
         4      1.3E+08         1.5E+08           46.0
         5      1.3E+08         1.5E+08           46.0
         6      1.9E+07         2.2E+07           47.4
         7      1.8E+07         2.1E+07           46.4
         8      1.2E+08         1.4E+08           45.5
```

Table 3-5 Upper Bound capacity of the Soil, (sum of T-Z for all layers).

```
--------     P I L E        R E A C T I O N     F O R C E S        --------

     Pile        X-for      Y-for      Z-for     Utiliz (Soil Axial)
        1      2.347E+05 -3.751E+05  3.312E+06          2 %
        2      5.968E+03 -1.068E+05  9.167E+05          4 %
        3     -2.192E+03 -1.009E+05  8.961E+05          4 %
        4     -1.703E+05 -3.554E+05  3.123E+06          2 %
        5      2.501E+05  4.130E+05  3.711E+06          3 %
        6      6.211E+03  1.209E+05  1.062E+06          5 %
        7     -3.144E+03  1.222E+05  1.048E+06          5 %
        8     -2.217E+05  4.019E+05  3.712E+06          3 %


     Pile        X-for      Y-for      Z-for     Utiliz (Soil Axial)
        1      2.339E+06 -3.744E+06  3.301E+07         25 %
        2      5.920E+04 -1.014E+06  8.736E+06         40 %
        3     -2.250E+04 -9.764E+05  8.681E+06         36 %
        4     -1.704E+06 -3.548E+06  3.113E+07         21 %
        5      2.519E+06  4.151E+06  3.722E+07         25 %
        6      6.307E+04  1.135E+06  1.003E+07         45 %
        7     -3.213E+04  1.143E+06  9.867E+06         46 %
        8     -2.233E+06  4.037E+06  3.721E+07         26 %

     Pile        X-for      Y-for      Z-for     Utiliz (Soil Axial)
        1     -4.508E+06  1.448E+07 -2.115E+07         19 %
        2     -3.466E+05  2.843E+06 -5.907E+06         31 %
        3     -2.781E+05  3.222E+06 -7.312E+06         34 %
        4      5.937E+05  2.144E+07 -3.710E+07         30 %
        5      9.555E+06  2.313E+07  9.905E+07         67 %
        6      4.617E+05  3.774E+06  1.815E+07         82 %
        7      2.522E+05  3.895E+06  1.729E+07         81 %
        8     -4.720E+06  2.865E+07  1.139E+08         79 %
```

Table 3-6 Pile force (axial), Relative utilization of soil is printed per pile.

3.7 Member imperfections

Member buckling is described in a separate document, “Member Buckling in USFOS” and is found on the web.

3.7.1 Element Imperfection / Dent

If the automatic imperfection (CINIDEF) is used, the orientation of the imperfection may change from case to case, (if for example the member load is based on wave load).

The initial dent inserted automatically by USFOS is placed on the side of the section, which will get increased compression when the member buckles.

3.7.2 Member Imperfection

By default, the imperfection is applied on element-by-element as an initial element deformation. With reasonable element length, (approx same length as face-face length of the physical member minus stubs), the initial imperfection will behave as intended. However, if of different reasons, the member is sub-divide into several elements, the shape will be as shown in Figure 3-14, (the imperfections are exaggerated). For the lower row, the imperfections are applied for the entire member, and nodes are moved.

The definition of a “member” is group of elements, (three groups are shown).

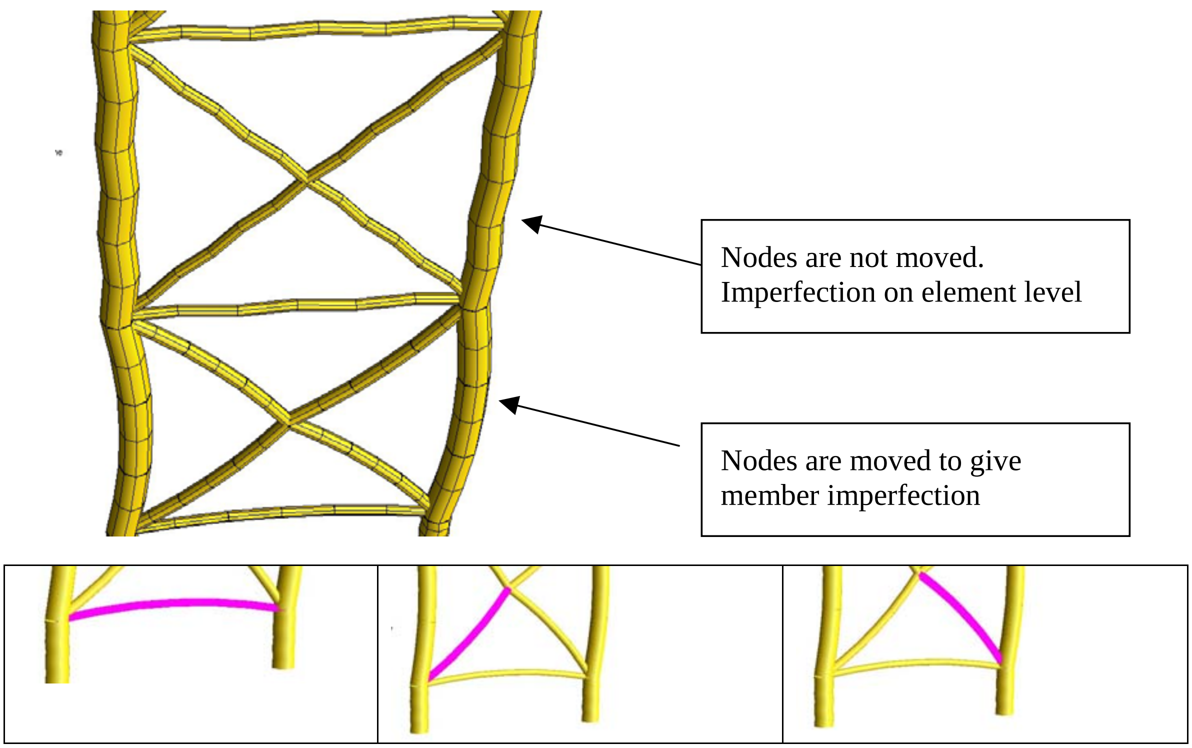

Figure 3-14 Member imperfection (lower bay) and element imperfection (exaggerated)

The member imperfection could be defined in two ways:

1. Specifying a buckling curve (CINIDEF) see Table 3-7
2. Referring to a user defined imperfection (GIMPER) see Table 3-8

```
'                          LC
CINIDEF  API_WSD  MembLoad   12

'                  ImpGrp   ListTyp  Ids ....
Member  Imperfect   Auto    Group      1
```

Table 3-7 Defining API Column Buckling curve for member group -1.

```
'         impgroup impshape angle  Offset   dent1  dent2  dentMid
GIMPER      10       0      0.0  0.0015    1e-3   1e-3   1e-3     '

'                  ImpGrp   ListTyp   Ids ....
Member  Imperfect    10      Group     1
'
```

Table 3-8 Defining imperfection for member group -1.

If alternative 1 (CINIDEF) with reference to a column-buckling curve is selected, the key parameters are printed for each element. Elements belonging to same member group get same key data, (see for example elements: 2439, 2441, 2443 and 2445).

```
     ----      M E M B E R     B U C K L I N G    I N F O      -----
        ( Based on physical member length and k-factor = 1 )

          Minimum Imperfection (in CINIDEF) :     0.050%
          Maximum Imperfection              :     1.000%

      ElemID    Ncr        IniDef[%]   Physical    Reduced      Column
             (for k=1)                  Length   Slenderness    Curve
      2439   1.147E+05       0.217       1.512       0.032     API_WSD
      2440   5.327E+04       1.000       1.512       0.028     API_WSD
      2441   1.147E+05       0.217       1.512       0.032     API_WSD
      2442   5.327E+04       1.000       1.512       0.028     API_WSD
      2443   1.147E+05       0.217       1.512       0.032     API_WSD
      2444   5.327E+04       1.000       1.512       0.028     API_WSD
      2445   1.147E+05       0.217       1.512       0.032     API_WSD
      2446   5.327E+04       1.000       1.512       0.028     API_WSD
      2447   3.721E+04       1.000       2.024       0.038     API_WSD
      2448   1.145E+05       0.253       2.024       0.043     API_WSD
      2449   1.145E+05       0.253       2.024       0.043     API_WSD


      4560   1.683E+04       0.246       0.839       0.040     API_WSD
      4561   1.683E+04       0.246       0.839       0.040     API_WSD
      4570   3.325E+04       0.194       0.728       0.027     API_WSD
      4572   6.666E+03       0.169      20.671       1.128     API_WSD
      4573   6.666E+03       0.169      20.671       1.128     API_WSD
      4593   1.682E+04       0.249       0.866       0.041     API_WSD
      4594   1.682E+04       0.249       0.866       0.041     API_WSD
      4595   1.682E+04       0.248       0.857       0.041     API_WSD
```

Table 3-9 Print of Member Buckling key parameters.

3.8 Strain Calculation

USFOS beam element ***does not*** use strain as a basis for the element, (it is based on plastic hinges with plastic deformations and rotations). However, since a certain plastic strain level often is used as an acceptance criterion, (f ex USERFRAC), USFOS will estimate the strain level based on the force- and plastic displacement level.

A new and improved algorithm is implemented in version 8-6. This method is based on the incremental plastic displacement, (i.e. change from $step_{(i-1)}$ to $step_{(i)}$).

The extent of the plastic zone depends on the material parameters. USFOS 8-6 has some pre-defined strain model material data, but the user may also specify the data using the new command STRAINMOD.

By default, the material data for S355 steel are assigned to all materials, but the user may change the defaults for selected material IDs.

The actual data used in the simulation are printed in the “out” file.

```
' -------------------------------------------------------------------
'     User defined  Input.
' -------------------------------------------------------------------
'
'                Key        SubKey        Value     ListType     ID
 StrainModel     UserDef    SigURatio      1.25       mat        1
 StrainModel     UserDef    EpsRef         0.13       mat        1
'
'
' -------------------------------------------------------------------
'     Using built in values
' -------------------------------------------------------------------
'                Key       ListType     ID
 StrainModel     S355        mat        2
 StrainModel     S420        mat        3
```

Table 3-10 Assigning Strain Model Data to the materials.

```
   ----     S T R A I N    M O D E L   D A T A         -----

  MAT_ID    Type           SigyRatio    EpsRef      EpsS
      1    UserDefined      1.250       0.130      20.000
      2    S355             1.300       0.150      10.000
      3    S420             1.200       0.120       8.000
```

Table 3-11 Print of Strain model data for each material.

The strain assessment procedure is described in a separate document:
“*Strain Assessment in USFOS*”, which is found on the web.

3.9 Miscellaneous

3.9.1 Spring Property scaling

This options is used to activate / de-activate springs gradually during the analysis.

- Increase : Spring is “dead” until actual load case is activated.
- Decrease : Spring is active, and will become “dead” when load case is completed

```
'               KeyWord     LCase     Material
 SpriScale     Decrease      2         1000

'
 CUSFOS    10    100   1    1
'
'          lc      inc  max  n    min
            1      0.1   1  100   0.001
            2      0.1   1  100   0.001  ! Dummy Load. Controls spring
            1      0.1   2  100   0.001  !
'
```

Table 3-12 Decrease Spring material to zero during dummy load case 2.

```
'               KeyWord     LCase     Material
 SpriScale     Increase      2         1000

'
 CUSFOS    10    100   1    1
'
'          lc      inc  max  n    min
            1      0.1   1  100   0.001
            2      0.1   1  100   0.001  ! Dummy Load. Controls spring
            1      0.1   2  100   0.001  !
'
```

Table 3-13 Increase Spring material from zero to 100% during dummy load case 2.

3.9.2 Friction Element

A simple friction element is introduced. It has an elastic – perfect plastic behaviour for a certain load level. The friction element has following options:

- Constant friction unaffected by force level
- Friction force depends on force level

3.9.3 Extended Shell element results

By default, 5 shell results are stored on the raf-file. It is possible to increase the total number of different shell results using the SWITCHES command. See Table 3-14 and Table 3-19.

```
 1 :   Plastic Utilization
 2 :   Equivalent Strain, Upper Side
 3 :   Equivalent Strain, Lower Side
 4 :   Von Mises  Stress, Upper Side
 5 :   Von Mises  Stress, Lower Side

 6 :   Sxx  Membrane
 7 :   Syy  Membrane
 8 :   Sxy  Membrane
 9 :   Sxx  Bending
10 :   Syy  Bending
11 :   Sxy  Bending
12 :   Sxx  Upper Side
13 :   Syy  Upper Side
14 :   Sxy  Upper Side
15 :   Sxx  Lower Side
16 :   Syy  Lower Side
17 :   Sxy  Lower Side
18 :   Sig-1
19 :   Sig-2
```

Table 3-14 Definition of the 19 possible shell results.

3.9.4 Element eXtra masses

If the user wants to specify distributed masses on beam elements, the new X_ELMASS could be used. This mass will contribute both on inertia and will be multiplied with the gravity acc (if activated) and become a distributed load.

The input syntax is shown below, and could be specified “Elem-by-elem” only.

```
'           Mass   ListTyp   Elem
 X_ElMass   1000    elem      10
```

Table 3-15 Specifying eXtra Beam Mass.

3.9.5 Extended Dynres_G

In order to check that the system mass matrix is defined correctly, the Dyres_G options shown in Table 3-16 could be used. The system mass matrix (inertia) in X- Y- and Z directions are available. This option could be useful for structures going in/out of water, (changed hydrodynamic added mass), and in cases where masses (weight option) are added/removed during the simulation.

```
SysMassX : Plot of Total System (inertia) mass. X-component
SysMassY : Plot of Total System (inertia) mass. Y-component
SysMassZ : Plot of Total System (inertia) mass. Z-component
```

Table 3-16 New Dynres_G options: Plot of System Mass

3.9.6 New Dynres_X command

A new “DynRes” command is introduced. It has the format described in Table 3-17 with a “keyword” followed by the actual value.

```
DynRes_X  KeyWord   Value
```

Table 3-17 New command: Dynres_X. General Format

Often it could be useful to see the user-defined time histories. (f ex ground motion histories). The keyword “TimeHist” followed by the time history ID will generate a plot of the history.

In the example below, the time history controlling the self weight (ID=1) and the ground motion history (ID=111) are speficied.

```
‘            keyword   ID
DynRes_X  TimeHist     1 ! Plot of Self weight TimeHist
DynRes_X  TimeHist   111 ! Plot of Ground motion TimeHist
```

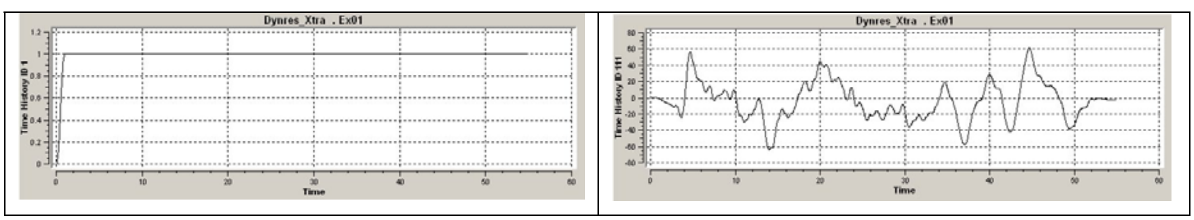

Table 3-18 Plot of User’s Input Time Histories.

3.10 SWITCHES, (Special Options).

The command "SWITCHES" was introduced in 8-5 to switch on special options and is extended in version 8-6. Following "Switches" commands are available:

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	860
WaveData	*TimeInc*	val	Time between each hydrodyn calc.	every
	NoDoppler	-	Switches OFF doppler effects.	ON
	NoStore		Switches OFF storing of wave data for visualiz.	ON
	TidalLevel	Level	Specify Tidal Level	0
	Accuracy	val	Change accuracy. 0: old accur, 1: new accur	1
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*	Case/stp	Writes deformed FE model at given case/stp	OFF
	LinDepAlt	-	Writes ZL-springs for each BLINDP2	Off
Solution	*FracRepeat*	MxRep	Max fracture repeates	10
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
WindData	*ReynDep*	ON/OFF	Switch to Reynold-number dependent Cd	OFF

Table 3-19 SWITCHES options

3.11 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

3.12 New/modified input commands

Since last main release (8-5), following input identifiers are added/extended:

X_ELMASS	:	New command	: e*Xtra Element Mass per length*
PILEOPT	:	New command	: *Pile/Soil Options*
DYNRES_X		New command	: *Extended dynamic results*
STRAINMOD		New command	: *Define Strain Model Data*
MEMBER		New command	: *Member options (imperfection)*
SPRISCAL		New command	: *Scaling spring properties*
CHJOINT	:	Extended command	*: ISO & API joint curves.*
SWITCHES	:	Extended command	*: See above.*

3.13 Documentation

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

- USFOS strain assessment : New document
- Member Buckling in USFOS : New document
- Buoyancy and Hydrodynamic Added Mass : New document
- User's manual : Updated document