MEMO

USFOS
Reality Engineering

USFOS AS

Phone: +47 905 05 717

www.USFOS.com

Enterprise No.: NO 986 827 374 MVA

MEMO CONCERNS	FOR YOUR ATTENTION	COMMENTS ARE INVITED	FOR YOUR INFORMATION	AS AGREED
Release Notes **USFOS Version 9-0**				
DISTRIBUTION Members of USFOS user group				X

FILE CODE	CLASSIFICATION
	Confidential
REFERENCE NO.	

PROJECT NO.	DATE	PERSON RESPONSIBLE / AUTHOR	NUMBER OF PAGES
	2022-11-01	Tore Holmas	32

Release Notes

USFOS 9-0, Nov 2022

1 Introduction

The current official version of USFOS is version 9-0 with release date 2022-11-01. The release contains the following:

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

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

2 Changes in version 9-0

Comparison of 9-0 vs. older USFOS versions could give somewhat different results due to modifications of the algorithms and default parameters.

2.1 Joint Capacity

2.1.1 NORSOK

USFOS 9-0 has the latest revision of NORSOK. The keyword is “NOR_2021”

```
'              node     el1         El2      Geo      Rule
 Chjoint       100      220         200       0       NOR_2021
```

Older versions of NORSOK are available using the following keywords:

- NOR_R2 : Norsok N-004 Rev2
- NOR_R3 : Norsok N-004 Rev3

2.1.2 ISO

USFOS 9-0 has the latest revision of ISO. The keyword is “ISO_2020”

```
'              node     el1         El2      Geo      Rule
 Chjoint       100      220         200       0       ISO_2020
```

The 2007 version of ISO is available using the keyword: ISO_2007.

2.1.3 Default curve

If no “rule” is specified, Norsok 2021-01, (NOR_2021) is used. ISO and NORSOK give same response.

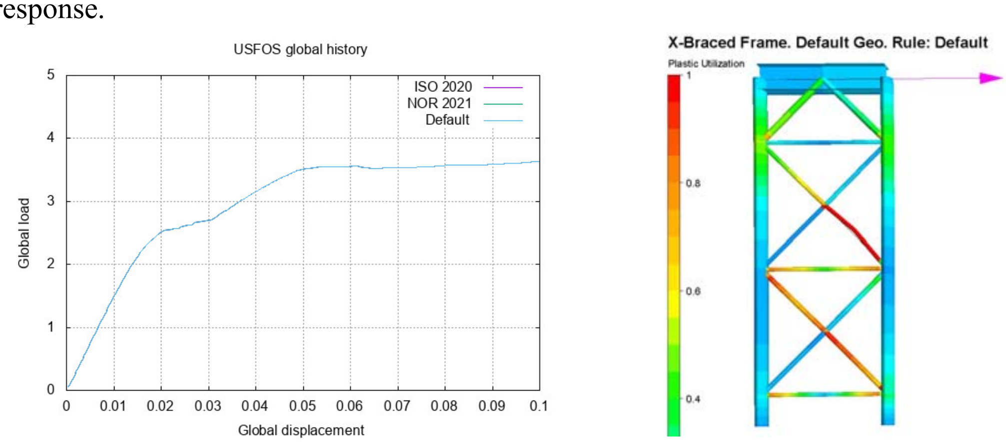


Figure 2-1 - ISO, Norsok (and default) curves give same response

2.2 Member buckling

2.2.1 ISO 19902

According to a conservative interpretation of ISO 19902 the combined loaded member (axial and bending) follows a straight line. The line ends at the axes for a certain level below 1.0. This assumption is not necessarily the best estimate on how “reality” behaves. (If the alternative interpretation is wanted, just specify NORSOK since the shape of the interaction curve is the only difference in the USFOS implementation)

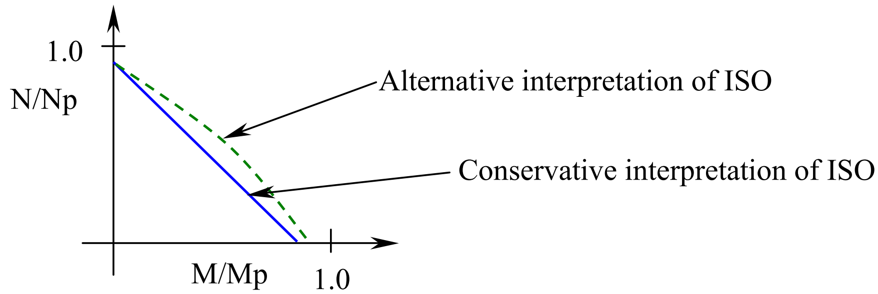


Figure 2-2 – Interaction between normalized moment and normalized axial.

However, if the user wants to use the conservative ISO assumption, the CINIDEF curve ISO_19902 is specified.

```
''
'          Curve      LoadPattern    LoadCase
 CINIDEF   ISO_19902    BaseShear       2
'
```

The responses for the X-braced frame show that choice of buckling curve has impact on the global response.

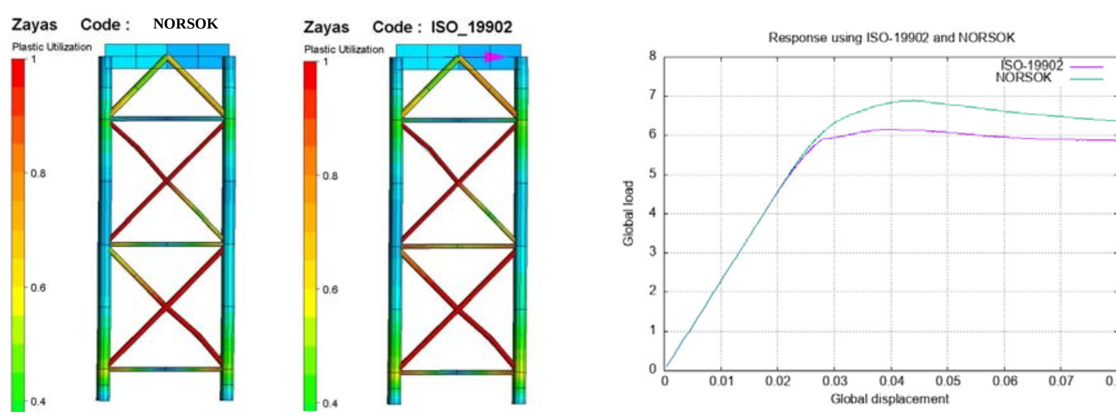


Figure 2-3 - Plastic utilization and global response curve.

2.2.2 Fire according to Eurocode 3 – EN 1993-1-2:2005 E

When PUSHDOWN is used (recommended), a “fire” buckling curve is required. The earlier versions used EUROCODE curve-C for fire. (Keywords: Either “Euro_3C” or “Fire”). Version 9-0 has an updated column-buckling curve. The keyword “fire” is used to activate the new curve.

```
'
'          ColumnCurve  Pattern  LoadCase
 cIniDef       Fire       Base       2
'
```

The difference between the old “fire curve” (Euro_C) and the updated is shown in Figure 2-4. Version 9-0 gives slightly lower resistance compared with the old implementation. For high temperatures, the differences are less.

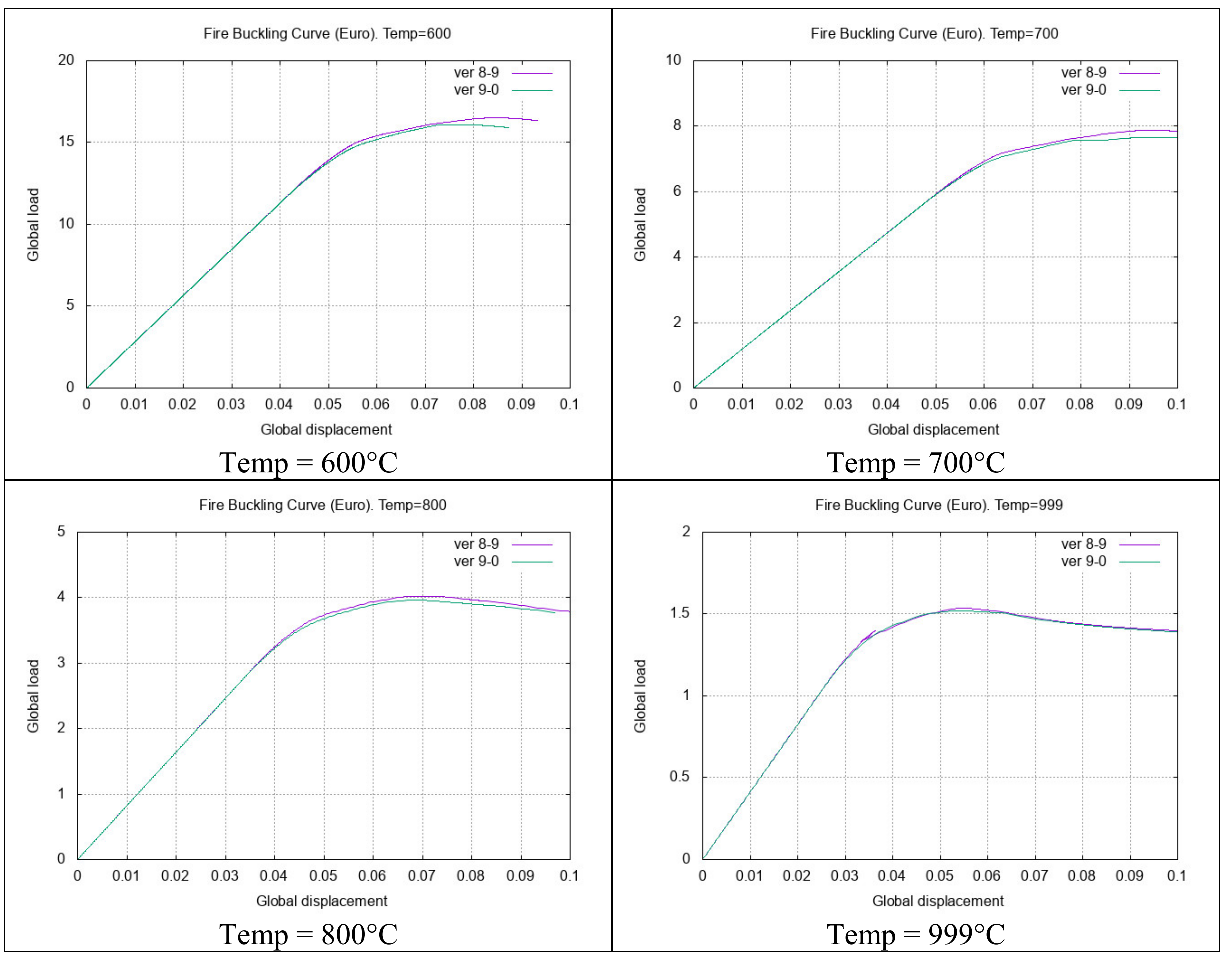


Figure 2-4 - PushDown response. Version 8-9 vs. version 9-0. Different temperatures.

2.3 Element response

2.3.1 Shell Element

The non-linear shell element was developed for shell buckling purpose. The yield function was based on an assumption of combined bending and axial stresses. For pure axial stress, the element would under-estimate the resistance, (i.e.: is conservative).

In version 9-0 this is changed. The user may experience higher axial resistance if old cases are re-run. It is possible to switch back to the old formulation using the command:

```
Switches  ShellOpt AxCorrection OFF   ! Switch back to old formulation
```

3 News in USFOS version 9-0 - 2022.

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 9-0 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 9-0 32bit windows. However, version 9-0 becomes available by downloading the central modules (similar to Alternative-2 above):

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

3.2.3 LINUX

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

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

On Ubuntu, the library is installed using the command:

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

3.2.4 MACOSX

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

The simulation engine has also been built for the “M1” processor, OSX Monterey. The gFortran compiler is used. Contact us for access to the USFOS “engine” implementation.

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 9-0 is “3.2” for the Win-64bit and LINUX versions. The 32bit version of XACT.exe is not changed since v8-7, but the updated 32bit “dll” gives extended functionality.

3.3.1 "PanCake" utilization

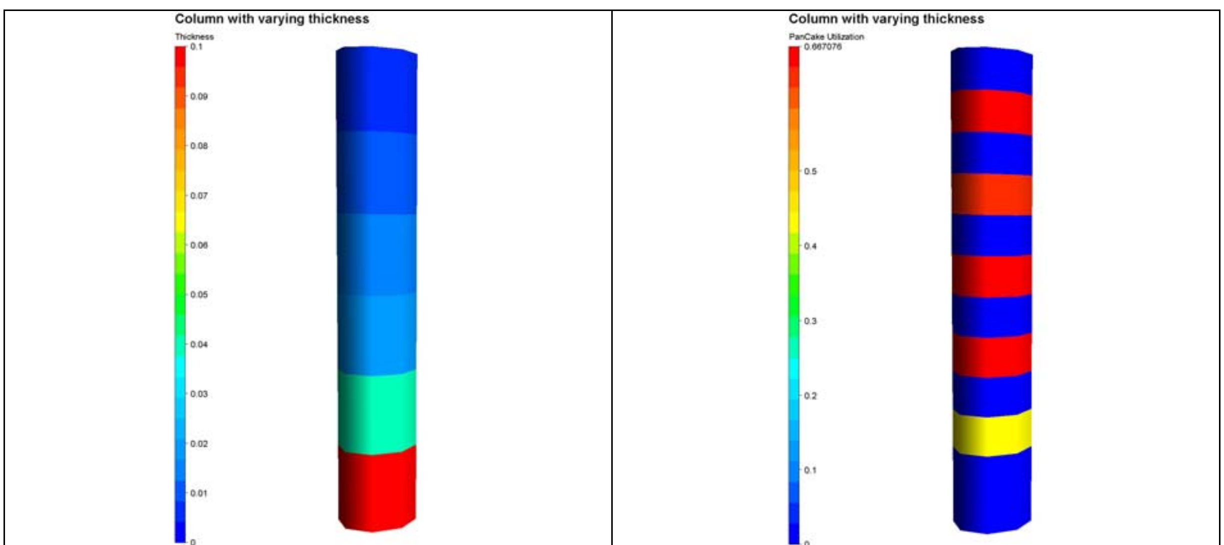
The Results for Beam Elements are extended with the "PanCake Utilization". If USFOS has been run with the "pancake" switch ON, the utilization is available. The Fringe Range goes from minimum to maximum utilization. (If no switch is activated, all elements get 0 utilization and the range goes from 0 to 1).

The pancake utilization is computed where USFOS finds a thickness change, and the thinner element will be checked. In the image, the lower end of the thinner element gets utilization>0.

If the max utilization is below 1.0, the pancake failure mode is not governing. The example has max utilization of 0.67, and this means that the pancake buckling does not take place.

Figure 3-1 - Extended Beam Results. Pancake Utilization is added

NOTE: XACT will find thickness changes automatically. The element (of the two with common node) with the smaller wall thickness will be checked for "pan cake". The other element (with thicker wall) will not be checked.

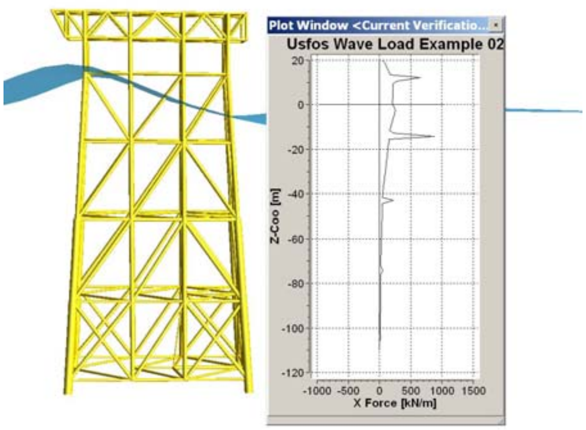


Thickness variation | PanCake utilization

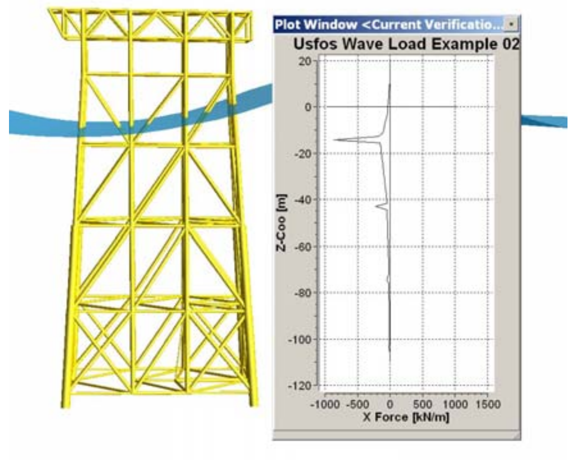
Figure 3-2 - Column with varying thickness. PanCake utilization (right).

3.3.2 Total Wave Load distribution

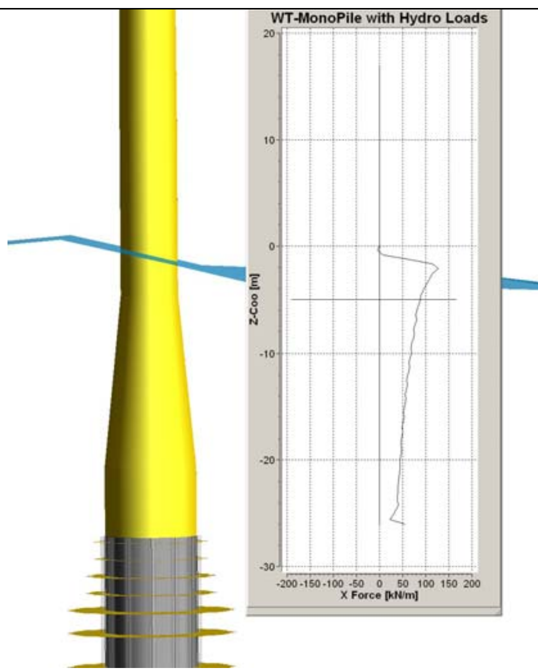
The total wave load computed by USFOS (WAVEDATA) in a time domain analysis (commands: static or dynamic), are available in XACT. The depth profile of the ***total*** wave load is visualized for the chosen step. Elevations with more structures are easily seen for the jacket example.



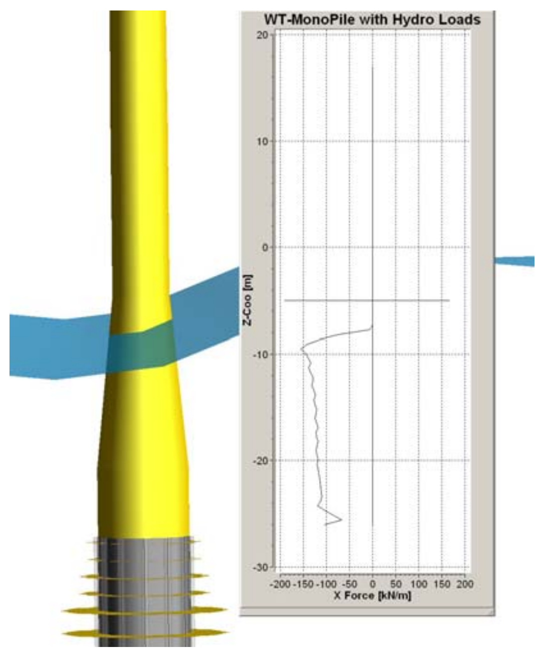
Wave phase -1
Note the higher loads at the horizontal frames



Wave phase - 2
Note the higher loads at the horizontal frames



Wave phase -1
Note the smooth distribution



Wave phase -2

Figure 3-3 - Depth Profile of TOTAL wave load for two different time steps.

3.3.3 Hp/A Ratio (for PFP evaluation)

In connection with fire analysis and evaluation of need for passive fire protection (PFP), the parameter: HpA is central when the necessary PFP thickness is chosen. HpA gives the cross section perimeter (i.e. surface) to cross section area (mass) ratio. A small ratio means the less vulnerable for heating.

The HpA ratio is found under Results/Global/Verify.

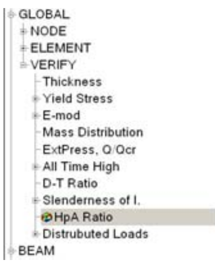


Figure 3-4 - The HpA Ratio is found under Global / Verify.

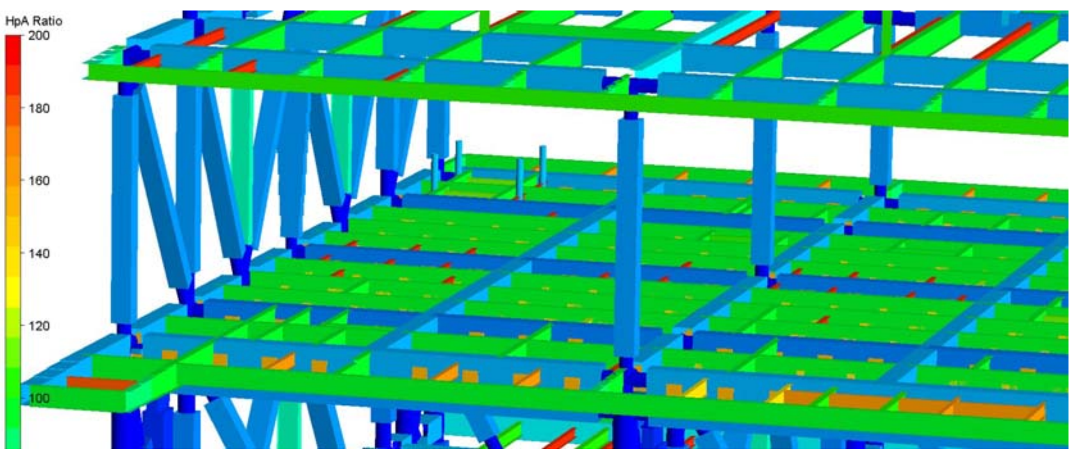


Figure 3-5 - Visualization of the Hp/A ratio in XACT.

3.3.4 History Plot of Beam Element Stress

The plot of stress as a result is in general limited to the elastic state. If the element yields, the linear stress distribution over the cross section used in the graphical plot (Beam Stress) is no longer valid. The stresses presented becomes higher.

So far this has also been the case when the History Plot of element stress are presented.

In version 9-0, the plot of element stress accounts for the yield stress of the actual element as shown in the image below, where the bending stress (dof-5) is plotted. The yield stress is 355MPa for this element.

NOTE!
Since this stress plot shows the stress for one degree of freedom at the time, the plot is strictly valid for cases where one force component (here the My moment) dominates the cross section axial stress.

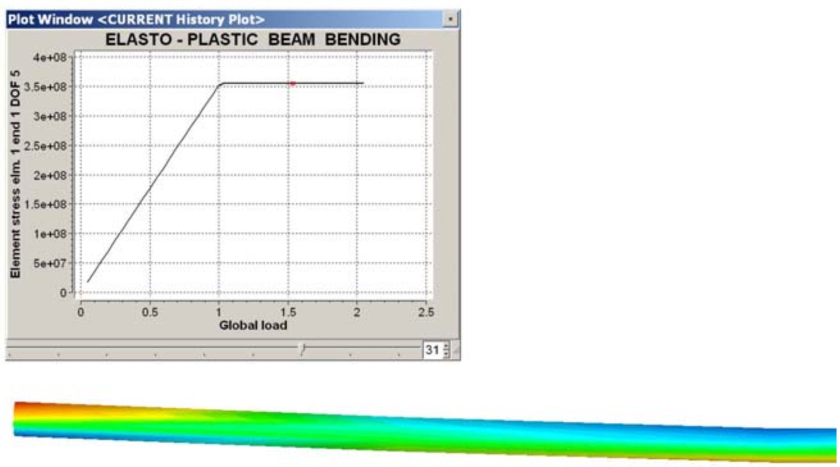


Figure 3-6 - Plot of Element Stress accounts for the yield stress.

3.4 Local Buckling

3.4.1 Bending Dent

The formulation of local denting of the tube wall is a function of the axial compression force in the pipes element. In the typical cases for frame structures with tension/compression members this criterion for dent growth is sufficient.

However, if there are cases with significant bending it could be of interest to include the bending as a trigger for the local dent. In v 9-0 it is possible to switch to an alternative dent formulation, where the dent growth depends on both the axial compression force and the bending.

It is done either element by element (command ELMPAR) or using the global SWITCHES command to apply on all pipes.

```
 '          Key       Value     ListTyp   Ids
  ElmPar  BendDent     ON        Elem     1
 '
```

Figure 3-7 - Switching ON the alternative formulation on one specific element

```
Switches    LocDent    BendDent    ON
```

Figure 3-8 - Switching ON the alternative formulation for all pipes.

It is recommended to introduce the alternative formulation in a “controlled manner”, which means on the elements with significant bending. In addition: always compare the response from the new formulation with the original formulation, (which has been used for more than 3 decades).

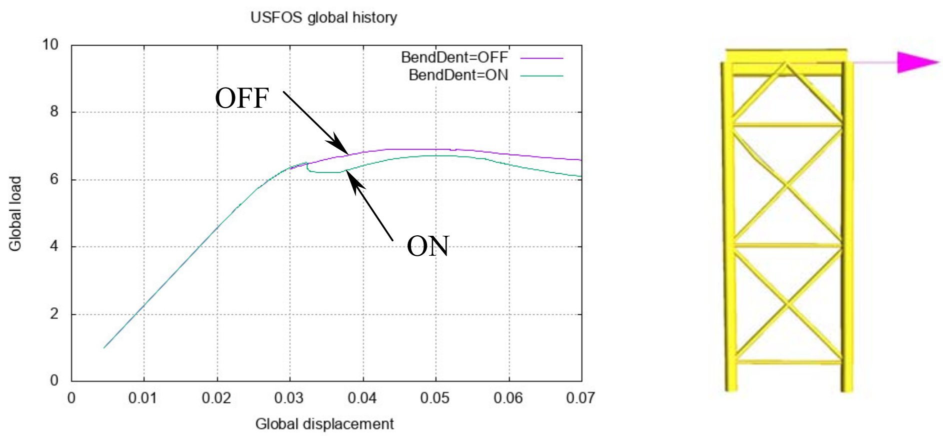


Figure 3-9 - Response with and without Bend Dent option for All

3.4.2 “Pancake”

Observations after extreme weather has shown that pipes with high D/T ratios could get a special local buckling mode: the so-called “pancake failure”, see Figure 3-10. The example shows the transition from 40mm to 60mm of pipes with outer diameter 2.0m.

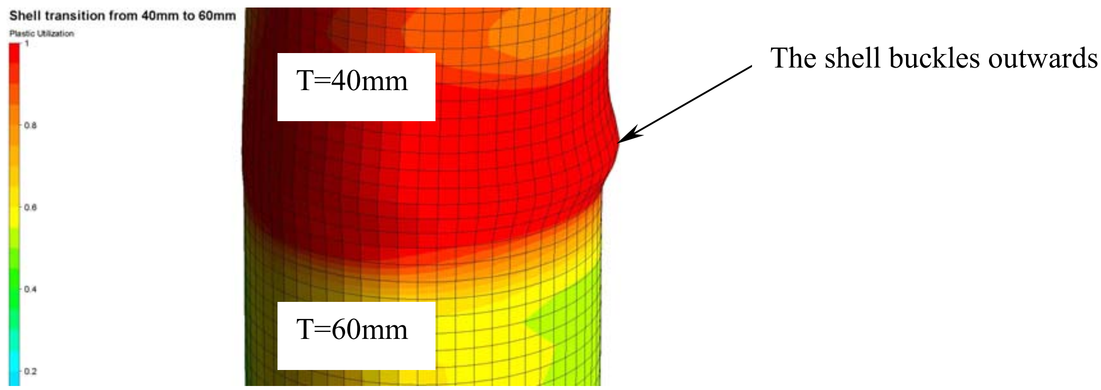


Figure 3-10 - "Pancake" buckling at the transition from 40mm to 60mm

USFOS beam element is extended to account for such “pancake” buckling. It is by default OFF, so the user has to switch ON this option using the following “Switches” command:

```
Switches Solution PanCake ON
```

If the pancake utilization exceeds 1.0, a message is printed to the terminal and to the out file as shown below.

```
                                                  12   YIELD    MID
   1   202    7.853  -0.004   2.202E-01   5.828E+04   MAX   DISPL    INCR
                                                  18   pCake    END2
   1   203    7.849   0.001   2.273E-01   6.051E+04   MAX   DISPL    INCR
                                                  11   YIELD    END1
```

```
     8   0 -0.38(-0.08)  -0.43(-0.14)  -0.32(-0.00)
     9   0 -0.33(-0.04)  -0.49(-0.24)  -0.56(-0.34)
    10   0 -0.52(-0.28)  -0.42(-0.15)  -0.34(-0.04)
    16                                               PanCake Fail at   end 2
    16   6 -0.33(-0.04)  -0.28(-0.00)  -0.29(-0.00)        +---0---0
    17   7 -0.29( 0.00)  -0.28(-0.00)  -0.20(-0.00)        0---0---0
```

Figure 3-11 - Print to Terminal and to OUT file

When the "PanCake Failure" is detected (pancake utilization > 1), the following happens with the element:

- The cross section capacity degrades.
- The degradation continues each step after the first detected. In practice this means that the element fractures gradually.

Figure 3-12 shows a simply supported beam, where the left half-part has D/T ratio 50 and the right part has D/T=100.

This means that the thickness ratio between thick and thin pipe is 2.0. A pipe with D/T=100 and a thickness ratio of 2.0 means that the "pan cake" capacity becomes ~87% relative to the perfect case, (with no thickness change).

The degradation takes some steps, and the peak shoots somewhat over the point where the pCake failure was detected. Then the beam starts the unloading (which will continue to ~zero). See also a comparison with and without the PanCake option (Figure 3-16 on next page).

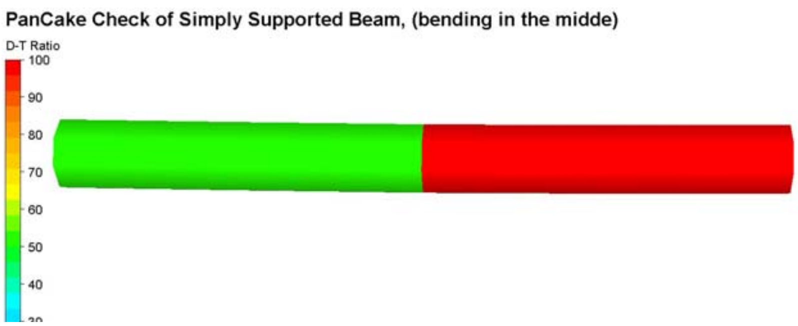


Figure 3-12 - Simply supported beam with load in the middle.

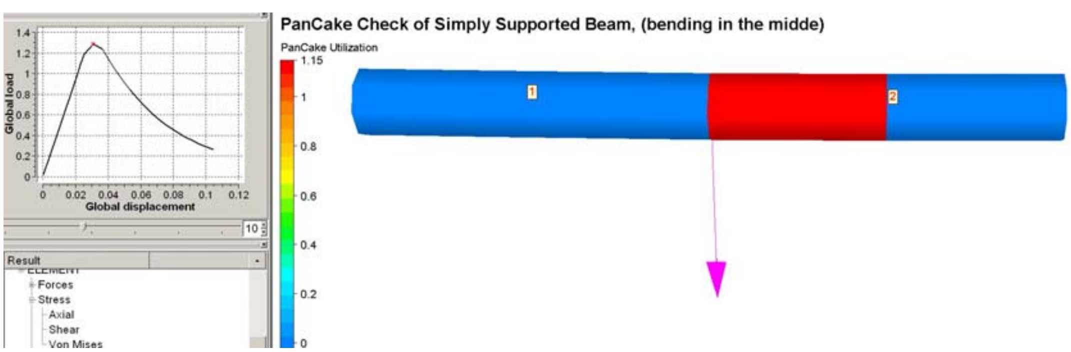


Figure 3-13 – PanCake utilization > 1 for the thin-wall element.

The comparison clearly shows how the conventional model keeps the resistance for increasing displacement. The model with pan cake fails.

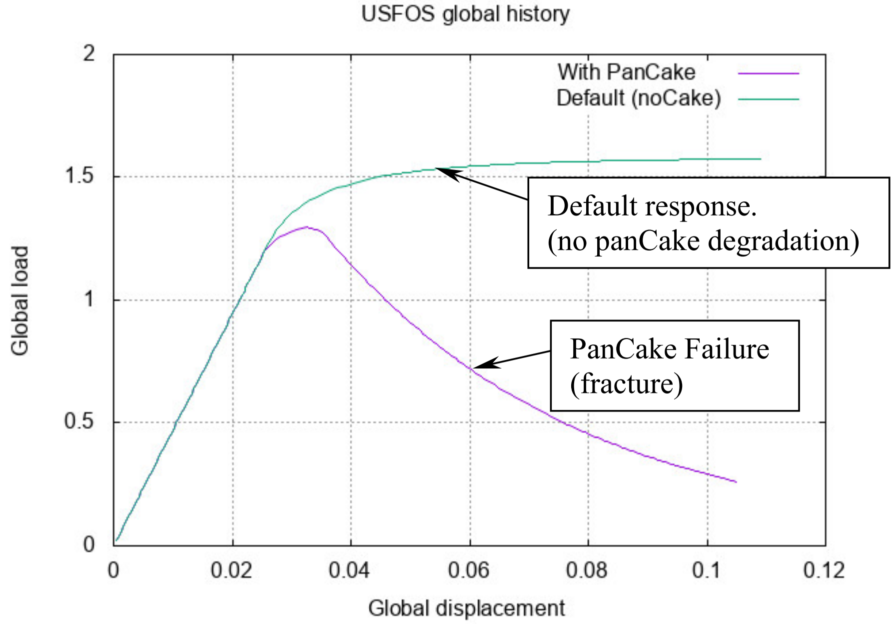


Figure 3-14 - Global Response. Comparison with and without Pan Cake switch.

Figure 3-15 shows a 2-D X-braced frame with a horizontal force at the top. The legs have inner piles, which means that the leg to the left gets compression (since the pile has tension).

For demonstration purpose, the legs have thickness change in the middle.

In this case, the pancake utilization exceeds 1.0, and this has some impact on the global response, see Figure 3-16 for comparison Switch = ON/OFF.

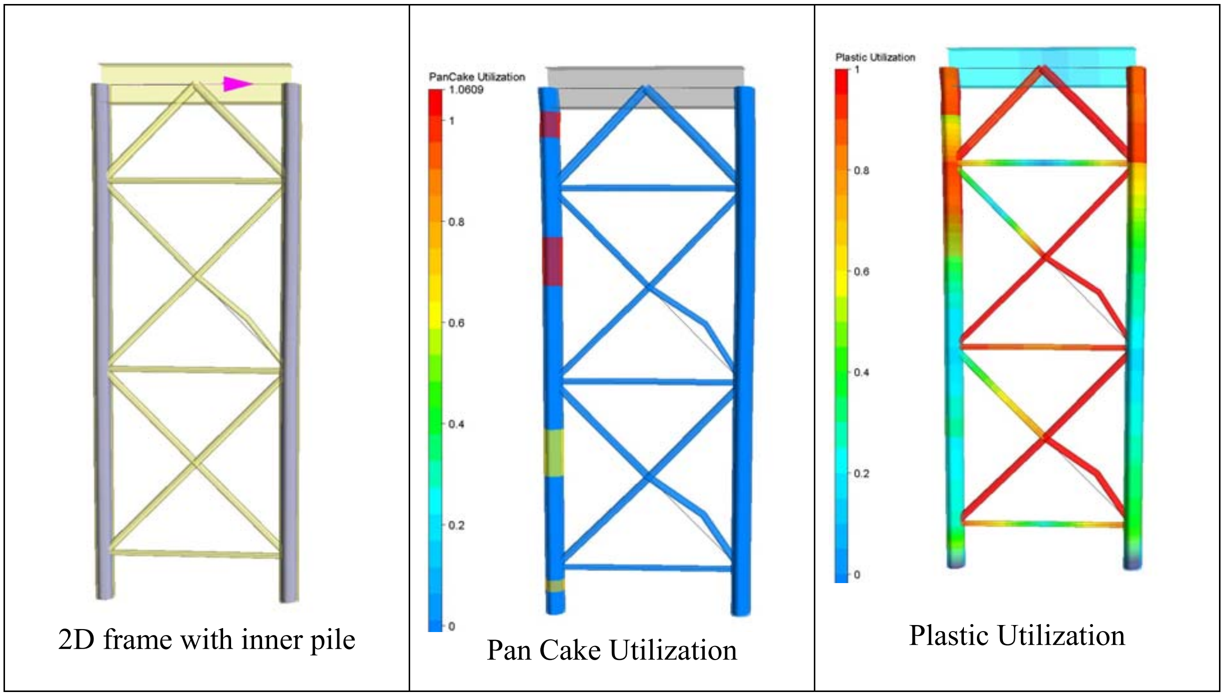


Figure 3-15 - 2D frame with inner piles.

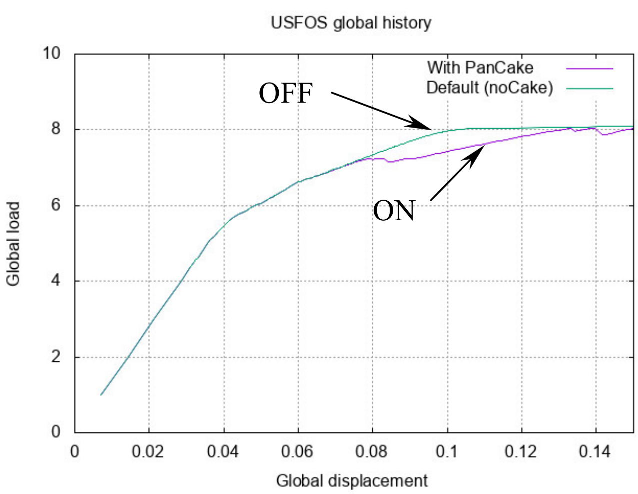


Figure 3-16 - Response with and without pancake option.

If a large number of simulations are performed, it could be useful to run the utility tool BUCCAKE. The tool will check all RAF files given to the tool (the raf file names are stored in a text file). The tool gives max utilization per RAF file and the “n” highest utilization per RAF file in addition.

Table 3-1 - Print from Utility Tool "BucCake"

```
------------------------------------------------
------------------------------------------------
------------------------------------------------
-----         B u c  C a k e            ------
-----  Checks pipes for pancake buckling  ------
-----   Version 0.9-01 / Feb 26, 2021     ------
-----              Usfos As             ------
------------------------------------------------
------------------------------------------------
# Checks Entire Simulation


# File  Name    pCakeUtil       Elem    Level    GlbDisp
   1  res         1.061          18     7.897     0.251


# =====================================================
# File no    1        :  bug.raf
# -----------------------------------------------------
  Load Case           :         1
  Highest Load Level  :      8.103
  Global Displacement :      0.174

        Elem    pCakeUtil  Level
          18     1.061     7.897
          16     1.050     6.999
          20     0.811     8.017
          22     0.754     8.103
           1     0.000     0.000
           2     0.000     0.000
           3     0.000     0.000
           4     0.000     0.000
           5     0.000     0.000
           6     0.000     0.000

#   --------  e o f --------
```

3.5 Hydrodynamics

3.5.1 Concentrated (nodal) added mass

Concentrate hydrodynamic mass could be assigned to nodes. This is done using the command “HydMass” with option “Node”. The masses are referring to the Global Coordinate System.

```
'        type     opt      Mass[kg] ListType IDs
HydMass   NODE    XTransl   1E4      Node     9  ! 10,000kg in X-dir. Node  9
HydMass   NODE    ZTransl   3E4      Node     9  ! 30,000kg in Z-dir. Node  9
HydMass   NODE    XTransl   1E4      Node    10  ! 10,000kg in X-dir. Node 10
```

In this example, 10,000kg is added in X-dir and 30,000kg in Z dir for node 9. Node 10 gets 10,000kg extra added mass in X-direction.

In the print file (“res.out”), the total concentrated additional drag areas are printed together with the other hydrodynamic masses.

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

           Mass item                   GlbX        GlbY        GlbZ
                                      [ kg ]      [ kg ]      [ kg ]
           HydroDyn Added Mass :   2.737E+05   2.737E+05   0.000E+00
           Internal Fluid Mass :   0.000E+00   0.000E+00   0.000E+00
           Marine Growth  Mass :   0.000E+00   0.000E+00   0.000E+00
           Nodal Added    Mass :   2.000E+04   0.000E+00   3.000E+04

           Total               :   2.937E+05   2.737E+05   3.000E+04
```

By default, only the inertia is accounted for. However, if the ”Coeff” option is used in addition, the wave acceleration forces are computed.

The coefficients are given for X-, Y and Z directions. The default coefficient is Zero, so only coefficients > 0 need to be given.

```
'
'           type     opt     Coeff  ListType   IDs..
 HydMass   NODE   Coeff_X   1.0    Node       9  10  ! Cm = 1.0 in X-dir
 HydMass   NODE   Coeff_Z   1.1    Node       9      ! Cm = 1.1 in Z-dir
'
```

The mass coefficients for the nodes with specifications are printed in the “res.out” file:

```
                 ----  C o n c e n t r a t e d   M a s s  C o e f  ----

           Node                CmX          CmY          CmZ
              9            1.000E+00    0.000E+00    1.100E+00
             10            1.000E+00    0.000E+00    0.000E+00
```

For the nodes/dofs with Coefficients specified, the force from a concentrated mass is computed as:

Force = Cm x Acc_{REL}, where

Acc_{REL}, is the instantaneous *relative* acceleration between the node and wave particle.

If a node goes in and out of water, and since the node has no extent, a smoothing procedure is introduced. It ensures a gradual increase/decrease of both the added mass and the mass forces.

3.5.2 Concentrated (nodal) drag

Concentrate drag could be assigned to nodes. This is done using the command “HydroXtra” with option “NodeDrag”. The “Glob” parameter means that the drag areas are referring to the Global Coordinate System.

The drag areas are given as Cd x Area for the three axis directions.

The formula: Fx = ½ ρ CdA_X (U_X U_{RES}) is used for computing of forces in the global X-direction, where:

U_X is the relative velocity in X-direction at actual node
U_{RES} = SQRT ($U_X^2 + U_Y^2 + U_Z^2$): The length of the velocity vector.

```
'
'              key        opt      CdA_X  CdA_Y CdA_Z  ListType    IDs...
 HydroXtra  NodeDrag    Glob     1000   1000   0       Node        5  7
```

In the example, 1000m^2 is used for the drag in X- and Y directions and no drag in Z direction. This extra drag is assigned to nodes 5 and 7.

As Figure 3-17 shows, the total concentrated force from the actual example is unaffected by the direction of the wave/current.

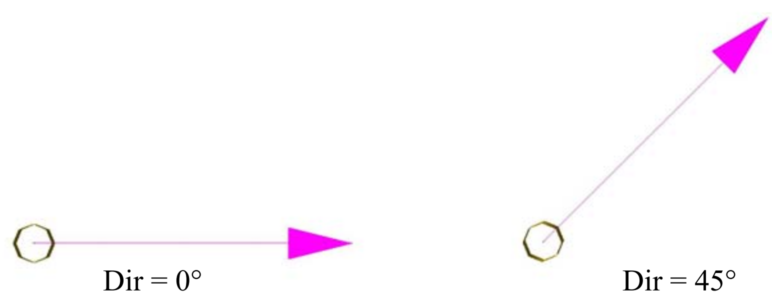


Figure 3-17 - The total drag (length of force vector)

In the print file (“res.out”), the total concentrated additional drag areas are printed. (located just after the sum of CdA’s from the beam elements).

```
'
     ----  Additional CdA [m2] from Nodal Drag Areas     ----

                X           Y           Z
           2.000E+03   2.000E+03   0.000E+00
```

3.5.3 Wave Kinematics Reduction. 3D Profile.

If the wave kinematics reduction option "WAVE_KFR" is used, the same reduction factor is used for all depths and all directions: X, Y and Z.

With the WAVE_KFR *profile*, it is possible to use different factors for the different depths. USFOS will then use the interpolate value based on the specified table. The element's initial position is used.

The selected factors are printed in the "res.out" file. The printed table is extended with factors for X- Y- and Z-directions. (Ref 3D_profile below).

```
'
'          Type       Z         Kfr
 Wave_KRF  Profile   10         1.0
                      0         1.0
                     -5         0.8
                    -10         0.7
                    -20         0.5
                    -90         0.5
'
```

```
'
             Elem     Wave     Integr   Marine  Marine Int   Dir    Wave   Wave
              No      Kinem    Points   Growth  Growth Fluid Dep    Kinem  Kinem
                      Red-X             Thickn. Dens.  Dens  Coeff  Red-Y  Red-Z
                 1     0.50         2     0.00   1025     0     0   0.50   0.50

                 8     0.60         2     0.00   1025     0     0   0.60   0.60
                 9     0.80         2     0.00   1025     0     0   0.80   0.80
                10     1.00         2     0.00   1025     0     0   1.00   1.00
'
```

If different reduction factors are needed for the different degrees of freedom, the 3D_Profile option could be used.

```
'
'                            Z    KRF_X      KRF_Y     KRF_Z
 Wave_KRF    3D_Profile   20.0   1.00       0.00      0.00
                           0.0   1.00       0.00      0.00
                          -7.0   1.00       0.00      0.00
                         -20.0   1.00       0.00      0.00
                         -90.0   1.00       0.00      0.00
'
```

3.5.4 SpoolWave. Wave Height. (Max Crest to Trough)

The “SpoolWave” option is extended to find the maximum wave height. This is defined as the maximum crest-to-trough. The “time-before-peak” specifies the time where the max crest occurs in this max-height wave. Always ensure to have sufficient “time-before-peak” to initialise the structure.

```
' ======================================================================
'   Search for the highest peak within a 1 hour storm, with steps =0.5s.
'   Spool the wave up to 20 sec before the peak.
' ----------------------------------------------------------------------
'
'            TimeBeforePeak   Order  dT  StormLength   Crit
 SpoolWav          20           1    0.5    3600      Height
'
```

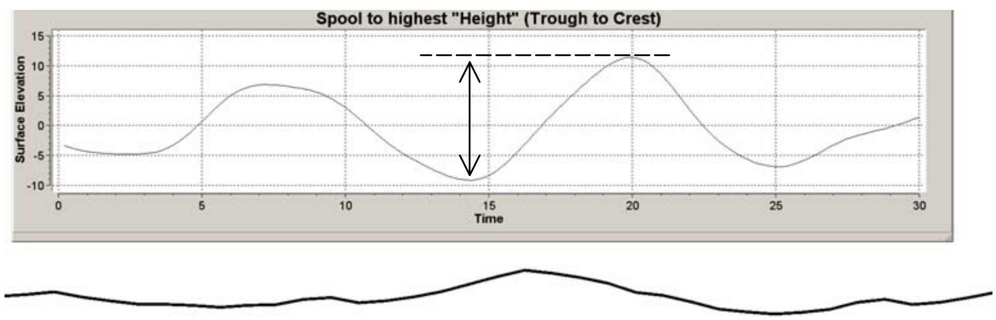


Figure 3-18 - Wave Elevation History. Max Height after 20sec

3.6 Solution

3.6.1 Hinge Restrictions on short elements

The USFOS element was designed for handling the physical member with only one beam element. When needed, a plastic hinge is introduced in the ends and at the middle.

However, the user often let the model pre-processor (e.g. GeniE) sub-divide the member in to many short elements. This could lead to a series of plastic hinges close to each other (like a “bicycle chain”), and the solution could fail.

It is therefore introduced a new option, which gives better numerical stability. It works as follows:

- The user specifies a certain L/D ratio (max length where the new option is used)
- USFOS will create a list of such short elements.
- If a plastic hinge is inserted on an element on this list, the other two hinges are flagged with restrictions. No more hinges within that short element is possible.
- A message is printed on the terminal each time such restrictions are introduced.
- After termination, a list of the elements, which have been restricted, is printed (see below). The syntax is the same in this print as used for the manual “PlastHin” option, where “1” means that hinge was suppressed. “0” tells where the hinge was introduced.

The option is *activated* using the following command:

```
'
'                              L/D
 Switches Solution HingeLim  3.0
```

Elements with “L/D ratio” shorter than 3.0 will be considered in this example.

```
                 --------------------------------------------------------
                 ----    H i n g e    L i m i t a t i o n s        ----
                 ----            S u m m a r y                     ----
                 ----                 o f                          ----
                 ----    R e s t r i c t e d   H i n g e s         ----
                 --------------------------------------------------------

                       Element       Hing1       Hing2      HingMid
                         1026          1           0           1
                         1028          1           0           1
                         1023          0           1           1
                         2124          1           0           1
                         2308          0           1           1
                         2134          0           1           1
```

3.6.2 Check of "crazy" Global Displacement

If, of some reason, the solution fails, the computed global displacement could end as "not a number", NaN or "Inf". When this number is stored on the RAF file, it "messes up" the global disp/load plot in XACT for all steps thereafter.

In USFOS 9-0, there is a check for such "rotten" global displacements, and, if detected, USFOS will terminate the simulation without storing that step.

3.7 Foundation

3.7.1 Spring Displacement (SpriDisp and MovSpri). Time Domain

A new one-node spring to ground is available in USFOS: a “moving spring”. In practice it means that the ground, to which the spring is attached, could move.

In the simple example below, the ground moves according to earthquake history. Figure 3-19 describes the input:

- Springs are defined in the two frame supports
- The spring is linear with stiffness given in global coordinate system
- The spring motion (i.e. ground motion) is defined using the new SPRIDISP command (load case 10).
- The input is identical to “NODEDISP” except that it is referred to a spring element and not a node.

NOTE! This spring requires iterations.

```
'                                       Define Spring supports
'           elm   nod   mat
 Sprng2Gr  10120  120  2000
 Sprng2Gr  10130  130  2000
'                                       Define Special Spring Material

'            ID     Type    opt  s11 s22 s33   s44  s55  s66
 Material   2000  MovSpring Lin  5e7 1e8 1e8   1e8   0   1e8

'                                       Define Spring movements

'           LC   Elem   DofCode     Value(s)
 SpriDisp   10  10120      1        0.001
 SpriDisp   10  10130      1        0.001
```

Figure 3-19 - Definition of the new spring displacement option.

For comparison:
If the conventional spring to ground were used, the “old” NODEDISP will prescribe nodes 120 and 130 to follow the motion defined in the time history, 1:1.

```
'           load-case   node-no   DofCode  Value(s)
 NODEDISP       10        120        1     0.001
 NODEDISP       10        130        1     0.001
'
```

Figure 3-20 - Definition of the “old” prescribed node displacement.

The differences between the two cases (SPRIDISP vs. NODEDISP) are:

- When SPRIDISP is used, the foundation has elastic stiffness also in the motion degree of freedom (in this case, X-direction, which has a stiffness of 5.0E7)

- When NODEDISP is used, the foundation has no flexibility in X-direction. The motion will follow the earthquake time history 1:1. And: there is no possibility for vibrations of the prescribed degree of freedom.

Figure 3-21 shows a comparison of the frame top for the two cases. SPRIDISP gets more response. The displacements in the foundation node of the frame are compared in Figure 3-22. It shows that the SPRIDISP will allow the foundation to move more than the prescribed motion. The higher stiffness the spring has in X-direction, the closer will the two responses become.

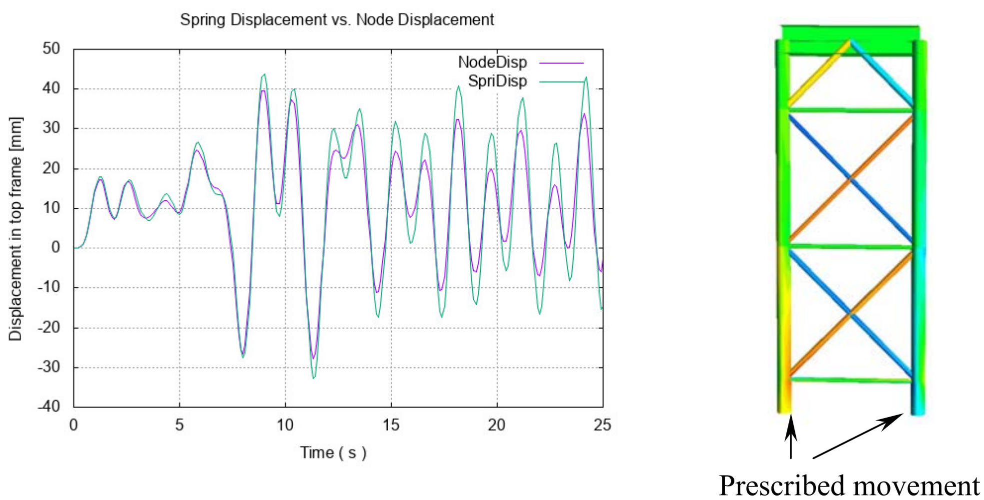


Figure 3-21 - Displacement history of frame top. SPRIDISP vs. conventional NODEDISP.

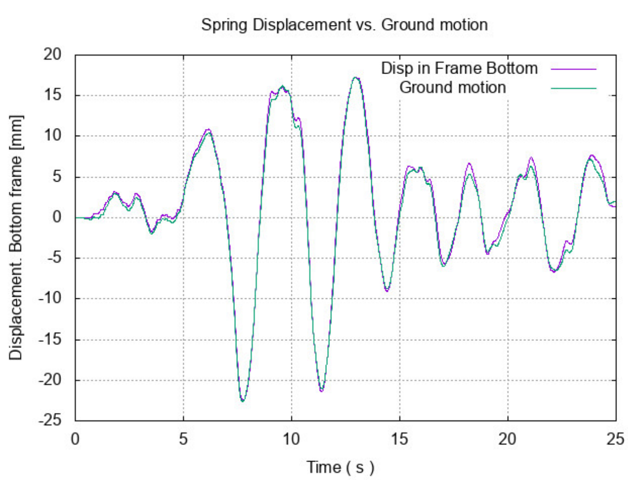


Figure 3-22 - Displacement history of frame bottom. SPRIDISP vs. conventional NODEDISP

3.8 Misc

3.8.1 Element Temperature. Material input

If the same temperature needs to be assigned to many elements, the ELEMTEMP option “Mat” could reduce the input as shown below.

```
'
'              LC   Type    Mean    GradY   GradZ ListTyp    Mat ID...
 ElemTemp       4     1      800    0.0     0.0     Mat       1  2  3
'
```

All elements with material 1, 2 or 3 will be given a mean temperature of 800.

3.8.2 AccField. Centrifugal Acceleration

If the effect of rotation should be modelled, the ACCFIELD extension “Centrif” could be used.

The example shows a wind turbine blade exposed to typical lift forces. The real rotor spins with a speed around 10 RPM, and the acceleration field, ($a = \omega r^2$), will set up tension forces in the blade, (see Figure 3-24). The rotation speed is here set to 1.0 rad/s. (corresponds to ~10 RPM).

Load case 5 will represent this “centrifugal” force.

```
'
'             LoadCase KeyWord  Type   Omx   OmY OmZ  X   Y   Z
 AccField        5     Centrif  Stat    0     0   1   0   0   0
'
'
```

Figure 3-23 - "Centrifugal" forces. Rotation about Z-axis, 1 rad/s about point (0,0,0).

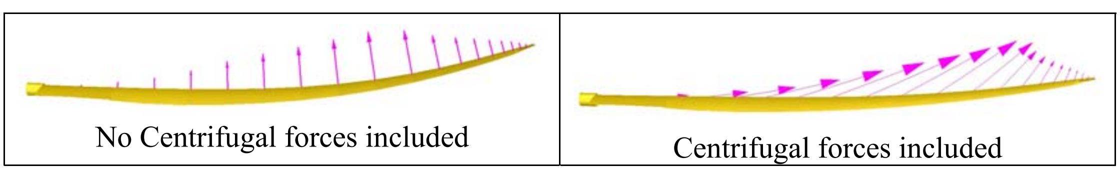


Figure 3-24 - Applied loads. With and without centrifugal forces.

3.9 SWITCHES, (Special Options).

The command "SWITCHES" was introduced in 8-5 to switch on special options and is extended in version 9-0. Following "Switches" commands are available, (**sub keys** in bold are new):

KeyWord	SubKey	Value	Description	Default
General	*IndefLimit*		Min / Max imperfection (in CINIDEF).	0.05 / 1%
Defaults	*Version*	ver	850: switch to version 8-5 defaults	890
WaveData	*TimeInc*	val	Time between each hydrodyn calc.	Every
	NoDoppler	-	Switches OFF Doppler effects.	ON
	NoStore		Switches OFF storing of wave data for visualize.	ON
	TidalLevel	Level	Specify Tidal Level	0
	Accuracy	val	Change accuracy. 0: old accur, 1: new accur	1
	SeaDim	X , Y-dim	Specify size of sea surface used in xact	2 λ
	StreamOrd	order	Stream Function order	10
	SecOrder	ON/OFF	Switch on Second order wave selected members	OFF
	KC_CdDepth	ON/OFF	Reference Z. ON: Elem location, OFF: MWL	ON
NodeData	*DoublyDef*	ON/OFF	ON: Accept doubly defined nodes with same coo	OFF
StatusPrint	*MaxElem*	val	Max element in status print	10
Iterations	*RLF_Calc*	-	Activate "Residual Load Factor" method	OFF
Write	*FE_Model*	IDAdd Case stp	Writes deformed FE model at given case stp	OFF
	LinDepAlt	-	Writes ZL-springs for each BLINDP2	Off
Solution	*FracRepeat*	MxRep	Max fracture repeat	10
	PlateEdge	ON/OFF	Avoiding I-girder to buckle about weak axis if the beam element is attached to a plate element	OFF
	Impact	*UnLoFact*	Load factor during unloading after boat impact	0.02
	PanCake	ON/OFF	Account for "pancake" failure	OFF
	HingeLim	L/D	Restrict hinges for short elements. Recommended	OFF
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.

KeyWord	SubKey	Value	Description	Default
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
Joint	*ShortCan*	ON/OFF	Detect and account for short can effect	OFF
	EccUpdate	ON/OFF	"Repair" joint ecc. to avoid short joint elements	OFF
	EyeLift	Val	Location of joint surface node. 1.0 is on leg surf.	1.2
FE_Model	*Hing2Elm*	ON/OFF	Replace BEAMHING with ZL-spring	OFF
	Hing2Elm	HingStf	Specify Stf of "fixed" dofs	Estim.
	Hing2Elm	ReleaseS	Specify Stf of released dofs.	0.0
	Hing2Elm	IDAdd	Specify number to be added to generated IDs	77E6
Soil	*DiscVisual*	Val	Specify PY and TZ relative weight factor for size	1 100
DentPlot	*Store*	ON/OFF	Stores dent depth to be visualized in XACT	OFF
ShellOpt	***AxCorrect***	ON/Off	Corrects under prediction of Axial resistance	ON

3.10 Updates Usfos and Utility Tools

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

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

3.11 New/modified input commands

Since last main release (8-9), following input identifiers are added/extended:

SPRIDISP :	New	*: Spring Displacement*
HYDROXTRA:	New	*: Special Hydrodynamic data*
MATERIAL	Extended	: *MovSpri (used with SpriDisp)*
SPOOLWAV	Extended	: *Wave Height (Crest to trough)*
WAVE_KRF	Extended	: *Profile: Different reduction in space.*
SWITCHES :	Extended	*: See above.*
ACCFIELD	Extended	: *Centrifugal forces.*
DYNRES_G	Extended	: *Distance between nodes.*

3.12 Documentation

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

- User's manual : Updated document (user's)
- Joint Capacity Theory : Updated document (theory)