USFOS
Reality Engineering

USFOS AS

Phone: +47 905 05 717

www.USFOS.com

Enterprise No.: NO 986 827 374 MVA

MEMO

MEMO CONCERNS

Release Notes
USFOS Version 8-5

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	PERSON RESPONSIBLE / AUTHOR	NUMBER OF PAGES
	2010-01-01	Tore Holmas	22

Release Notes

USFOS 8-5, Jan 2010

CONTENTS:

1 Introduction

The current official version of USFOS is version 8-5 with release date 2010-01-01. 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.

2 News in USFOS version 8-5 - 2010.

2.1 Introduction

The new features are described by examples located on the web site, and reference to the actual example(s) will be given for the different new options.

2.2 How to upgrade your USFOS version

From release 8-4, USFOS could be upgrades in two different ways:

- Alt 1: Download the new "*setup.exe*" and u-install/install USFOS, (same as for release 8-4). 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).

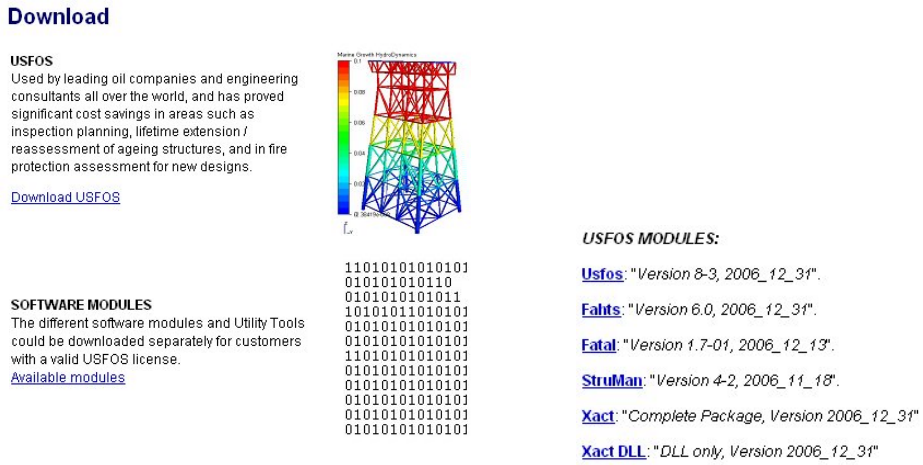

Figure 2-1 Download complete USFOS installation setup or the modules one by one

2.3 Enhanced Graphical User Interface

The graphical user interface (*xact*) has been enhanced since last year's release. The GUI version released together with USFOS 8-5 is "2.5". Check under help/about to ensure that the latest version is installed.

2.3.1 Vector Results

A new result type is introduced. It is called "Global Vector", and at present, the following sub-menus are implemented:

- Verify
  - Nodal Mass
    - Fixed (NodeMass defined by the user)
    - Movable ("Weights". New Option)
    - All (Sum of Fixed masses and movable)
  - Applied Load (The instantaneous magnitude of the applied loads, which is the accumulated "right hand side" of the equation system. Only translation degrees of freedom are shown).
  - Nodal Loads (Basic Node Loads defined by the user)

Figure 2-2 shows the "Applied Loads" for an example with a rotating vector field applied. The Maximum Length of the vector is adjusted relative to the size of the structure or given absolute scaling factors. The vector scaling could be adjusted (Verify/ Vector Settings, see Figure 2-3). It is also possible to define a "filter", where vectors bigger than a specified value are removed. This could be useful if the different forces have very different size, (f ex hydrodynamic forces vs. topside weight).

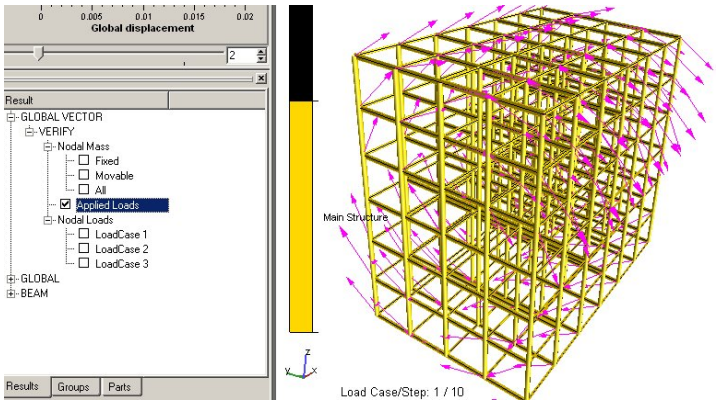

Figure 2-2 Vector Results. Applied Loads, Rotating Acceleration field

By default, following settings are used:

- Absolute scaling factor (computed by xact)
- Line thickness = 1
- Pink colour
- Arrow heads visualized
- No filtering

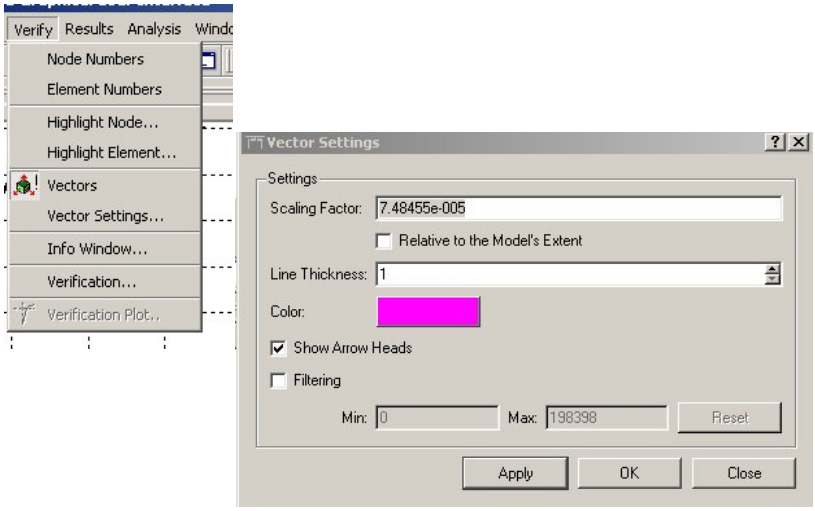

Figure 2-3 Vector Results. Vector Settings

2.3.2 Line Modus / Part visualization.

The part visualization is extended as follows:

- A separate “tab” where visualization of the different parts are switched ON/OFF
- An extra part, named “BeamCenterLine” is automatically created. This is a “line-model” (not surface), going from end-1 to end-2 of the flexible part of a beam element, (offsets are included).

By switching OFF the surface visualization, the line model appears. Selection of elements and nodes (f ex in connection with history plot or verification) could be done by point/click also on the line representation.

It is recommended to utilize the part definitions to enhance the information of the model. (See example on the web: *http://www.usfos.no/examples/usfos/misc/partdata1/index.html*).

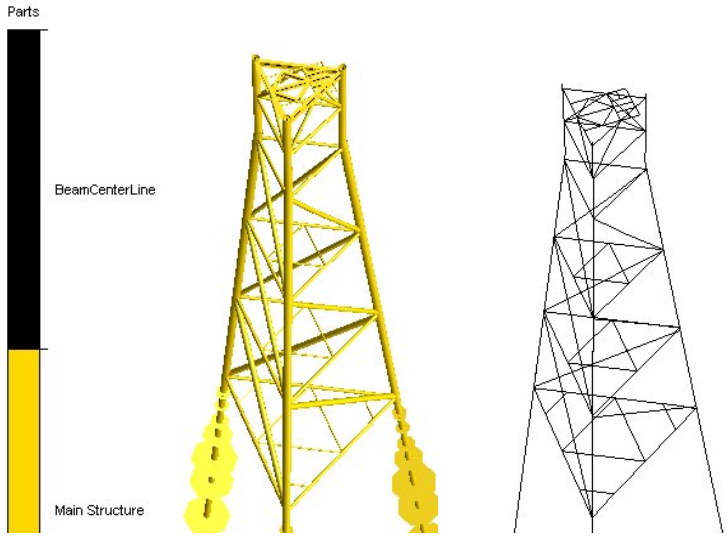

Figure 2-4 Line modus visualization. Image to the right shows “*BeamCenterLine*” only.

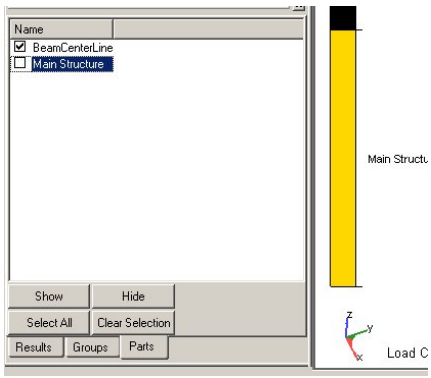

Figure 2-5 Use the “Parts” tab to switch ON/OFF visualization of parts.

2.3.3 Material plot.

The “Material Plot” presents the normalized interaction between bending moment and axial forces, which his central for the USFOS beam element cross-section capacity. The “old” material plot visualization is modified slightly. Higher point density gives a smoother graph and better impression of the physical representation of the cross section strength parameters.

It should be emphasized that these plots are generated in the post processor. The point of interest on the surface curve is the *instantaneous force* point (small rectangle). The rest of the surface curve is made in order to get the overview.

The force point path describes the actual combination of bending moment and axial force from step 1 and up to the actual step.

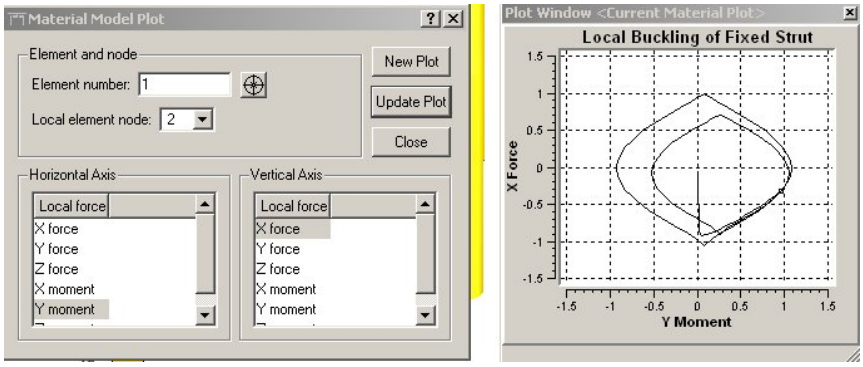

Figure 2-6 Improved quality of the material surface plot.

2.3.4 Dent Depth plot.

When a local dent is introduced on the pipe wall, the cross section capacity will be influenced. This is visualized as a “kink” in the yield surface. (see Figure 2-7).

The development of the dent growth could be visualized from the History Plot / Element / Dent depth). Both the current depth and the permanent could be visualized).

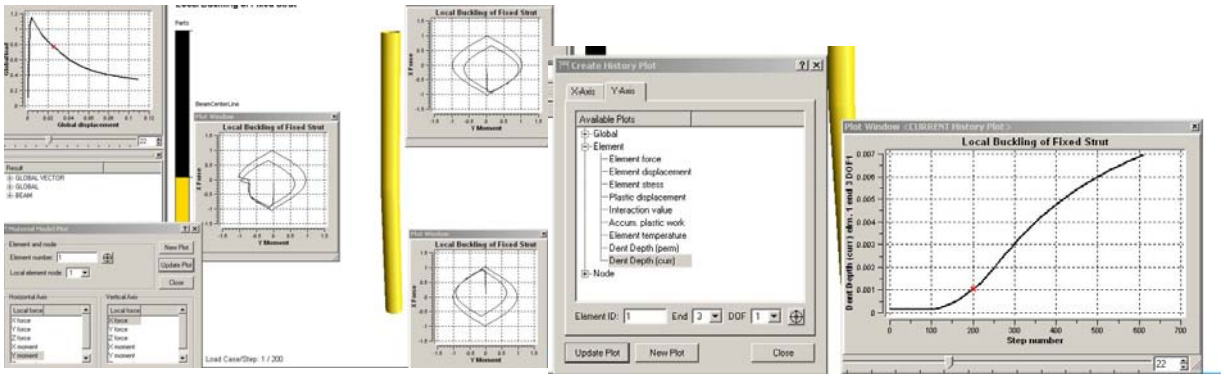

Figure 2-7 Visualization of Current Dent Depth of middle section of the column.

2.4 Hydrodynamics

The hydrodynamic module has been improved and extended in release 8-5. A new manual is also available on the web: *http://www.usfos.no/manuals/usfos/theory/documents/Usfos_Hydrodynamics.pdf*

The extensions covers the following:

2.4.1 Irregular Waves / User defined wave spectrum

If special wave energy spectra should be used in connection with irregular wave simulations, the "User" options should be used. The energy S(ω) vs. ω given as discrete points, where **ω** is the angular frequency. Figure 2-8 describes one example, where the energy is plotted as function of angular frequency (which is the input syntax) and as a function of period [s], which often is easier to interpret. To ease the checking, USFOS prints an echo of the read-in energy spectrum as function of both angular frequency and period. An example is found on the web, (see link below).

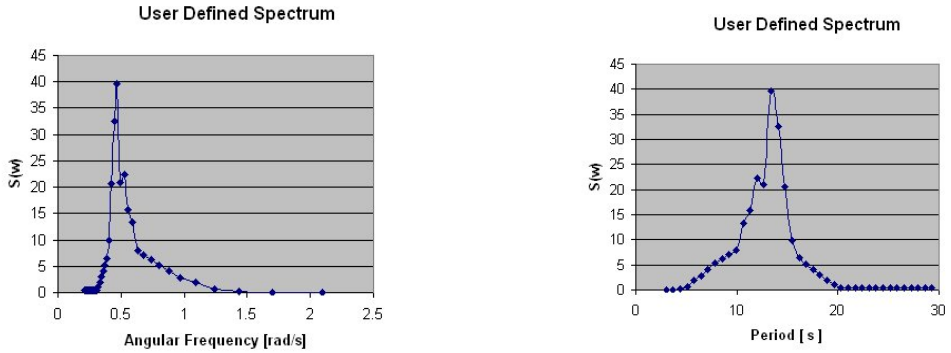

Figure 2-8 Wave energy Spectrum vs angular freq (left) and vs Period (right)

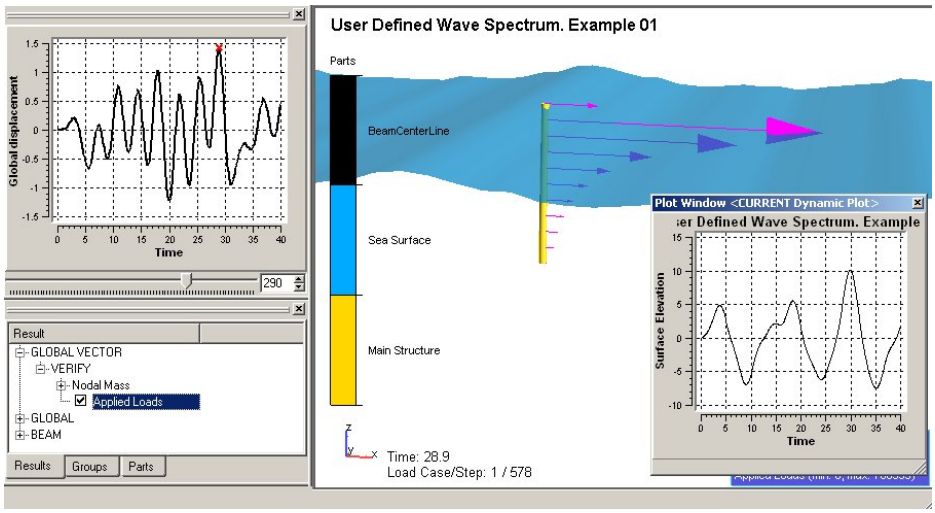

Figure 2-9 Visualization of instantaneous forces acting on the cylinder.

See: *http://www.usfos.no/examples/usfos/hydrodynamics/wave_spect2/index.html*

2.4.2 MacCamy Fuchs

If the “MacCamy Fuchs” correction for large volume cylinders should be included, this could be specified element-by-element using the following command:

```
'            KeyWord   Value   ListType  ID's
 HydroPar  McCamyFucs    ON     Elem ....
 HydroPar  McCamyFucs    ON     Mat  ....
 HydroPar  McCamyFucs    ON     Elem ....
```

Or just

```
 HydroPar  McCamyFucs    ON     All    ! Apply to all beams
```

For further description, see manual:

http://www.usfos.no/manuals/usfos/theory/documents/Usfos_Hydrodynamics.pdf

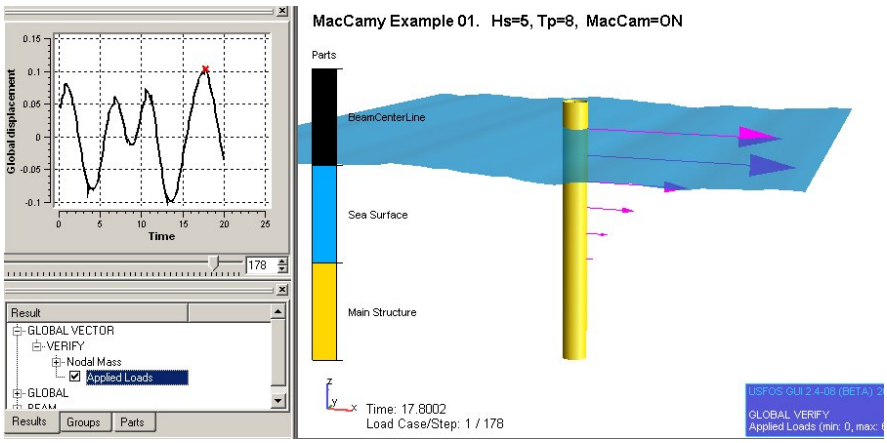

Figure 2-10 Large volume cylinder. McCamy – Fuchs correction ON

The example is found on the web, see:

http://www.usfos.no/examples/usfos/hydrodynamics/MacCamy_Fuchs/index.html

2.4.3 Slamming forces on Tubular Members

If slamming is an issue for elements in the splash zone, the following command

```
'          KeyWord   Value   ListType  ID's
 HydroPar  SlamCalc   ON      Elem ....
```

switches ON the special slamming calculation on the selected beam elements. (the other "*ListTypes*" are also available).

The following procedure is used:

- When the wet surface is moving upwards relative to the pipe, the slamming calculation is activated.
- Only pipes with less inclination than 15° are checked
- Slamming calculations are performed for every wave integration point (ref `WAVE_INT`)
- The submerged part of the cross section, S, is computed, and the S/D ratio is used to compute the current drag coefficient. See Figure 2-11 and Figure 2-12 .

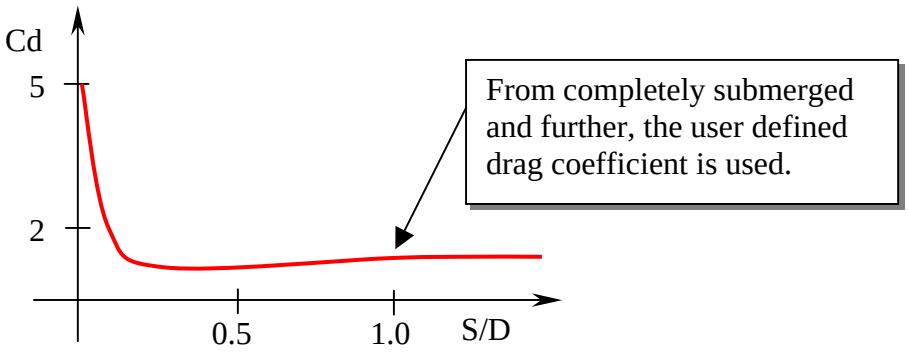

Figure 2-11 Drag coefficient as a function of submerged/Diameter ratio.

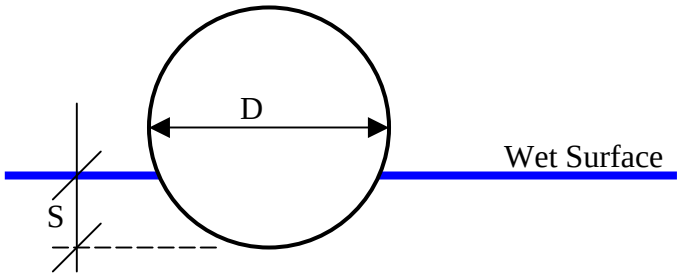

Figure 2-12 Definition of *submerged* part , S.

Figure 2-13 and Figure 2-14 describe a wave hitting a member. In the first case, the member is oriented normal to the wave crest, and the slamming forces will start at the left end and “travel” towards the right side. In the other case, the member is oriented parallel with the wave crest, and the slamming will be introduced over the entire member simultaneously.

The main advantage with this feature is to get a precise and continuously updated drag coefficient instead of using an artificial high coefficient for the entire member throughout the simulation.

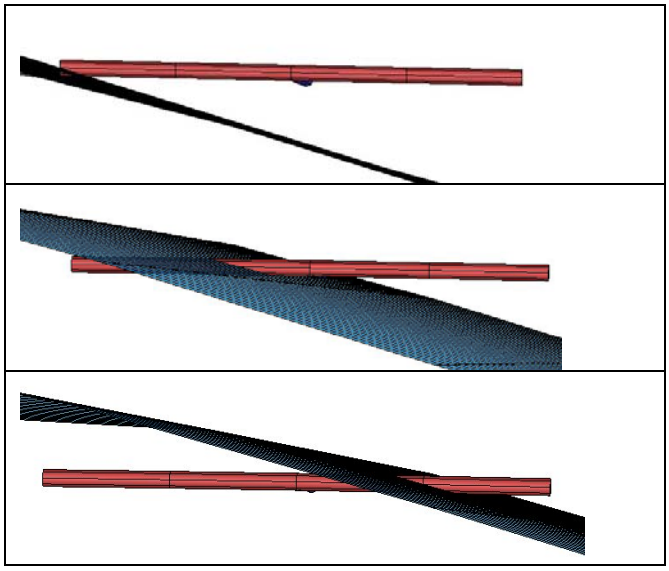

Figure 2-13 Member oriented normal to the wave crest.

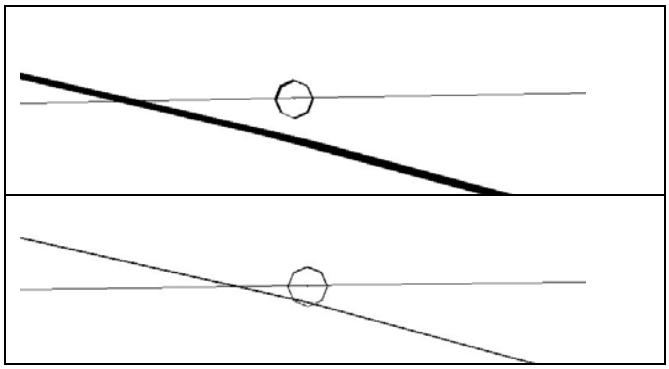

Figure 2-14 Member oriented parallel with the wave crest.

Figure 2-15 presents the member reaction forces at end-1 (left) and end-2 (right) of the member, which his oriented normal to the wave crest. The two upper plots have no slam calculation included, (Cd=1.0 always), and the two lower plots represents the simulation with slamming ON.

As seen from the plots, approx 20% higher forces are observed for the case with slam calculation ON. (180 kN vs 150kN and 150kN vs 120kN).

Other cases could give different results, but it should be emphasized that the observed forces during this kind of dynamic simulation are relatively small compared with other more conservative methods, (like using the peak slam Cd, which could be in the order of 5 as a constant coefficient).

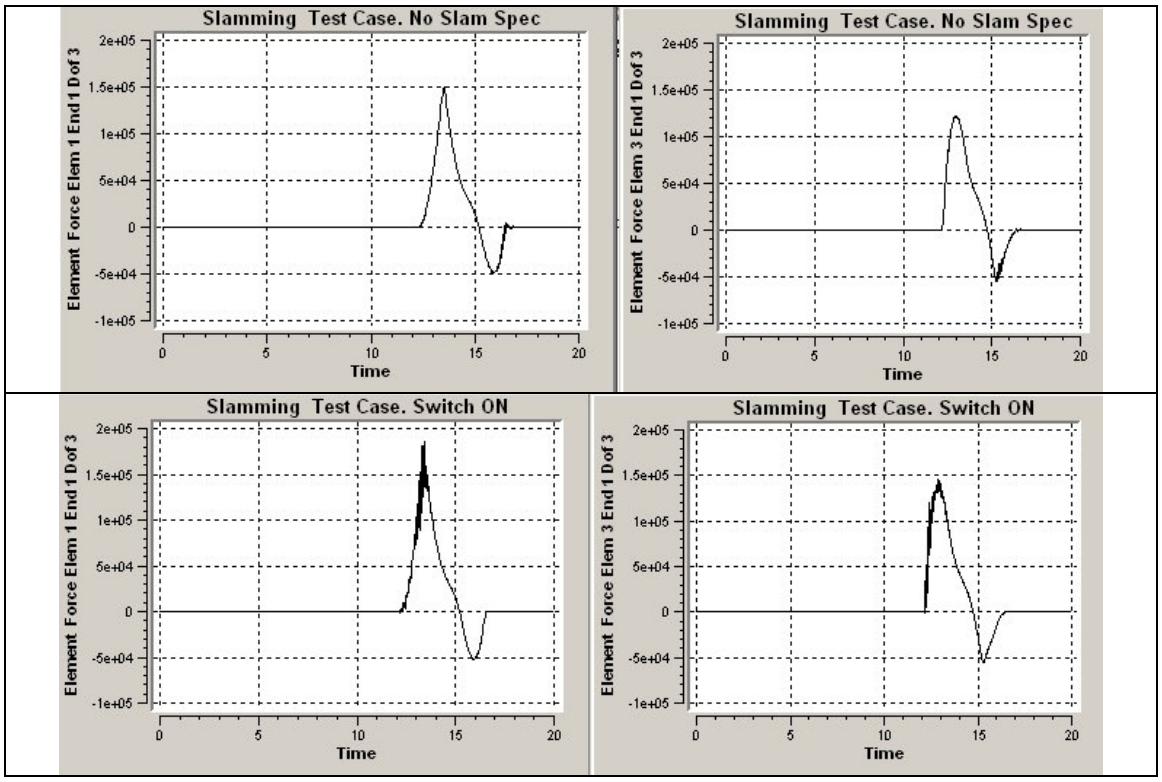

Figure 2-15 Vert. Forces at End 1and 2 of parallel members. Without/With SlamCalc ON

See: *http://www.usfos.no/examples/usfos/hydrodynamics/slam1/index.html*

2.4.4 Irregular waves, general.

The irregular wave module, (which is activated using the wave type `SPECT`), has been improved. Minor difference between results from version 8-4 is therefore expected. The changes are made within the following:

- Wave energy integration methods
- Finite depth corrections

2.4.5 Activation of hydrodynamic module for specified time increment

Some hydrodynamic options (like the buoyancy based on surface pressure) are relatively time consuming. Depending on problem, the time interval between each update of hydrodynamic force could be set longer than the time increment used in the dynamic simulation in general.

The command:

```
SWITCHES  WaveData  TimeIncrement  0.1
```

….means that the hydrodynamic forces are computed every 0.1 second unaffected by smaller dT used in the general dynamic simulation.

2.5 Joint Model. NORSOK

A new capacity curve is added to the joint option: Norsok N-004, Revision 2. The example is found on the web following address:

http://www.usfos.no/examples/usfos/misc/Jnt_Norsok_1/index.html

2.6 Soil and Earthquake

If earthquake simulations should be performed for a structure with pile foundation, the motion histories should be applied to the soil. The motions will then result in forces in the pile-soil interaction elements and thus be transferred to the structure.

The soil motion could be defined in two ways:

- Soil between specified Z-coordinates are given the actual motion histories (this example)
- The soil around a specified pile gets the actual motion (next example).

Different soil layers could be assigned different motion histories if necessary, see Figure 2-16.

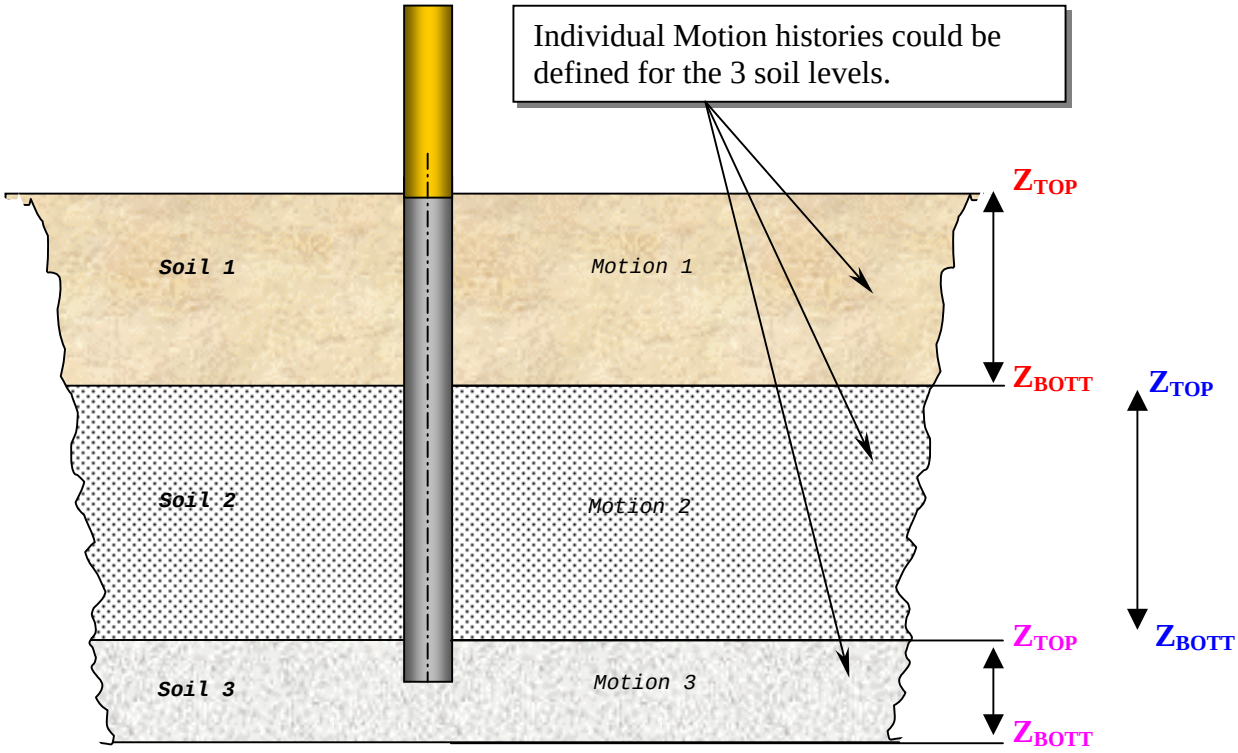

Figure 2-16 Definition of individual motion histories for the 3 different soil layer

See : *http://www.usfos.no/examples/usfos/foundation/SoilQuake01/index.html*

2.7 Subsidence of the Soil

Special foundation problems like uneven subsidence could be simulated using the SoilDisp option, where the soil around one pile is given a certain motion.

The options could be used for both *static* and *dynamic* simulations.

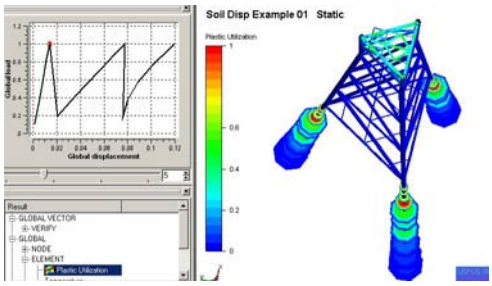

Figure 2-17 Prescribed Displacement of the soil around one pile

See : *http://www.usfos.no/examples/usfos/foundation/Subsidence01/index.html*

For more information, see the user's manual and the example:

2.8 Eigenvalue calculations

A completely new eigenvalue calculation package is implemented. The new package is activated if the new input format is specified. (The old package is still available when the old input syntax is used). The command EIGNVAL is repeated for each "keyword". The new syntax is:

`EIGENVAL` *`KewyWord Value(s)`*

Where the keywords are like:

- Time (specification of the time when the analysis is performed)
- NumberOf (specification of number of vectors to compute)
- ModeScale (scaling of eigenvectors for better visualization in xact)
- Algorithm (selection of either Sub Space Iteration or Lanzcos solvers)

The new module makes it possible to perform eigenvalues also of structures without a static solution (f ex a floating structure).

http://www.usfos.no/examples/usfos/misc/Eigenval01/index.html
http://www.usfos.no/examples/usfos/misc/Eigenval02/index.html

2.9 Special Damping Options

The new commands are introduced in version 8-5:

- DampData (defining damping properties)
- ElemDamp (assigning damping properties to selected elements)

By default, the damping is a global setting, which is assigned to all elements, (stiffness and mass proportional damping). The new feature makes it possible to specify different damping properties for the different elements (and materials).

The damping properties, (Rayleigh mass- and stiffness coefficients), could be common for all degrees of freedom within the elements, (the *Rayl_All* option), but it is also possible to use different damping f ex in axial and bending, (using *Rayl_Ind*).

The new, individual damping option requires more computer operations and should therefore not be used if all elements are using same damping coefficients.

```
'          DmpID           a1  a2
 DampData  1001  Rayl_All  0  5E-5       ! common for 6 dofs
```

```
'          DmpID           a1  a2
 DampData  1002  Rayl_Ind  0  5E-5       ! Ax
                           0  5E-5*1     ! Shy
                           0  5E-5*100   ! Shz
                           0  5E-5       ! Mx
                           0  5E-5*100   ! My
                           0  5E-5*1     ! Mz
```

Table 2-1 Definition of Damping Coefficients.

```
'          DmpID  ListType     Ids..
 ElemDamp  1001    All               ! Assign to all first
 ElemDamp  1002    Mat     200 300   ! Assign to special thereafter
```

Table 2-2 Assigning Damping Coefficients to elements.

2.10 Acceleration Fields

Up to now, only translation acceleration fields have been available (command: `GRAVITY`). A new load command, `AccField` , makes it possible to specify rotating acceleration fields, where the intensity increase with increasing radius from a certain center. Figure 2-18 demonstrates an acceleration field around global X-axis, and the corresponding forces, (`F=ma`), are visualized in Figure 2-1 using the new ”Vector Result” option. Figure 2-19 describes the structural response.

Note that the new AccField is consistent with the existing Gravity:

Forces are applied in the direction of the field.

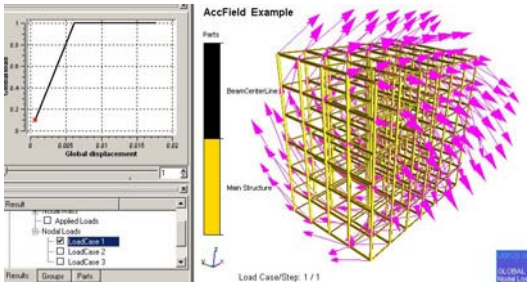

Figure 2-18 Rotating Acceleration Field. Visualization of Forces

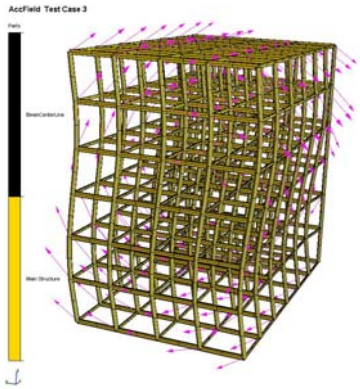

Figure 2-19 Response on the Rotating Acceleration Field about Global X-axis

See: *http://www.usfos.no/examples/usfos/basic_loads/AccField01/index.html*

2.11 Time Dependent Weights (masses)

The new "WEIGHT" option could be used if masses are changing or moving during a dynamic simulation, (for example increasing or moving ballast during a marine operation). Each "weight" is a concentrated mass, and is assigned to a time history.

The example below demonstrates the simulation of a mass travelling from end-1 to end-2 of a member. Since USFOS is forming and assembling all system matrices (mass, damp and stiff) every step, the instantaneous magnitude of the weight will have impact on the mode shapes, dynamic response, (and load vectors if acceleration fields, gravity or accfield, are defined).

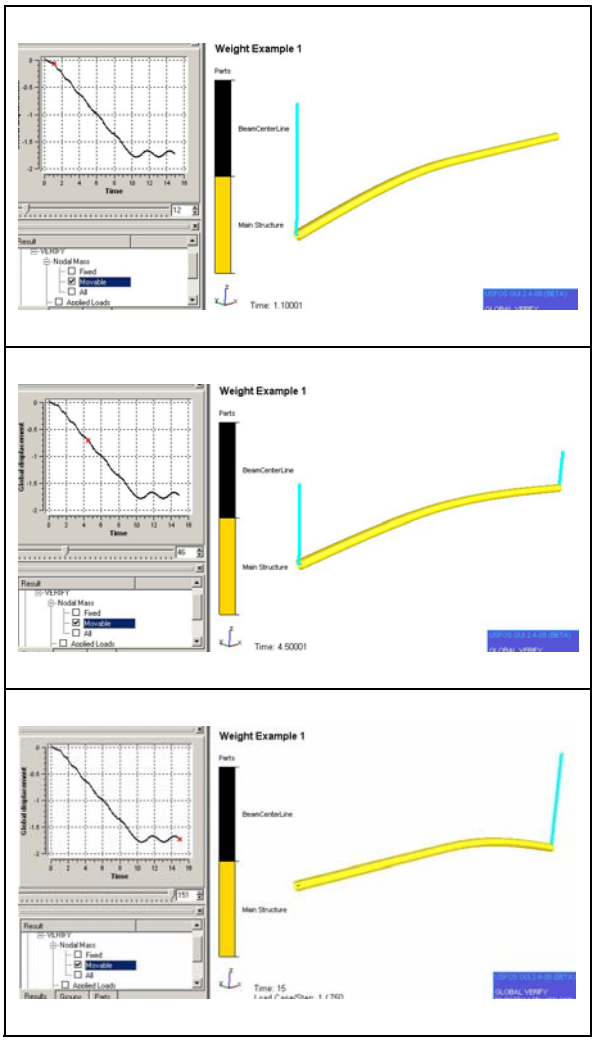

Figure 2-20 Weight moves from end-1 towards end 2

See : *http://www.usfos.no/examples/usfos/basic_loads/Weight01/index.html*

2.12 SWITCHES, (Special Options).

A new command: “SWITCHES” is introduced to switch on special options.

Following “Switches” options are available:

KeyWord	SubKey	Value	Description
WaveData	*TimeIncr*	“val”	Specify time increment different from simulation time increment
	noDopper	-	switch OFF Doppler effect
	noStore	-	Do not store wave data. Actual if extremely space consuming wave models, like “grid wave” are used. No wave visualization is then available.
StatusPrint	*MaxElem*	“num”	Specify more than the default 10 elements in the status print file (*res_status.text* file)
Write	*FE_Model*	-	Dump the FE-model including the internal generated elements. Dump file format : “ufo”.
Iteration	*RLF_Calc*	ON	Activate a special “Residual Load Factor” algorithm during iterations.
NodeData	*DoublyDef*	ON	Accept doubly defined nodes if the coordinates are identical. Could be useful if several self-contained, smaller models assemble a total model.

Table 2-3 SWITCHES options

3 Linux Versions


All USFOS modules are available on LINUX 64 bit. The different USFOS modules and utility codes are found on the following web address:

http://www.usfos.no/download/Linux/index.html

A simple installation description is found on the download page.
http://www.usfos.no/download/Linux/files/Usfos_on_Linux.pdf

4 Mac OSX Versions


All USFOS "*engine*" modules (except the GUI, xact), are available on Mac OSX 64 bit. Mac OSX is a UNIX workstation, and the installation procedure is identical to LINUX.

The different USFOS modules and utility codes are found on the following web address:

See :
http://www.usfos.no/download/Mac_OSX/index.html

5 Misc

5.1 Utility Tool Fact

A simple “User’s manual” is available for the simple utility tool “fact”. Both fact and the manual are available from “xact” (File/Utilities).

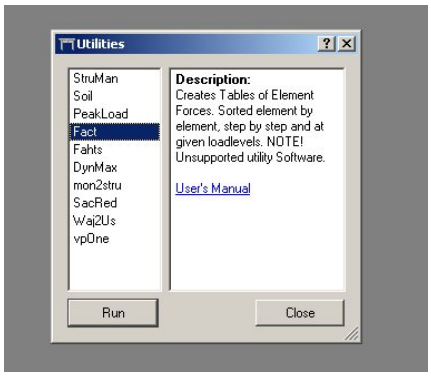

Figure 5-1 Utility Tool: Fact

5.2 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

6 New/modified input commands

Since last main release (8-4), following input identifiers are added/extended:

ACCFIELD	:	New command	: *Acceleration Field*
SOILDISP	:	New command	: *Soil displacement (earthquake / subsidence)*
DAMPDATA	:	New command	*: Special damping input.*
ELEMDAMP	:	New command	*: Assign special damping to elements.*
WEIGHT	:	New command	*: Time dependent weights (masses).*
CHJOINT	:	Extended command	*: NORSOK joint curves.*
WAVEDATA	:	Extended command	*: Revised User Defined Wave Spectrum.*
EIGENVAL	:	Extended command	*: New algorithms, extended possibilities.*