USFOS AS

Phone: +47 905 05 717

www.USFOS.com

Enterprise No.: NO 986 827 374 MVA

MEMO

MEMO CONCERNS

Release Notes
USFOS Version 8-3

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
	2006-12-31	Tore Holmas	18

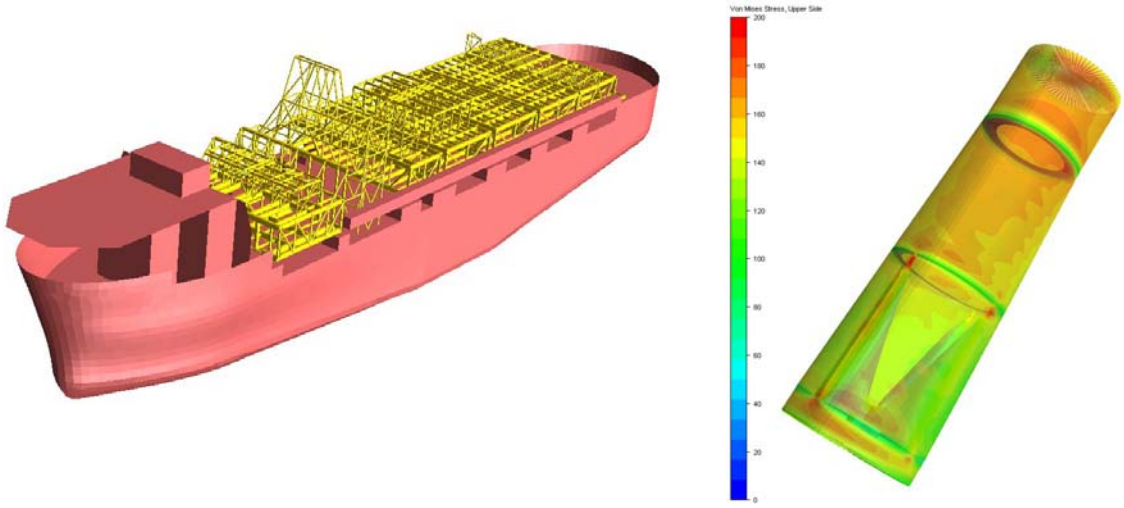
Release notes
USFOS 8-3, Dec 2006

1 Introduction

The current official version of USFOS is version 8-3 with release date 2006-12-31. The release contains 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 8-3 - 2006.

2.1 Introduction

Some of 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 8-3 USFOS could be upgrades in two different ways:

- Alt 1: Download the new "*setup.exe*" and u-install/install USFOS, (same as for release 8-2). 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 (the manual) 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).

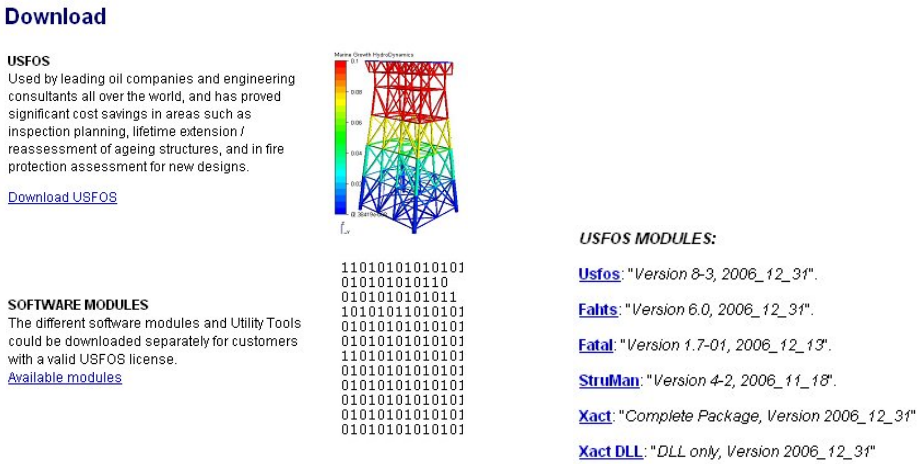


Figure 2-1 Download complete USFOS installation setup or the modules one by one

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 8-3 has version 2-3. Check under help/about to ensure that the latest version is installed.

The system details of your PC could be printed (Details….), and this is valuable information for us in case you have problems with the graphics.

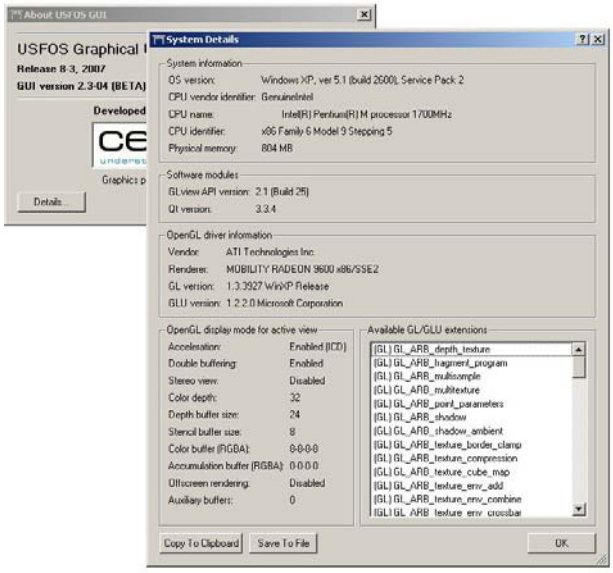


Figure 2-2 Help/About for release 8-3. Print of system details

2.3.1 Print of current file

The current opened file is printed on the main bar. The file path is not printed (unless the file is opened from a command window and path is specified). Both model files and result files are printed.

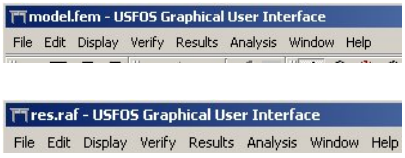


Figure 2-3 Actual opened file (model or result-file) is printed on the main bar

2.3.2 Modified Analysis Control for USFOS

The “Analysis” menu is extended, and the “old” Analysis Control” is now found under “USFOS Analysis Control”.

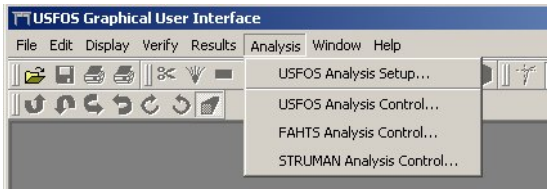


Figure 2-4 Extended “Analysis” menu

Only minor changes are made for the analysis control:

- Adjustable fields for “Editors” and “output” including hide/show output and editor
- “Open *.out file”. This means that the “res.out” is opened automatically after the simulation is completed.
- Default memory size is increased from 25 to 100 (mill words).
- Extra “Command line” to be used in special cases where USFOS asks for more information than the normal 4 fields. (relevant for special versions only).

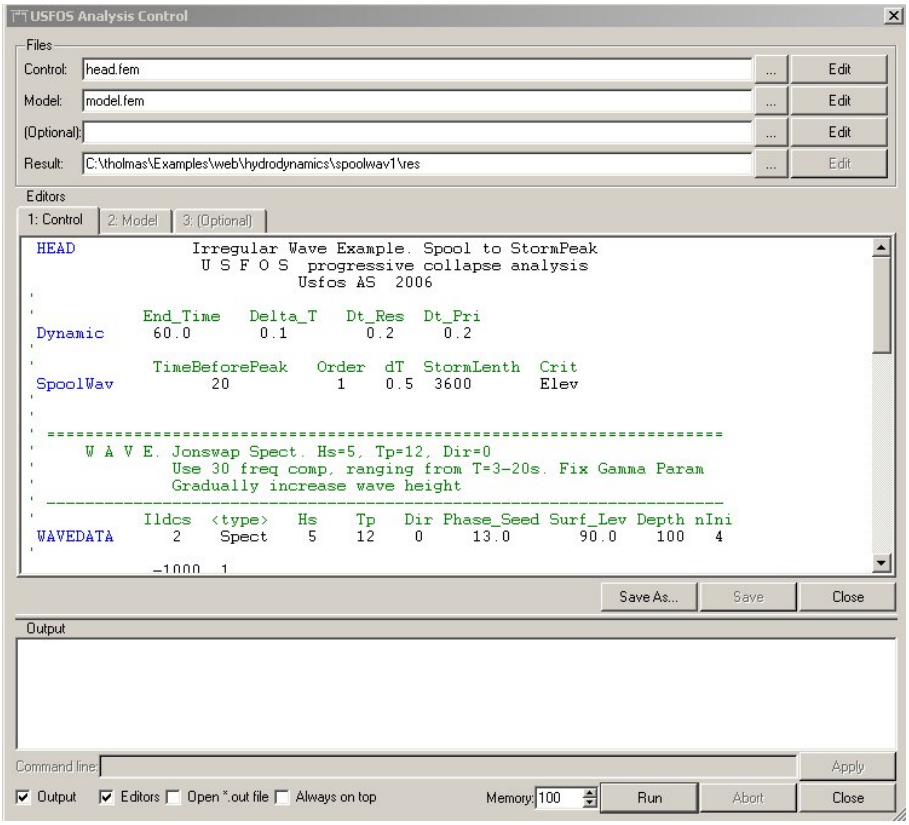


Figure 2-5 Modified Analysis Control.

2.3.3 New Analysis Control for FAHTS

The fire module FAHTS (**F**ire **A**nd **H**eat **T**ransfer **S**imulations), which produces temperature histories for USFOS, could be run from the GUI similar to USFOS. The analysis control is found under *Analysis*. The functionality with editor etc should be well known from USFOS. The main difference is the use of results from KAMELEON FIREEX (KFX), which is needed by FAHTS. The file extension of the KAMELEON to FAHTS files is **.k2f**

FAHTS produces a "*raf* file", (in addition to the "*beltemp*" file), and this file is opened automatically after the FAHTS simulation is completed.

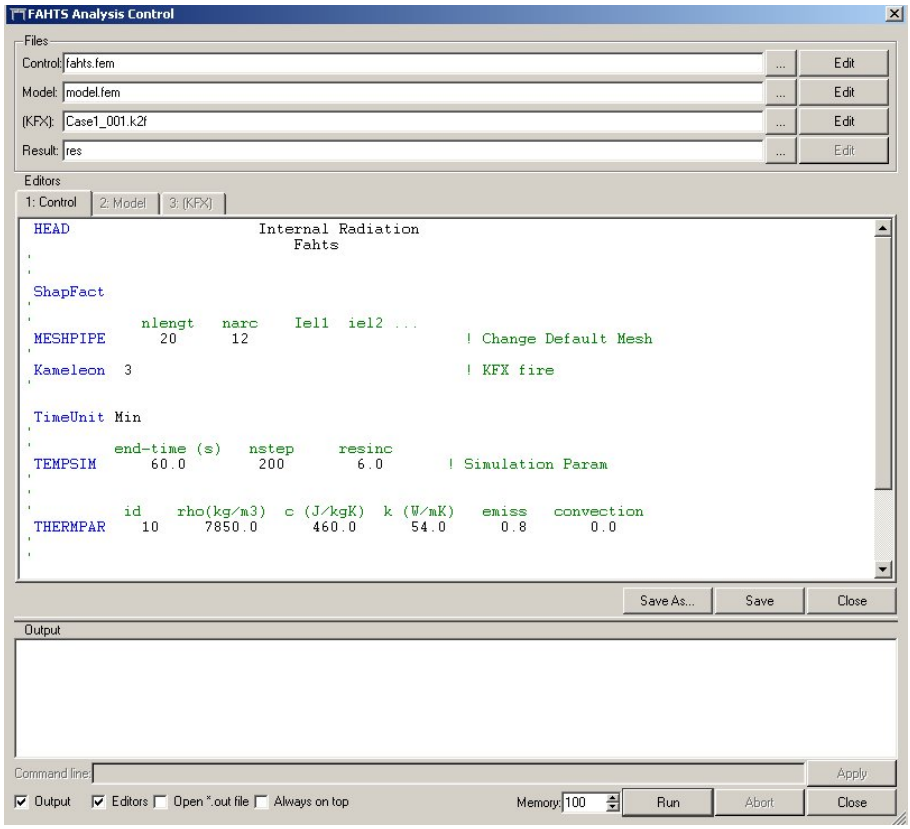


Figure 2-6 Analysis Control for FAHTS .

2.3.4 New Analysis Control for STRUMAN

A dedicated "analysis control" for the utility tool STRUMAN. (**Stru**ctural File **Man**ipulator) is included similar to USFOS and FAHTS. If the STRUMAN operation results in a raf file, (export to RAF in the STRUMAN control file), the file is opened automatically after STRUMAN is completed.

STRUMAN is a multipurpose tool, and often the results are printed on the out-file. Therefore the "Open *.out file" is active by default.

Some STRUMAN operations require interactive input from the user, and the "command line" is used for this communication. When the text is typed in, "Enter" or the "Apply" button is used. Please note that the general text editor, which opens automatically after completion of STRUMAN (opens res.out) could be used to inspect other result files from STRUMAN (for example a converted file, *res_ufo.fem*).

A typical operation could be as follows:

- Open model file (open for example a SESAM model)
- Press the "edit" button on the "Control" file
- Type in the actual STRUMAN command (for example ufo, convunit etc)
- Save control file (Save As)
- Run STRUMAN
- Open res_ufo.fem from the editor (which pops up automatically).

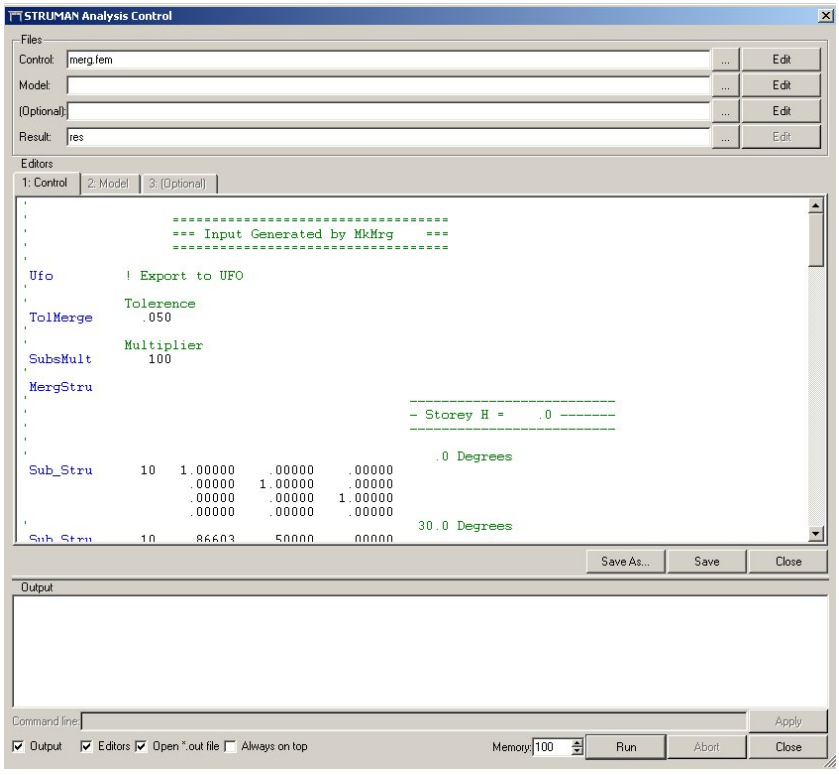


Figure 2-7 Analysis Control for STRUMAN.

2.3.5 New Run Utilities Control

The general utility control is re-designed in the 8-3 release. The re-design is done of the following reasons:

- Assist the user to "browse" to the wanted file folder (directory) and then define the "Working Directory" (Or "Start in" folder). This means that *input* and *output* will be taken from the actual directory. (F ex error messages, result files etc).
- Having an editor available for write/read text files.
- Specification of special "input" to the utility tool, (f ex codchk 200, where 200 means the memory size, etc.). The default parameter(s) is specified in the "Parameter field" and could be modified by the user. If the tool has no parameter to specify on the startup, the Parameter field is "grey".

The "Select" is used to define working directory. The current directory is printed. The "browse" option is used to find the actual input files.

The utility tools may ask for other kinds of input (like *yes* or *no*), and the same "Standard Input" line is used. Please note that "Enter" gives same effect as pressing the "Apply" button.

No files are opened automatically after the utility tool is completed, (use the editor or your own favourite editor).

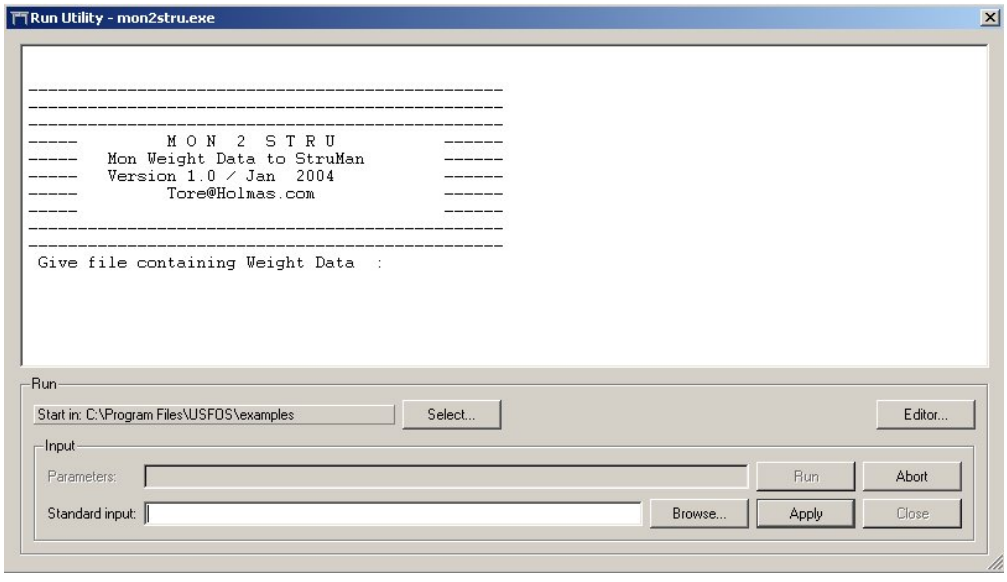


Figure 2-8 Utility Control

2.3.6 Highlight Groups

If groups (same as sets in SESAM), are defined, following could be done:

- Specification of group labels (**Name** *Group* …). The group label is printed on the group overview (see Figure 2-9).
- The selected groups are highlighted. Please note that the arrow keys (up/down) could be used to browse through the groups. The normal use of "shift" and "ctrl" is also possible (to define several groups, see Figure 2-10).

The exit from the "highlight group" (get rid of the pink elements), just click in the white field outside the group labels.

Group Names defined in SESAM (set name) are transferred to USFOS automatically.

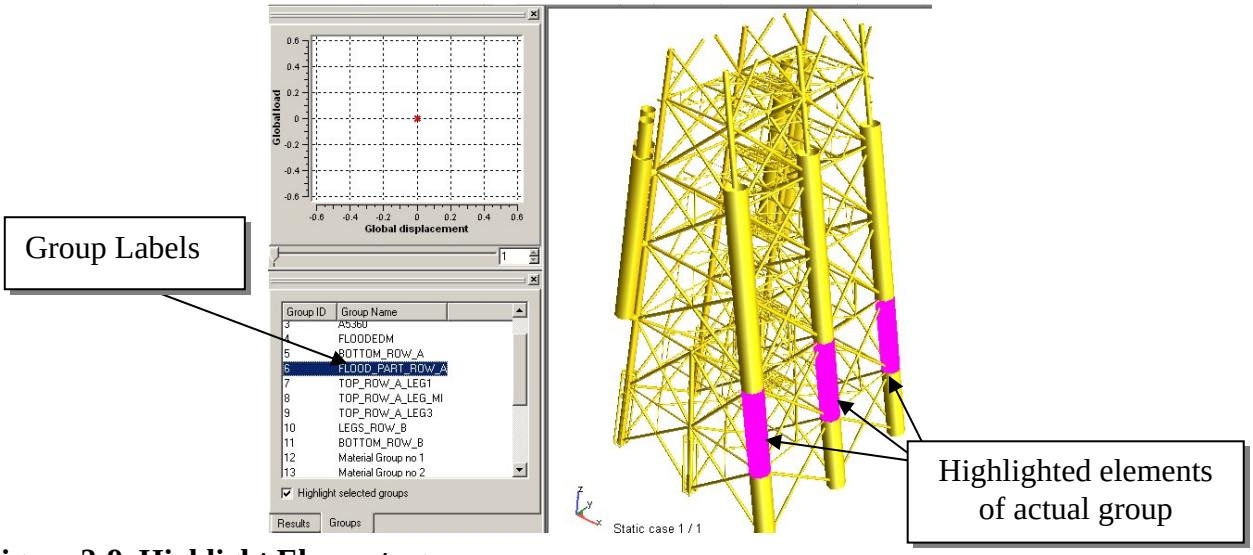


Figure 2-9 Highlight Elements groups

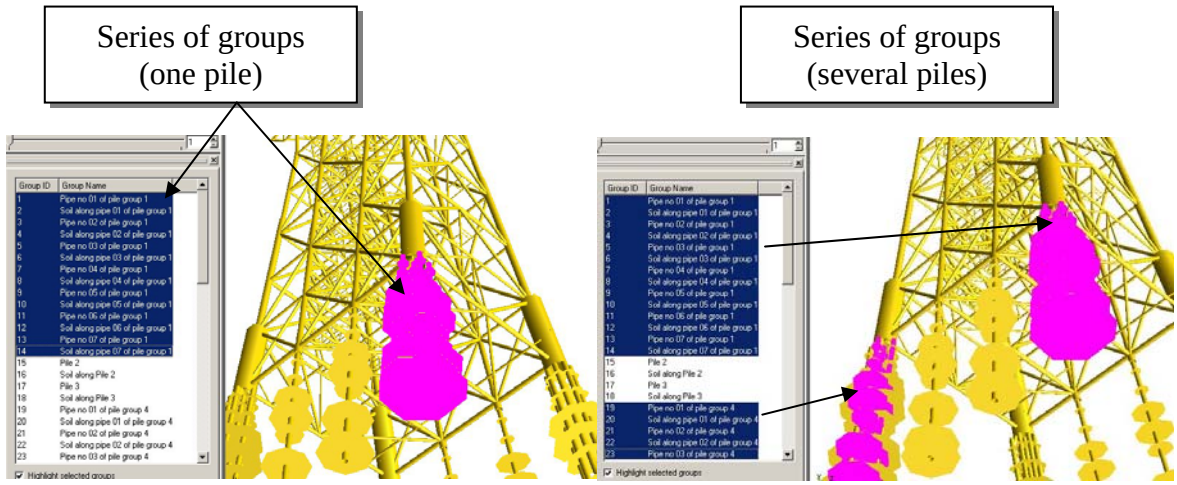


Figure 2-10 Highlight several groups simultaneously (pile elements)

2.3.7 User Defined Parts.

In order to divide the structure into several separated *parts*, the user defined part option is implemented. The reason for defining parts could be:

- Specifying special colour on the different elements (f ex white turbine blades, red FPSO hull, etc).
- Specify that the actual structural part should be transparent. (F ex transparent riser)
- Do *not* apply fringes, for example on dummy elements, which are defined for visualization purpose only (f ex the brown mud line, see Figure 2-11)

The colours are specified as the RGB mix, the gradual transparency is specified from 0 (100% solid) to 255 (completely invisible). The smooth shading and apply fringes are switches (0/1). See also: www.USFOS.com : (http://www.usfos.no/examples/usfos/misc/partdata1/index.html)

- Colour (RGB, 0-255). See web example for colour definitions (X11)
- Transparency (0-255)
- Smooth shading (0/1)
- Apply fringes (0/1)

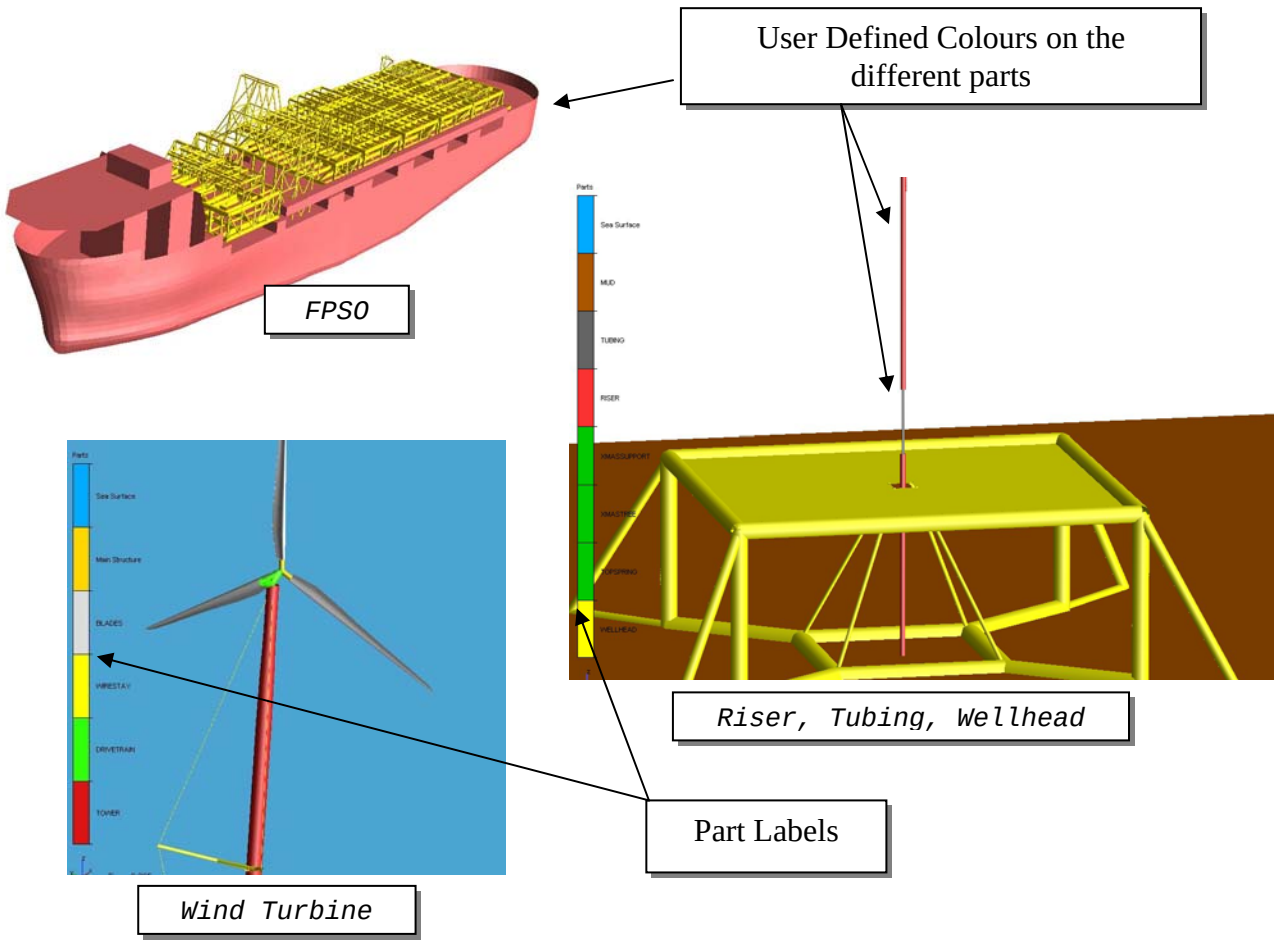


Figure 2-11 User defined Parts and colours.

The part attributes dialogue is found under “Display”, and is used to adjust the part colours etc, (change the initial definitions). If one part should be removed, (for example the sea surface), click on the “visible” filed (on/off). Transparency could be adjusted interactively. Different examples are shown in Figure 2-13 to Figure 2-16.
By default, the structural elements belong to part “Main Structure”. If hydrodynamics is specified, the “Sea Surface” will be defined autmatically. Similar if Ice Impact is defined, see Figure 2-14.

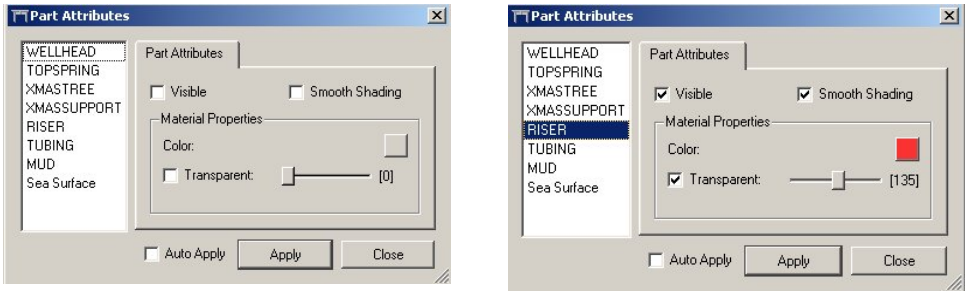


Figure 2-12 Part Attributes Dialogue.

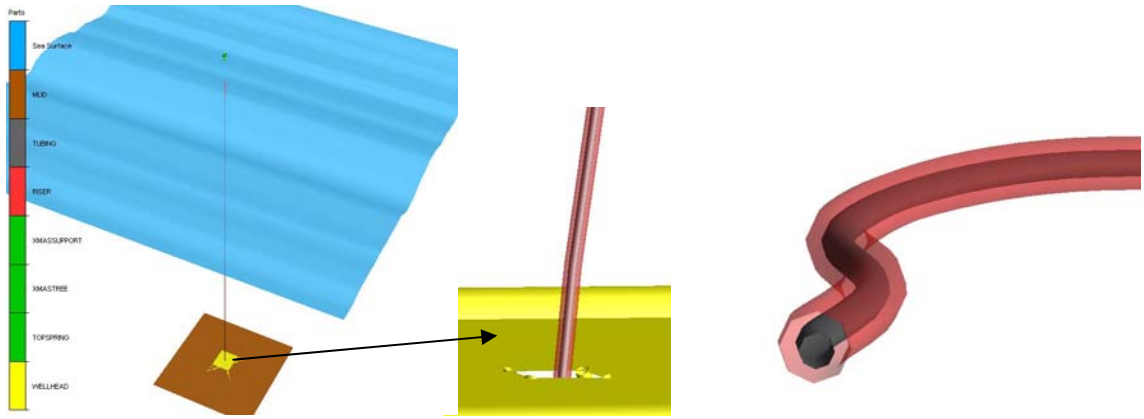


Figure 2-13 Transparent Riser showing the inner Tubing.

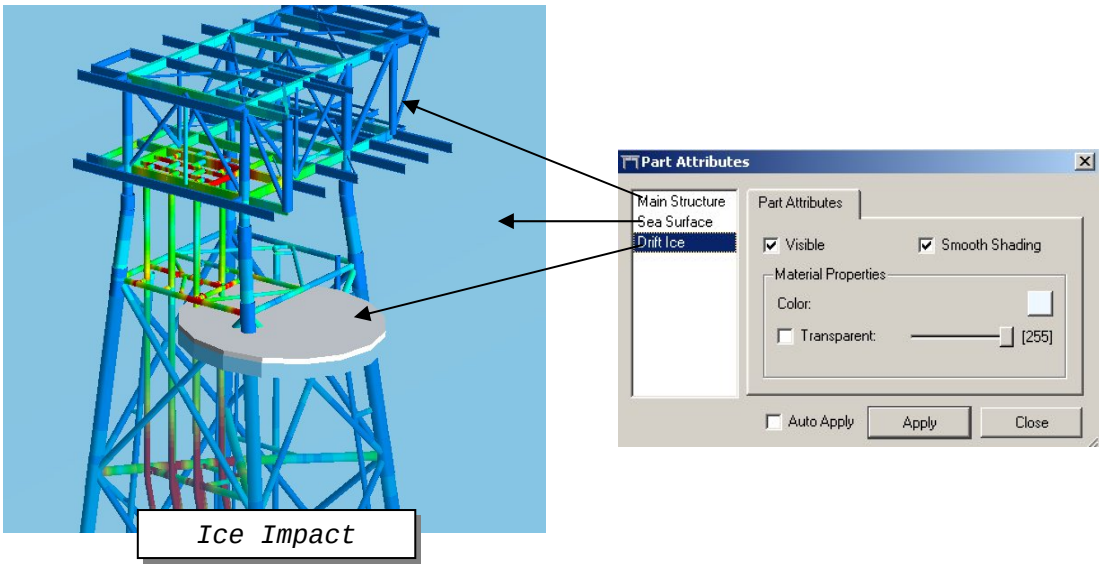


Figure 2-14 Automatic Generated Parts.

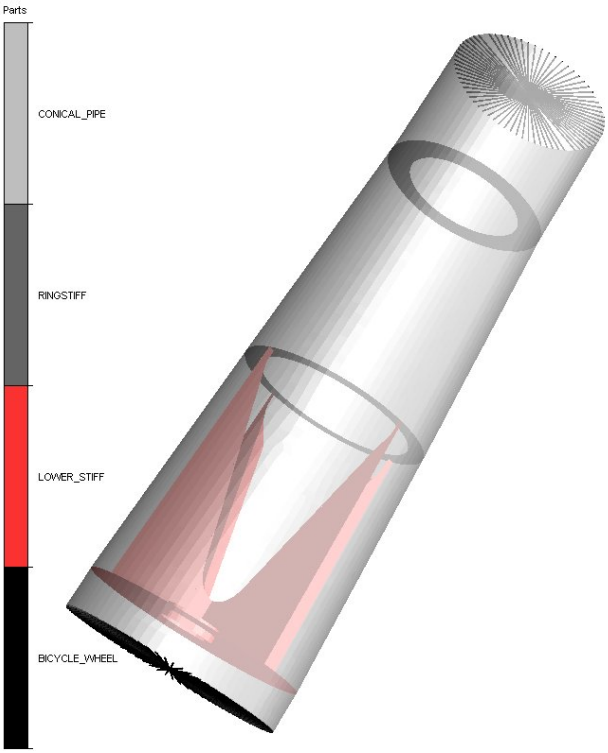


Figure 2-15 Transparent Conical Shell showing inner stiffeners

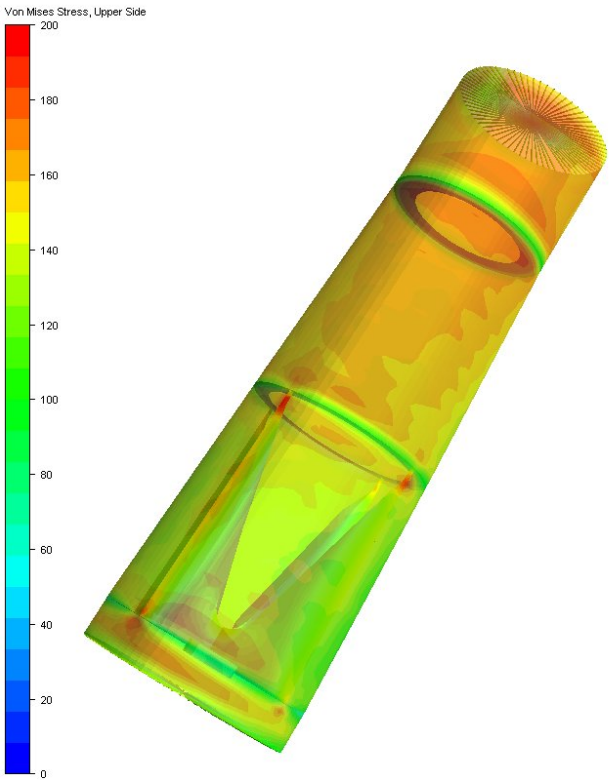


Figure 2-16 Result Fringes (stress) applied on transparent structure.

2.3.8 Editor

The context sensitive editor is available from the File menu. When the editor is opened from the analysis control (integrated in the dialogue box), it's searched for files with extension **.fem.** When the editor is opened from File/Open Text file, all file types are searched for (*.*).

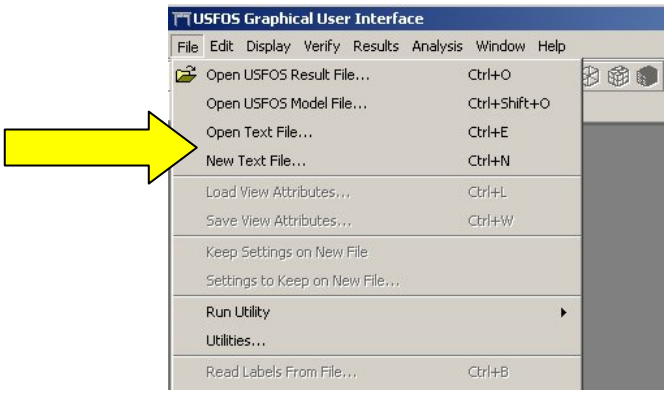


Figure 2-17 Opening the editor (existing or new) general text file (*.*).

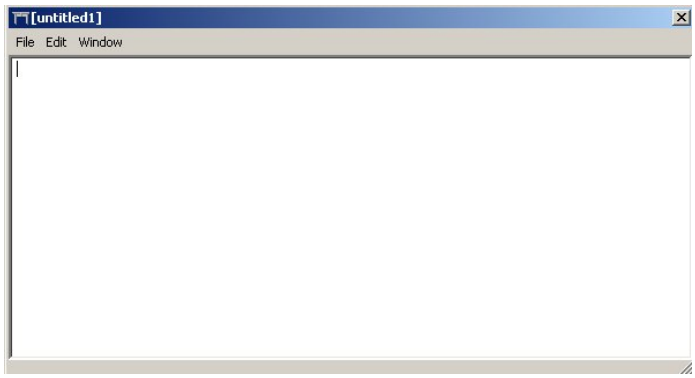


Figure 2-18 Context sensitive editor. No extension preference.

2.4 Irregular Waves

The **wavedata** command is extended from regular waves to also cover irregular waves. Different kinds of wave energy spectrum are available:

- Jonswap
- PM (Piesron-Moskowitz)
- User Defined

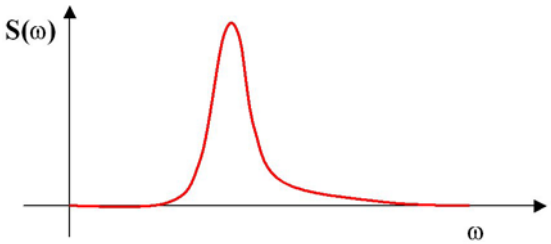


Figure 2-19 Wave energy spectrum

See www.USFOS.com : http://www.usfos.no/examples/usfos/hydrodynamics/wave_spect1/index.html

2.5 Spool to extreme wave. (ULS)

If irregular sea states are defined and for example a 3 hour extreme should be checked (ULS), the new "**SpoolWav**" command will search for the n^{th} highest peak and "spool" the wave up to a specified time before the actual peak. The analysis will then start the actual time before the peak and rapidly hit the extreme without wasting simulation time.

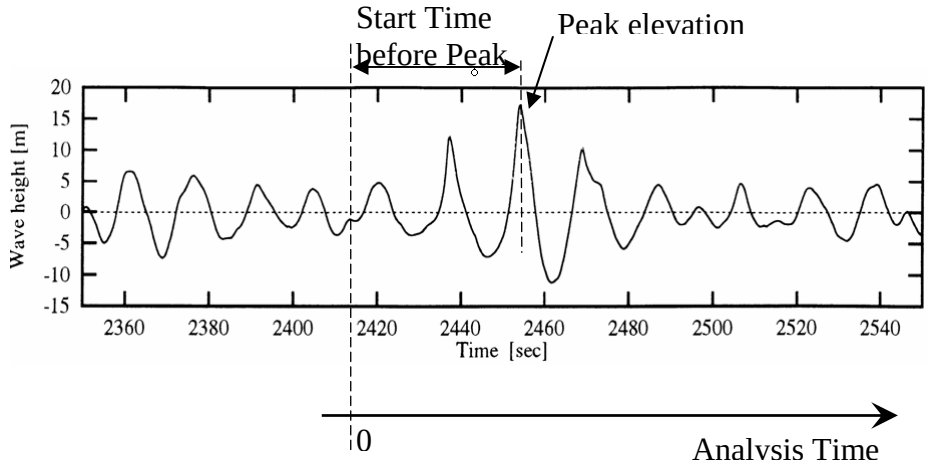


Figure 2-20 Search for peaks and spool the wave .

See www.USFOS.com : http://www.usfos.no/examples/usfos/hydrodynamics/spoolwav1/index.html

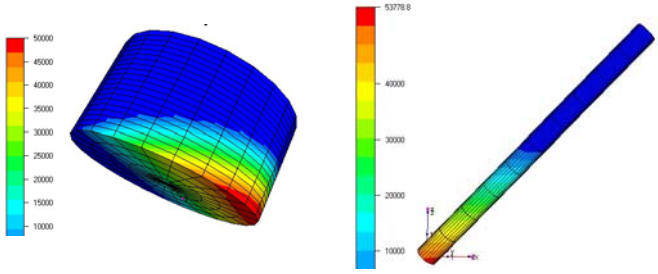
2.6 Non Hydrodynamic Elements

In some cases, elements below the sea surface should be by-passed during the hydrodynamic calculations. This could for example be the tubing inside a riser etc.

The command NONHYDRO is used to specify the actual elements.

See www.USFOS.com : http://www.usfos.no/examples/usfos/hydrodynamics/nonhydro1/index.html

2.7 Buoyancy calculation. Predefined Fill history (free surface calculation)



During marine operation simulations (installation, upending float up, etc) controlling the buoyancy for selected elements could be done used using either the "old" *buoyhist* option or the new IntFluid. This new option offers the user to define the following:

- Density of the internal fluid
- Fill rate history (predefined time history only in the present implementation)

The free surface is computed continuously, and instabilities like the example shown below are detected. Note that the 'panel' buoyancy formulation has to be used for the actual element(s).

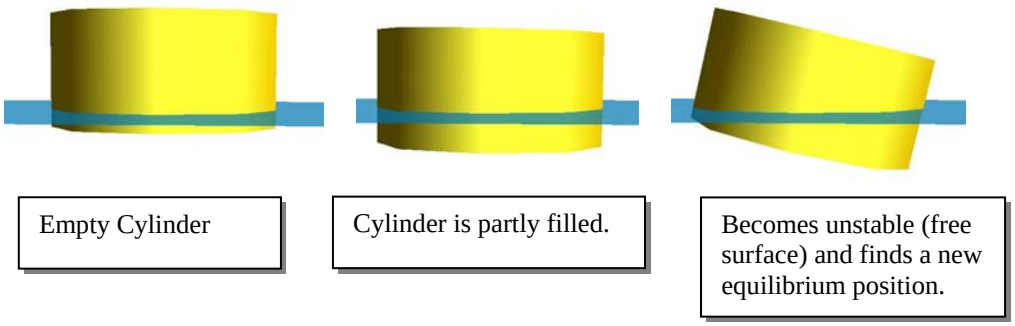


Figure 2-21 Three stages of a floating cylinder.

See www.usfos.com : http://www.usfos.no/examples/usfos/hydrodynamics/intfluid1/index.html

2.8 Zero Length 2 node Springs. (Tubing-In-Riser, Hinges, etc)

If contact between independent parts of a structure should be modeled (for example tubing inside a riser), the zero length 2 node non-linear spring is useful of following reasons:

- The initial coordinate system of the zero length spring is identical to the beam element's coordinate system.
- Orientation of the spring follows the rotation of the main structure, (f ex the riser)
- Modeling of contact with a certain "slack" before the spring resists, (or just linear spring stiffness).

The relatively complicated definition of local coordinate systems could be made using one single command, **ElmTrans**. By using for example the "material" ID list option, a series of springs (referring to same material) could be defined in one line.

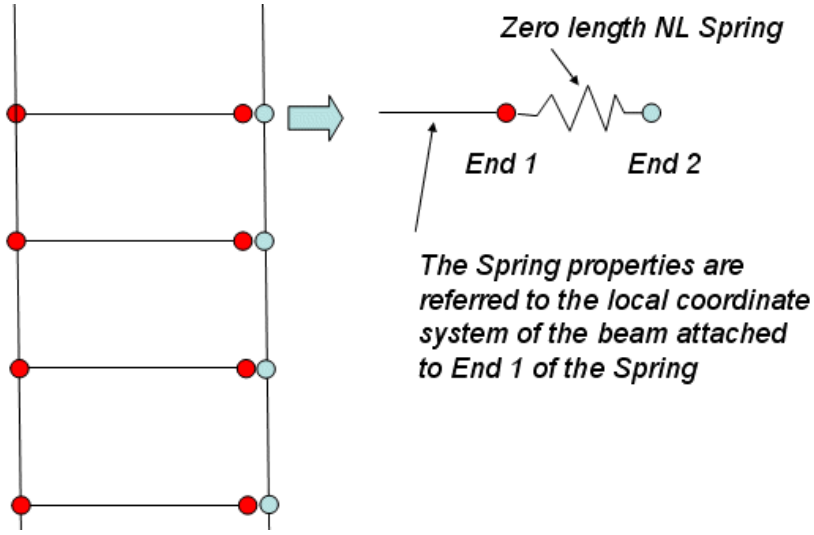


Figure 2-22 Schematic description of Zero length 2 node springs.

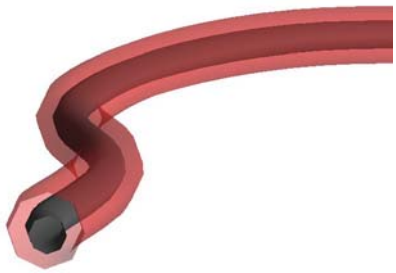


Figure 2-23 Transversal connection between independent tubing- and riser elements.

See www.usfos.com : http://www.usfos.no/examples/usfos/misc/elmtrans1/index.html

2.9 Local Masses (Co rotated Torsion masses etc)

If concentrated masses follow the local coordinate system of an element (for example torsion mass of a turbine blade), the **LocNMass** option should be used. This option defines following:

- The initial orientation of the concentrated mass
- How to update the reference system

See www.usfos.com : http://www.usfos.no/examples/usfos/misc/localmass1/index.html

2.10 Combining Load-Cases into new cases

The STRUMAN command CombLoad is implemented in USFOS. This makes it possible to combine (scale and add) several load cases into one new load case. Combload could be repeated if several combination should be performed in the same analysis.

Example:

```
'        New_Case Old_Case  Factor
COMBLOAD     3        4       1.3
                      3       2.0
                     10       0.7
                     13      -0.55
```

Generates a *new* load case 3 as follows:

```
l_case3 = (l_case4 * 1.3 + l_case3*2.0 + l_case10*0.7 - l_case13 *0.55)
```

The original loads, (case 4, 3, 10 and 13), are ***cleared*** after all combinations are processed.
NOTE. This option should be used for basic loads only (NodeLoad, BeamLoad, Pressure, Gravity

2.11 Group and Part Labels (the NAME command)

The new ***Name*** option makes it possible to define labels to both groups and parts. The input structure is as follows for assigning name for parts and groups respectively:

```
NAME     Part  Part_ID   List_Type  IdList …
NAME     Group Group_ID  List_Type  IdList …
```

The names of the actual groups/labels appears in the graphical post processor.

See www.usfos.com : http://www.usfos.no/examples/usfos/misc/partdata1/index.html

3 New/modified input commands

Since last main release (8-2), following input identifiers are added/extended:

BUOYFORM	:	New command	: *Buoyancy formulation*
INTFLUID	:	New Command	*: Internal fluid fill ratio, free surface*
NONHYDRO	:	New Command	*: Non hydrodynamic elements*
WAVEDATA	:	Extended command	*: Irregular wave. Several spectrums*
SPOOLWAV	:	New Command	*: Spool to extreme wave during storm*
ELMTRANS	:	New Command	*: Transformation of zero length 2-node spring.*
LOCNMASS	:	New Command	*: Node mass referring to local beam system*
PARTDATA	:	New Command	*: Definition of Part Attributes*
PARTELEM	:	New Command	*: Assign elements to parts*
NAME	:	New Command	*: Definition of label for Parts and Groups*
GWF	***:***	New Command	: *Global Wind/Wave Field (GridWave)*
SYSDAMP	***:***	New Command	: *Switch for forming separate* $\boldsymbol{C}_0$ *matrix*
COMBLOAD	***:***	New Command	: *Combining (scale) Basic loads into new cases*