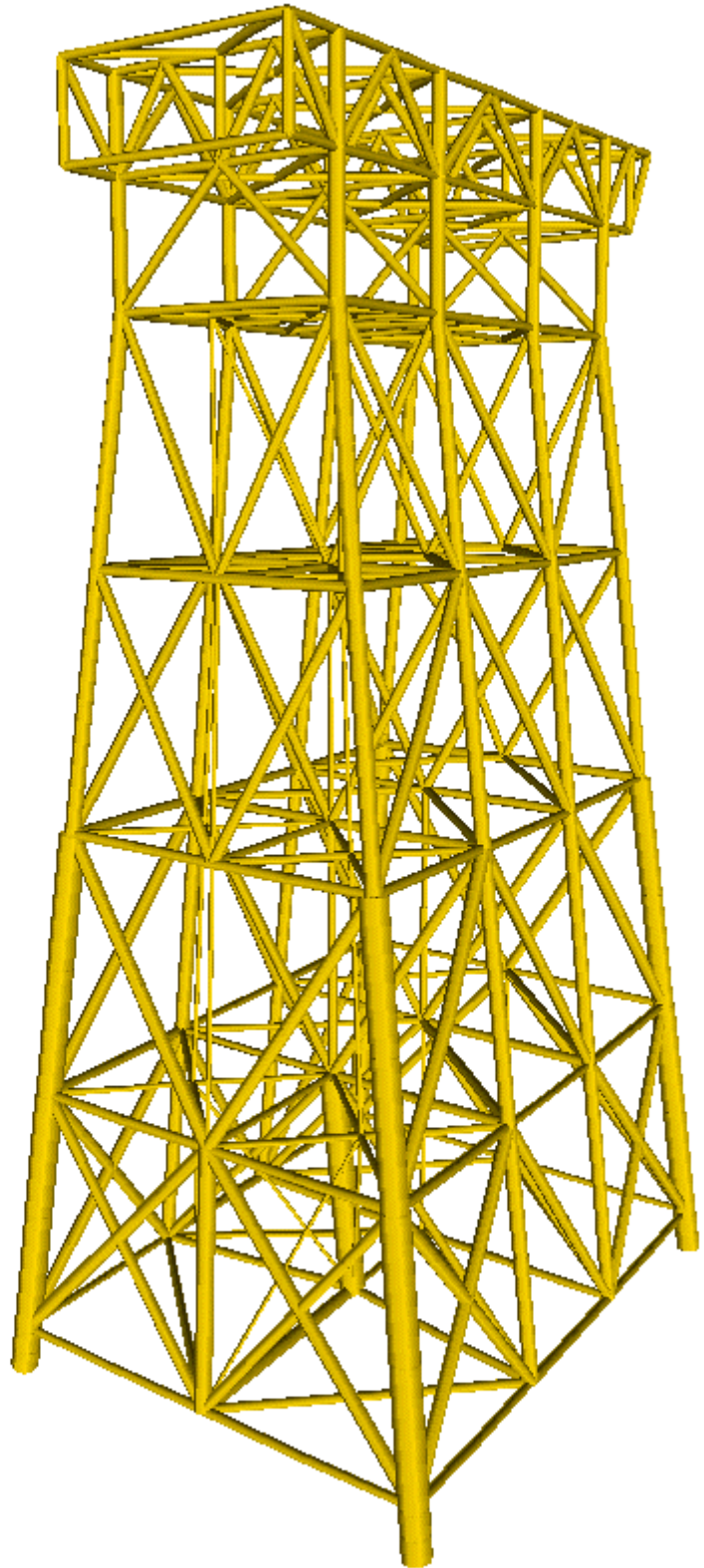


USFOS Getting Started



Structural Engineering
Marintek
SINTEF GROUP

Table of Content

1	UNDERSTANDING USFOS	1-4
1.1	BASIC FEATURES	1-4
1.2	THEORETICAL BASIS	1-6
1.2.1	<i>Continuum Mechanics</i>	1-6
1.2.2	<i>Finite Element Formulation</i>	1-7
1.2.3	<i>Shape Functions</i>	1-9
1.2.4	<i>Plasticity Formulation</i>	1-11
1.3	IMPLEMENTATION	1-15
1.3.1	<i>Load Specification</i>	1-15
1.3.2	<i>Load step scaling</i>	1-16
1.3.3	<i>Sign of load increment</i>	1-17
1.3.4	<i>Elastic 'Spring-Back'</i>	1-18
1.3.5	<i>Load control algorithm</i>	1-19
1.3.6	<i>Equilibrium iterations</i>	1-19
1.3.7	<i>Plastic hinges</i>	1-20
2	RUNNING USFOS	2-1
2.1	SYSTEM ARCHITECTURE.....	2-1
2.2	MEMORY ALLOCATION	2-1
2.3	INPUT	2-3
2.4	OUTPUT	2-5
2.4.1	<i>Global history output</i>	2-6
2.4.2	<i>Analysis-print-file</i>	2-7
2.4.3	<i>Analysis Status file</i>	2-8
2.5	PRACTICAL CONSIDERATIONS.....	2-13
2.5.1	<i>Load specification</i>	2-13
2.5.2	<i>Analysis spesification</i>	2-13
2.5.3	<i>Analysis verification</i>	2-14
2.5.4	<i>Potential problem areas</i>	2-16
3	“LINEAR” MODEL, NON-LINEAR ANALYSIS (“MODEL REPAIR”).....	3-1
3.1	LARGE MODELS	3-1
3.2	UTILIZING GROUP DEFINITIONS	3-2
4	WAVE LOADING	4-1
4.1	LOAD MODULE	4-1
4.2	EXTREME WAVE CALCULATION/AUTOMATIC MEMBER IMPERFECTIONS	4-2
5	JOINT MODELLING	5-1
5.1	JOINT CAPACITY CHECK	5-1
5.2	JOINT DEFORMATION CONTROL	5-2
5.3	JOINT CLASSIFICATION / MSL JOINT CHARACTERISTICS	5-3
6	FOUNDATION MODELLING	6-1
6.1	NONLINEAR SPRING	6-1
6.2	PILE/SOIL INTERACTION	6-2
6.2.1	<i>Pile – Soil / Automatic generation of piles and soil capacity</i>	6-3
6.3	SPUDCAN ELEMENT.....	6-5

7	DYNAMIC ANALYSES	7-1
7.1	INTRODUCTION	7-1
7.2	INPUT	7-2
7.3	DYNAMIC ANALYSIS RESULTS. TIME SERIES	7-3
8	EFFICIENT USE OF USFOS	8-1
8.1	GENERAL	8-1
8.2	ADJUSTING THE UNIX KORN SHELL WINDOW (WINDOWS NT/2000 INSTALLATIONS ONLY).....	8-1
8.3	SOME UNIX COMMANDS	8-4
8.4	EXAMPLE 1, FIXED USFOS INPUT FILE NAMES	8-5
8.5	EXAMPLE 2, VARYING USFOS INPUT FILE NAMES	8-6
8.6	EXAMPLE 3, ASSEMBLING INPUT FILES BEFORE USFOS ANALYSIS	8-8
8.7	EXAMPLE 4, USING THE SED EDITOR TO MODIFY MASTER INPUT FILES.....	8-10
8.8	EXAMPLE 5, PROCEDURE FOR ELEMENT REMOVAL (REDUNDANCY ANALYSIS)	8-15
9	WORKSHOPS	9-1
9.1	WORKSHOP I, USFOS ELEMENT FORMULATION, ANALYSIS CONTROL	9-1
9.1.1	<i>Workshop I a: Elasto-Plastic Beam Bending.....</i>	<i>9-2</i>
9.1.2	<i>Program input.....</i>	<i>9-2</i>
9.1.3	<i>Workshop I b: Elasto-Plastic Column Buckling.....</i>	<i>9-4</i>
9.1.4	<i>2D Frame Analysis</i>	<i>9-8</i>
9.2	WORKSHOP II, USFOS JACKET PUSHOVER	9-3
9.2.1	<i>Structure Description.....</i>	<i>9-3</i>
9.2.2	<i>Loads.....</i>	<i>9-5</i>
9.2.3	<i>FEM Model.....</i>	<i>9-7</i>
9.2.4	<i>Traditional pushover, loads from WAJAC.....</i>	<i>9-7</i>
9.3	WORKSHOP III SHIP IMPACT	9-11
9.3.1	<i>Program input.....</i>	<i>9-11</i>
9.3.2	<i>Analyses</i>	<i>9-11</i>
9.3.3	<i>Documentation.....</i>	<i>9-12</i>
9.3.4	<i>General comments :</i>	<i>9-12</i>
9.3.5	<i>Longitudinal Impact on Leg A4</i>	<i>9-12</i>
9.4	WORKSHOP IV RESIDUAL STRENGTH ANALYSIS	9-14
9.4.1	<i>Program input.....</i>	<i>9-15</i>
9.4.2	<i>Analyses</i>	<i>9-15</i>
9.4.3	<i>Documentation.....</i>	<i>9-15</i>
9.4.4	<i>General comments</i>	<i>9-15</i>
9.5	WORKSHOP V PILE AND SOIL MODELLING WITH USFOS (IMP COURSE).....	9-16
9.5.1	<i>Description of the structure</i>	<i>9-16</i>
9.5.2	<i>FEM model.</i>	<i>9-17</i>
9.5.3	<i>Foundation.....</i>	<i>9-18</i>
9.5.4	<i>Load modelling for reliability analysis.....</i>	<i>9-20</i>
9.5.5	<i>Lateral load on the jacket.</i>	<i>9-21</i>
9.5.6	<i>Program input.....</i>	<i>9-22</i>
9.5.7	<i>Analyses</i>	<i>9-23</i>
9.5.8	<i>Documentation.....</i>	<i>9-23</i>

1 UNDERSTANDING USFOS

1.1 Basic features

USFOS is a numerical tool for ultimate strength and progressive collapse analysis at space frame structures. The formulation includes nonlinear geometry and nonlinear material properties. The basic idea of the program is to use only one finite element per physical element of the structure, i.e. to use the same finite element discretization as in linear, elastic analysis.

- USFOS operates on element stress resultants, i.e. forces and moments.
- Material nonlinearities are modelled by plastic hinges at element midspan and at element ends.

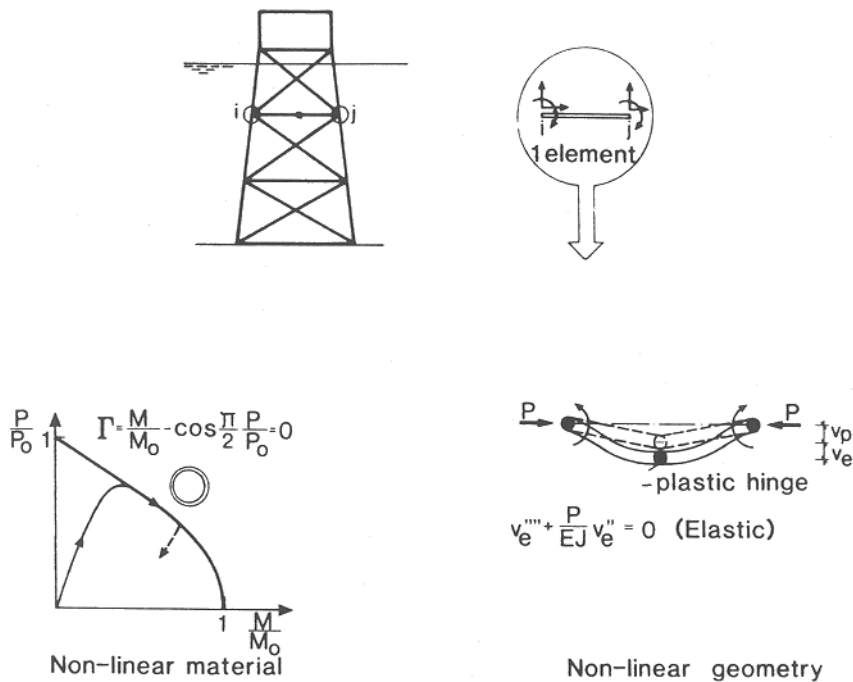


Figure 1.1-1 USFOS basic concepts

- Effects of large displacements and coupling between lateral deflection and axial strain are included by using nonlinear strain relations (Green strain). This gives a very accurate representation of element behaviour, including membrane effects and column buckling.
- Material models are included both for elastic-perfectly-plastic behaviour and gradual plastification-strain hardening characteristics. The first yield and fully plastic capacities are represented by yield surfaces based on plastic interaction between element forces.
- The load is applied incrementally.
- The load increment is automatically reversed if global instability is detected.
- The effect of initial deformations and local buckling are included for beam elements.
- Joint capacity checks and joint behaviour is implemented according to the API rules.
- Member rupture, and redistribution of forces from ruptured element is fully integrated in the analysis procedure.
- Hydrodynamic loads may be specified directly, without using a separate hydrodynamic load generation program.
- Pile and Soil data may be specified directly without using a separate pile-soil-interaction program

1.2 Theoretical basis

The formulation behind the program is valid for large displacements, but restricted to moderate strains. USFOS follows an updated Lagrange formulation.

1.2.1 Continuum Mechanics

The formulation is based on Green strains, defined by

$$\epsilon_x = u_{,x} + \frac{I}{2} u_{,x}^2 + \frac{I}{2} v_{,x}^2 + \frac{I}{2} w_{,x}^2 \quad (1.1)$$

For moderate element deflection, the von Karman approximation applies, and ϵ_x simplifies into

$$\epsilon_x = u_{,x} + \frac{I}{2} v_{,x}^2 + \frac{I}{2} w_{,x}^2 \quad (1.2)$$

The stiffness formulation of USFOS is derived from potential energy consideration or the virtual work principle. For an elastic beam element the internal strain energy reads

$$U = \frac{I}{2} \int_0^l EA \left(u_{,x} + \frac{I}{2} v_{,x}^2 + \frac{I}{2} w_{,x}^2 \right)^2 dx + \frac{I}{2} \int_0^l (EI_z v_{,xx}^2 + EI_y w_{,xx}^2) dx \quad (1.3)$$

where the first integral comes from axial straining and the last integral represents bending. The expression in the first parenthesis is the element strain, ϵ_x . The total displacement is decomposed into axial displacement $u(x)$ and lateral deflection $v(x)$ (and $w(x)$ in three dimensions). Torsion is not included in the variational formulation, but is added directly into the element stiffness matrix.

The potential of external loads is written as

$$H = - \left(F_i u_i + \int_0^l q_x u dx + \int_0^l q_y v dx + \int_0^l q_z w dx \right) \quad (1.4)$$

The total potential for an elastic element is now

$$\Pi = U + H \quad (1.5)$$

Total and incremental equilibrium equations are established by taking the first and second variation of the strain energy and the potential of the external work. The first variation of internal strain energy comes out of equation (1.3), as shown in equation (1.6).

$$\begin{aligned} \delta U = & \int_0^l EA u_{,x} \delta u_{,x} dx + \int_0^l EI_z \left(v_{,xx} \delta v_{,xx} - \frac{N}{EI_z} v_{,x} \delta v_{,x} \right) dx \\ & + \int_0^l EI_y \left(w_{,xx} \delta w_{,xx} - \frac{N}{EI_y} w_{,x} \delta w_{,x} \right) dx - \int_0^l (N + EA u_{,x}) \delta u_{,x} dx \end{aligned} \quad (1.6)$$

The first term of (1.6) is the linear contribution from axial strain. The two next integrals represents bending deformation, including the influence of axial forces ('membrane effects'). These terms are represented by the 'Livesly stability functions' in the stiffness matrix. The

last integral comes from the nonlinear axial strain contribution from lateral deflections and gives a correction to the equilibrium axial loads. Equation (1.6) is the basis for calculating internal equilibrium forces to be compared with external loads during equilibrium correction.

Denoting by Δ the increment between two close configurations, the variation of increment in strain energy is given in equation (1.7).

$$\begin{aligned}
 \delta\Delta U = & \int_0^l EA \Delta u_{,x} \delta u_{,x} dx \\
 & + \int_0^l EI_z \left(\Delta v_{,xx} \delta v_{,xx} \frac{N}{EI_z} \Delta v_{,x} \delta v_{,x} \right) dx + \int_0^l EI_y \left(\Delta w_{,xx} \delta w_{,xx} \frac{N}{EI_y} \Delta w_{,x} \delta w_{,x} \right) \\
 & + \int_0^l EA \left(\Delta u_{,x} v_{,x} \delta v_{,x} + \Delta v_{,x} v_{,x} \delta u_{,x} \right) dx + \int_0^l EA \left(\Delta u_{,x} w_{,x} \delta w_{,x} + \Delta w_{,x} w_{,x} \delta u_{,x} \right) \\
 & + \int_0^l EA \Delta v_{,x} v_{,x}^2 \delta v_{,x} dx + \int_0^l EA \Delta w_{,x} w_{,x}^2 \delta w_{,x} dx \\
 & + \int_0^l EA \left(\Delta v_{,x} v_{,x} w_{,x} \delta w_{,x} + \Delta w_{,x} w_{,x} v_{,x} \delta v_{,x} \right) dx + \text{higher order terms}
 \end{aligned} \tag{1.7}$$

1.2.2 Finite Element Formulation

The incremental stiffness is obtained by introducing interpolation functions ('shape functions') for element displacements.

$$\begin{aligned}
 u(x) &= \phi^T q_u \\
 v(x) &= \phi^T q_v \\
 w(x) &= \phi^T q_w
 \end{aligned} \tag{1.8}$$

The variation of increment in strain energy can now be written as

$$\begin{aligned}
\delta \Delta U = & \delta \mathbf{u}^T \int_0^l EA \phi_{u,x} \phi_{u,x}^T dx \Delta \mathbf{u} \\
& + \delta \mathbf{v}^T \int_0^l EI_z \left(\phi_{v,xx} \phi_{v,xx}^T \frac{N}{EI_z} \phi_{v,x} \phi_{v,x}^T \right) dx \Delta \mathbf{v} \\
& + \delta \mathbf{w}^T \int_0^l EI_y \left(\phi_{w,xx} \phi_{w,xx}^T \frac{N}{EI_y} \phi_{w,x} \phi_{w,x}^T \right) dx \Delta \mathbf{w} \\
& + \delta \mathbf{v}^T \int_0^l EA \phi_{v,x} v_{,x} \phi_{u,x}^T dx \Delta \mathbf{u} + \delta \mathbf{u}^T \int_0^l EA \phi_{u,x} v_{,x} \phi_{v,x}^T dx \Delta \mathbf{v} \\
& + \delta \mathbf{w}^T \int_0^l EA \phi_{w,x} w_{,x} \phi_{u,x}^T dx \Delta \mathbf{u} + \delta \mathbf{u}^T \int_0^l EA \phi_{u,x} w_{,x} \phi_{w,x}^T dx \Delta \mathbf{w} \\
& + \delta \mathbf{v}^T \int_0^l EA \phi_{v,x} v_{,x}^2 \phi_{v,x}^T dx \Delta \mathbf{v} + \delta \mathbf{w}^T \int_0^l EA \phi_{w,x} w_{,x}^2 \phi_{w,x}^T dx \Delta \mathbf{w} \\
& + \delta \mathbf{w}^T \int_0^l EA \phi_{w,x} w_{,x} v_{,x} \phi_{v,x}^T dx \Delta \mathbf{v} + \delta \mathbf{v}^T \int_0^l EA \phi_{v,x} v_{,x} w_{,x} \phi_{w,x}^T dx \Delta \mathbf{w}
\end{aligned} \tag{1.9}$$

Arranging the parameters in the order u, v, w the separate terms of the elastic stiffness matrix may be determined

$$\mathbf{K}_T = \begin{bmatrix} k_{uu} & k_{uv} & k_{uw} \\ k_{vu} & k_{vv} & k_{vw} \\ k_{wu} & k_{wv} & k_{ww} \end{bmatrix} \tag{1.10}$$

The following expressions emerge for the sub-matrices :

$$\begin{aligned}
k_{uu} &= \int_0^l EA \phi_{u,x} \phi_{u,x}^T dx \\
k_{vv}^I &= \int_0^l EI_z \left(\phi_{v,xx} \phi_{v,xx}^T \frac{N}{EI_z} \phi_{v,x} \phi_{v,x}^T \right) dx \\
k_{ww}^I &= \int_0^l EI_y \left(\phi_{w,xx} \phi_{w,xx}^T \frac{N}{EI_y} \phi_{w,x} \phi_{w,x}^T \right) dx
\end{aligned} \tag{1.11}$$

These are the diagonal sub-matrices that also are present in the secant stiffness matrix (ref (1.6)). The subsequent terms comes from large rotations and are nonlinear contributions.

$$\begin{aligned}
k_{vu} &= \int_0^l EA \phi_{v,x} v_{,x} \phi_{u,x}^T dx = k_{uv}^T \\
k_{wu} &= \int_0^l EA \phi_{w,x} w_{,x} \phi_{u,x}^T dx = k_{uw}^T
\end{aligned} \tag{1.12}$$

These two are coupling matrices between axial and lateral deformation and are linear in rotation.

Finally, the diagonal sub-matrices for deflection k_{vv} and k_{ww} get additional contributions that are second order in rotation.

$$\begin{aligned} k_{vv}^{\text{II}} &= \int_0^l EA \phi_{v,x} v_{,x}^2 \phi_{v,x}^T \\ k_{ww}^{\text{II}} &= \int_0^l EA \phi_{w,x} w_{,x}^2 \phi_{w,x}^T \end{aligned} \quad (1.13)$$

The last integral of eq. 9 give coupling matrices between the two directions of deflection:

$$k_{vw} = \int_0^l EA \phi_{w,x} w_{,x} v_{,x} \phi_{v,x}^T dx = k_{vw}^T \quad (1.14)$$

1.2.3 Shape Functions

The shape function used for the transverse displacement field,

$$v(x) = \phi^T q \quad (1.15)$$

is taken as the exact solution to the 4th order differential equation of a beam subjected to end forces :

$$\phi^T = [\cosh kx, \sinh kx, x/L, 1] \quad (1.16)$$

for positive N (compression)

$$\phi^T = [\cos kx, \sin kx, x/L, 1] \quad (1.17)$$

for negative N (tension).

The value of k is given by

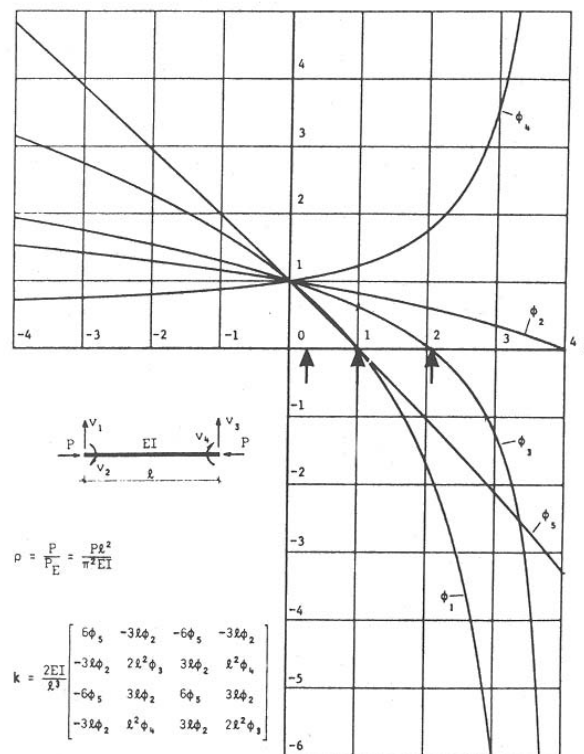
$$k^2 = \frac{|N|}{EI_z} \quad (1.18)$$

Similar expressions are used for the displacement fields $w(x)$ and $u(x)$.

The generalized constants q are then determined by the boundary conditions for the beam.

The advantage of using the above shape functions is that all integration in the element stiffness expression can be carried out analytically, and the element stiffness matrices presented as closed-form expressions. Furthermore, the "quality" of the shape functions allows for a very simple modelling; one element between each joint is normally sufficient to simulate the nonlinear column behaviour with satisfactory accuracy.

As USFOS employs exact element displacement functions satisfying the governing differential equation, USFOS should predict the elastic buckling load for the three cases in Figure 1.2-1 . It is observed that the element equilibrium equations becomes singular at the exact Euler buckling formulas.



Examples:

$$k = \frac{2EJ}{l} \begin{bmatrix} 2\phi_3 & \phi_4 \\ \phi_4 & 2\phi_3 \end{bmatrix}$$

$$\det k = 0 \text{ for } \phi_3 = \frac{1}{2} \phi_4$$

$$P_{\text{crit}} = P_E$$

$$k = \frac{4EJ}{l} \cdot \phi_3$$

$$\det k = 0 \text{ for } \phi_3 = 0$$

$$P_{\text{crit}} = 2.04 P_E$$

$$k = \frac{2EJ}{l^3} \begin{bmatrix} 6\phi_5 & 3l\phi_2 \\ 3\phi_2 & 2l^2\phi_3 \end{bmatrix}$$

$$\det k = 0 \text{ for } 12\phi_5\phi_3 = 9\phi_2^2$$

$$P_{\text{crit}} = 0.25 P_E$$

Figure 1.2-1 Elastic Column Buckling

1.2.4 Plasticity Formulation

Material nonlinearities are modelled by yield hinges. Plastic hinges may be inserted at element ends or at element midspan. In the latter case, the original element is divided into two subelements. The extra nodal point is introduced automatically and eliminated by static condensation before adding into the global stiffness matrix.

The behaviour of the hinges is governed by plastic flow theory. The basic assumptions of plastic flow theory can be summarized as follows :

- 1) There exists a yield condition which can be illustrated by an initial yield surface
- 2) There exists a flow rule, relating plastic strain increments to stress increment
- 3) There exist a hardening rule, relating the extension of the yield surface to the amount of plastic deformation

$$\Gamma = f \left(\frac{N}{N_P}, \frac{Q_y}{Q_{yP}}, \frac{Q_z}{Q_{zP}}, \frac{M_x}{M_{xP}}, \frac{M_y}{M_{yP}}, \frac{M_z}{M_{zP}} \right) - 1 = 0 \quad (1.19)$$

In USFOS, the yield condition (or plastic capacity of a cross-section) is represented by a plastic interaction function/yield surface for stress resultants

For a tubular section, the plastic interaction function is given as

$$\begin{aligned} \Gamma &= f \left(\frac{N}{N_P}, \frac{M_y}{M_P}, \frac{M_z}{M_P} \right) - 1 \\ &= \cos \left(\frac{\pi}{2} \frac{N}{N_P} \right) - \frac{\sqrt{M_y^2 + M_z^2}}{M_P} = 0 \end{aligned} \quad (1.20)$$

when torsion and shear forces are neglected.

The function is defined so that $\Gamma = 0$ for all forces giving full plastification of the cross section. $\Gamma = -1$ is the initial value of a stress-free cross section. In principle, a state of forces characterized by $\Gamma > 0$ is illegal.

The flow rule is given by

$$\Delta \mathbf{v}^P = \begin{bmatrix} \mathbf{g}_1 & 0 \\ 0 & \mathbf{g}_2 \end{bmatrix} \begin{bmatrix} \Delta \lambda_1 \\ \Delta \lambda_2 \end{bmatrix} = \mathbf{G} \Delta \lambda \quad (1.21)$$

where

$$\mathbf{g}_i^T = \frac{\partial \Gamma}{\partial S_i} = \left[\frac{\partial \Gamma}{\partial N}, \frac{\partial \Gamma}{\partial Q_y}, \frac{\partial \Gamma}{\partial Q_z}, \frac{\partial \Gamma}{\partial M_x}, \frac{\partial \Gamma}{\partial M_y}, \frac{\partial \Gamma}{\partial M_z} \right]_i \quad (1.22)$$

and index i refers to beam end 1 and beam end 2. These equations state that the plastic displacements are normal to the yield surface. The 'direction' of the plastic displacements (plastic elongation vs. plastic shear or plastic rotation) are given by the surface normal of the yield surface, $\Delta \mathbf{g}_i$, at the current force state, S_i . The magnitude of the plastic displacements are given by a scalar factor, $\Delta \lambda$.

The hardening rule describes loading from one plastic state to another plastic state. When a plastic hinge has been introduced, the state of forces should move from one plastic state to another plastic state, following the yield surface so that $\Gamma = 0$. For an elastic - perfectly plastic material model, this can be expressed as

$$\begin{aligned} \Delta\Gamma = \frac{\partial\Gamma}{\partial N} \Delta N + \frac{\partial\Gamma}{\partial Q_y} \Delta Q_y + \frac{\partial\Gamma}{\partial Q_z} \Delta Q_z + \frac{\partial\Gamma}{\partial M_x} \Delta M_x + \frac{\partial\Gamma}{\partial M_y} \Delta M_y + \frac{\partial\Gamma}{\partial M_z} \Delta M_z \\ = \mathbf{g}^T \Delta \mathbf{S}_i = 0 \end{aligned} \quad (1.23)$$

or

$$\mathbf{G}^T \Delta \mathbf{S} = 0 \quad (1.24)$$

The elastic stiffness expression for the beam element is expressed as

$$\Delta \mathbf{S} = \mathbf{K}_T \Delta \mathbf{v}^E \quad (1.25)$$

To determine the elasto-plastic stiffness expression, the total displacement increment is separated into an elastic and a plastic component

$$\Delta \mathbf{v} = \Delta \mathbf{v}^E + \Delta \mathbf{v}^P \quad (1.26)$$

The stiffness equations can then be expressed as

$$\begin{aligned} \Delta \mathbf{S} &= \mathbf{K}_T (\Delta \mathbf{v} - \Delta \mathbf{v}^P) \\ &= \mathbf{K}_T \Delta \mathbf{v} - \mathbf{K}_T \mathbf{G} \Delta \lambda \end{aligned} \quad (1.27)$$

when the flow rule is introduced.

Pre-multiplying with $\mathbf{G}^T$, the right-hand side takes the form of the hardening rule

$$\begin{aligned} \mathbf{G}^T \Delta \mathbf{S} &= \mathbf{G}^T \mathbf{K}_T \Delta \mathbf{v} - \mathbf{G}^T \mathbf{K}_T \mathbf{G} \Delta \lambda \\ &= 0 \end{aligned} \quad (1.28)$$

and the plastic increment can be solved

$$\Delta \lambda = (\mathbf{G}^T \mathbf{K}_T \mathbf{G})^{-1} (\mathbf{G}^T \mathbf{K}_T \Delta \mathbf{v}) \quad (1.29)$$

Substituting $\Delta \lambda$ into eq. (1.27), an expression for the elasto-plastic stiffness of the beam can be determined

$$\begin{aligned} \Delta \mathbf{S} &= \mathbf{K}_T \Delta \mathbf{v} - \mathbf{K}_T \mathbf{G} \Delta \lambda \\ &= \mathbf{K}_T \Delta \mathbf{v} - \mathbf{K}_T \mathbf{G} (\mathbf{G}^T \mathbf{K}_T \mathbf{G})^{-1} \mathbf{G}^T \mathbf{K}_T \Delta \mathbf{v} \\ &= (\mathbf{K}_T - \mathbf{K}_T \mathbf{G} (\mathbf{G}^T \mathbf{K}_T \mathbf{G})^{-1} \mathbf{G}^T \mathbf{K}_T) \Delta \mathbf{v} \end{aligned} \quad (1.30)$$

or

$$\Delta \mathbf{S} = \mathbf{K}_T^{EP} \Delta \mathbf{v} \quad (1.31)$$

The considerations above are valid for a plasticity formulation where the cross section is either elastic, or fully plastic. That is, only one yield surface is used.

A plasticity model which accounts for partial plastification and strain hardening is formulated according to the bounding surface concept. This model employs two interaction surfaces - one yield surface and one bounding surface. The yield surface bounds the region of elastic cross sectional behaviour; when the force state contacts the yield surface this corresponds to initial yielding in the cross section. The bounding surface defines the state of full plastification of the cross section. This surface has the same shape as the yield surface.

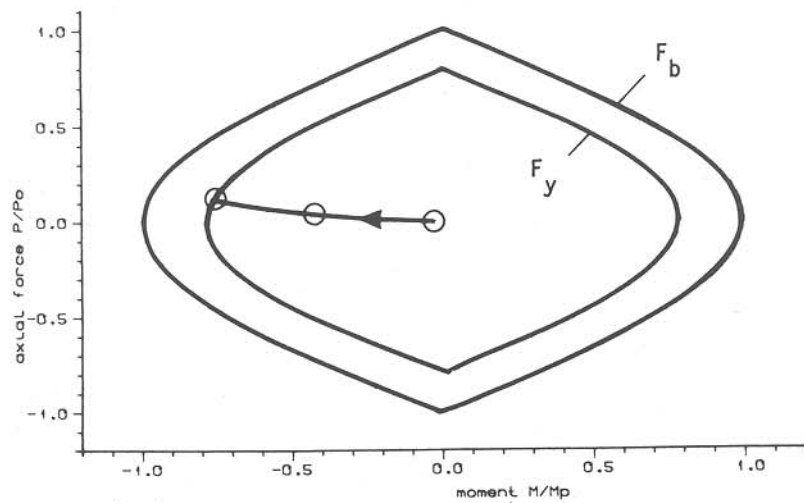
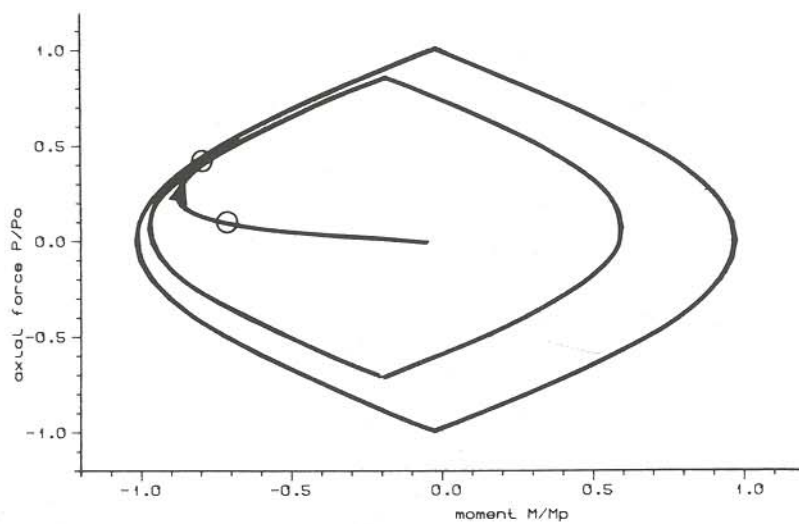
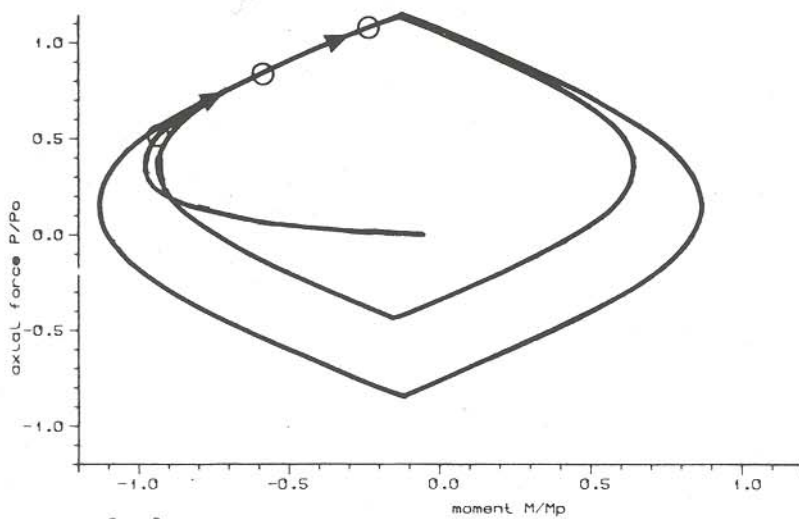
Figure 1.2-2 illustrates the yield and bounding surfaces for a tubular cross section plotted in the m_2 - n - plane. When the cross section is loaded, the force point travels through the elastic region and contacts the yield surface (upper plot). This represents first fibre yield in the cross section. At this stage a yield hinge is introduced.

When further loading takes place the yield surface translates such that the force state remains on the yield surface ($\Gamma_y = 0$) (middle plot). The bounding surface also translates, but at a much smaller rate. This is used to model strain hardening, according to a kinematic hardening model.

The translation of the yield surface, which approaches the bounding surface during the loading process, provides for a smooth transition from initial yield to full plastification.

In the bottom plot, the force state has reached the bounding surface; the cross section has reached full plastification. From this stage the force state remain on the bounding surface and both surfaces will translate in contact.

Figure 1.2-3 Relates the multidimensional illustration in stress resultant space to a uniaxial stress-strain curve.

**[a]****[b]****[c]****Figure 1.2-2 Two-surface plasticity model**

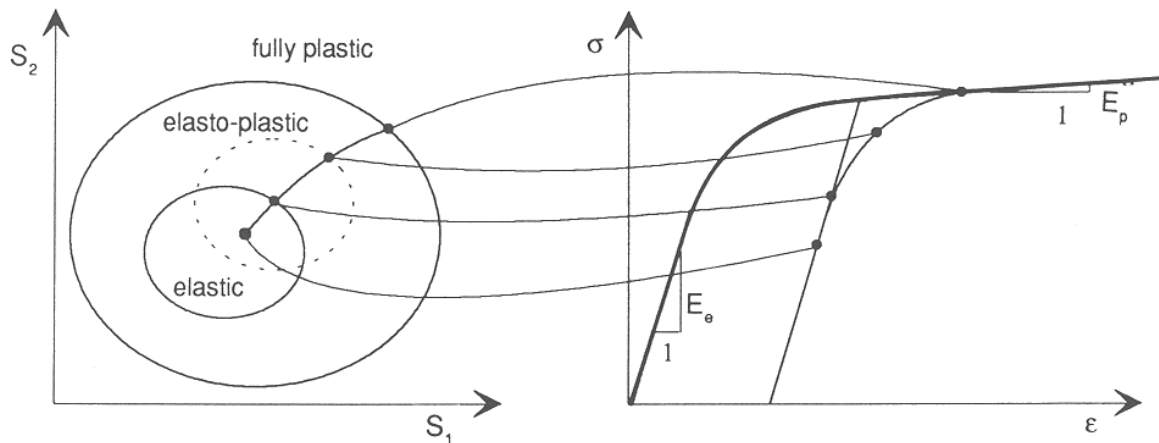


Figure 1.2-3 Analogy between multidimensional stress-space and uniaxial stress-strain curve

1.3 Implementation

According to the updated Lagrangian formulation, the load is applied in steps, and the system stiffness equations are solved at every step. After each step, the structural configuration is updated - element forces, nodal coordinates etc. are updated, and plastic hinges are introduced if necessary. Thus, each step constitutes a full, linear analysis, based on the updated information from all previous analysis steps.

A combined incremental, iterative loading algorithm is implemented. As default, a pure incremental procedure is adopted. Equilibrium iterations may be specified by the user.

Global instability/collapse is detected by a formulation based on the *Current Stiffness Parameter* in combination with a *Determinant Criterion*. When instability is detected, USFOS reverses the sign of the load increment, and the analysis proceeds into the post-collapse range.

1.3.1 Load Specification

The user supplies the basic *load cases* as input to USFOS. In the run commands to the program, the load cases are combined into *load combinations* (if necessary). Then, the user specifies the *loading history* for the analysis (which loads are to be applied, in which order, and how large increments are to be used ...)

The loads are incremented a given number of steps, up to a specified load level, or until a defined displacement is reached. The load is incremented on top of the previous loads, i.e. each load increment is added to the accumulated load of all previous load steps. Thus, the calculated results at each step are the combined results of the total load history prior to and including that step.

The results of different load cases may not be superposed, since the response of the structure is highly history dependent. In nonlinear analyses, the actual loads must be combined.

1.3.2 Load step scaling

In each load increment, the program first applies the load increment specified by the user. Then, USFOS checks the following criteria, and scales the load step down if necessary:

- Introduction of plastic hinges
- Exceedance of the user defined maximum displacement increment
- Adjustments during equilibrium iterations

If yielding occurs in any cross section, the load step is scaled so that the element forces comply "exactly" with the yield surface.

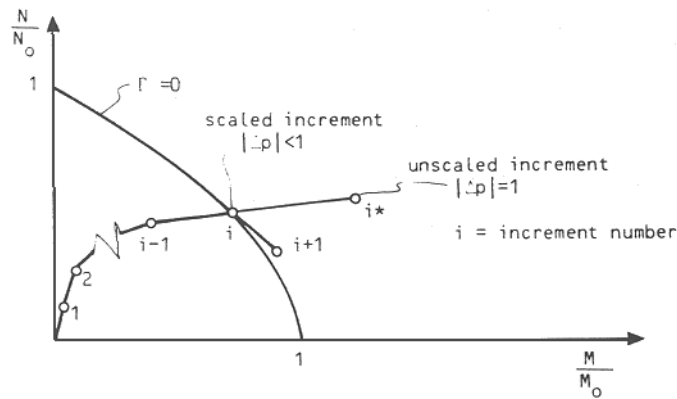


Figure 1.3-1 Increment scaling due to introduction of plastic hinges

By this procedure only one hinge is detected per load increment. To avoid unreasonably small step length in case of frequent occurrence of hinges, the user may specify a minimum load step for the scaling. In this way, "exact" scaling to the yield surface is not always possible and several hinges may be inserted during one load increment.

In regions where the current stiffness parameter is small, very large incremental displacements may result. This may reduce the accuracy of the analysis, as shown in Figure 1.3-2

To control too large displacements, the user may define a "global" displacement of the structure, and set a limit to the size of the displacement increments. The global displacement is specified as a weighed sum of some characteristic degrees of freedom, supplied by the user¹⁾:

$$\Delta r_{glob}^i = \frac{\sum \Delta r_k^i \cdot \omega_k}{\sum \omega_k}, \quad k = 1, NCNODS \quad (1.32)$$

Δr_k^i = displacement increment for control displacement k at step i

ω_k = weight factor associated with control displacement k

The load step will be scaled down if the control displacement increment of the current step exceeds *mxpdis* times the control displacement of the initial load step of that load combination.

¹⁾ If only one degree of freedom is specified, the control displacement will not be normalized. I.e. the control d.o.f will only be multiplied by the weight factor.

$$\Delta r_{glob}^i \leq mxpdis \cdot \Delta r_{glob}^l \quad (1.33)$$

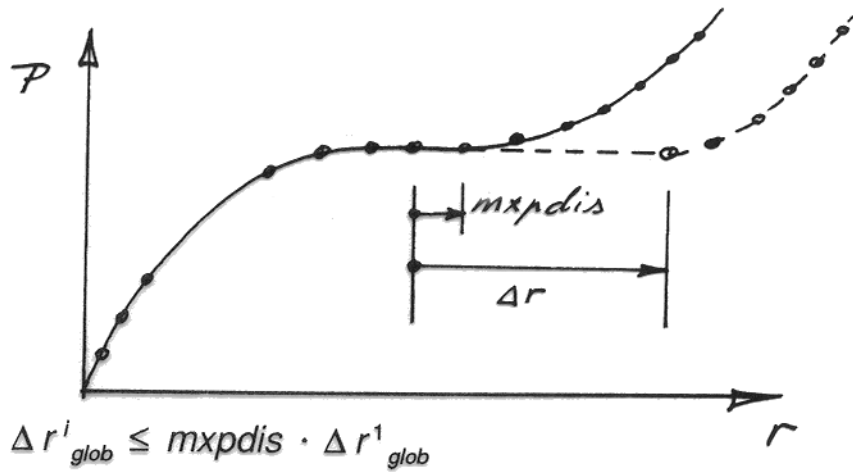


Figure 1.3-2 Scaling by maximum control displacement

1.3.3 Sign of load increment

The sign of the load increment is governed by the Current Stiffness Parameter and by the determinant of the tangential stiffness matrix.

1.3.3.1 Determinant of Stiffness Matrix

The stiffness matrix determinant is the 'classical' stability criterion for nonlinear analyses. As long as the determinant is positive, the stiffness matrix is positive definite, and the structure is 'stable'. As the load increases, the structural response becomes more and more nonlinear and the determinant will decrease (for softening systems). Zero determinant signifies a global instability point or a bifurcation point, and a negative determinant (one or more negative terms on the stiffness matrix diagonal) represent an unstable structure.

1.3.3.2 Current Stiffness Parameter

The Current Stiffness Parameter is defined by

$$S_p^i = \frac{(\Delta r^l)^T \cdot \Delta R^l \cdot (\Delta p^i)^2}{(\Delta r^i)^T \cdot \Delta R^i \cdot (\Delta p^l)^2} \quad (1.34)$$

where Δr and ΔR are incremental displacements and forces. Δp is the relative load increment size at each load step.

The Current Stiffness Parameter is a normalized parameter representing the stiffness of the structure during the deformation. It may be regarded as the incremental work carried out in the first load step, divided by the incremental work at load step no. i . Thus, the current stiffness parameter will have an initial value of 1.0. For stiffening systems (membrane effects) it will increase. For softening systems, it will decrease.

A small absolute value of the current stiffness parameter will represent an unstable structure, the instability point having 0.0 Current Stiffness. The Current Stiffness will be negative in the post-collapse range.

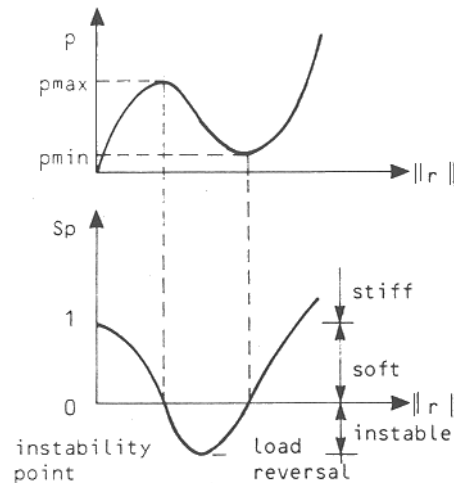


Figure 1.3-3 Current stiffness parameter

1.3.4 Elastic 'Spring-Back'

The elastic spring-back problem is characterized by an extremely "brittle" behaviour. At a specific level, the load drops, accompanied by a temporary reduction in displacement. The structure may later regain stiffness, and the deformations increase.

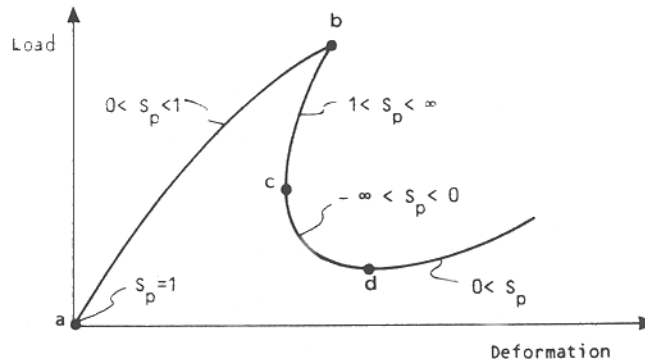


Figure 1.3-4 Elastic spring-back problem

Along path b-c the load should be reduced. The Current Stiffness is positive, but with a higher value than the initial stiffness ($/ 1$). The spring-back behaviour is often characterized by a large Current Stiffness, $S_p > 1$, in combination with a negative stiffness matrix determinant. This is included in the load control algorithm of USFOS. The user may change the value of the parameter *cmax*. A Current Stiffness larger than *cmax* will be interpreted as spring-back, and the sign of the load increment will be reversed. Currently, *cmax* is set to a high value (999), i.e. only the determinant criterion is active in detecting spring-back.

1.3.5 Load control algorithm

The following load control algorithm implemented in USFOS:

Positive increment,	$\Delta p^i \geq 0$	IF	The tangential stiffness matrix has no negative diagonal terms
		AND	$S_p^i > 0.0$
Negative increment,	$\Delta p^i < 0$	IF	The tangential stiffness matrix has one or more negative diagonal terms
		OR	$S_p^i < 0.0$
		OR	$S_p^i > cmax$

Figure 1.3-5 Load control algorithm

1.3.6 Equilibrium iterations

The pure incremental algorithm generally causes a drift-off from the 'true' solution path. (Each step is a solution of the tangential stiffness matrix - each step will move at a tangent to the 'true' curve.) Corrections for this deviation can be taken care of by specifying equilibrium iterations on the unbalance between external loads and internal forces after each load step.

In the USFOS, the tangent stiffness matrix is updated after each iteration. New plastic hinges are inserted if so should be necessary. Elastic unloading in yield hinges is not allowed.

An arc length iteration procedure is implemented, with a special algorithm for passing load limit points or bifurcation points. Instead of keeping the external load level fixed during iterations, the external load and displacement vectors vary according to a prescribed function in the 'load-displacement space'. In the current formulation, the loads and displacements are forced to move along a plane normal to the direction of the original load and displacement increment. This is illustrated in Figure 1.3-6.

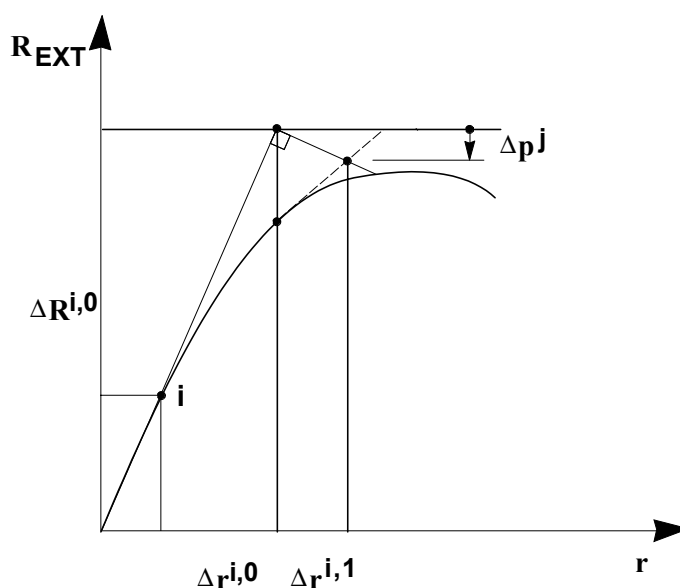


Figure 1.3-6 Arc length iterations

Iterations have converged when the change in iterational load and displacement becomes smaller than a specified limit. This is expressed by the following parameters :

$$\delta_{r_{it}}^j = \frac{\|\Delta r^{i,j}\|}{\|\Delta r^{i,0}\|} \quad \delta_{R_{it}}^j = \frac{\|\Delta R^{i,j}\|}{\|\Delta R^{i,0}\|} \quad (1.35)$$

where $\Delta R^{i,j}$ and $\Delta r^{i,j}$ are the load and displacement vectors at iteration j of step i , and $\Delta R^{i,0}$ and $\Delta r^{i,0}$ are the load and displacement increments for step number i .

The convergence parameters compare the changes in load/displacement at each iteration, with the corresponding changes during the initial increment (iteration zero). Thus, the test values start from 1.00, and should be steadily reduced as the unbalanced forces vanish.

The actual convergence criterion is defined by the parameter *epsit*. Iterations are terminated when the above expression is satisfied, or if the maximum number of iterations is performed.

In addition, iterations are terminated if a load limit point or bifurcation point is detected: The Current Stiffness and the stiffness matrix determinant are calculated at each iteration. A load limit point or bifurcation point is detected if either the Current Stiffness or the stiffness matrix determinant changes sign from one iteration to the next. If a limit point or a bifurcation point is detected, the iterations at the current load step are terminated. The results from the last iteration are accepted as the results of the load step even if equilibrium has not been obtained.

1.3.7 Plastic hinges

When a plastic hinge has been introduced, the state of forces should move from one plastic state to another plastic state, following the yield surface so that $\Gamma = 0$. However, in each increment, the element forces will move at a tangent to the yield surface. The state of forces will depart from the yield surface, as shown in Figure 1.3-1

If the pure incremental solution procedure is used, this yield surface departure will lead to $\Gamma > 0$. Small load steps should be used to keep this 'drift-off' small. However, the iterative procedure include a correction to bring the cross section force state back onto the yield surface. As long as the iteration process converge, the forces will always remain on the yield surface.

2 RUNNING USFOS

2.1 System architecture

The USFOS analysis system consists of three main program modules.

- The USFOS analysis module performs all numerical calculations, and generates two (or more...) files of analysis data. The analysis-print-file (.out) is a text file containing general analysis results; the Analysis-data-file (.raf) is a binary file containing structure data and analysis results data. This file is as a result database for XFOS and POSTFOS. In addition, global analysis results are logged on terminal or batch-output device.
- XFOS is an interactive system for visualization and presentation of USFOS analysis results. Three dimensional pictures of the analyzed structure may be presented in colours at selected deformation states in order to investigate the collapse process of the structure. XFOS also generates XY-plots of global structural behaviour as well as element history results. Colour pictures/plots are generated in PostScript format for plotting or text document inclusion. XFOS accesses the USFOS binary result database through POSTFOS.
- POSTFOS a module designed to extract data from the USFOS binary result database. POSTFOS is command oriented, with extensive built-in HELP functions. POSTFOS generates text files of selected analysis results. POSTFOS is mainly used through XFOS, but can also be used as a standalone program to extract data to tables or for plotting outside XFOS.

2.2 Memory allocation

Both USFOS and POSTFOS will allocate a default amount of your computers memory at startup. Sometimes this is not enough to read your (big) model. To allocate more (or less) than default memory at startup, simply add a number after the command, ex:

```
>usfos 50
```

this will start USFOS with 50 million words of memory.

To change the amount of memory POSTFOS allocates when started from XFOS you have to edit the *Xfos* file in your HOME directory. Change the parameter *postfos size* to suit your needs.

Table 2.2-1 System files

	Type	Content
USFOS Input		
ANALYSIS CONTROL FILE	FEM	Control parameters for the USFOS nonlinear analysis. Generated manually.
STRUCTURE MODEL FILE	FEM	Finite element idealization of the structure. Generated manually or by preprocessor program.
LOAD FILE	FEM	Structural loads. Generated manually or by load generation program.
USFOS Output		
ANALYSIS PRINT FILE	OUT	Print of analysis results : Input verification, global history output or output of each load step.
ANALYSIS DATA FILE	RAF	Structure data and analysis results of each load step.
POSTFOS Output		
POSTFOS PRINT FILE(S)	PRI	Printed tables of USFOS analysis results.
POSTFOS PLOT FILE(S)	PLO	Plot data of USFOS analysis results.

2.3 Input

The USFOS analysis module reads data from one, two or three input files. With exception of the control parameters for the non-linear analysis, the user is free to organize the data on these files. The specific content of each file is not important, as long as all data are present on the files used.

Only the filenames are input to USFOS. The filetypes are predefined by the program system; the files **MUST** have filetype *fem*.

User input to USFOS is read from text files. Two input formats for structure and load data are currently supported directly:

- USFOS reads structure and load input prepared for the SESAM program system directly.
- A simplified USFOS specific format (UFO) may also be used for structure and load input.

In the current version of the User's manual, one chapter describing the UFO file format is added. The UFO file format is used to describe the same type of information, which normally is described in SESAM file format, and has been used since 1994 by non-SESAM users. The type of information is: Nodal ID's, Coordinates and Boundary conditions, Element ID's, connectivity and properties etc. USFOS recognises the file format automatically, and the results are unaffected by the structural/load file format used. **However, mixing commands from the two input formats are not possible.**

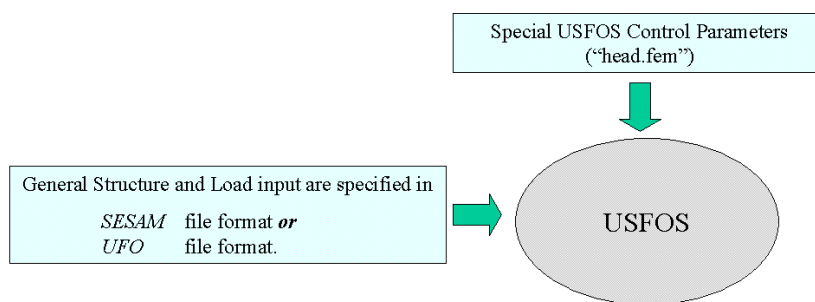


Figure 2.3-1 Input files to USFOS

STRUMAN, a converting tool to handle other input format is available free of charge on a “as is” basis (no formal support is given, STRUMAN is still considered to be a SINTEF in-house program).

Since USFOS requires only one element per physical element, a structural model developed for LINEAR analyses may be used more or less directly in the USFOS nonlinear analysis. Little extra input is needed.

All control parameters and additional input for nonlinear analysis may be specified separately, in the Analysis Control File. Structure data can also be read from this file, but is usually given on one or two separate files. The specific content of these files is not important, as long as all data are present. For convenience, these files are labeled "Structure File" and "Load file".

Figure 2.3-2 shows interactive program initiation. User input is underlined.

```
$ RUN <USFOS>USFOS

      - U S F O S -
      Progressive Collapse
      Analysis of Frame Structures
      Version 5.3 / Release 92-02-01
      SINTEF div of Structural Engineering

      Control   file   prefix : ZAYAS-FRAME-HEAD
      Structure file prefix : ZAYAS-FRAME-STRU
      Load      file   prefix :
      Result    files  prefix : ZAYAS-FRAME
```

Figure 2.3-2 Interactive running.

2.4 Output

The main results of a USFOS analyses are

- Ultimate collapse load or critical collapse temperature
- Energy absorption
- Load displacement relations at any nodal point
- Element forces at all load levels
- Formation of plastic hinges
- Redistribution of forces

The results are presented in the following way:

- As plots and images, presented by the graphical post-processor XFOS
- As printed tables, presented by the POSTFOS module
- As analysis print-out on the Analysis Print File generated by USFOS during the analysis
- As on-line print-out to terminal or batch-output stream

2.4.1 Global history output

The global history output gives an overview of the total, global behaviour of the structure during loading. Load, displacement, energy and structural stiffness are listed at each loadstep, with the formation or removal of element plastic hinges. The global history output may be written to the batch-out stream during analysis, or may be generated by POSTFOS.

The formation of plastic hinges is listed at each load step. This is of particular interest to determine the redistribution of forces throughout the structure, and to isolate the elements that trigger the final collapse.

The global history output is shown in Figure 2.4-1. Each term is briefly commented in Table 2.4-1.

===== U S F O S A N A L Y S I S R E S U L T S =====								
U S F O S Z A Y A s FRAME S I N T E F progressive collapse analysis div of Structural Engineering								
USFOS load comb.	Load step	Load level	Current stiff.	Control displ.	Energy absorb.	Elem. no.	Event type	Event pos.
1	1	1.000	1.000	4.380E-03	8.760E+01			
1	2	2.000	1.000	8.760E-03	3.504E+02			
1	3	3.000	1.000	1.314E-02	7.885E+02			
1	4	4.000	1.000	1.752E-02	1.402E+03			
1	5	5.000	1.000	2.190E-02	2.190E+03			
1	6	5.500	1.000	2.409E-02	2.650E+03			
1	7	5.923	1.000	2.595E-02	3.074E+03			
						6	PLAST	END1
1	8	6.099	.948	2.676E-02	3.270E+03	8	PLAST	END2
1	9	6.489	.887	2.869E-02	3.754E+03	1	PLAST	END2
1	10	6.500	.865	2.874E-02	3.769E+03			
1	11	6.535	.865	2.892E-02	3.815E+03	3	PLAST	END1
1	12	6.585	.837	2.918E-02	3.884E+03			
1	13	6.609	.836	2.931E-02	3.917E+03	3	PLAST	MID
1	14	6.631	.785	2.943E-02	3.949E+03	3	UNLOD	END1
						1	PLAST	END1
						1	PLAST	MID
						1	AXIAL	MID
						2	PLAST	MID
						2	AXIAL	MID

Figure 2.4-1 Global history output.

Table 2.4-1 Terminology, global history table

USFOS load combination:	Load combination number, or basic load case number.
Load step:	Number of times the initial load has been incremented.
Load level:	Relative load level of the current load combination and load step. The load level is "local" within each combination, starting from zero when a new load combination is specified.
Current stiffness:	Structural stiffness. The initial stiffness is 1.0. Decreasing value represents a decreasing stiffness in the structure.
Control displacement:	Equivalent displacement of the structure. The displacement is calculated as a balanced average of selected displacements.
Energy absorption:	Accumulated external work absorbed by the structure. This is the total energy of all load combinations.
Element number:	Element identification number.
Event position:	Position where a plastic hinge is formed/removed.
END1	First element end
END2	Second element end
MID	Element midspan
JNT1	Joint at first element end
JNT2	Joint at second element end
Event type:	Change of element status.
YIELD	The forces has reached first fibre yield of the cross section, and a yield hinge is formed.
PLAST	The forces has reached the full plastic capacity of the cross section.
UNLOD	The element has unloaded and the cross section has returned to the elastic state.
AXIAL	The element forces have reached the full plastic tension capacity of the member. A membrane element is introduced, accounting for geometric stiffness of the member.
FRACT	Fracture is detected in the member.
JOINT	The full capacity of the joint has been reached and the joint is yielding.
MIN STEP LENGTH:	Attempt to scale the load step below the minimum size specified by the user.
MAX DISPL INCR:	Load step scaled due to large displacement increments.

2.4.2 Analysis-print-file

The Analysis Print file is a text file generated during USFOS analysis. The file contains input verification data and analysis results at each load step.

The amount of print is controlled by the user through the parameters *inprint* and *outprint* of the *CPRINT*-record, governing input verification print and analysis output, respectively.

The amount of input verification data is governed by the input parameter *inprint*.

The minimum amount of print is shown in Figure 2.4-2. This is key-parameters, load control data, displacement control data and element imperfection data (if any).

Additional input verification data is

- Structural Data
 - Nodal point data
 - Material data

- Element data
 - Spring characteristics
 - Local coordinate system data
 - Cross sectional data
- Load Data
 - Distributed element loads
 - Nodal point loads
 - Gravity loads
- Internal F.E.M parameters
 - Element degrees of freedom
 - Nodal degrees of freedom
 - Nodal point connectivity

An example of structural data is shown in Figure 2.4-3.

2.4.3 Analysis Status file

USFOS creates a text file at the end of the analysis, (with name f.inst. jacket_status.text if jacket was the result file prefix). The status file gives a brief overview of the analysis:

- Time/Load level for first yield, first plastic hinge, first buckling, first element exceeding a specified utilization threshold.
- Number of negative pivot-elements.
- Number of new hinges in one step.
- Iteration convergence information.
- List of the 10 first elements yielding, getting plastic hinges, buckling and exceeding the specified utilization threshold.

```

=====  A N A L Y S I S   P A R A M E T E R S   =====

              Z A Y A s  FRAME
      U S F O S  progressive collapse analysis
      S I N T E F  div of Structural Engineering

Number of input lines read           = 220

Number of nodal points               = 13
Number of structural elements        = 23
Number of springs to ground          = 0
Number of shell property elements    = 0
Number of overlaps                   = 0
Number of damaged elements           = 0
Number of materials                  = 3
Number of cross sections             = 5
Number of spring characteristics     = 0
Number of linear dependencies        = 0
Number of element imperfection groups = 0
Number of local element coord. systems = 23
Number of local nodal coord. systems = 0
Number of eccentricity vectors       = 0
Number of load cases                 = 1
Number of temperature fields         = 0
Number of load combinations          = 0
Number of control nodes              = 1
Number of steps in post-collapse     = 15

Numerical accuracy equation solver   = 1.00E-20
Numerical accuracy interaction surface = 1.00E-01

Combined shape function, load level = .050
Max recalculations due to unloading = 5
Elastic spring-back introduced at CSTF > 1.20E+00
Local dent formulation used
Restart data stored at intervals     = 1

-----  L O A D      C O N T R O L      D A T A      -----
USFOS   Load   Max.   Max.   Min.
load    scaling load  no. of  displ.
comb.   factor  level  steps  step

      1      1.000   5.000    0    .010
      1      .500   6.500    0    .010
      1      .050   .000   20    .005
      1      .100   8.000   40    .010

-----  D I S P L A C E M E N T      C O N T R O L      -----
      External      Global      Displacement
      node          displacement
      no.            direction    weight
                                factor

      1              X            1.000

-----  A L L O C A T E D      D A T A      S P A C E      -----
                                Integer      Real
                                data         data

Total          200000      2500000
Used           10743       19747

Max no. of elements to develop
plastic hinges at mid-span is      4305
(requires additional data storage)

```

Figure 2.4-2 Input verification

```

-----
NPEX      N O D A L      P O I N T      D A T A      -----
          NP              X              Y              Z              BOUN.COND.

   1       1              .000000          .000000          8.382000
   2       2              1.524000          .000000          8.382000
   3       3              3.048000          .000000          8.382000
   :       :              :              :              :
  11      11              3.048000          .000000          .762000
  12      12              .000000          .000000          .000000  X X X X  X
  13      13              3.048000          .000000          .000000  X X X X  X

-----
E L E M E N T      D A T A      -----
ELEX  ELNO  ELTYP    GEOM    MATER    NP1    NP2    LCOOR  ECCEN1  ECCEN2

   1     1    BEAM     4       1       6       5       1       0       0
   2     2    BEAM     4       1       7       6       2       0       0
   3     3    BEAM     4       1       4       6       3       0       0
   :     :     :       :       :       :       :       :       :
  21    21    BEAM     1       2       8      11      21       0       0
  22    22    BEAM     1       2      10      12      22       0       0
  23    23    BEAM     1       2      11      13      23       0       0

-----
L O C A L      C O O R D I N A T E      S Y S T E M S      -----
ELEX  LCNO      LOCAL-X      LOCAL-Y      LOCAL-Z

   1     1      .707      .000      .707      .000      1.000      .000      -.707      .000      .707
   2     2      .707      .000      .707      .000      1.000      .000      -.707      .000      .707
   3     3      .707      .000      -.707      .000      1.000      .000      .707      .000      .707
   :     :       :       :       :       :       :       :       :       :       :
  21    21      .000      .000     -1.000     -1.000      .000      .000      .000      1.000      .000
  22    22      .000      .000     -1.000     -1.000      .000      .000      .000      1.000      .000
  23    23      .000      .000     -1.000     -1.000      .000      .000      .000      1.000      .000

-----
G E O M E T R Y      P A R A M E T E R S      -----
GEO  TYP      Area      Ixx      Iyy      Izz      Sh. area      Sh. area
          mod-x      mod-y      mod-z      y-axis      z-axis

   1  PIPE  7.101E-03  1.782E-04  8.908E-05  8.908E-05
          1.125E-03  7.160E-04  7.160E-04  3.552E-03  3.552E-03

   :    :    :       :       :       :       :       :

   5  I/H  1.570E-01  1.449E-04  6.045E-02  1.519E-02
          3.435E-02  9.372E-02  3.761E-02  8.128E-02  3.418E-02

-----
M A T E R I A L      P A R A M E T E R S      -----
MAT  TYP      Youngs      Poiss.      Yield      Density      Thermal
          modul.      ratio      stress      Density      expan.

   1     1      2.100E+11  3.000E-01  2.480E+08  7.850E+03  1.400E-05
   :     :       :       :       :       :       :
   3     1      2.100E+11  3.000E-01  3.240E+08  7.850E+03  1.400E-05

```

Figure 2.4-3 Structural data

The load step output in the analysis-control-file gives detailed information of the structural response. At each load step the load, energy and stiffness are listed; interaction function values and status of selected elements; total displacements of specified nodes and global reaction forces of fixed nodes.

Formation/removal of plastic hinges at each element is commented.

The accumulated displacements are printed for all nodes included in the "global displacement" defined by the user.

Load step 1 / 19									
===== I N C R E M E N T A L S O L U T I O N =====									
Z A Y A s FRAME									
U S F O S progressive collapse analysis									
S I N T E F div of Structural Engineering									
USFOS load combination no = 1									
Load step no. = 19									
Load increment scaled to minimum step length									
Load increment = .050									
New load level = 5.939									
Current stiffness parameter = .409									
Solution accuracy parameter = 2.800*E-00005									
Determinant of tangential matrix = 7.980*E 00523									
Energy absorbtion = 3.699*E 00003									
----- I N T E R A C T I O N F U N C T I O N V A L U E S Fb(Fy) -----									
ELEM	ES	Node1	Midspan	Node2					
1					Yield at end 1				
1	5	-.19(.10)	-.19(.09)	-.19(.04)	0---0---+				
2	7	-.15(.17)	-.18(.11)	-.06(.06)	0---0---0				
3					Unloading at end 1				
3					Unloading at midspan				
3					Yield at end 1				
3	7	-.22(.01)	-.29(.04)	-.22(.15)	0---*---0				
4	7	-.25(.03)	-.29(.04)	-.23(.06)	0---0---0				
5					Yield at end 1				
5	1	-.32(.00)	-.33(-.01)	-.34(-.02)	0---+---+				
6	1	-.28(.00)	-.33(-.01)	-.33(-.01)	0---+---+				
7					Yield at end 2				
7	2	-.34(-.03)	-.32(.00)	-.32(.00)	+---+---0				
8					Unloading at end 2				
8	5	-.31(.00)	-.32(.00)	-.28(-.02)	0---0---+				
9	0	-.64(-.47)	-.72(-.57)	-.74(-.60)					
10	0	-.73(-.58)	-.70(-.55)	-.64(-.46)					
13	0	-.60(-.50)	-1.00(-1.00)	-.60(-.50)					
20	0	-.78(-.70)	-.84(-.78)	-.57(-.43)					
21	0	-.78(-.70)	-.84(-.78)	-.57(-.43)					
22	0	-.51(-.35)	-.71(-.60)	-.90(-.84)					
23	0	-.51(-.35)	-.71(-.60)	-.90(-.84)					
----- G L O B A L T O T A L D I S P L A C E M E N T S -----									
NODE		X-dis	Y-dis	Z-dis	X-rot	Y-rot	Z-rot		
1		2.872E-02	.000E+00	1.772E-03	.000E+00	1.210E-03	.000E+00		
----- G L O B A L R E A C T I O N F O R C E S -----									
NODE		X-for	Y-for	Z-for	X-mom	Y-mom	Z-mom		
12		1.219E+05	.000E+00	-6.534E+05	.000E+00	1.104E-11	.000E+00		
13		1.157E+05	.000E+00	6.534E+05	.000E+00	1.440E-11	.000E+00		
TOTAL:		2.376E+05	.000E+00	2.910E-10					
Load step 1 / 19 cpu = 1.90 sec, Total accumulated cpu = 32.68 sec									

Figure 2.4-4 Load step output

Table 2.4-2 Terminology, load step output

ELEM:	Element identification number.
ES:	Element status.
INTERACTION FUNCTION VALUES:	These values represents the accumulated stress level of each element position with the value -1.00 in the initial, stress-free configuration, and the value 0.00 when first fibre yield or the full plastic capacity is reached. The primary columns concerns the full plastic capacity (Value 0 when the full plastic capacity is reached); the secondary column (in brackets) reaches zero on first fibre yield.
Yield at ...:	Short comments when the status of an element is changed
Plastic hinge at ...	
Unloading at ...	
Tension failure mode	
Fracture at ...	
Capacity lim. at ...	
... end 1	: Position where the element status changes
... end 2	
... midspan	
... joint 1	
... joint 2	
0---+---+	Representation of the element status, corresponding to the value 'ES'.
0	Yield hinge inserted.
+	Position where element forces are checked for plasticity.
*	Plastic hinge removed at element midspan. Internally, the element is still divided in two sub-elements.
0- -0- -0	Plastic tension failure. The axial tension force has reached the plastic capacity, and a membrane element is inserted.
GLOBAL TOTAL: DISPLACEMENTS	This table shows the total accumulated displacements up to and including the current load increment.
GLOBAL REACTION : FORCES	This table show the total accumulated reaction forces up to and including the current load increment.
NODE:	Nodal point identification number.

2.5 Practical Considerations

2.5.1 Load specification

The user supplies the basic *load cases* as input to USFOS. In the run commands to the program, the load cases are combined into *load combinations* (if necessary). Then, the user specifies the *loading history* for the analysis (which loads are to be applied, in which order, and how large increments are to be used ...)

The loads are incremented a given number of steps, up to a specified load level, or until a defined displacement is reached. The load is incremented on top of the previous loads, i.e. each load increment is added to the accumulated load of all previous load steps. Thus, the calculated results at each step are the combined results of the total load history prior to and including that step.

The results of different load cases may not be superposed, since the response of the structure is highly history dependent. In nonlinear analyses, the actual loads must be combined.

2.5.2 Analysis specification

In nonlinear analyses the accuracy of the results depend on the size of the load steps. The load steps may be large as long as the structure behaves "linearly". The more nonlinearly the structure behaves, the smaller the load steps should be. That is, the optimum load specification is closely linked to the nonlinear characteristics of the structure itself.

In the USFOS formulation, this problem is partly solved by the automatic load scaling used when plastic hinges are introduced, but the user still have to supply sensible values for the size of the initial load increment and for the minimum load step in the load scaling algorithm.

The correct size for these parameters will be a compromise between accuracy and time/cost, and the right load specification will often be determined through an iterative process as outlined below:

- 1) Determine an initial load history, based on the prior knowledge of the structural behaviour (e.g. linear elastic analyses).
- 2) Check the global behaviour of the structure, if significant and sudden redistribution of forces seems to occur at any load level. Check if the load increments at this load level are small enough to capture these effects.
- 3) Check the interaction function at the plastic hinges whether the Γ -values are at an acceptable level, or if the state of forces show significant exceedance of the yield surface.
- 4) Determine at which load level the analysis accuracy deteriorates.
- 5) If analysis results beyond this load level is required, then specify a new load history and restart the analysis.

REPEAT FROM STEP 2.

For offshore structures the global behaviour is normally fairly linear up to, and a bit beyond the factored, characteristic load ('design' load). To ensure that nonlinear effects are properly activated, an initial load step of 0.10-0.30 of the (unfactored) characteristic load is recommended.

Depending on the structure (4 leg jacket, 8 leg jacket, K-braces, X-braces etc.) the behaviour becomes more and more nonlinear from about 1.5 - 2.5 of the characteristic load and up to the maximum load. The collapse load typically varies from around twice the characteristic load for slender, 4 leg structures up to about three-four times the characteristic load for 8 leg jackets.

In general, the size of the load step should reflect the degree of nonlinearity in the response. As the load increases, the load increment and the minimum step length should be reduced.

2.5.2.1 Iterations

Equilibrium iterations may be specified to ensure equilibrium between external loads and internal element forces. For pushover analyses, the pure incremental formulation will in most cases give satisfactory results, and require less computational time. Moreover, use of iterations introduce additional complexity and new potential error sources (iteration divergence etc ...).

However, the iterative procedure include a correction to the yield surface. As long as the iterations converge, the forces will always be brought back to the yield surface.

Thus, use of iterations will be a priority between computational time / simplicity on the one hand, and accuracy / easier analysis verification on the other hand.

In most cases, use of equilibrium iterations seem to be beneficial.

2.5.2.2 Minimum step length

When yielding occurs, the load step is scaled so that the state of forces comply 'exactly' with the yield surface. In the nonlinear range, several cross sections may yield almost simultaneously, resulting in far too small load steps. To ensure reasonable progress in the analysis, the 'minimum step length' parameter is introduced. However, the minimum step size should not be specified too large.

If more than, say, 5 hinges are introduced in a 'MIN STEP LENGTH'-step, this may indicate that the step is too large. The 'true' yielding process may not be identified (how yielding in one member influence the load redistribution to other members), and the correct failure mechanisms may not be initiated.

As already stated, finding the right load specification will often be an iterative process.

2.5.3 Analysis verification

2.5.3.1 Iteration Convergence

Iterations may be specified to ensure equilibrium between external loads and internal element forces. To verify the numerical accuracy of the analysis, the iteration test parameters should be checked for each load step. The test parameters are printed to the 'OUT'-file for each iteration at every load step. To ensure the required numerical accuracy, the final values should be below the input convergence criterion.

Note that the program may terminate iterations without having reached convergence. This is done to avoid known divergence situations, identified by the following criteria:

- If the Current Stiffness parameter changes sign during iterations
- If the Determinant changes sign during iterations

If iterations fail to converge, observe the following :

- 1) Verify that the solution converges in the next step, or that number of steps before next iteration convergence is limited. With iterations, we can control any deviation from the 'true' solution, but any step without iterations (or iteration divergence) will introduce an 'error' in the solution.
- 2) The solution may fail to converge, but still show a steady, stable behaviour of the test values. The final value (after max number of iterations) may be close to, but not quite below the specified convergence criterion. This indicates a stable solution, but you should make sure the solution converges after a limited number of steps.
- 3) On the other hand, the test values may increase severely, or may jump up and down. This can be a sign that the solution has broken down, and the further results should be regarded with great caution. Probably, a minor or major discrepancy have been introduced in one of the previous steps, leading to uncontrolled behaviour.

If you have problems due to the iteration divergence, modify the control parameters for the steps prior to the divergence.

- Reduce step size
- Reduce the minimum step size, *minstp*
- Reduce max displacement increment, *mxdpdis*
- Increase the number of iterations
- Strengthen (reduce) the convergence criterion

2.5.3.2 Γ -values (Interaction Function Values)

The Γ -values should be checked at every step. Γ -values above 0.00 (1.0 in XFOS..) imply that the solution have deviated from the 'true' solution, and may be considered as if that member's load carrying capacity has been overestimated by the same value.

Generally, if iterations are applied successfully the Γ -values should be equal to zero. (The exception is AXIAL FAILURE members.)

For pushover analyses, some deviation from the yield surface can be tolerated, depending on the importance of the actual member in the global load carrying behaviour of the structure. As a rule of thumb, the Γ -values should usually not exceed 0.05 for primary members (e.g. legs and primary braces of jackets), whereas values up to 0.20 might be acceptable for secondary members. However, this must be considered in each case against the use of the analysis results and the necessary accuracy of the results.

To verify the analyses, observe the following steps :

- 4) Check that the Γ -values are small.
- 5) If there are non-zero Γ -values in any step, check that are within acceptable limits, or that they become zero in the succeeding step(s) if iterations are specified.
- 6) Determine at which load level the deviation starts. Are accurate results beyond this load level required ?
- 7) If 'yes', then the analysis has to be resumed with modified control parameters at this load level :

Reduce step size

Reduce the minimum step size, *minstp*

Reduce max displacement increment, *mxdpdis*

2.5.3.3 Number of Hinges per Load Step

Number of yield hinges introduced in one load step should not be too large. Yielding in one member influence the load redistribution to other members and thus, the formation of new hinges. If too many new hinges are introduced in a 'MIN STEP LENGTH'-step, the 'true' yielding process may not be identified, and the correct failure mechanisms may not be initiated.

Preferably no more than, say, 5 hinges should be introduced per step.

2.5.4 Potential problem areas

2.5.4.1 Repeated Plastification/Elastic Unloading

A known problem in USFOS is repeated plastification/ elastic unloading of specific members. This is termed as "false on-/off loading".

When USFOS detect unloading of a member during the load increment, the stiffness matrix for this element is re-calculated, and the system stiffness matrix reassembled. The load step is then repeated, and all members are checked for yielding, as usual. The problem occurs when yielding is detected in the same element that just unloaded. The load step is scaled to 'zero' (at least to minimum step size), and a plastic hinge is introduced. In the next step, the procedure may repeat itself. Specific elements may keep on loading on/off for a significant number of load steps, 'clogging up' the analysis.

Of course, during significant redistribution of forces within the structure, or at bifurcation points, the analysis will often need some steps of on/off to 'hit' the correct failure path. But if one member 'stops' the analysis, this can be a significant obstacle.

To circumvent this problem, a procedure has been implemented to identify members with repeated on/off loading, and to prevent them from stopping the analysis.

- The user can specify an 'acceptable' number of subsequent load steps with plastification/ elastic unloading for one element
- If this number is exceeded, the element is prevented from unloading in the subsequent steps (but new hinges may still be introduced)
- The restriction is removed the first time the element goes through a load step without trying to unload
- The restriction is also removed on the first step of every new load vector (each new CUSFOS/CICYFOS-line). In particular, all elements are free to unload when the external load is reversed

In general, such a restraint introduces artificial restraints in the solution, and should be used with care. But, the error introduced will often be less than the inconvenience of 'clogging up' the analysis.

If you still get problems related to repeated 'on/off', observe the following steps :

- 8) Check if there is any indication of a bifurcation point in the previous load steps
- 9) If not, set a reasonable number of 'on/off's before 'locking' members
- 10) Check if they continue on/off at the next load vector (next CUSFOS/CICYFOS line) -in that case the locking is probably OK

- 11) If they unload, and stay that way after the next load vector, this indicates real, physical unloading, and that the previous locking may have introduced errors in the solution.
- 12) Decide if you can let the member move free until this load vector, or terminate the previous load vector earlier.

2.5.4.2 Tension Failure

For a member yielding in pure tension, increased loading should result in increased axial forces (due to strain hardening), and reduced, or constant, bending moments. I.e., cross sectional forces should move in direction of the positive N axis of the yield surface.

However, the plasticity formulation states that the forces should move along the yield surface, from one point on the yield surface, to another point on the yield surface. And at pure tension yielding, the yield function is singular. The force point may cross from one quadrant of the yield surface, over to the other quadrant, instead of moving in direction of the positive N axis. In the next step, it may cross back over again.

A special membrane element is therefore implemented to model pure tension yielding of a member. The membrane element is automatically introduced when the axial force exceeds 98% of the tension yield force ('AXIAL' failure mode). The element accounts for the geometric stiffness of the member, i.e. conserves the axial stiffness of the member, and allows transformation back to a beam element if the member is unloaded.

However, if the two-surface material model is used (gradual plastification of the cross section), then the 'over-crossing' can still occur at the tip of the inner surface (the yield surface).

In some cases, this over-crossing may reduce the accuracy of the analysis. It is therefore recommended to check any elements with 'AXIAL' failure if any over-crossing has occurred. If the relative magnitude of the bending moments is small, the analysis is probably OK. If the over-crossing is too large, the analysis should be repeated with smaller loadsteps in that area.

USFOS scales the load step when ordinary yield hinges are formed. But in the present version, the load step is not scaled when the element reaches AXIAL FAILURE. The Γ -values may jump far off the yield surface. Therefore, the Γ -values of AXIAL FAILURE members must be checked separately. It may be necessary to use extremely small steps to pass a point where an AXIAL FAILURE element has 'taken off' from the yield surface.

2.5.4.3 Bifurcation

Sometimes the solution fails to detect the correct failure path during traversal of a bifurcation point. This is not a general problem, but can occur for specific structures.

In some cases, this can be identified by erroneous development of element forces. For example, compression members get increasing axial forces, even after a three-hinge mechanism has been formed and buckling is expected. In this case, the analysis should probably have performed unloading of the structure at some previous load level.

In other occasions, repeated plastification/elastic unloading of single members may be an indication that the solution is proceeding along an erroneous failure path.

To verify the analyses, observe the following steps :

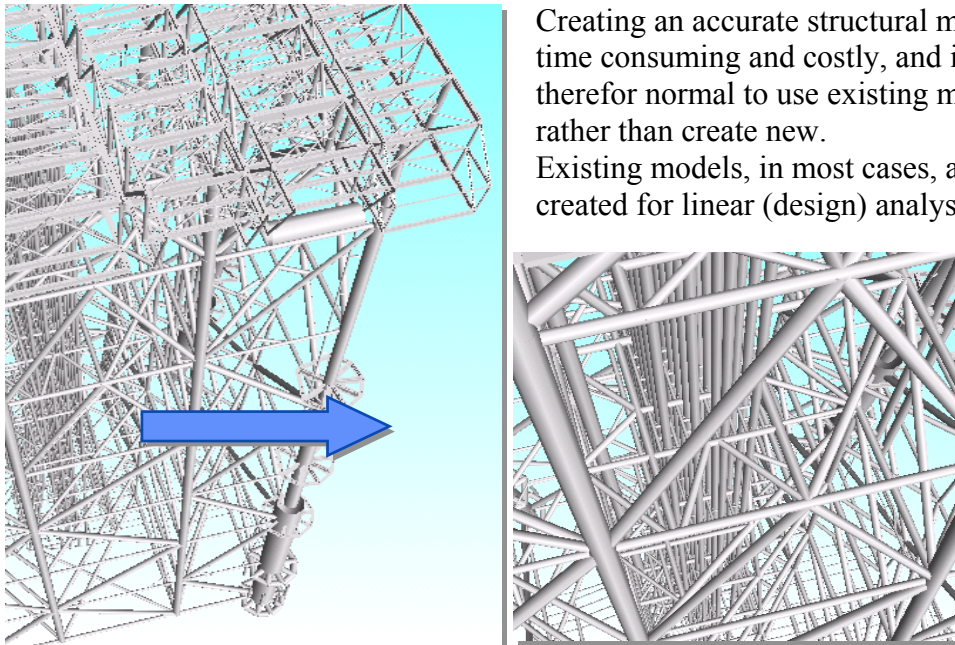
- 1) Ensure that the determinant criterion is active and that the *cmax* parameter has a large value (e.g. 999)

- 2) Check the development of the Current Stiffness parameter and the determinant. If the sign changes simultaneously, the solution is stable, and USFOS should follow the correct path.
- 3) If the determinant changes sign, but the Current Stiffness parameter remains positive, this may signify that the analysis has passed a bifurcation point.

To improve the bifurcation point traversal, the procedure with buckling mode injection may be employed.

3 “LINEAR” MODEL, NON-LINEAR ANALYSIS (“Model Repair”)

3.1 Large Models



Creating an accurate structural model is time consuming and costly, and it is therefor normal to use existing models rather than create new.

Existing models, in most cases, are created for linear (design) analysis.

Figure 3.1-1 Large Challenge for Non Linear Analysis

Seldom, existing models are created with non-linear analysis in mind, and substantial work has to be done before it's suited for non-linear problems. As computers are getting faster, the model size may increase correspondingly. But, modification of models means in practice manual work, and the bigger models, the more man hours have to be spent in order to 'repair' the linear model. A few years ago, a typical jacket structural model consisted of 500-1000 members. Today the same structure is represented by 5000-10000 members. An increasing part of the model is *non structural* members introduced of different reasons in the linear analysis, see Figure 3.1-1 for typical example.

If possible, the original structural model should become “read only”, and an “intelligent filter” should transfer the ‘linear’ model into a model accepted by the non-linear tool, see Figure 3.1-2.

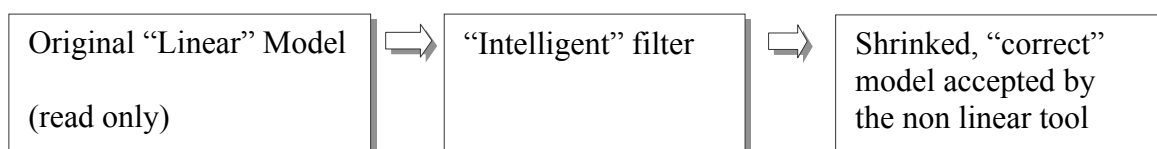


Figure 3.1-2 Preferred “Model Repair” solution

Often, the original (linear) model will not run at all, the analysis fails due to lack of boundary conditions, etc. To be able to inspect the structure in XFOS, the use of the dynamic load procedure is a useful intermediate solution, see Table 3.1-1. In an early modelling stage, the gravity loading is sufficient load to ensure that all elements are connected, boundary conditions correct, etc.

<i>Dynamic</i>	<i>0.1</i>	<i>0.025</i>	<i>0.1</i>	<i>0.1</i>					
<i>LoadHist</i>	<i>1</i>	<i>1</i>							
<i>TimeHist</i>	<i>1</i>	<i>Points</i>	<i>0</i>	<i>0</i>	<i>1</i>	<i>1</i>	<i>1000</i>	<i>1</i>	

Table 3.1-1 Using dynamic load procedure

Table 3.2-3 shows the group definition used on a ‘real’ example, and it’s here defined 5 groups, which all use geometry ID’s to identify the elements. The general cross sections and the small diameter pipes ($D < 300\text{mm}$) are grouped, because elements referring to those beams are the typical secondary members, which should be removed from the analysis model.

3.2 Utilizing Group definitions

- ❑ Groups are introduced in the latest USFOS version (7-7).
- ❑ A group is identified by its ID, which is a number (up to 8 digits).
- ❑ Elements become “members of” groups, and the same element may participate in several groups.
- ❑ The nodal points, to which the elements are attached, becomes “members of” the actual group.

The groups are referred to in connection with assigning properties to elements, which will ease the input (reduce the amount of input lines). In xfos its possible to include/exclude groups in the structural image (Edit/Clip/Group).

Elements are defined “members of” a group using the GROUPDEF command. The element may be identified through:

- ❑ Element ID
- ❑ All elements referring to given material ID’s
- ❑ All elements referring to given cross section geometry ID’s
- ❑ All elements ‘members of’ existing groups

The actual way of defining the elements is specified using the parameters “Elem”, “Mat”, “Geo” or “Group” as shown in Table 3.2-1.

<i>'</i>	<i>ID</i>	<i>Type</i>	<i>{ ID-List }</i>
<i>GroupDef</i>	<i>888</i>	<i>Elem</i>	<i>10 20 30</i>
<i>GroupDef</i>	<i>88881</i>	<i>Mat</i>	<i>1</i>
<i>GroupDef</i>	<i>88</i>	<i>Geo</i>	<i>5</i>
<i>GroupDef</i>	<i>8</i>	<i>Group</i>	<i>88881 88</i>
<i>'</i>			

Table 3.2-1 Defining element groups using of the GROUPDEF command

If wanted, extra nodes could be defined “members of” an actual group, and the command “groupnod” is used for this purpose, see Table 3.2-2. This command is used in connection with ‘guiding’ loads from non structural members towards (kept) structural nodes.

'	Group ID	Nodes.....
GROUPNOD	8	1 2 3

Table 3.2-2 Assigning (extra) nodes to a group using the GROUPNOD command

When the groups are defined, one single NONSTRU command will remove all the “members of” the actual groups from the analysis model (but loads are kept).

```

-----
Specify Groups. (which should become nonstructural)
-----
      Type      List.....
GroupDef 1000  Geo

- GenBeams

10101 10228 10229 10230 10231 10251 10252 10352 15198 15199
16106 16129 16193 16194 16195 16196 16197 16198 16199 16206
16229 16293 16294 16295 16296 16297 16298 16299 16306 16329
16393 16394 16395 16396 16397 16398 16399 16406 16429 16493
16494 16495 16496 16497 16498 16499 16506 16529 16593 16594
16595 16596 16597 16598 16599 16606 16629 16693 16694 16695
16696 16697 16698 16699 17529 17592 17593 17594 17597 17598
17606 17629 17693 17694 17695 17696 17697 17698 17535 17600
17634

GroupDef 2000  Geo

- Pipes 1

19107 19108 16202 16302 10253 16102 16402 17502 17602 16502 16602
10102 10104 15110 19106 15186 10106 10105 10107 19105 15111 15106 19104 20110
10113 10360 10111 15185 15112 15107 10109 10112 19103 20095 20096 20094 15114
15113 20097 15191 20099 20098 16607 10117 16407 16307 16207 16107 16507 10365
10243 17607 10367 10118 10114 10116 10119 20111 10122 20124 16213 16212 16214
16109 16110 16209 16114 16210 16112 16113 17509 17510 17511 16614 16610 16612
16613 17512 17612 17613 17614 17610 17513 17514 17609 16609 16314 16409 16410
16313 16309 16310 16312 16412 16512 16513 16514 16510 16413 16414 16509 10121
10120 19102 16617 16616 15189 17517 17516 15115 16516 16416 16417 16316 16317
16517 15108 16216 10123 10125 10126 16217 17617 17616 16117 16116
16218 16119 16318 16118 16618 20085 16619 17619 17618 16519 20122 16419 16418
20112 16518 17620 16120 16420 16320 16520 16620 16220 10127 19101

GroupDef 3000  Geo

- Pipes 2

10102 10104 10105 10106 10107 10109 10111 10112 10113
10128 10130 10131 20072 20113 20114 10181 19109
20075 20076 20077 20073 20074 20080 20082 20080
20082 10185 10186 10102 10253 16202 16302 16402 16502 16602
17502 17602 10114 10365 10183

GroupDef 16319  Geo 16319
GroupDef 16219  Geo 16219

-----
SPECIFY Groups 1000, 2000 and 3000 NonStructural.
-----

```

Table 3.2-3 Shrinking model using the GROUPDEF and NONSTRU commands

If the definition of the bounding surface (the gbound command) is left out for general sections, default values are used and a warning is printed, see Table 8.4-2. The default values are shown in the same table.

```
* Warning. GBOUND input not specified for General Beam: 10101. Default used.
* Warning. GBOUND input not specified for General Beam: 10228. Default used.
* Warning. GBOUND input not specified for General Beam: 10229. Default used.
* Warning. GBOUND input not specified for General Beam: 10230. Default used.
```

GBOUND	10101	0.8	1.0	0.6	1.0
--------	-------	-----	-----	-----	-----

Table 3.2-4 Default “Gbound” data assigned to general beams

When element groups are defined, the contents of the different groups are listed in the *.out* file, see Table 3.2-5. In the actual example, group no. 1000 is defined through geometry ID's, and the specified ID's are listed first (similar if the group was defined through material ID's).

Next, the elements, which are “members of” group no 1000 are listed, and finally, all nodal point, to which the element are connected to are listed.

```
-----  G R O U P   D E F I N I T I O N S  -----

G R O U P   label                : "Geometry Group no    1000"
Contains following Geometries:
10101    10228    10229    10230    10231    10251
10252    10352    15198    15199    16106    16129

17535    17600    17634

.....elements ..... :
5001     5002     5003     5004     5005     5006
5007     5008     5009     5010     5011     5012
5013     5014     5015     5016     5017     5018
5019     5020     5021     5022     5023     5024
5025     5026     5027     5028     5029     5030
5031     5032     5033     5034     5035     5036

78614    78615    755507    755508    726550    726551
726500    726501

.....and nodes ..... :
54531    54834    54935    54930    54535    54536
54837    54938    54936    54538    54539    54841
54942    54939    54542    54543    54844    54950
54943    54550    54856    54957    54557    54558
54859    54961    54958    54561    54562    54863
```

Table 3.2-5 Print of group data: geometries, elements and nodes on the *.out* file.

The example shown in Figure 3.2-1, represents a first stage in a model repair procedure. The entire structure is still “structural”, but members are grouped as specified above. By using the Edit/Clip/Group command in xfos, it's possible to visualise the different groups (include/exclude). The image to the right shows the full model, and by excluding all groups as seen in the “Specify Clip Group” menu, the image to the right appears.

If the NONSTRU command in Table 3.2-3 is activated (note that the # passives the command) only the elements in the image to the right remains structural, but loads are attracted on the full structure (image to the left).

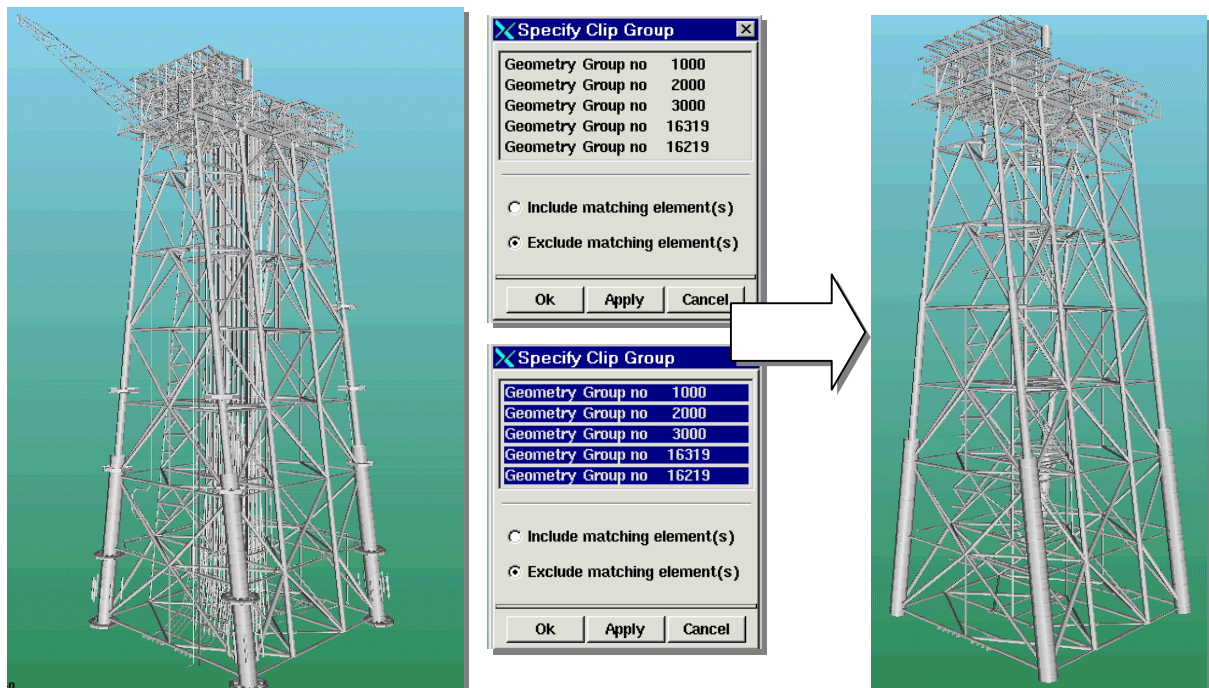


Figure 3.2-1 Edit / Clip / Group

Useful USFOS commands for the “model repair” work:

- ❑ GROUPDEF : Define element groups
- ❑ GROUPNOD : Add nodes to groups (guide loads towards nodes)
- ❑ NONSTRU : Define elements nonstructural
- ❑ STRUCTEL : Define elements structural (override NONSTRU for some elem.)
- ❑ LIN_ELEM : Define element linear elastic (with and without elastic buckling)

4 Wave Loading

4.1 Load Module

In connection with dynamic analysis of structures exposed to loads which are dependent on the structural response, it is not possible to pre-define the load history. The loads must be calculated during the analysis.

A new load module has been developed and implemented in USFOS.

This new module is designed to handle:

- Hydrodynamic loads
- Aerodynamic loads
- User defined load routines

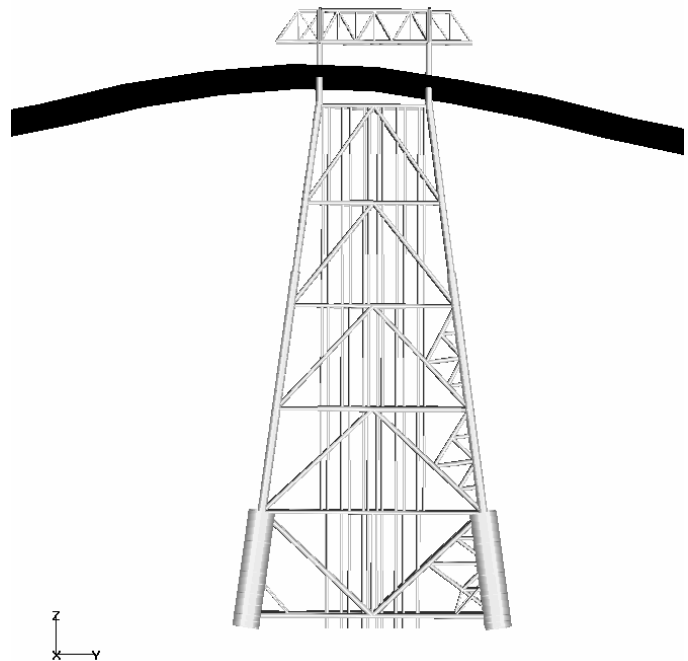
In connection with the hydrodynamic load module, the following are implemented:

- **Hydrodynamic coefficients**, C_d and C_m .
The coefficients may be defined by depth profiles and/or element by element.
- **Marine Growth**.
The marine growth thickness is defined by a depth profile.
- **Buoyancy**.
The buoyancy is calculated during the analysis which means that elements in the splash zone become buoyant/non buoyant as the surface moves up and down.
- **Flooded members**
- **Current**.
The current is defined by speed, direction and depth profile.
- **Kinematics Reduction Factor**
- **Hydrodynamic Damping**

The wave theories implemented in USFOS are:

- Airy, extrapolated
- Airy stretched
- Stoke's 5'th (Skjelbreia, Hendrickson)
- Stream Function Theory (Dean, Dalrymple)

Current and wave must be combined in the same loadcase, and it is possible to combine several (basic) waves to an irregular seastate.



In XFOS the sea surface elevation is visualized as a 'carpet' with dimensions 2*wave length in X- and Y- direction. The surface elevation which in fact presents the travelling of the wave accounts for the actual current, (which increases/decreases the propagation of the waves). An irregular seastate is visualized adding components from each basic wave to the resulting surface elevation.

The **aerodynamics** implemented is a part of a Ph.D. study within dynamic response of slender structures exposed to fluctuating wind. The study is not yet completed. For more information, please contact us.

The **user defined load** routines open for a possibility for the users to link their own load routines with USFOS. For more details, please contact us.

4.2 Extreme Wave calculation/Automatic member imperfections

Modules for calculation of hydrodynamic forces are included in USFOS. This means that using separate wave load pre processor is not needed. Using the USFOS hydrodynamic in connection with *static* "push over" analysis will typically contain following:

- ❑ Specify the actual wave (type, height, period, direction...)
- ❑ Specify the corresponding current (if any)
- ❑ Switch on buoyancy (optional)
- ❑ Specify criterion to be used for selecting worst wave position (max base shear or max overturning moment)

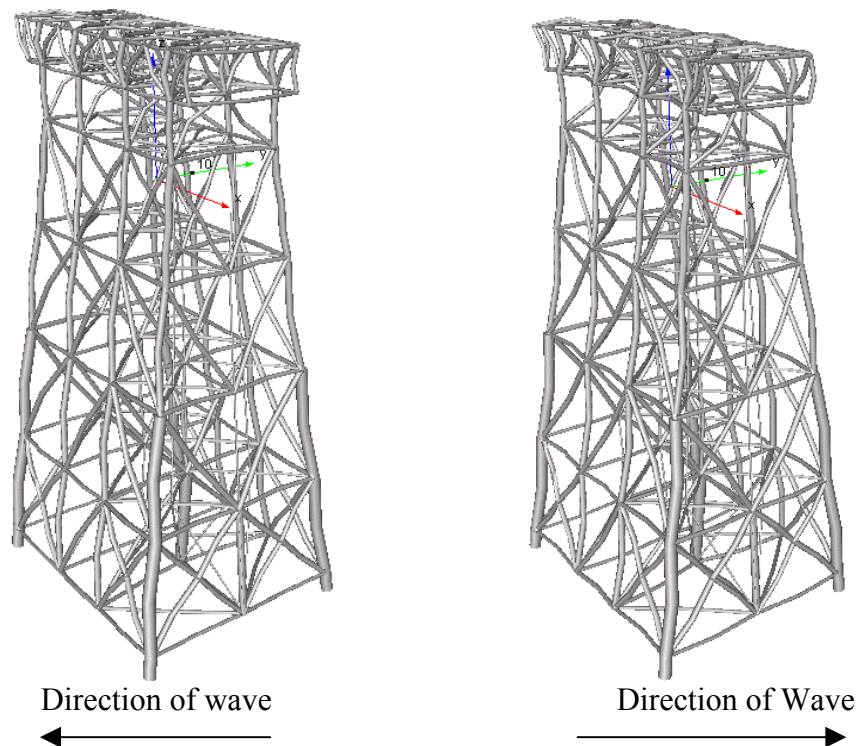


Figure 4.2-1 Automatic member imperfection according to wave force direction

USFOS will then step through the actual wave and identify the worst wave position (the position causing the highest base shear or overturning moment). The hydrodynamic forces from this wave phase (position) are saved (in memory) to be used in the pushover analysis. The calculated *buoyancy* forces are possible to separate from the other hydrodynamic forces, and the user may specify how to use the buoyancy forces, (add to an existing deadweight loadcase etc.).

Applying member imperfections, one by one, is a time consuming task, but by using the new option **CINIDEF**, the correct member imperfection is applied automatically for all beam elements. The most common buckling curves are available defining the size of the imperfection, (see User's manual Ch. 6). The *direction* of the imperfections follow the direction of the loads for a specified load case.

In Figure 4.2-1, the jacket to the right is exposed to waves with direction 45°, while the jacket to the left is exposed to a wave with opposite direction (225°). It is seen that the direction of the imperfections are opposite in the two cases (size is scaled).

All necessary input is shown in Table 4.2-1, and it should be noted that these few commands replace 1000's of input lines and use of separate wave load pre-processor / load files.

Comments to the input given in Table 4.2-1 (see also example folder wave_maxwav):

- ❑ Load case 1 is used for "dead weight" and calculated buoyancy
- ❑ Load case 2 is used for the extreme wave

Load case 1 is not scaled beyond factor 1.0 (that's why the calculated buoyancy forces is separated from the other hydro. forces and added to this load case). Load case 2 forces are scaled to platform collapse.

- ❑ The direction of the member imperfections (CINIDEF par. no 2 and 3) follows the direction of the member forces defined by load case 2 (which is the calculated wave forces).
- ❑ The size of the imperfection (CINIDEF par. no 1) is calculated according to "Chen column curve".
- ❑ A Stoke 5'th wave with height 25m, period 16s, 45° direction is applied. The sea surface is located for global Z-coordinate=0.0. Water depth is 100m.
- ❑ A current profile with peak value 2 m/s is defined with same direction as the wave. From depth 20m (Z=-20m relative to the sea surface), the current is reduces linearly.
- ❑ The actual wave is 'stepped through' the structure with time increment 1 s. The wave position giving the highest *base shear* in the interval Time = 0 -20s is used in the "push over" analysis.

NOTE As all hydrodynamic calculations are using SI units, the forces are calculated in N (Newton). If f ex. MN is used as force unit, the wave forces must be scaled before they are used in the "pushover" analysis. The command **WAVMXSCL <factor>** is used, (see also User's manual, Ch 6). In the current example, the wave forces are scaled with a factor 1.3 (just for demo purpose).

- ❑ For both the buoyancy forces and the wave forces, it is possible to print the calculated forces to separate files, but in the example, printing is switched off (nowrite).

```

-----
'   Lcomb 1 is gravity loads and static deck loads+calculated buoyancy,
'   Lcomb 2 is Stoke Wave 45 deg diretion
-----
'
'   nloads  npostp  mxpstp  mxpdis
CUSFOS      10      15      1.00   0.05
'   lcomb    lfact   mxld    nstep   minstp
'   1        1.0     1.0     10      0.05   ! Dead + Buoyancy
'   2        0.5     3.0     50      0.001  ! Wave
'   2        0.1     6.0    100      0.001  ! Wave
'
-----
'   Apply automatic out of straightness.
'   Use loads from Waves (lcase 2)
-----
'
'   Size      Pat      LoadCase
cInidef      70       1       2
'
-----
'   Separate Bouyancy from wave forces.
'   Add Buoyancy to load case 1
-----
'
'   lCase      Option
BUOYANCY      1      noWrite
'
'   - Define Wave:
'
'   Ildcs <type> H      Period  Direction Phase  Surf_Lev Depth
WAVEDATA     2  Stoke 25.0   16.0    45      0.0     0.0    100
'
'   Ildcs Speed Direction Surf_Lev Depth  [Profile]
CURRENT      2     2     45      0.0    100     0.0  1.0
'                                     -20.0  1.0
'                                     -100.0 0.0
'                                     -110.0 0.0
'
-----
'   Identify Worst Phase (Max Base Shear) and do not create a loadfile
-----
'
'   Criterion  dT      EndT      Write
MaxWave      Baseshear 1.0     20.0   noWrite
'
-----
'   Scale the Wave load. This option is required when Force Unit is not N.
'   (generated wave loads are always using Newton).
'   In this demo case, scale by 1.3 :
-----
'
WavMxScl     1.3

```

Table 4.2-1 Input for automatic wave calculations and automatic member imperfections

5 Joint Modelling

5.1 Joint capacity check

Depending on the joint geometry, the capacity of the connection brace/chord is less than the brace capacity. This means that the brace can not be utilized 100 %. In conventional joint models the limitations in load transfer through the chord surface are neglected.

The user specifies the nodes where tubular joint capacity should be considered. USFOS then calculates the geometry of the tubular joints and introduces extra elements, nodal points, geometries and materials in the finite element model.

The capacities are calculated according to API.

Figure 5.1-1a describes the user defined finite element model of a tubular joint, and Figure 5.1-1b describes the "modified" input model.

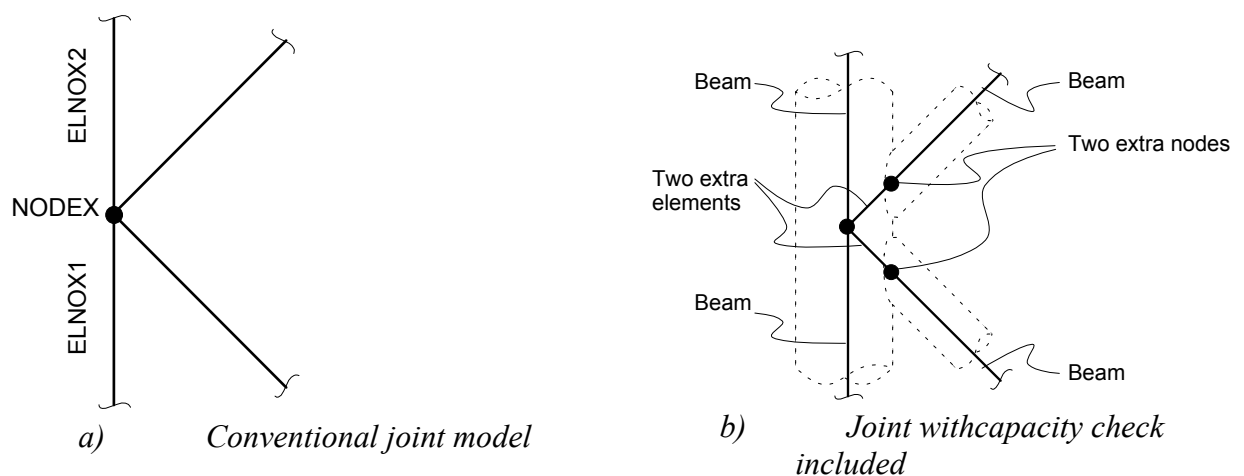


Figure 5.1-1 Joint capacity modelling

The numbering of the extra nodes and elements are as follows, see Figure 5.1-2.

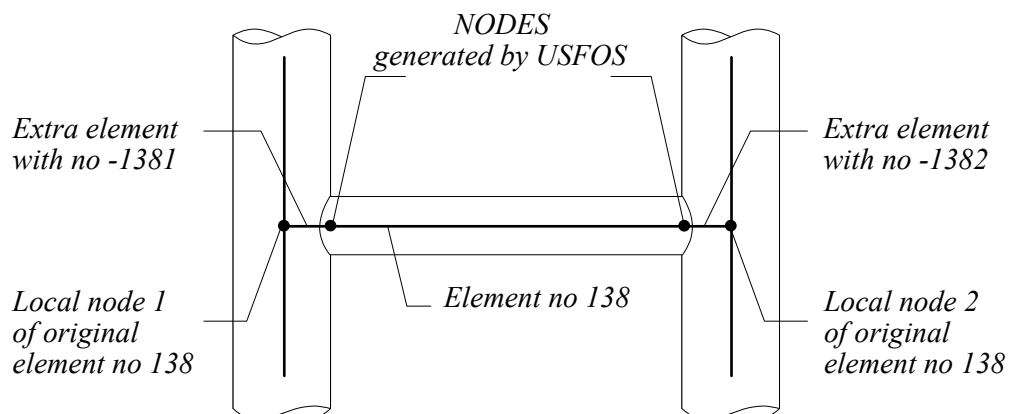


Figure 5.1-2 Numbering of extra elements generated by USFOS

Extra element at end 1 of the actual number gets the member number plus one extra digit with value 1. At end two, the extra digit has value 2.

Note! All elements and nodes generated by USFOS have negative sign.

The material and geometry numbering starts from the highest user defined material and geometry numbering.

Properties of the extra "stub" elements

The material properties are set equal to the properties of the actual chord, but hardening is not permitted.

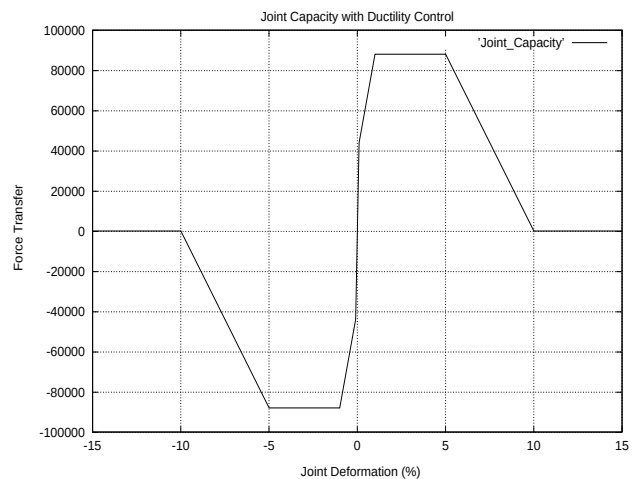
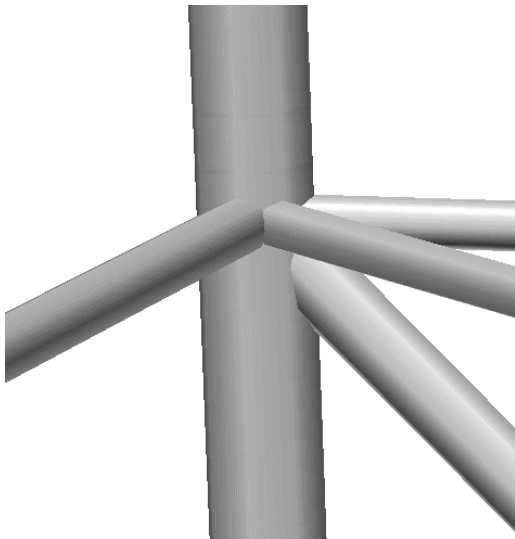
Fracture is excluded for the joint (no limit of the magnitude of the tension strain).

The cross-section parameters:

- Cross sectional area
- Plastic resistance moment about local Y-axis
- Plastic resistance moment about local Z-axis

are derived from the API capacity formulas. The other cross sectional parameters are set equal to the ones for the actual brace.

5.2 Joint deformation control



The joint capacity option is extended to have a control of the **joint deformation**, not only the force level to be transferred through the brace/chord connection.

The force - displacement characteristics (P_d curve) of the individual brace/chord connections are derived from the actual peak capacities (according to f.inst. API) as follows:

- Deformation = 0.1% of chord diameter defines 'yielding'.
- Deformation = 1.0% of chord diameter defines peak value.
- Deformation = 5.0% of chord diameter defines end of peak value.
- Deformation = 10 % of chord diameter defines joint fracture.

The generated curves are printed to the '.out' file, and in XFOS the peak capacities are printed using the Verify/Element/Information option. The joint behaviour is inspected in XFOS using the Result/History plot, and by selecting Element displacement vs. Element force for **end 2** of the joint spring elements, the load through the joint is visualized. This new option is controlled by the new USFOS command: JNT_FORM.

The peak capacities are easily scaled up and down using the command JSURFSIZ, (sensitivity studies, cracked joints, reinforced joints, etc.).

In addition, a *new user defined* joint capacity option is implemented. This option allows the user to 100% control the P_d curves of any brace/chord connection.

5.3 Joint classification / MSL joint characteristics

This write-up is a preliminary description of the implementation of MSL joint formulation in USFOS, for use with the β -release of the new feature.

The MSL equations are implemented with ductility limits and “post-rupture” unloading for tension loading, but with no ductility limits for compression loading.

Joint failure in tension invokes the “FRACTURE” option in USFOS.

Joint utilisation will be visualised by colour fringes in **Xfos**

The following shows the input required to include MSL joint characteristics in the analysis of a 2D K-frame. The input is described in more detail below.

```

'-----
' Joint properties defined by MSL curves and plasticity formulation:
'-----
'
JNT_FORM      3      ! 0=beam stub  1=P-delta spring  3=plasticity model
JNTCLASS      1      ! 0=OFF        i>0 : interval for (re)classification
'
'
' CHJOINT      nodex   chord1   chord2   Can   Rule   CapLevel GammaQf
'              7       6       7       0    MSL    mean     1.0

```

Table 5.3-1 USFOS control input activating MSL joint classification

Comparison between the USFOS analysis and alternative joint models and tests results are presented in Figure 5.3-2.

Each time joint (re)classification is performed, the following information is printed to the .out file.

Load step 1 / 60

===== J O I N T C L A S S I F I C A T I O N =====

2D K -F R A M E
U S F O S progressive collapse analysis
S I N T E F div of Structural Engineering

Specified capacity

Joint ident.

USFOS load combination no = 1
Load step no = 60
Load level = 462.683

NODE ID	Capacity rule	Chord diameter	Chord thickness	Chord yield str.
7	MSL mean	1.680E-01	4.500E-03	2.780E

ID	(deg)	Type	brace	Cap/Qf	Cap/Qf	Cap/Qf
4	60	97% K 3% Y 100% =>	5	.016	4.122E+05 3.877E+05 4.114E+05 .93 3.747E+05 1.00	2.584E+04 2.584E+04 2.584E+04 .85 2.584E+04 1.00
5	60	K	4	.016		

100% Y capacity

Combined, 97%K + 3%Y capacity

Q_r factors

Table 5.3-2 Print from the MSL routines on the <res>.out file.

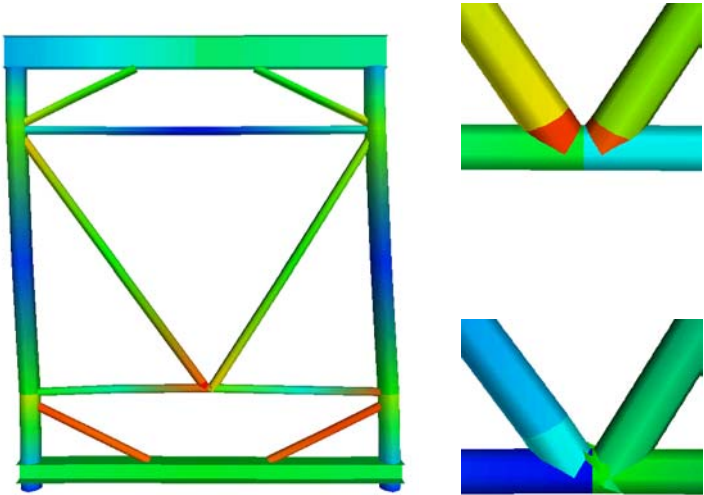


Figure 5.3-1 2D K-frame

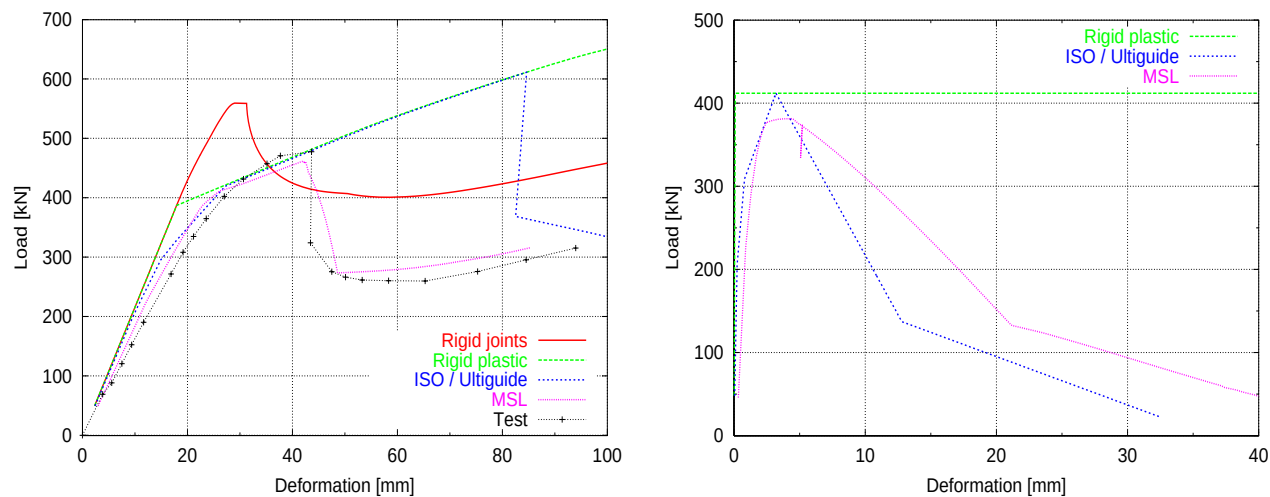


Figure 5.3-2 2D K-frame *Load – deformation curves*

6 Foundation Modelling

6.1 Nonlinear spring

A general nonlinear spring element is available in USFOS. The spring has 6 uncoupled degrees of freedom. The behaviour of each degree of freedom is defined by discrete $P - \delta$ points, see Figure 6.1-1. Hyperelastic material behaviour (loading and unloading follows the same curve), and an elasto-plastic material behaviour with kinematic hardening are available.

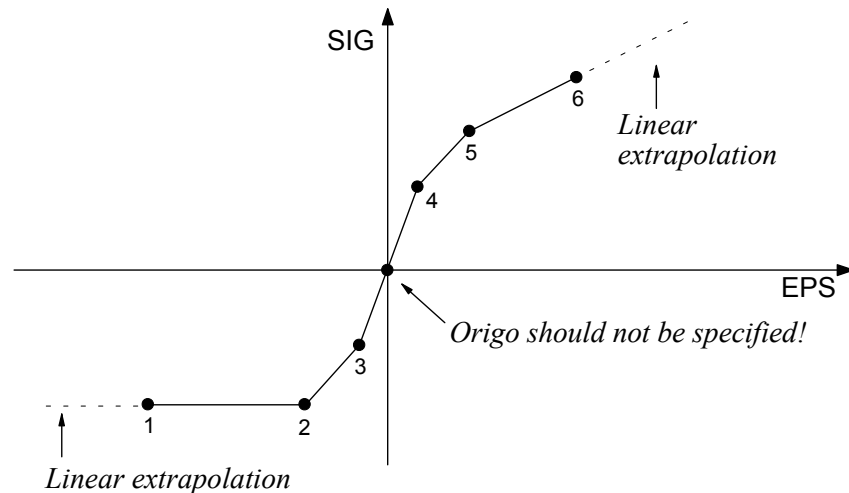


Figure 6.1-1 Definition of spring properties by discrete points

The curve should be straight through origo, i.e. do not break the curve at origo.

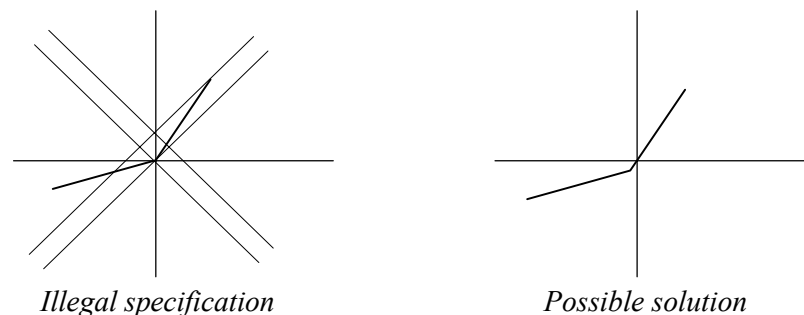


Figure 6.1-2 Example of legal and illegal spring definition

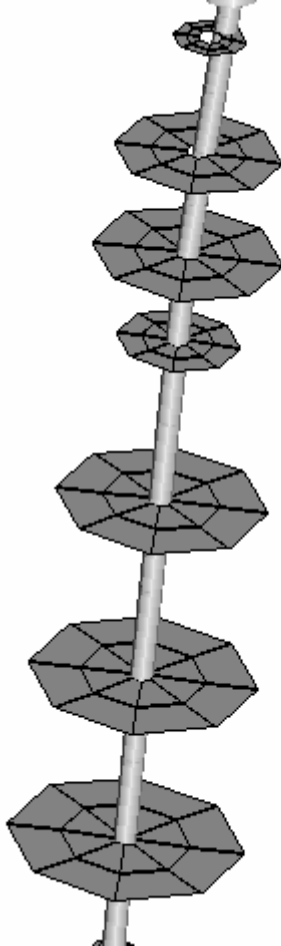
Both 1 node (spring to ground) and 2 node spring elements are available. The input accounts for the lack of nonlinear preprocessors and therefore the following data handling are performed:

If the linear spring to ground (SESAM element no 18), refers to a nonlinear spring definition ("MREF"), the element will be handled by USFOS as a 1 node nonlinear spring to ground.

If the 2 noded beam element (SESAM no 15) refers to the nonlinear spring definition (MREF), the element will be handled as a 2 node nonlinear spring.

6.2 Pile/soil interaction

In the analysis of fixed offshore structures a proper modelling of the interaction between soil and structure, both the static and dynamic case, is of major importance. The purpose of this activity was to improve the models for soil-structure interaction. To simplify the user input of the pile geometry and soil properties, a specific pre-processing function is implemented in USFOS. This saves the user from defining the detailed geometry of the pile-spring model.



The soil-pile interface material behaviour, i.e. the spring characteristics is implemented according to a general plasticity formulation. The model is 3-dimensional in the sense that both lateral and axial springs are applied to each node.

The user defines the soil characteristics for each soil layer by P-Y, T-Z and Q-Z curves. With this information, combined with the user's definition of the pile location, type (single or group) and dimensions (diameter and thickness), USFOS generates finite elements (beam and spring). The size of this foundation model varies from approx. 100 - 1000 elements depending on number of piles and number of soil layers.

In XFOS the pile is visualized with discs representing the soil behaviour at the different levels, and the size of the discs reflects the relative strength of the soil. Both soil deformations and utilization are visualized.

6.2.1 Pile – Soil / Automatic generation of piles and soil capacity

The automatic generation of piles and corresponding soil capacities is a powerful option, which requires a few input lines only. The user's structure ends at "mud line", and all elements below mud line are generated automatically by USFOS, see Figure 6.2-1. In Table 6.2-1 overleaf, the necessary commands used to produce the foundation model shown in the figure are given.

See also the example in folder PSI_2.

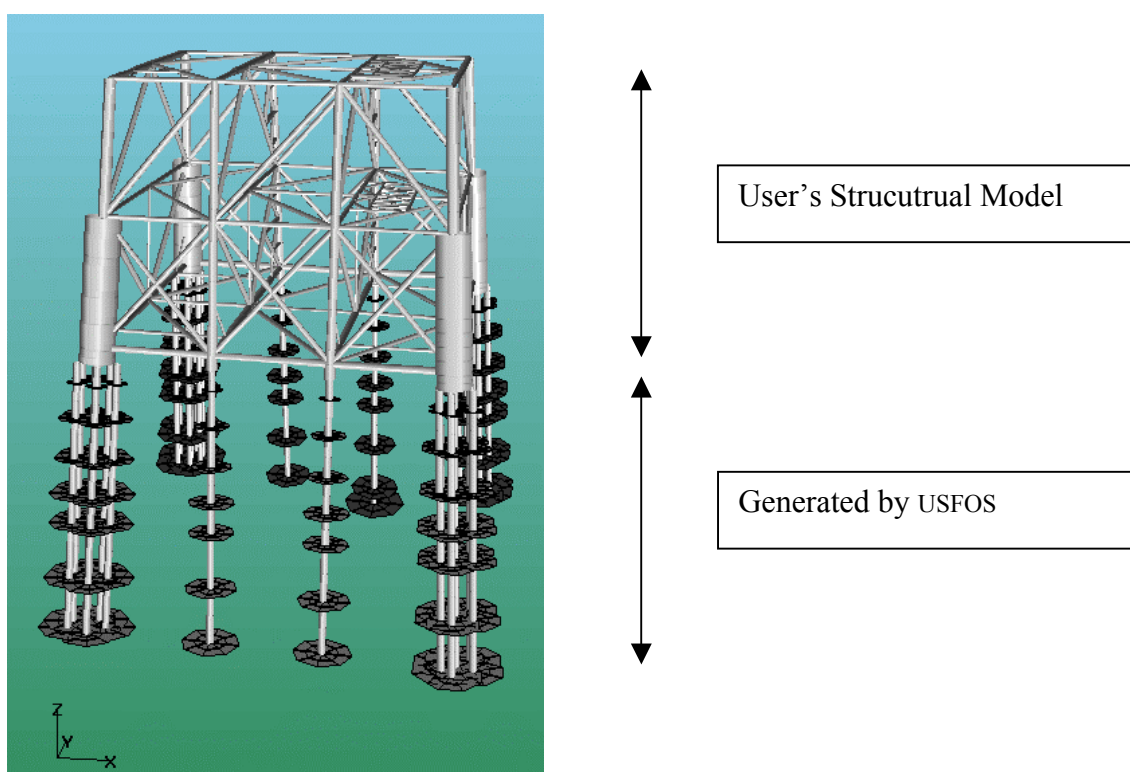


Figure 6.2-1 Automatic generation of piles and soil capacity

Comments to the input in Table 6.2-1:

- ❑ The foundation consists of 4 pile clusters, each with 7 piles, and 4 single piles.
- ❑ This foundation is defined as 8 **PILE** elements, which refer to one of the two **PILEGEO** records.
- ❑ **PILEGEO** number 1 consists of 7 pipes with diameter 1.22m. The individual positions are specified through local Y- and Z-co ordinates referring to the **PILE** local axis.
- ❑ The **PILE** local x-axis goes (downwards) from the pile head towards the pile tip.
- ❑ **PILEGEO** number 2 is a single pile, here defined as a group with only one pipe in the centre of the pile element axis. (The single pile option could also been used, see UM Ch 6).
- ❑ For all the 8 piles, the same soil exists (refer all to the same **SOILCHAR** record)
- ❑ The **SOILCHAR** is specified with 3 clay layers and 3 sand layers. However, in order to obtain a reasonable element density in the rather thick sand layer no. 2 (-24.1 to -48.8m), the same soil

property (no. 501) is referred to three times. (The soil spring is inserted in the middle of the layers defined under SOILCHAR.

- The soil strength is calculated according to API 1993 by specifying the geotechnical data in the command **API_SOIL**.

'	Elem	ID	np1	np2	Soil	ID	Pile_mat	Pile_geo	lcoor	Imper	
PILE	1	1	1001	10	99	1	0				
PILE	2	2	1002	10	99	2	0				
PILE	3	3	1003	10	99	2	0				
PILE	4	4	1004	10	99	1	0				
PILE	5	5	1005	10	99	1	0				
PILE	6	6	1006	10	99	2	0				
PILE	7	7	1007	10	99	2	0				
PILE	8	8	1008	10	99	1	0				
'											
'	ID	Type	Do	T	Npile	Y_loc	Z_loc				
PILEGEO	1	2	1.22	0.05	7	0.0	0.0				
						2.1	2.1				
						3.0	-1.4				
						0.6	-2.5				
						-1.65	-1.65				
						-2.5	0.6				
						-1.5	2.7				
'	ID	Type	Do	T	Npile	Y_loc	Z_loc				
PILEGEO	2	2	1.22	0.05	1	0.0	0.0				
'											
'	ID	Type	Z_Mud	D_ref	Ffac	Lfac	Z1	Z2	API_Soil	ID	
SOILCHAR	10	API	-93.725	1.0	1.0	1.0	-1.0	-5.2	101	! Clay	
							-5.2	-12.5	201	! Clay	
							-12.5	-18.3	301	! Clay	
							-18.3	-24.1	401	! Sand	
							-24.1	-28.3	501	! Sand	
							-28.3	-42.7	501	! Sand	
							-42.7	-48.8	501	! Sand	
							-48.8	-67.0	601	! Sand	
'											
'	ID	Type	load	Gam	Plug	Su	eps50	APIJ	Tresf	QPLim	iDyn
API_SOIL	101	SoftClay	Static	9500	1	50E3	0.013	0.25	0.74	0.2E6	0
API_SOIL	201	StifClay	Static	9500	1	120E3	0.012	0.25	0.72	1.2E6	0
API_SOIL	301	StifClay	Static	9500	1	150E3	0.010	0.25	0.73	1.0E6	0
API_SOIL	401	StifClay	Static	9500	1	190E3	0.019	0.25	0.75	1.9E6	0
'											
'	ID	typ	load	Gam	Plug	Phi	Delta	rNq		QPLim	iDyn
API_SOIL	501	Sand	Static	8000	0	33	22	22		1.4E7	0

Table 6.2-1 Input for automatic calculation of piles and soil capacities

6.3 SpudCan Element

A Non-linear SpudCan element specially designed for Jack-Up Structures has been available for some time, but until recently only sand models were implemented.

The implementation was revised in 2001, and the current implementation (available in usfos version 7.8) uses capacity- and interaction formulas from the SNAME RP /1/.

- Both sand and clay models are now implemented.
- Elastic stiffness is taken from SNAME RP, but with embedment corrections as described in the commentary to SNAME RP /1/.
- The nonlinear rotational spring stiffness correction is based on plastic rotations as described by Van Langen /2/, as the secant stiffness correction suggested by SNAME RP is not suitable for USFOS (we use a tangent stiffness approach)

A new input card, **SPUDMAT**, has been introduced. This can be used to give input both for sand and clay. The old input card **MSPUD** is now obsolete, but will still be available for compatibility with old models.

Also new in 2001 is visualization of the spudcan element in XFOS.

The element needs equilibrium iterations, Spudcan elements should not be used without equilibrium iterations switched on.

```

-----
Define Spud model:  3 Spring 2 ground elements
-----

- Elements:

Elem ID      Node ID      Material ID
SPRNG2GR      1          359          2
SPRNG2GR      2          380          2
SPRNG2GR      3          405          2

-----

Spudcan material parameters
-----

Sample Clay parameters :

Mat_No | Typ | V_pre | R_eff | D_emb | As_lat
SPUDMAT  2 | Clay | 48.9E+06 | 7.0 | 12.2 | 102.6

| Gv | Gh | Gr | Pois. | cuo
5.39E+06 | 5.39E+06 | 5.39E+06 | 0.5 | 26.98E+03

| adh_fac | Suction_flag | backfill_flag | [ C8 | C1 | C7 ]
0.9 | 0 | 0 | 1.0 | 0.5 | 1.0

```

Table 6.3-1 Input for SpudCan elements with *clay* material parameters

```

-----
' Define Spud model:  3 Spring 2 ground elements
-----
'                                     - Elements:
'
'      Elem ID      Node ID      Material ID
SPRNG2GR      1      359      2
SPRNG2GR      2      380      2
SPRNG2GR      3      405      2
'
' -----
' Spudcan material parameters
' -----
' Sample Sand parameters :
'
'      Mat_No| Typ | Vpre | R_tot | Apex_ang | Gv | Gh | Gr | Pois.
SPUDMAT      2   Sand 10.0E7 10.00  85.0   50e06 50e06 50e06 0.25
'
'      Fr_ang | Cohe. | E_U_W | [ C8 | C1 | C7 ]
'      35.0   0.0   0.01E+06 1.0   0.3   1.4
'

```

Table 6.3-2 Input for SpudCan elements with *sand* material parmeters

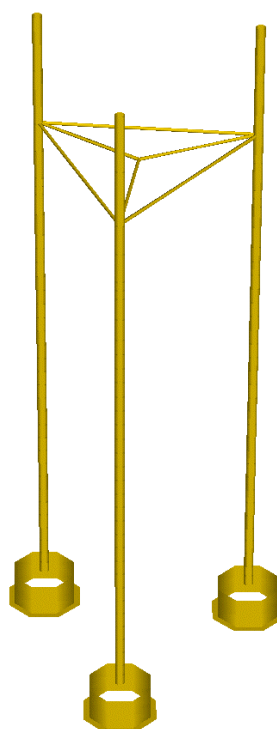


Figure 6.3-1 Simplified Jackup structure with spudcan-elements.

References:

- /1/ SNAME (1997): Recommended practice for site specific Assessment of mobile Jack-Up units, Rev 1. The society of naval architects and marine engineers, Jersey City Nj
- /2/ Langen H van, Hospers B (1993) "OTC 7302 Theoretical model for determining rotational behaviour of spud cans" *Proc. Offshore Technology Conference*, Houston, Texas, May 1993. Dynamic Analysis

7 Dynamic analyses

7.1 Introduction

Dynamic analysis can be performed for given load-time histories and for ship collision. In the latter case, the impact velocity of the ship mass is treated as the initial condition for a free vibration problem.

Two options exist for the mass of the structural element;

- **Consistent mass**, based on interpolation functions for the linear 3D beam. Thus, it is not truly consistent with the displacement shape function used in USFOS, but accurate enough for most practical purposes.
- **Lumped mass**, yielding a diagonal mass matrix. In this case the rotational masses are scaled by a factor denoted "rotmass". The scale factor should be fairly low in order to maintain accuracy for high frequency modes. By default this is set equal to 0.01.

Linear damping may be given in the form of **Rayleigh damping** with one term proportional to the system mass and one term proportional to the system stiffness. Generally, the mass-proportional term damps the lower modes of vibration and the stiffness-proportional term damps the higher modes of vibration. The two proportionality constants can be calibrated such that a desired damping level may be obtained at two frequencies. It should be born in mind, however, that the Rayleigh damping terms will often be of minor importance because since the effective damping will be predominated by hysteretic material behaviour in plastic hinges.

The **numerical integration** scheme is based upon the **HHT- α method**, which condenses to the **Newmark- β** method for $\alpha=0$. The property of the α -parameter is to introduce artificial damping of the higher order vibration modes, which is beneficial for the accuracy of the solution.

In order to obtain **numerical stability** during integration the step length has to be adjusted such that it is less than a prescribed fraction of the fundamental eigenperiod of the system. For a system with a large number of dofs, the highest natural period may become very small. This restriction requires many more time steps than needed for accuracy, especially when low mode response is governing. Hence, it is recommended to use an unconditionally stable algorithm. For The HHT- α **unconditional stability** is obtained when the following conditions are met:

$$-\frac{1}{3} < \alpha < 0$$

$$\gamma = \frac{1}{2}(1 - 2\alpha)$$

$$\beta = \frac{1}{4}(1 - \alpha)^2$$

where β and γ are the free parameters in the Newmarck- β method. Generally $\alpha=-0.3$ is recommended when the HHT- α method is used.

The integration may be performed with normal **direct integration** or with the **predictor-corrector approach**. In the latter case the displacement and velocity at the next step are first predicted on the basis of the known displacements, velocities and accelerations at the present step, assuming implicitly that the acceleration at the next step is equal to zero. This is performed without any need for solving system equations. Then, the accelerations at the next step is solved iteratively by means of the dynamic equilibrium equation, and the predictor velocities and - displacements are updated accordingly.

The predictor corrector approach is convenient because a scaling of the step length may be carried out in the predictor phase. At least one equilibrium iteration has to be carried out in order to determine the acceleration at the next step.

With the direct integration approach a pure incrementation can be carried out. However, no scaling of the time step is performed. With respect to CPU consumption, the direct integration with no iteration and the predictor-corrector method with one iteration should be comparable because both methods employ one solution of system equation.

7.2 Input

According to the dynamic input, all load control is controlled by a parameter **time**, and the loads to be applied at the different times are specified using **time histories**.

A time history is a scaling-factor/time curve as shown in Figure 7.2-1.

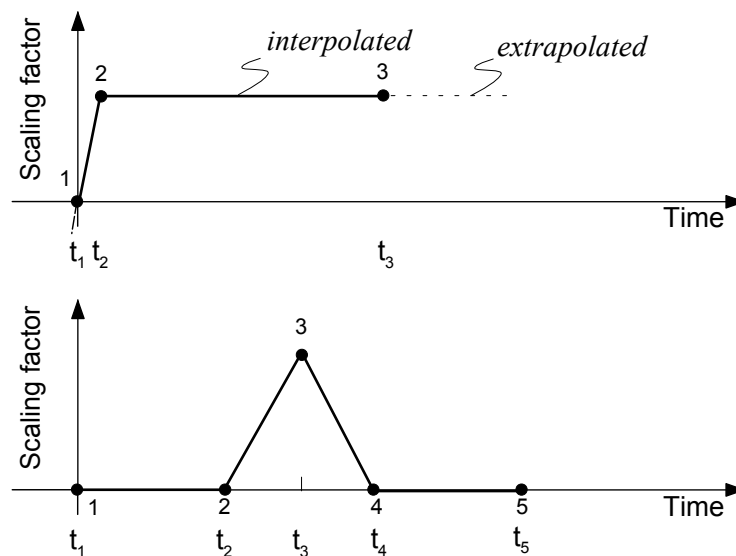


Figure 7.2-1 Time history examples

The upper time history is a typical 'apply dead loads' history. The loads connected to this time history are scaled up to the actual level at time t_2 and then be kept constant the rest of the analysis. For times greater than t_3 , the extrapolated line through the two last points is used.

The lower time history example may be an 'apply impact load' history. The loads connected to this time history is 'sleeping' up to time t_2 where the loads are scaled to the actual level at time t_3 . Then the load is reduced, (causes negative load increments internally in USFOS), until time t_4 is reached from where the load level is kept equal to zero.

A **loadvector** combined with a **time history** is called a **load history**, and an 'unlimited' number of load histories may be defined. A loadvector may be combined with several time histories, and a time history may be combined with several load vectors.

The records used to define the analysis are:

- DYNAMIC** : Defines Δt (time increment) to be used within a time interval defined by the time terminating the interval, see Figure 7.2-2.
- TIMEHIST** : Defines a time history identified by an ID and described by discrete points.
- LOADHIST** : Defines a load history by connecting a loadvector to a time history.

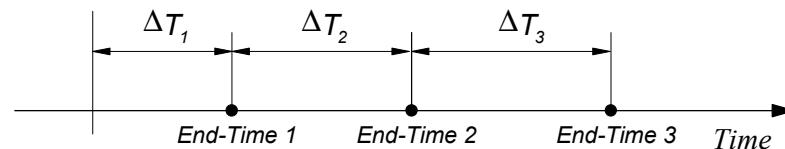


Figure 7.2-2 Specification of time increment to be used with the different time interval

7.3 Dynamic Analysis results. Time Series

A dynamic analysis may involve a large number of analysis steps (1000 - 100 000), and saving of analysis results is then a challenge. It is then necessary to select a few results, which could be saved every analysis step, while the rest of the results could be saved more seldom. In this way, the user obtain following:

- ❑ High density on the time series of the selected (most important) results
- ❑ Acceptable density on the results presented in XFOS for inspection of the global behaviour of the structure (f ex generation of animation etc).

The few, selected result quantities are stored in a separate file with extension *.dyn* in addition to the usual *.raf* file. The dynamic results are accessed from XFOS through the result/dynamic_result dialogue box, see Figure 7.3-1.

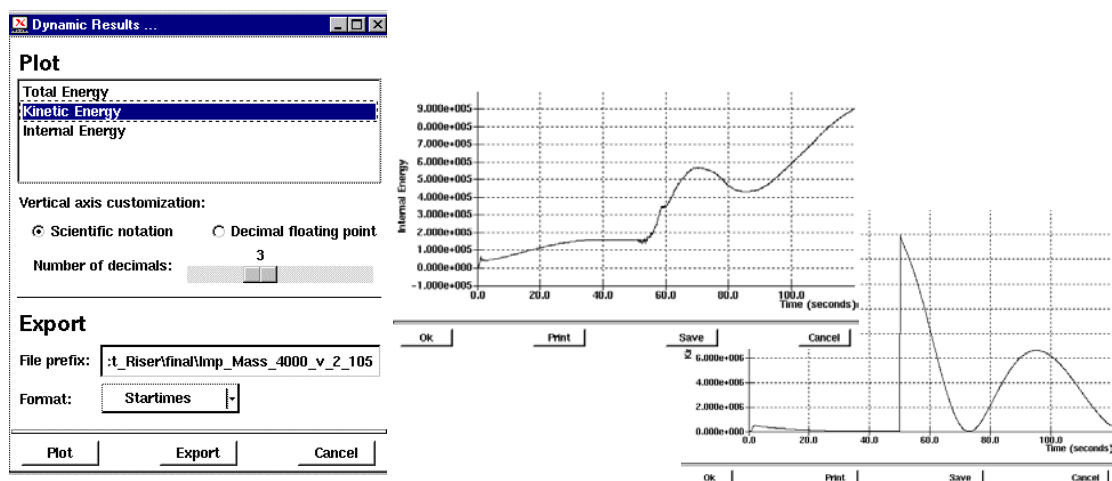


Figure 7.3-1 Selecting Dynamic Results from XFOS

Following results are

- ❑ NODAL
 - Displacement
 - Velocity
 - Acceleration
 - Relative displacement (between two nodes)
- ❑ ELEMENT
 - Displacement
 - Force
- ❑ GENERAL
 - Internal Energy
 - Plastic Energy
 - Kinetic Energy
 - Total Energy

See Table 7.3-1 for example of use:

'	Type	Node_ID	Dof		
DYNRES_Node	Dis	10	1		
DYNRES_Node	Dis	130	1		
DYNRES_Node	Vel	130	1		
DYNRES_Node	Acc	130	1		
'					
'	Type	Node_ID	Dof	Node_ID	Dof
DYNRES_Node	RelDis	10	1	130	1
'					
'	Type	Elem_ID	End	Dof	
DYNRES_Elem	Disp	20	2	1	
DYNRES_Elem	Force	20	1	1	
	Type				
DYNRES_General	Wint				
DYNRES_General	Wplast				
DYNRES_General	Wkin				
DYNRES_General	Wtot				

Table 7.3-1 Input for "Dynamic result" saving

See also in the example folders:

- ❑ dyn_drop
- ❑ dyn_imp
- ❑ dyn_imp2
- ❑ dyn_quak

8 Efficient use of USFOS

8.1 General

Seldom, only one USFOS analysis is performed for a given problem. The more typical use is repeated runs due to several load cases, parametric (sensitivity) study, model change, etc.

In cases where many USFOS analyses should be performed, well organising of both input and output files is important. There should be no doubt about “what was the parameters used for this particular result plot” and so on. It is highly recommended to **not** use one input file set, which is modified over and over again until all cases are run, because:

- ❑ Possible confusion about input parameters used
- ❑ Difficult to repeat the analyses after a time
- ❑ Requires manual editing before each new run, impossible to automate

It's better to plan and organise the USFOS analysis in a way that makes it possible to, ultimately, perform hundreds of analyses using only one, (magic) command. One solution (among several) is using UNIX scripts, and the following sections will describe this solution.

USFOS (even on Windows NT) runs in a UNIX environment, and all procedures described in the sections below are running on “all” computer platforms. However, some differences may occur, (f. ex: *C:/TEMP* on PC and */tmp* on standard UNIX).

The next sections will deal with use of UNIX commands typed in from the keyboard in the “old fashion way”. It's therefor worth spending some minutes adjusting the UNIX command prompt window.

8.1.1 Adjusting the UNIX korn shell window (Windows NT/2000 Installations Only)

Before you start using the UNIX korn shell, it's recommended to modify slightly the layout. Figure 8.1-1 shows the default window with white text an black background and with size 24 lines / 80 columns). To modify the window, point on the (blue) top frame of the window, and press the right hand button. The menu Figure 8.1-2 appears.

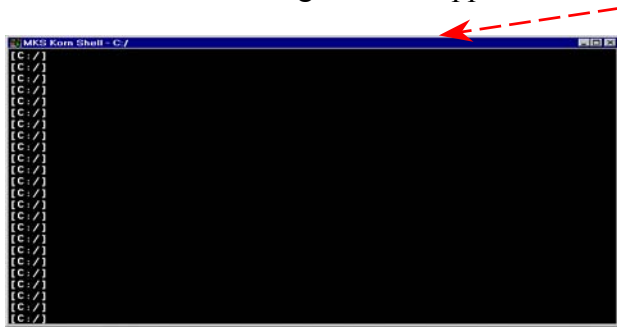


Figure 8.1-1 The default NutCracker Window layout

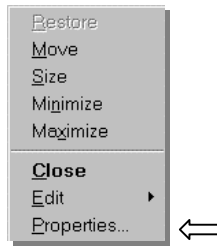


Figure 8.1-2 Menu

Select *Properties* and the “*select colors*” menu shown in Figure 8.1-3 appears.

Select *screen text* and *screen background* among the indicated colours. The light grey background together with black text is a good combination.

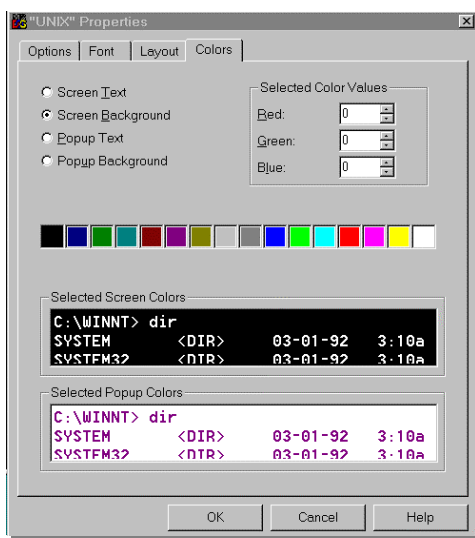


Figure 8.1-3 Defining screen- and text colour

The default window has no screen buffer (has no scroll bar), but the buffer sizes in vertical (number of lines) and horizontal (number of columns) are possible to specify under the *layout* menu, see

Figure 8.1-4. Type in (or use the arrow) the actual sizes, which here is set to 132/2048. The window size when it pops up is set to 80/40.

When the OK button is pressed, the menu shown in Figure 8.1-5 appears. Select “*Modify Shortcut*” to save the settings permanently.

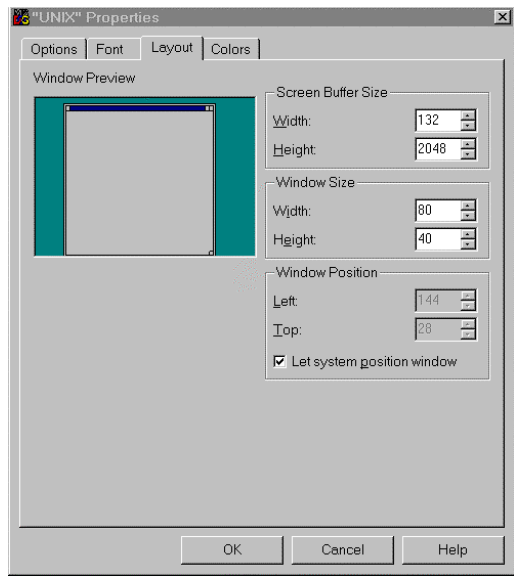


Figure 8.1-4 Defining window layout

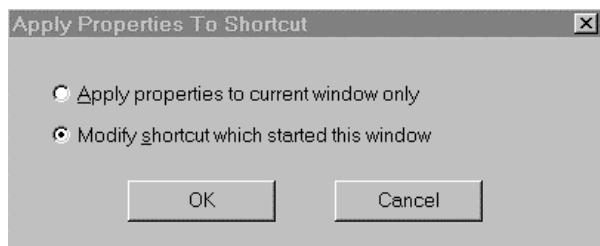


Figure 8.1-5 Selecting permanent modification of the short cut

The UNIX window will from now on look like the one in Figure 8.1-6 with two scroll bars (and it's resizable) and a comfortable colour.

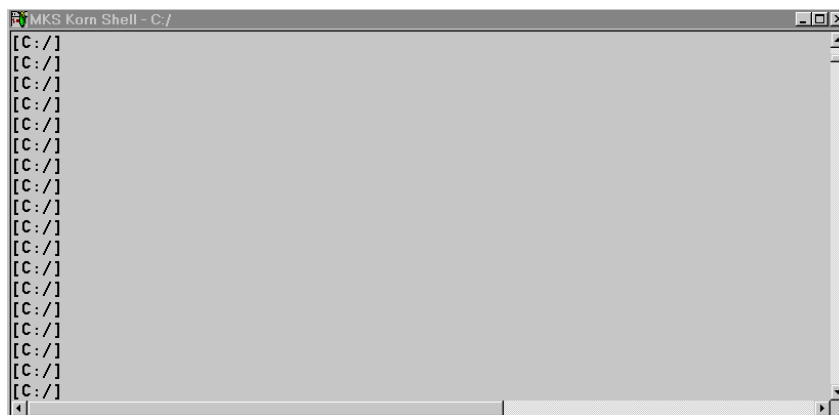


Figure 8.1-6 The modified NutC window with scroll bar.

8.2 Some UNIX commands

The procedures described in the examples below require that the users knows some UNIX commands, and in the following a brief summary of the commands used in the scripts is given:

Command / Argument	Description	Use
<i>cp</i>	<i>Copy one file into another</i>	<i>cp "from file" "to file"</i>
<i>mv</i>	<i>Rename a file or directory</i>	<i>mv "from name" "to name"</i>
<i>cat</i>	<i>dump the content of a file to screen</i>	<i>cat "file"</i>
<i>cat ></i>	<i>dump the content of file 1 into file 2</i>	<i>cat "file1" > "file2"</i>
<i>cat >></i>	<i>dump content of file1 behind existing content of file 2 (append)</i>	<i>cat "file1" >> "file2"</i>
<i>mkdir</i>	<i>create a directory (folder)</i>	<i>mkdir "directory name"</i>
<i>cd</i>	<i>change directory</i>	<i>cd "directory name"</i>
<i>..</i>	<i>directory path. (one level up)</i>	<i>cd ..</i>
<i>../..</i>	<i>directory path. (two levels up)</i>	<i>cd ../..</i>
<i>../dir_name</i>	<i>directory path (one level up and one down)</i>	<i>cd ../case2</i> <i>cp "file1" "../case2/file2"</i>
<i>\$NAME</i>	<i>Environmental variable with name NAME</i>	<i>cp \$MASTER/file1 file2</i>
<i>echo \$NAME</i>	<i>"Show me the content of the environmental variable whit name NAME"</i>	<i>Will be used in the examples below</i>
<i>sed</i>	<i>"Stream Editor"</i>	<i>Will be used in the examples below</i>
<i>rm</i>	<i>Delete file(s)</i>	<i>rm file1</i> <i>rm file1 file2 file3</i>
<i>rmdir</i>	<i>Delete directory</i>	<i>rmdir directory_name</i>
<i>ls</i>	<i>List files</i>	<i>ls</i>
<i>ls *.fem</i>	<i>List all files with extension .fem</i>	

Table 8.2-1 UNIX commands overview

8.3 Example 1, Fixed USFOS input file names

The simplest example on a UNIX script (which saves you for tediously typing) is a file with name **go1** containing following:

```
$USFOS_HOME/bin/usfos 15 << ENDIN
head
stru
load
res
ENDIN
```

Table 8.3-1 Content of script file: "go1" with 3 fixed USFOS input files

Explanation:

The variable `USFOS_HOME` is set during installation of USFOS on both UNIX and NT computers. It contains the file path of the root of the actual USFOS version. By prefixing the variable name with `$`, the contents of the variable name becomes available for use in connection with any UNIX command.

“`$USFOS_HOME/bin/usfos`” is the address to the USFOS code, and by adding 15 after the file name, a workspace of 15 mill is required.

The “`<< ENDIN`” defines that the usual screen input/output is given between in the lines between `<< ENDIN`

and

`ENDIN`

The name “`ENDIN`” is an arbitrarily chosen name of the label.

In a usual USFOS run, it's first asked for the control file name prefix, which here is set to “head”. Further it's asked for the structural and load files, which here are “stru” and “load” respectively. Finally, USFOS asks for the result file prefix, which is set to “res”.

By typing **go1** USFOS will start, use the input files head.fem, stru.fem and load.fem, and store the results in files with prefix: *res*. All input files must be located on the same directory as the script file **go1**, and results are stored in the same directory.

As USFOS accepts input from one, two or 3 files, it's possible to leave up to two file names blank as shown in Table 8.3-2, where the ‘load’ file is left out.

```
$USFOS_HOME/bin/usfos 15 << ENDIN
head
stru

res
ENDIN
```

Table 8.3-2 Content of script file: "go2" with only “head” and “stru” input files

It is possible to access files located on other directories than the directory where the script **go** is located (and started from). Table 8.3-3 describes the case where some files are located on different directories:

```
$USFOS_HOME/bin/usfos 15 << ENDIN
head_intact_nw_100yr
../model/intact_stru
../loads/nw_100yr
D:/temp/res_intact_nw_100yr
ENDIN
```

Table 8.3-3 Content of script file: "go3" with input files located on different directories

In this case, the control file (head_intact_nw_100yr.fem) is located on same directory as the script file (and where the script is started from). The structural file (stru.fem) is located in the directory *model* (which is located on same level, besides, the current directory), and the file is named *intact_stru.fem*.

The load file is located on an other directory (also on same level as the other two) with name *loads*, in a file with name *nw_100yr.fem*

The results are saved on the D: disc, on a directory named *temp*, and file *res_nw_100.raf*.

The third variant of the “fixed name script”: **go3** indicates a first try to organise an analysis series involving several versions of the structural file, (f ex *intact* and *damaged*), and several loads (f ex *nw_100yr*, *nw_1000yr*, *sw_100yr*, *sw_1000yr*, etc).

This leads to the next example, which will give an example on how a slight modified **go3** could be used for many different analyses.

8.4 Example 2, Varying USFOS input file names

The “fixed name script”, **go3** described above is slight modified. Instead of defining the file names 100%, some of the file name is substituted by the keywords **\$1** and **\$2**. It's possible to give input parameters to UNIX scrips, and **\$1** is parameter no. 1, **\$2** is parameter no. 2 etc...

```
$USFOS_HOME/bin/usfos 15 << ENDIN
head_$1_$2
../model/$1_stru
../loads/$2
D:/temp/res_$1_$2
ENDIN
```

Table 8.4-1 Content of script file: "go" with varying input file names

By typing:

go intact nw_100yr

the same analysis as described under example 1, **go3** will be performed.

The **\$1** variable will be expanded to **intact** inside the script, and **\$2** will be expanded to **nw_100yr**, which gives the actual file names:

```
Control file : head_intact_nw_100yr
Struct file  : ../model/intact_stru
Load file    : ../loads/nw_100_yr
Result file   : D:/temp/res_intact_nw_100yr
```



A script file may not only refer to UNIX commands, it's possible to refer to other script files as well. This leads to next level in script programming: defining a top level script, which refers to user defined script(s).

If f ex. one analysis series should consist of a number of different structural conditions, different load directions and – conditions, the following script named **run_all** would run through all 16 cases without need for any human interference.

```
# -----
# -- Script for running: - 2 structural conditions, --
# --                      - 4 load directions and   --
# --                      - 2 load conditions        --
# -- Totally 2x4x2=16 cases.                        --
# -----
#
#      Structure      Load
go    intact         nw_100
go    intact         sw_100
go    intact         se_100
go    intact         ne_100
#
go    intact         nw_10000
go    intact         sw_10000
go    intact         se_10000
go    intact         ne_10000
#
go    damaged        nw_100
go    damaged        sw_100
go    damaged        se_100
go    damaged        ne_100
#
go    damaged        nw_10000
go    damaged        sw_10000
go    damaged        se_10000
go    damaged        ne_10000
# ----- End of Script File -----
```

Table 8.4-2 Content of level 2 script file: "run_all", which refers to "go".

8.5 Example 3, Assembling input files before USFOS analysis

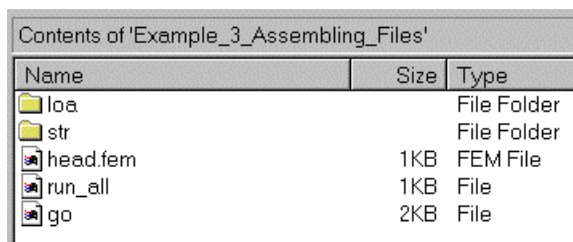
In the previous examples, all input files were complete before the script was executed. In many cases, only a small fraction of the entire input is different from one case to another. Instead of making lots of copies of near 100% equal files, the key in this example is to show how the input files could be composed by common information + some special information.

Common information:

- ❑ Control file, : head.fem
- ❑ Main structure located in : str/Main_Structure
- ❑ Main load located in : loa/Main_Load

Special information:

- ❑ Support Structure : str/Spring_Support_1 and _2
- ❑ Special Load : loa/Node*i*_Load



Name	Size	Type
loa		File Folder
str		File Folder
head.fem	1KB	FEM File
run_all	1KB	File
go	2KB	File

Figure 8.5-1 Content of file folder *before* running script "run_all".

The idea is as follows:

- ❑ Use the control file *head.fem* in all cases.
- ❑ Compose a structural file consisting of the common *Main_Structure* and the special support, and assemble the complete structural model in the file *stru.fem*.
- ❑ Compose a load file, which should consist of the common load file *Main_Load* and the special nodal load, and collect all load info in the file *load.fem*.
- ❑ Create a new, unique directory (below current directory) for each case with informative name reflecting the actual case.
- ❑ Run USFOS and save *stru*- and *load* files + result files on the actual directory.
- ❑ Create script *go* for running on case, and *run_all* for running all 6 combinations

In Table 8.5-1 the script with name *go* is described in detail as it appears in the example folder. Lines starting with the sign # is comment lines, and may appear anywhere in the script file except between << *ENDIN* and *ENDIN*. (It is recommended to use *comments*, both in scripts and in the USFOS input files).

Firstly, the *cp* command is used to copy the main structure to the file *stru.fem*. Next, the selected support structure is appended to the *stru.fem* using the *cat >>* command. Similar is done for the load file assembly.

A unique directory for each case is created using the **mkdir** command, and the directory name (with prefix *Case_*) contains information about both support and load. USFOS is started with 15 mill and results are saved in the actual Case directory using the result file prefix **res** for all cases (the directory contains information about the different cases). Finally, the actual *stru.fem* and *load.fem* are moved into the actual Case directory using the **mv** command. (Note that if only directory name is defined in connection with the **mv** command, the file name will be unchanged in the new directory, just moved.)

```
# =====
# -- Script for assembling USFOS input and run USFOS --
# -- Usage: go par1 par2 --
# -- par1 : Support Structure --
# -- par2 : Load definition --
#-----
#
#                                     - Copy Main Structure into
#                                     file stru.fem and add
#                                     selected support:
cp   str/Main_Structure             stru.fem
cat   str/$1                        >> stru.fem
#
#                                     - Copy Main Load into
#                                     file load.fem and add
#                                     selected load:
cp   loa/Main_Load                  load.fem
cat   loa/$2                        >> load.fem
#
#                                     - Run USFOS and save results
#                                     in unique directories:
#                                     .. Create Directory
mkdir Case_$1_$2
$USFOS_HOME/bin/usfos 15 << ENDIN
head
stru
load
Case_$1_$2/res
ENDIN
#
#                                     .. Move stru.fem and load.fem
#                                     into actual Case_Dir for
#                                     backup purpose.
mv   stru.fem Case_$1_$2
mv   load.fem Case_$1_$2
#
```

Table 8.5-1 Content of script file: "go" which assembles input files & runs USFOS

#	Support	Loa
go	Spring_Support_1	Node1_Load
go	Spring_Support_1	Node3_Load
go	Spring_Support_1	Node5_Load
#		
go	Spring_Support_2	Node1_Load
go	Spring_Support_2	Node3_Load
go	Spring_Support_2	Node5_Load
#		

Table 8.5-2 Content of script file: "run_all", which executes the script "go".

After the script **run_all** is completed, 6 new file folders (directories) are created, see Figure 8.5-2. All directories contain the actual, assembled input (stru and load) + the result files (res.*).

Contents of 'Example_3_Assembling_Files'		
Name	Size	Type
loa		File Folder
str		File Folder
go	2KB	File
head.fem	1KB	FEM File
run_all	1KB	File
Case_Spring_Support_1_Node1_Load		File Folder
Case_Spring_Support_1_Node3_Load		File Folder
Case_Spring_Support_1_Node5_Load		File Folder
Case_Spring_Support_2_Node1_Load		File Folder
Case_Spring_Support_2_Node3_Load		File Folder
Case_Spring_Support_2_Node5_Load		File Folder

Figure 8.5-2 Content of file folder *after* running script "run_all".

8.6 Example 4, Using the SED editor to modify master input files

In the previous example, the input to USFOS was composed by some common files + special files, and in all cases the content of the files were pre defined.

In the current example, another, and even more flexible solution is chosen. Instead of assembling 'pieces' of input, the *content* of the input file(s) are modified prior to the analysis. As the modification should be performed in a batch run, *a batch editor is necessary*. The UNIX shell on both UNIX workstations and the "NutCracker" UNIX shell on Win-NT offers the SED editor, the "Stream Editor".

The operation needed from the stream editor is the "REPLACE" or "SUBSTITUTE" command, where one character string should be replaced by another.

The (cryptic) UNIX command is wrapped into a file, which here is named *substitute*, Table 8.6-1, and which is used as follows:

Substitute "string_1" "string_2" *FileName*

In all connections where *string_1* occur on the specified file, it's replaced by *string_2*. The SED editor is case sensitive (differs between upper and lower case characters). Quotes must be used if blank character(s) occur in the strings.

```
sed "1,$ s/$1/$2/g" $3 > subst_string.temp
mv subst_string.temp $3
```

Table 8.6-1 Script "*substitute*", which utilises the *SED* editor for substituting strings.

With the powerful *substitute* script available, following operations should be done:

- ❑ Create only one master USFOS control file (which should be used for all cases)
- ❑ Use one structural file
- ❑ Run USFOS wave analysis for 8 different wave/current conditions.

As indicated in Figure 8.6-1, some files are present before the analyses are performed, and some are created during the analysis (executing the scripts defined in this section).

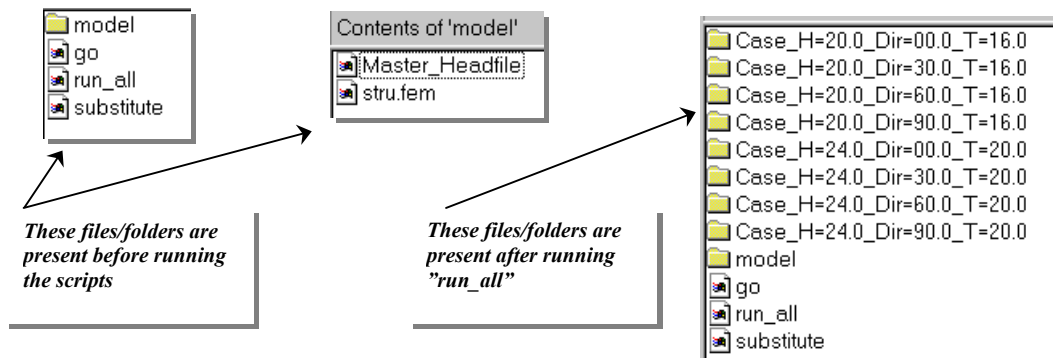


Figure 8.6-1 Files / Folders before and after running the scripts

Master Headfile, Table 8.6-2.

The file is an ordinary control file for USFOS, but some parameters are not yet set. Instead, the parameters are represented by arbitrarily chosen key words. In the actual study, the wave height, direction and period should be varied, and the keyword for the wave height is *WAVEH*, the keyword for direction is *DIRECT*, and the keyword for wave period is *PERIOD*.

Script file “go”, Table 8.6-3 :

The first operation in the script is creating a directory using the *mkdir* command, and all 3 parameters (wave- height, direction and period) are included in the directory name.

Next, the nearly complete USFOS control file (named *Master_Headfile* and located in directory *model*) is copied into the file *head.fem* on current directory. The script for substituting (named *substitute*) is used three times for replacing the keywords with the actual parameter values.

Then USFOS is run, and the same structural file (*stru.fem*) is used for all cases. Results are saved on the actual Case directory, and result prefix is *res*. When USFOS is finished, the (manipulated) *head.fem* is moved into the actual Case directory, (see Table 8.6-5 for example on modified head file).

```

HEAD      USFOS Extreme Wave. Height: WAVEH , Dir: DIRECT , T : PERIOD
          Progressive Collapse Analysis / JACKET model
          SINTEF 2000
,
,
,
,
          - Define Wave:
,
WAVEDATA  Ildcs <type>  H   Period  Direction Phase  Surf_Lev Depth
          2   Stoke WAVEH PERIOD  DIRECT   0.0    0.0    100
,
CURRENT   Ildcs  Speed Direction Surf_Lev Depth  [Profile]
          2      2    DIRECT    0.0    100    0.0  1.0
          -20.0  1.0
          -100.0 0.0

```

Table 8.6-2 “Master_Headfile” with keywords: WAVEH, DIRECT and PERIOD

```

# =====
# -- Script for assembling USFOS input and run USFOS --
# -- Usage: go Wave_Height Direction Period --
#-----
#
# .. Create Directory
mkdir Case_H=$1_Dir=$2_T=$3
#
#
# - Copy Master control file
#   into the current head
#   file:
cp model/Master_Headfile head.fem
#
# .. Substitute the string
#   "WAVEH" with the first
#   script parameter ($1)
substitute WAVEH $1 head.fem
#
# .. Similar for par. 2 & 3:
substitute DIRECT $2 head.fem
substitute PERIOD $3 head.fem
#
# - Run USFOS and save results
#   in unique directories:
$USFOS_HOME/bin/usfos 15 << ENDIN
head
model/stru

Case_H=$1_Dir=$2_T=$3/res
ENDIN
#
# .. Move head.fem
#   into actual Case_Dir for
#   backup purpose.
mv head.fem Case_H=$1_Dir=$2_T=$3
#

```

Table 8.6-3 Script file “go”

Script file “run_all” Table 8.6-4

The script file *run_all* starts *go* 8 times with different input parameters.

```
# =====
# -- Script for running 8 diffenent USFOS cases --
#-----
#
#
#      Wave Height   Wave/Curr Direction   Period
go      20.0         00.0                 16.0
go      20.0         30.0                 16.0
go      20.0         60.0                 16.0
go      20.0         90.0                 16.0
#
go      24.0         00.0                 20.0
go      24.0         30.0                 20.0
go      24.0         60.0                 20.0
go      24.0         90.0                 20.0
#
# ----- End of Script run_all -----
```

Table 8.6-4 Script file “run_all”

```
HEAD      USFOS Extreme Wave. Height: 20.0 , Dir: 00.0 , T : 16.0
          Progressive Collapse Analysis / JACKET model
          SINTEF 2000
,
,
,
,
,
          - Define Wave:
,
          Ildcs <type>   H   Period   Direction Phase   Surf_Lev Depth
WAVEDATA    2   Stoke  20.0  16.0    00.0    0.0    0.0    100
,
,
          Ildcs   Speed Direction Surf_Lev Depth   [Profile]
CURRENT      2     2     00.0     0.0    100      0.0  1.0
                                           -20.0  1.0
                                           -100.0  0.0
                                           -110.0  0.0
```

Table 8.6-5 USFOS control file modified by the SED editor.

After all 8 cases are run, 8 new directories are created (see Figure 8.6-1) containing the modified *head.fem* and the analysis results. Figure 8.6-2 shows results from one of the 8 analyses, and NOTE that the member imperfections (command CINIDEF) are applied automatically according to the actual wave load direction (which here is 30°).

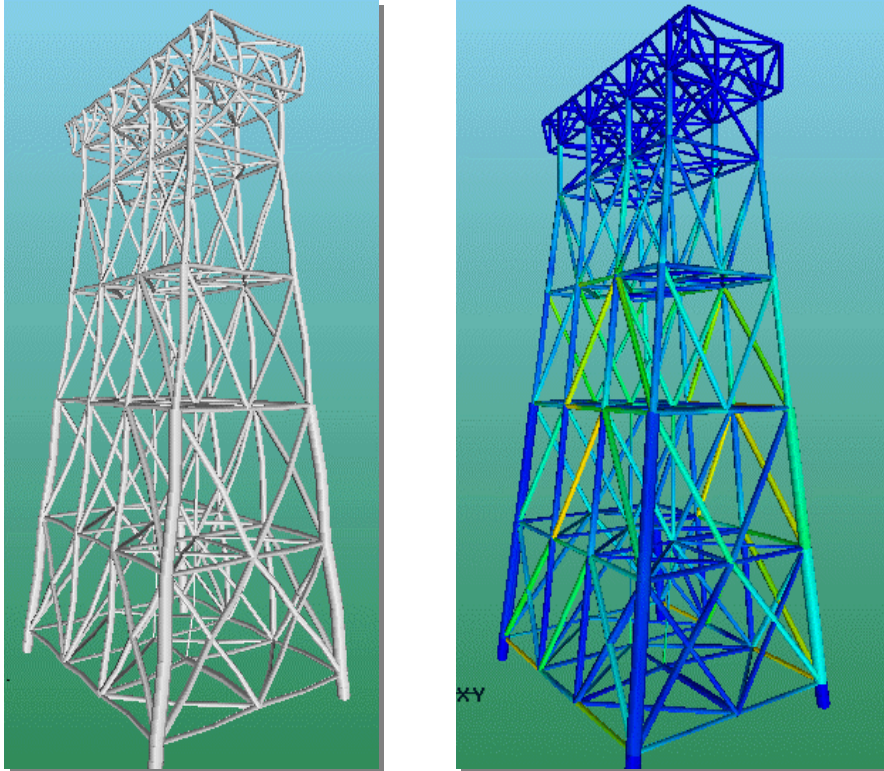


Figure 8.6-2 Case with H=20m, Dir=30deg and T=16s

8.7 Example 5, Procedure for element removal (redundancy analysis)

The final example solves following problem:

- ❑ Remove the structural members, one by one
- ❑ Use the same structural file and control file
- ❑ Save the results from the analyses in separate file folders

Figure 8.7-1 shows the content of the example folder before and after running the actual scripts. The scripts are organised in the *etc* folder, while the structural model is stored in the *model* folder. The content of the script files are described in Table 8.7-1, Table 8.7-2 and Table 8.7-4.

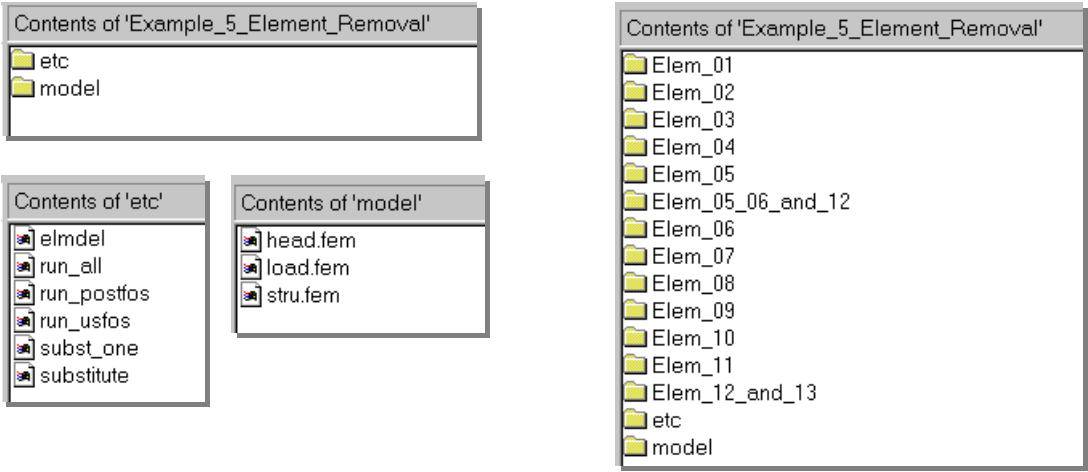


Figure 8.7-1 Files / Folders before and after running the script

```
#                                     - Define variable SCRATCH
#                                     (directory for Raf file storing)
export  SCRATCH=/tmp/scratch
#
#      Local Dir      Element to remove
elmdel  Elem_01        01
elmdel  Elem_02        02
elmdel  Elem_03        03
elmdel  Elem_04        04
elmdel  Elem_05        05
elmdel  Elem_06        06
elmdel  Elem_07        07
elmdel  Elem_08        08
elmdel  Elem_09        09
elmdel  Elem_10        10
elmdel  Elem_11        11
elmdel  Elem_12_and_13  12  13
elmdel  Elem_05_06_and_12  5  6  12
#
# ----- End of Run_All -----
```

Table 8.7-1 Script file “run_all”

The *run_postfos* script runs POSTFOS and creates the default history table, using the *define-history* and *print-history* commands. (Similar scripts could be created for extracting nodal displacements of selected nodes, element forces etc.)

```
$USFOS_HOME/bin/usfos << ENDIN
head
stru
load
$SCRATCH/res
ENDIN
```

```
$USFOS_HOME/bin/postfos << ENDIN

$1

define-hist,,,,,
print-hist,,,,,

q
ENDIN
```

Table 8.7-2 Scripts: “run_usfos” and “run_postfos”

Figure 8.7-2 shows the content of one automatically created file folder (named Elem_01), which contains the global history created by POSTFOS , the log files from the analysis and the different input and output files. Table 8.7-3 shows the content of the file *nonstru_elem.fem*, (which is created by the script), for two cases: To the left the case where element number 1 should become non structural, and to the right the case where elements 5,6 and 12 should be removed.

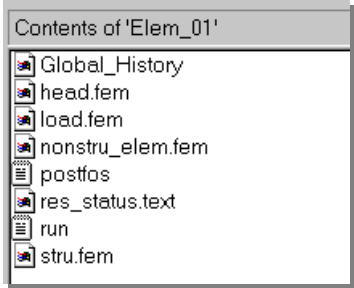


Figure 8.7-2 Files created automatically in folder *Elem_01*

```
'
'
' -----
' -- Nonstructural Members --
' -----
'
'              Type
NONSTRU  Element    01
'
' ----- E O F -----
'
```

```
'
' -----
'
```

Table 8.7-3 Automatically created files containing the NONSTRU comand.

```
#####
#   Author : Tore Holmas, SINTEF Group. Norway   #
#   Date   : 2000-03-18                         #
#####
#
if
  test "$#" -lt "2"
then
  echo '          *****'
  echo '          *                               *'
  echo '          *   Creates the directory "../Label"   *'
  echo '          *   creates a copy of usfos control file and *'
  echo '          *   adds necessary NONSTRU commands.      *'
  echo '          *                               *'
  echo '          *   Assumes structural file on ../model/stru.fem *'
  echo '          *                               *'
  echo '          *   Results are stored on file "$SCRATCH/res" *'
  echo '          *                               *'
  echo '          *   Usage: elmdel <Label> elem1 elem2 elem3 .. *'
  echo '          *                               *'
  echo '          *                               2re, March 2000 *'
  echo '          *****'
else
  echo " "
  echo "  Creates directory ../$1 "
  mkdir ../$1
  cd ../$1
  count="1"
  for i do
    if (test "$count" -gt "1")
    then
      echo "      Processing Element      : $i "
      if (test "$count" -eq "2")
      then
#          - Heading :
          echo "" > nonstru_elem.fem
          echo "" >> nonstru_elem.fem
          echo "" -- Nonstructural Members -- " >> nonstru_elem.fem
          echo "" >> nonstru_elem.fem
          echo "" >> nonstru_elem.fem
          echo "" Type " >> nonstru_elem.fem
          fi
#          - Add to file :
          echo " NONSTRU Element $i " >> nonstru_elem.fem
          if (test "$count" -eq "$#")
          then
#          - Tail:
          echo "" >> nonstru_elem.fem
          echo "" ----- E O F ----- " >> nonstru_elem.fem
          fi
#          - Update counter:
          count=`expr $count + 1`
        done
# =====
        echo " Grabbing USFOS master control file from ../model "
        cp ../model/head.fem .
        echo " Adds nonstru commands ....."
        cat nonstru_elem.fem >> head.fem
        echo " Creates Case identifier : $1 on head.fem "
        ../etc/substitute CASEID $1 head.fem
        echo " Grabbing USFOS stru & load file from ../model "
        cp ../model/stru.fem .
        cp ../model/load.fem .
        echo " and start USFOS "
        ../etc/run_usfos > run.log
        echo " and POSTFOS "
        ../etc/run_postfos $SCRATCH/res >> run.log
        echo " "
        echo " Saves Global History on current directory ....."
        echo " "
        echo " "
        echo " "
        mv $SCRATCH/res.pri Global_History
        mv $SCRATCH/res_status.text .
      fi
    fi
  done
fi
```

Table 8.7-4 Script file "elmdel"

9 Workshops

9.1 Workshop I, USFOS Element formulation, Analysis Control

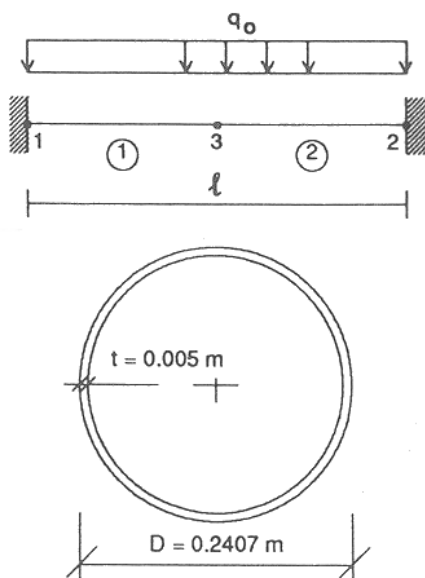
The purpose of this workshop is to get familiar with the USFOS program system, and to demonstrate the basic features of the program. Three simple models are presented, where the results can be verified by hand calculations or engineering judgment.

Theme	Relevant input
Non-linear Material	MISOIEP [MPLASMON, MPLASCYC] GBOUND SURF2OFF
Load Control	CNODES CCOMB CUSFOS CICYFOS
Analysis-solution-control	DETEROFF CITER
Imperfections	CINIDEF GIMPER GELIMP

9.1.1 Workshop I a: Elasto-Plastic Beam Bending

9.1.1.1 Objective

The purpose of this case is to demonstrate the membrane action of the USFOS beam element, and to investigate the different material models in USFOS. A tubular beam is loaded by transverse forces.



Beam length	L	10.0 m
Tube diameter	D	0.2407 m
Tube thickness	t	0.005 m
Yield stress	σ_y	330 MPa
Young's modulus	E	2.1×10^5 MPa
Reference load	q_0	10.0 kN/m
Section area	A	3.702×10^{-3} m ²
Moment of inertia	I_y	2.572×10^{-5} m ⁴
Moment of resistance	W_p	2.778×10^{-4} m ³
Axial yield load	N_p	1221.6 kN
Plastic moment cap.	M_p	91.67 kNm

Figure 9.1-1 Beam Model

9.1.1.2 Description

Plastic hinges form when the bending capacity of the critical sections is exhausted. When hinges are formed at both ends and at midspan, the beam forms a kinematic mechanism. From this stage, further loading is carried by axial tension forces as the beam deforms. When the mechanism is formed, the beam first deforms in a V-shape, with rotations concentrated to the plastic hinges and each half-beam elastic. For a further increase in loading, the beam deforms into a chain-link. The beam yields at the quarter-lengths, and the rotation angle at beam midspan is reduced (the midspan area is straightened out). The yield hinges at beam midspan unloads. A further loading will be carried almost entirely by axial straining, and the beam will enter a state of pure membrane action.

9.1.2 Program input

Control file head_orig.fem
 Structure file model_orig.fem
 Load file none
 Load case 1 Transverse load q_0

9.1.2.1 Analyses

1	Perform a nonlinear analysis of the beam. By default the Two-Surface plasticity model is used. 1. Familiarise with plotting of first node numbers and element numbers using Verify/Node numbers and Verify/element numbers 2. Activate the plastic utilisation fringing: Results/GLOBAL/ELEMENT/Plastic utilisation. Step up along Global load/global displacement curve and see how the fringe colour changes to red as the cross-section becomes plastic 3. Establish bending moment-axial force interaction for the middle cross-section (element 2 node 1) and end cross-section (element 1 node 1) by means of Results/Material plot/ - Horizontal axis: Z moment, Vertical axis: X force The diagram shows the bending moment-axial force history plotted in the contour of the yield - and fully plastic surface. Experience how the yield surface is reached first at element end, then at mid section, and see how the yield surface is “pulled” along the force point and becomes tangent to the fully plastic surface. Identify the initiation of first hinge in bending and the transition to fully plastic membrane tension. Why is the resistance virtually linear at this stage?	
2	Repeat analysis 1, but with the One-Surface plasticity model Compare results with base case; <i>Global load-global displacement</i> and <i>bending moment-axial force interaction</i>	use the record <i>SURF2OFF</i>
3	Analyse the beam under pure (nonlinear) elastic bending • Suppress formation of plastic hinges • OR specify very high yield stress (Recommended) • Compare results with the elasto-plastic case	use the <i>CELHINX</i> record use the <i>MISOIEP</i> record
4	Simulate pure plastic behaviour of the beam • Specify very high Youngs modulus, eg. increase by a factor of 10. • In order to avoid excessive calculation steps after creation of mechanism, it is recommended to increase maximum displacement in the low stiffness • How is the response now compared to the base case? Is it possible to compare with hand calculations?	use the <i>MISOIEP</i> record use CUSFOS/mxpdis =10. (control file)

9.1.2.2 Documentation

1. Take a hardcopy the P- δ plot displayed by XFOS ('global load - global deformation').
2. Use XFOS to generate plot of axial force vs. bending moment
3. Generate plots of axial force vs. global displacement and of bending moment vs. global displacement (for the one-surface plasticity model)

Plots can be pasted directly to Word-documents using the **Copy** option appearing on mouse right-clicking on the diagram, or plot arrays may be pasted to Excel worksheet by means of the **Copy as text** option, so as to enable plotting in a format at your own liking.

9.1.2.3 Comments to the analyses

USFOS include non-linear effects both due to material non-linearity (yielding) and due to geometric non-linearity (change of global geometry, and effect of internal forces in the structure).

To get a fair representation of the material non-linearities, the following points should be kept in mind :

- The *minimum load steps* should be 'reasonably' small. If the minimum step size is too large, the element forces will not be scaled back onto the yield surface. On the other hand, too small *min steps* may completely block the analysis of large, complex structures. But all in all, a too small *min step* is better than a too large.
- With the two surface plasticity model, there should be a reasonable number of steps between *first yield* and *full plasticity* at one cross section (some 5-10 steps should be OK). Some load steps are needed to activate the elasto-plastic transition of the model. If the transition is too abrupt, the two-surface model will 'degenerate' into the one-surface model (elastic-pure plastic behaviour). (Try with different load steps and see the difference on the M-N interaction ...)

The geometric non-linearities are a result of the deformations and forces built up during the proceeding analysis steps.

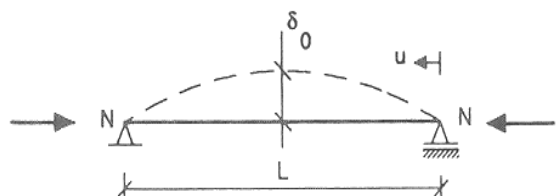
- To activate the geometric non-linearities, the user should specify some 5-10 load steps even in the elastic range. (Try the beam example with large steps and with small steps and compare ... If you use the one-surface model and keep same *min step*, the material non-linearities should be the same.)

The beam example is a typical 'stiffening' system, where the Current Stiffness parameter will increase during loading.

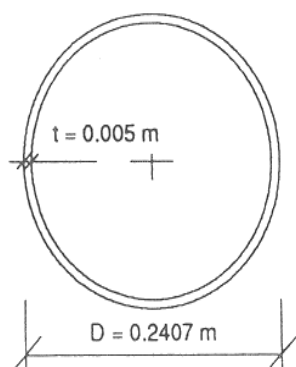
9.1.3 Workshop I b: Elasto-Plastic Column Buckling

9.1.3.1 Objective

The purpose of this case is to demonstrate column buckling by the USFOS beam element. A tubular beam-column is loaded in axial compression. The buckling load predicted by USFOS may be compared to hand calculations according to different codes. Further, the effect of initial deformations, the effect of local buckling and the effect of pre-existing dents may be studied.



9.1.3.2 Model



Column length	L	10.0 m
Tube diameter	D	0.2407 m
Tube thickness	t	0.005 m
Yield stress	σ_y	330 MPa

Young's modulus	E	2.1×10^5 MPa
Reference load	P_0	1.00 MN
Section area	A	3.702×10^{-3} m^2
Moment of inertia	I_y	2.572×10^{-5} m^4
Moment of resistance	W_P	2.778×10^{-4} m^3
Axial yield load	N_P	1221.6 kN
Plastic moment cap.	M_P	91.674 kNm
Euler buckling load	P_E	533.077 kN

9.1.3.3 Description

Due to the initial deformations, the column is loaded by combined bending moments and axial forces. Under elasto-plastic buckling, the member buckles when the beam-column form a kinematic mechanism. For a simply supported column, buckling occurs when a plastic hinge is formed at member midspan. The load is then reversed, and the column is unloaded into the post-buckling range. If a two-surface plasticity model is used, buckling takes place some load steps after first fiber yield, when the stiffness of the mid cross-section has been reduced enough for the column to become unstable.

9.1.3.4 Program input

Control file	:	head_orig.fem
Structure file	:	model_orig.fem
Load file	:	none
Load case 1	:	Axial load P_0

9.1.3.5 Analyses

1	Perform a buckling analysis of the column, with the following input : - initial deformation 0.001 (default) - local buckling formulation OFF (default) - Identify the buckling resistance - Create bending moment- axial force history diagram for mid cross-section using Results/History plot/ <i>X-axis</i>: Element force – Element ID:1, End: 3, DOF:5, <i>Y-axis</i>, Element force – Element ID:1, End 3, DOF: 1 Explain the interaction history	use <i>GELIMP</i> and <i>GIMPER</i> records use <i>DENT_OFF</i>
2	Repeat analysis 2, with initial deformations set to 0.002 Compare buckling resistance with base case.	use <i>GELIMP</i> and <i>GIMPER</i> records
3	Repeat with initial deformations set to 0.003 Compare buckling resistance with base case What can be concluded with respect to buckling resistance and initial imperfections?	use <i>GELIMP</i> and <i>GIMPER</i> records
4	Turn on the local buckling formulation, and repeat analysis 1 : - initial deformation 0.001 - local buckling formulation ON - How is the resistance-end deformation curve compared to base case?	use <i>GELIMP</i> and <i>GIMPER</i> records use <i>DENT_OFF</i>
5	Define an initial dent of 10% of the tube diameter, and repeat analysis 4. Compare with base case	use the <i>GIMPER</i> record

9.1.3.6 Documentation

Use XFOS to generate the following documentation :

- Global load vs. global deformation
- Axial force vs. bending moment
- Axial force vs. global displacement and bending moment vs. global displacement (For analysis 1, only)

9.1.3.7 Comments to the analyses

USFOS nonlinear effects: The same comments apply as for the beam example.

- To activate the nonlinear geometric terms of the element formulation, some 5-10 load steps should be applied before yielding takes place.

b) To activate the elasto-plastic transition of the plasticity model, there should be 5-10 load steps between first fibre yield and full plastification of the cross-section.

The predicted buckling load will show some variation with the load step size. In particular small load steps should be used around the peak load. (Try with different step sizes and see)

9.1.4 2D Frame Analysis

9.1.4.1 Objective

The purpose of this study is to investigate system effects and redistribution of forces in a relatively simple structure. Also, this case can be used to study the effect of the different plasticity models on global system behavior. The example structure is a two-story X-braced frame, loaded by a horizontal force at the top. The model is shown in Figure 9.1-2.

9.1.4.2 Model

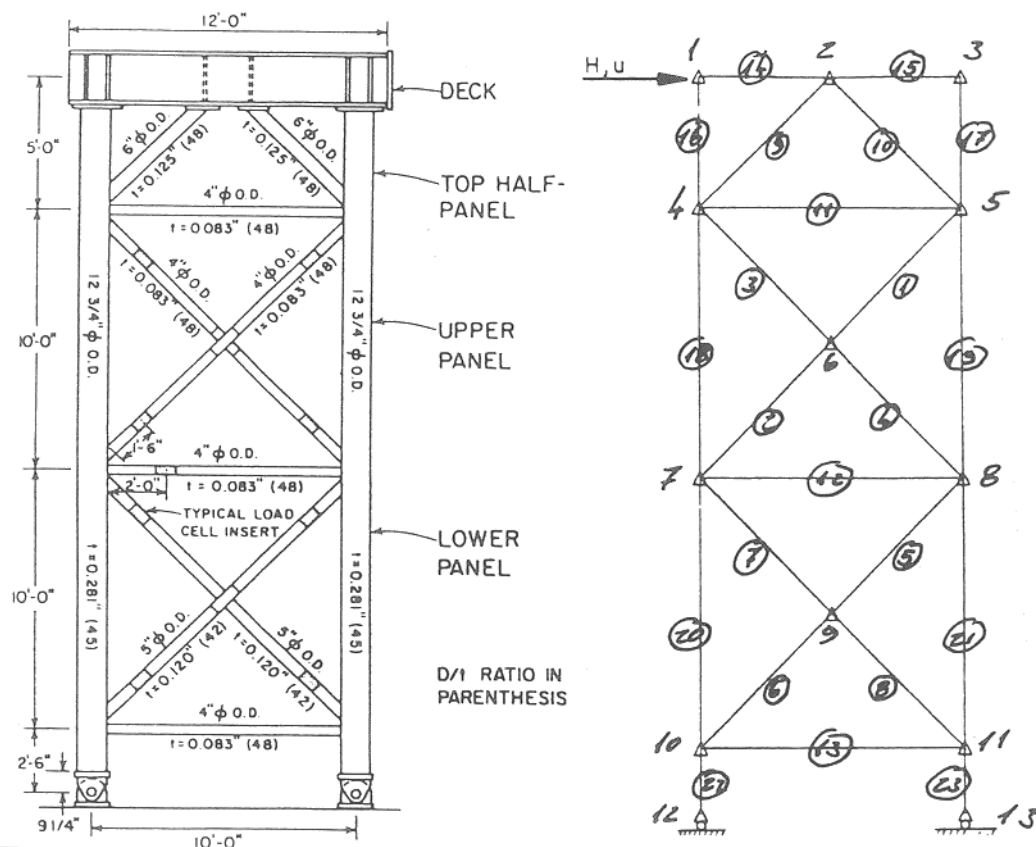


Figure 9.1-2 Plane frame structure

Yield stresses :

Braces	σ_y	248 MPa
Legs	σ_y	324 MPa
I-beam	σ_y	324 MPa

Reference H_0 40.0 kN
load (LC = 2)

Gravity load
(LC = 1)

9.1.4.3 Description

One compression brace of the upper panel fails when the external load reach a level of __.__. From this level, the load in the other compression brace decreases, and the load is transferred to the compression braces of the panel. At load level __.__, a compression brace of the lower panel fails. Again, the other compression brace is unloaded, and further loading carried by the tension braces. At load level __.__, the full plastic capacity of the tension braces is utilized. Any further loading is carried entirely by the legs.

9.1.4.4 Program input

Control file Head_orig.fem
 Structure file Stru_orig.fem
 Additional ~
 file
 Load case 1 Gravity load
 Load case 2 Horizontal load

9.1.4.5 Analyses

1.	Perform a pushover analysis of the frame, with the following input : No initial deformations. Local buckling formulation OFF. Load case 1 (gravity) to load level 1.0. Increasing Load case 2 Activate plastic utilisation. Magnify displacements by a factor of 2. Create axial force versus global displacement diagrams for element no's 4, 5 and 12. Step up along the curves and explain by physical reasoning the interrelationship between the curves. Identify maximum resistance of frame and buckling resistance of element 4.	use <i>DENT_OFF</i>
2.	Apply initial deformation equal to 0.002 times member length and perform pushover analysis as for the base case. Compare maximum resistance of the frame and buckling resistance of element 4 with the base case. Plot strain distribution in the frame using Result/BEAM/ELEMENT/Beam strain . Adjust fringe range to 0 – 0.15 using Results/Fringe range . Identify the deformation state with a maximum beam strain of 0.15.	use <i>CINIDEF</i>
3.	Activate local dent formulation and perform pushover analysis. Compare maximum resistance of the frame and buckling resistance of element 4 with base case. Identify initiation of local buckling on element 4 response.	deactivate <i>DENT_OFF</i>
4.	Experiment with step-size, and equilibrium iterations	modify <i>CUSFOS</i> activate/deactivate <i>CITER</i>

9.1.4.6 Documentation

Use XFOS to generate the following :

1. Global load vs. global deformation
2. Deformed geometry plots at first member failure, collapse and at final analysis step.
3. Axial force vs. global displacement for members #1, #4, #5, and #12.

9.1.4.7 Comments to the analyses

As stated previously the load step size should reflect the nonlinearity of the structure. This is not known in advance, and some trial analyses should usually be expected. Generally, the load steps may be relatively large in the linear range, and should be reduced as the response becomes nonlinear. Particularly in spring-back areas, both the load step size and the min step should be considered carefully. Generally speaking, the predicted P- δ curves moves at a 'tangent' to the true P- δ curve. Too large steps in a spring-back area could cause considerably 'drift' from the true solution.

As a general statement, the comments from the previous cases still apply:

- a) use some 5-10 steps in the linear range (to activate geometric nonlinearities)
- b) check the *min step* so that the analysis is not 'blocked' by too many load steps, but that the forces still are scaled to the yield surface. (Too many steps scaled to '*MIN STEP LENGTH*' too early in the analysis is usually a sign that the *min step* may have been set too large. Close to the peak load, however, it is inevitable that many steps are scaled to minimum.)
- c) check that at least 5-10 steps passes from *first yield* to *full plasticity* (to activate the elasto-plastic transition)

If the following question can be answered with YES, then the load specification is OK

"IS THE ANALYSIS DETAILED ENOUGH TO CAPTURE THE REDISTRIBUTION OF FORCES AT EVERY STAGE ?"

9.2 Workshop II, USFOS Jacket Pushover

9.2.1 Structure Description

A realistic North Sea jacket structure is analyzed. The structure is shown in Figure 9.2-1.

The structure is an 8 leg jacket, designed for a water depth of approx. 110 meters. The legs are arranged in a two by four rectangular grid, with the central pair of legs on the platform north side serving as launch runners. Overall dimensions a top elevation is 27 x 54 m, with launch legs twenty meters (20 m) apart. Overall dimensions at mudline is 56 x 70 m. Total height is 142 m, with horizontal bracings at 5 levels.

The module support frame (MSF) is a rectangular grid of built-up trusswork beams; 2 longitudinal and 4 transverse trusses. The trusswork is designed of built-up box sections. Overall dimensions are 27 x 68 meters. Trusswork height is 9.75 meters. The MSF is shown in Figure 9.2-2.

The jacket foundation is made up of four corner clusters with eight skirt piles in each group; no leg piles.

Longitudinal jacket frames are diagonal-braced, with X-braces between central and corner legs at the bottom bay. Transverse frames are K-braced, with the bottom K inverted to form a double-X. In the end frame by the conductor area (frame 4), the bottom K is substituted with two X-bracings. The horizontal levels are K-braced, with X-bracing in the conductor area.

Leg diameters range from 1.6 m at deck level to 3.0 m at elevation -104 m. Vertical braces range from 1.1 to 1.6 m diameter, horizontals 0.9-1.3 m, and horizontal braces 0.8-1.0 m.

The central legs have a D/t of 21-29 from elevation -14 to -43, a D/t of 50-67 from elev. -43 to -74, and 80 from elev. -74 to -104. Corner legs have a D/t of 9-12 at the upper bay, and 21-29 from elev. -14 to -104.

Typical D/t-values for vertical braces are 20-36, with one member at mudline with a D/t of 43.

D/t for the horizontals range from 26 to 48, with two members at mudline with D/t of 52.

Horizontal braces range from 20 to 33, with 40 as the highest for one member at mudline and one member at elevation -14.

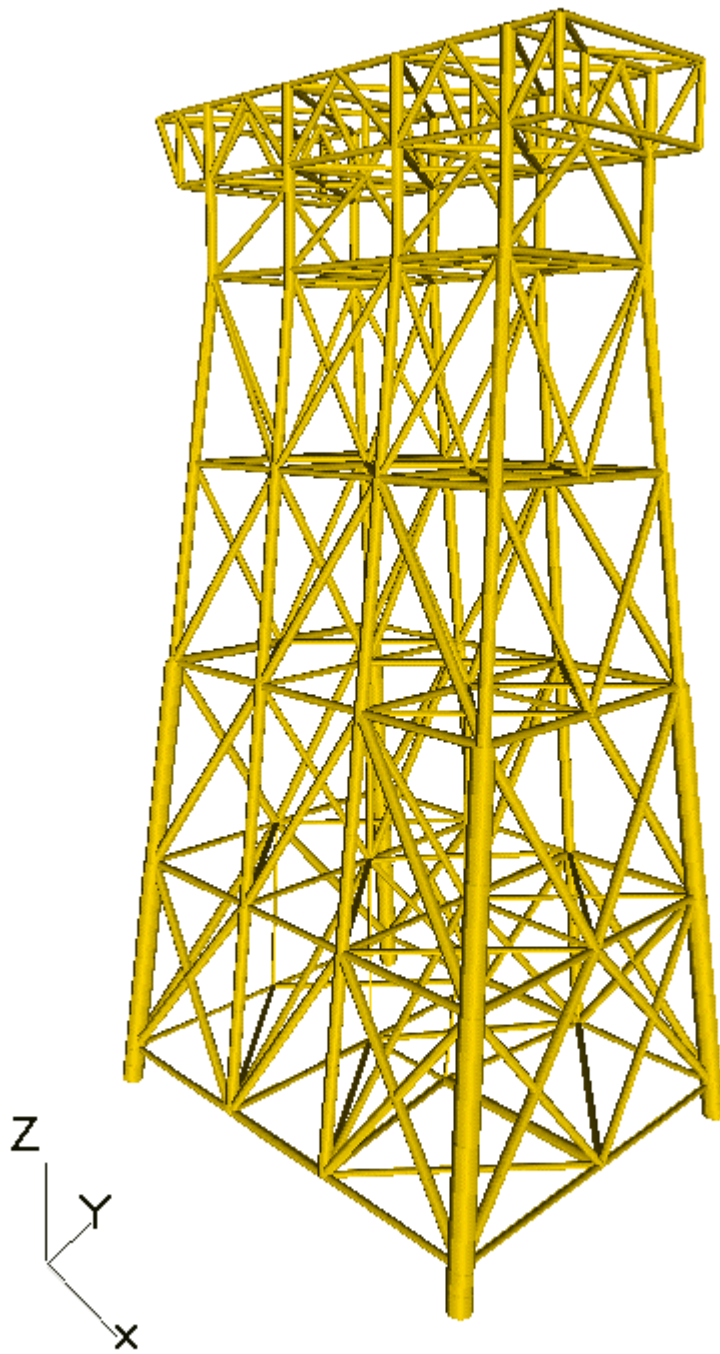


Figure 9.2-1 Jacket structure (Pile guides not shown)

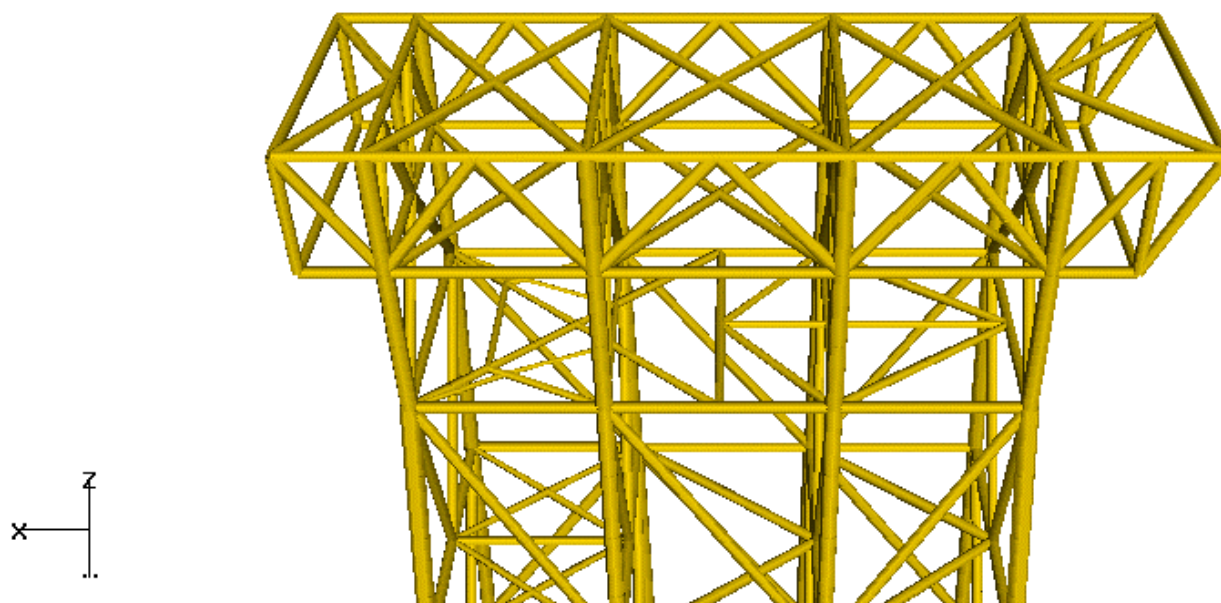


Figure 9.2-2 Module Support Frame

9.2.2 Loads

9.2.2.1 Environmental conditions

Representative environmental conditions are given in Table 9.2-1 to Table 9.2-4. Loading directions (platform directions) are defined in Figure 9.2-3.

Table 9.2-1 Wave (100 year return period)

	Direction							
	N	NW	W	SW	S	SE	E	NE
Wave height (m)	30.0	31.0	27.0	28.5	30.5	24.0	10.0	13.5
Wave period (s)	16.5	17.0	16.0	15.5	16.0	14.5	10.0	12.5

Table 9.2-2 Current (10 year return period)

	Direction							
	N	NW	W	SW	S	SE	E	NE
Mean water surface (m/s)	1.05	1.15	1.10	1.05	1.05	1.10	1.05	1.05
Elevation -50m (m/s)	.80	.85	.85	.80	.80	.85	.80	.80
Mudline (m/s)	.55	.60	.60	.55	.55	.60	.55	.55

9.2.2.2 Environmental loads

Wave loads are calculated using Stokes 5th order theory. Wave and current loads are represented as distributed (line) loads.

Module loads and wind loads are applied as concentrated forces/moments on the deck nodes.

Wave and current loads are calculated by the SESAM'80 program WAJAC. Environmental loads are determined for 8 platform directions. Loads are calculated for 100-year wave, 100-year wind and 10-year current, acting simultaneously.

Resulting global forces are listed below. Global load envelopes are shown in Figure 9.2-3.

Table 9.2-3 Global loads (base shear in MN)

	Direction							
	N	NW	W	SW	S	SE	E	NE
Total	123.8	131.1	96.0	109.9	124.8	85.5	23.1	36.5
Wa + Cu	116.8	122.2	90.4	101.0	117.8	76.6	17.5	27.6
Wind	7.0	8.9	5.6	8.9	7.0	8.9	5.6	8.9

Wa : Wave loading
 Cu : Current loading
 Wi : Wind loading

9.2.2.3 Static loads

Static loads 415.0 MN

Buoyancy 77.7 MN

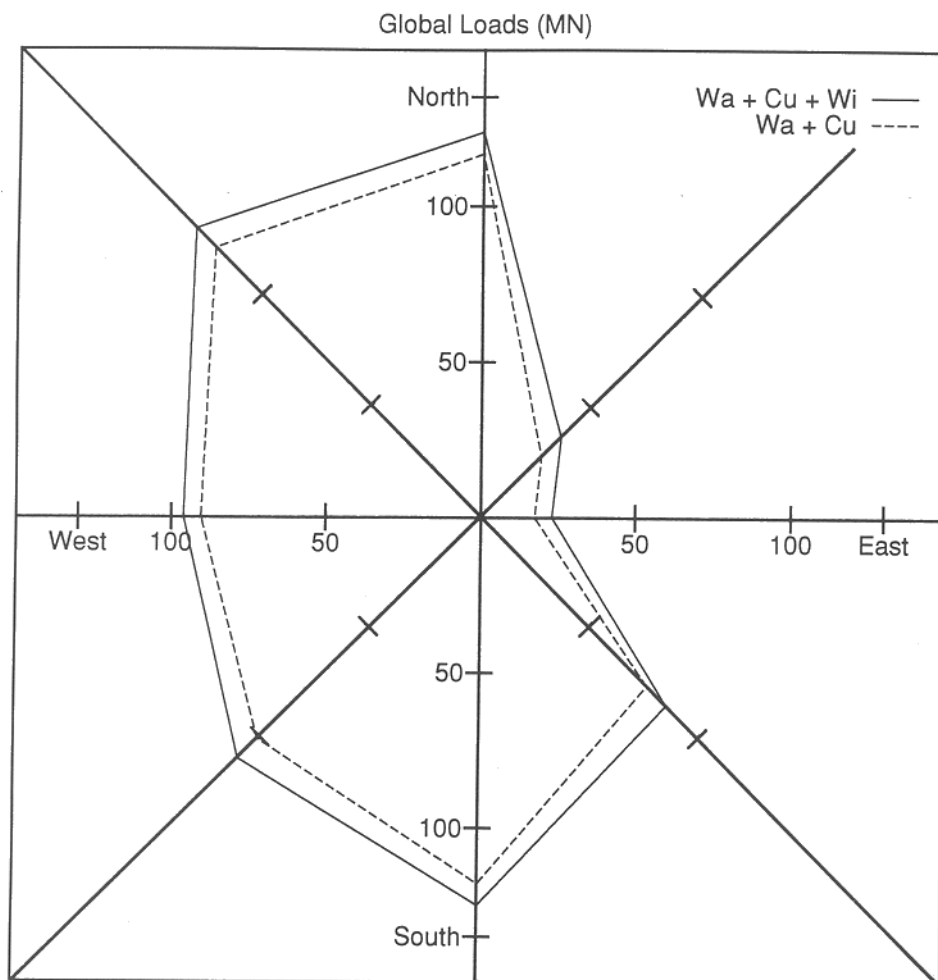


Figure 9.2-3 Global loads

9.2.3 FEM Model

The structure is modelled by 160 nodal points, and 455 elements. 20 elements are defined as non-structural. 19 materials and 119 cross-sections are defined.

Modules or topside facilities are not modelled. No joint cans or brace stubs are modelled.

Pile guides are modelled by equivalent elements. The equivalent stiffness of the pile guides and piles is computed and used. The wave load is modelled by equivalent hydrodynamic diameters.

Conductor guide frames and the conductor guide arrangement is simplified. Wave loads are modelled by equivalent element diameters.

The launch runners are modelled as an integrated part of the launch legs A2 and A3.

Conductors are lumped together as tubular elements with appropriate drag diameter and inertia diameter. Risers, caissons, J-tubes, boat bumpers, walkway and ladder are modelled as equivalent elements. These elements are defined as non-structural in the USFOS analyses.

Soil-structure interaction is modelled by linear springs.

9.2.4 Traditional pushover, loads from WAJAC

One pushover analysis is carried out for each platform direction.

Gravity loads, buoyancy and operational loads are incremented up to characteristic (unfactored) value. Then, environmental forces are applied incrementally until collapse.

The following events are identified for each loading direction:

- 1) Initial yielding, λ^Y
- 2) First member failure (member buckling or tension yielding), λ^F
- 3) Ultimate collapse load, λ^{ULT}

Table 9.2-4 summarizes the results of the pushover analyses. Load factors for characteristic events in each loading direction are listed. These are relative load levels, referred to the characteristic loading for each direction, ref Table 3.2.3.

Table 9.2-4 Characteristic load levels (load factors), pushover analyses

	Direction							
	N	NW	W	SW	S	SE	E	NE
λ^Y								
λ^F								
λ^{ULT}								

9.2.4.1 Program input

Control file	head-<direction>.fem	One file for each loading direction
Structure file	Model_orig.fem	One single file
Additional file	load-<direction>.fem	One file for each loading direction
Load case 1	Dead load + live loads	Listed on the <i>stru</i> -file
Load case 2	Buoyancy	Listed on the <i>load</i> -file
Load case 2	Wave + Current + Wind loads	Listed on the <i>load</i> -file

9.2.4.2 Analyses

1	• Perform a pushover analysis for the north loading direction, with the following input • no initial deformations • local buckling formulation OFF • Introduce the group definition command, see control file head-no.fem. One group is defined for non-structural elements (these elements take loads, only). • Activate non-structural elements • Activate group for pipes with diameter > 1.8 m • Activate group for material no 2. a. Visualise the various groups using RESULT/Groups command. b. Isolate various planes in the frame using the command Edit/Clip. c. Use the command Verify/Verification; tick off Node info and Element info and use the mouse to click on the structure, simultaneously pushing the Ctrl button, to obtain element and node information in a separate window.	deactivate <i>GELIMP</i> or <i>GIMPER</i> records activate DENT_OFF activate NONSTRU visible activate GROUPDEF 11 activate GROUPDEF 12
2	Perform one pushover analysis for each load direction. Identify ultimate resistance levels and identify governing collapse mechanisms. Is there any difference in the global load – global deformation curves for wave direction perpendicular to the short phase (West loading) and the long phase (North loading) of the platform. Give a physical explanation the response curves.	

9.2.4.3 Documentation

1. Generate P- δ plots from the final analysis of each direction
2. Take hardcopies of deformed geometry (with member utilization) at first member failure, max load and at the final analysis step.

3. Generate plots of axial force vs. global displacement (N- δ plots) for critical members in the failure mechanism (buckling members, tension failure members or failing leg members). The purpose of this is to document the redistribution of forces and the development of the final failure mechanism.

9.2.4.4 General comments

Now is the time for 'real life' analyses ...

We should utilize any prior knowledge about the structure :

- a) If the structure is designed according to *NPD regulations*, it will have elastic response up to a loading 1.6 times the characteristic load from the worst direction. For the other directions, we may assume elastic response up to approximately the same load level (in MN).

If the structure is designed according to *API*, then first member failure should coincide with 1.6 times the characteristic loading. First fibre yield may then occur at approximately 1.2 - 1.3.

- b) From first fibre yield to first member failure, there would only be a slight reduction in stiffness. The response would still be practically linear.
- c) For X-braced structures (or more generally, if the loading is carried by a statically indeterminate bracing system), we should expect a gradually softening behaviour up to a peak load, and then a gradual load reduction in the post-collapse range. Prior analyses indicate the reserve capacity from first member failure up to the collapse load to be in the order of 30%.
- d) For K- or diagonal braced structures (if the loading is carried by a statically determinate bracing system), the collapse load may be roughly equal to the first member failure load (also depending on the strength of the legs, of course). These bracing systems have very little redundancy, and may exhibit an extremely brittle behaviour; almost linear up to first member failure, system collapse only slightly above first member failure, and then a sudden reduction in capacity ('spring-back'). Several braces may fail in sequence, giving a series of spring-back failures until the final leg mechanism is activated.

9.2.4.5 Comments to the analyses

The general comments from the previous workshop still applies, and a few trial analyses or restarts should be expected.

- a) Check the size of load steps, to ensure that the nonlinear effects have been activated. For fixed offshore structures, the material nonlinearity is by far the dominating one, so the main verification lies in checking the Γ -values. (Check the Γ -values at some, characteristic positions along the P- δ curve - at max load, at the final analysis step etc.)
- b) Iteration convergence should be checked at every step. And since the iterations include a correction to the yield surface, convergence imply $\Gamma=0.0$ for all members except AXIAL FAILURE members.
- c) AXIAL FAILURE members must be checked separately. USFOS scales the load step when ordinary yield hinges are formed. But in the present version, the load step is not scaled when the element reaches AXIAL FAILURE. The Γ -values may jump far off the yield surface. (Load step scaling will be implemented in the coming version.)
- c) If the structure shows any sign of 'spring-back' behaviour, the step size just before, and during the spring-back must be carefully evaluated. Is the analysis detailed enough to capture the

redistribution of forces during spring-back? If not, or if in doubt, smaller load steps should be specified and the analysis restarted just before the spring-back. To answer the question, check for any sharp 'corners' on the $P-\delta$ curve, check the Γ -values during and just after the spring-back, and check if iterations have converged or performed normally during the spring-back.

9.3 Workshop III Ship Impact

According to the Regulations for offshore structures in Norwegian waters, the structure is to be designed for impacts with supply vessels with a displacement of 5000 tons and an impact speed of 2 m/s. Including hydrodynamic added mass this yields a kinetic energy of 14 MJ for sideways impact and 11 MJ for head-on and stern impact. Impact has to be considered all exposed members taking into account tidal effects, wave elevation, ship motion etc. The purpose of the present workshop is to check a jacket structure for various impact cases on jacket legs and braces. Analyses shall be executed for the following accident scenarios:

- 1) Longitudinal impact on leg A4, at elev. -1.0 meter
- 2) Transverse impact on leg A4, at elev. -1.0 meter
- 3) Brace impact, row 4, elev. -1.0 meter
- 4) Brace impact, row A, elev. -1.0 meter
- 5) Transverse impact on leg A3, at elev. -1.0 meter

Gravity loads, buoyancy and operational loads are applied up to unfactored, characteristic values. Then, the impact load is incremented until the specified impact energy is dissipated. Finally, the impact forces are stepped down.

Energy dissipation in ship and structure is determined. Permanent deformations and residual stresses are recorded, and may be used as input to a residual strength analysis (restart).

9.3.1 Program input

Control file	head- <i><case X></i> .fem	One file for each impact case
Structure file	Model_orig.fem	One single file
Load file	load.fem	One single file (with <i>north</i> wave loading)
Load case 1	Dead load + live loads	Listed on the <i>model_orig</i> -file
Load case 2	Buoyancy	Given in the <i>load</i> -file
Load case 3	Wave and current loads)	Given in the <i>load</i> -file
Load case 4	Ship impact loads	Given in the <i>head</i> -file, refer BIMPACT command. Impact energy shall be 14 MJ.

9.3.2 Analyses

1	Perform one collision analysis for each accident scenario. For case 1, include also a residual strength analysis (increment wave and current loads) of the damaged structure after impact.	
2	For case 1 perform a dynamic analysis of the ship impact. Use control file head_1d.fem	Define TIMEHIST and LOADHIST for dead load/buoyancy load and Ship impact loads

Apply dead+live loads statically	Use STATIC
Ship impact dynamically	Use DYNAMIC
Magnify displacements by a factor of 4 and activate plastic utilisation.. Generate an animation of the response, using Results/Animate	

9.3.3 Documentation

1. Generate P- δ plots for the impact location from each analysis. Verify that the energy dissipation is approximately 14 MJ. Use **Results/History/Nodal displacement/Nodal force**. Create axial force-global displacement plot for jacket legs in case of leg impact. Why does the axial force change? Identify mechanisms for redistribution leg axial force by generating plots of axial force vs. global displacement (N- δ plots) for key members in the impact area. How is the residual strength of the platform affected by the impact damage Case 1? Explain the result.
2. In case of brace impact investigate how the brace force is supported.
3. Take hardcopies of deformed geometry (with member utilization) at max impact load
4. Check strain values using **Result/Beam/Beam strain**. If necessary scale fringe reange.

9.3.4 General comments :

The ship collision analysis is essentially like any other pushover analysis. The major difference is that the user only specifies the general loading situation (e.g. impact energy, position and direction), and not the actual impact loads. Suitable impact load increments are calculated by USFOS. But the user still have to define the impact situation as a *load case*, and apply this load case in the USFOS load specification (the *CUSFOS/CICYFOS*-records) like any other load case.

USFOS calculates suitable impact load increments as a fraction of the force required to flatten the tube, or as a fraction of the mechanism load of the hit member. The impact increments are calculated so that initial yielding should occur after some 10-20 load steps.

When the specified impact energy is dissipated, USFOS reverses the impact load and unloads the structure.

The total energy dissipation is listed in the 'global history' table. This include energy dissipated by the ship, by the structure, and locally, by flattening of the tube. Energy dissipated by ship and structure each, is listed on the analysis print file (the OUT-file).

Residual forces and permanent deformations are stored on the RAF-file, like the results from any load step of a pushover analysis. This can be utilized for residual strength analyses by simply specifying a *restart* from the final step of the ship impact analysis. Then, the effect of residual forces and permanent deformations will be included in the residual strength analyses. Note that the environmental forces then will have to be read into USFOS in the initial analysis (read into the RAF-file, but not applied to the structure). It is not possible to read in new load cases in a restart analysis.

Alternatively and simpler; increment environmental loads directly after the impact case is finished; i.e. in the same analysis run as done in case 1.

9.3.5 Longitudinal Impact on Leg A4

An impact energy of 14.0 MJ is applied to corner leg A4 at elev -1.0 meter. Impact is specified in the longitudinal direction. Impact position: end 2 of member #732.

The ship is assumed infinitely stiff - all energy is absorbed by the structure

The applied impact forces can be absorbed by the structure, without danger of capsizing.

Permanent deformations at the point of impact is ____ m. Permanent deformations at deck level is ____ m.

Initial yielding occurs in the hit member, at ____% of the impact energy. Membrane action is activated in the hit member, and forces are redistributed to the surrounding structure. Member #____ fails at ____% of the impact energy, followed by member #____ at ____%.

The global response is fairly linear during the impact. Significant local deformations develop.

9.4 Workshop IV Residual strength analysis

The structure is analyzed in different damaged conditions. The residual strength of the structure is determined, compared to the collapse capacity in intact state.

Gravity loads, buoyancy and operational loads are incremented up to characteristic (unfactored) value. Then, environmental forces are applied incrementally until collapse.

The following damage conditions are analyzed :

- 1) Brace #261 damaged (row 4). Loading from west
- 2) Brace #363 damaged (row 4). Loading from west
- 3) Brace #463 damaged (row 4). Loading from west
- 4) Brace #355 damaged (row A). Loading from north
- 5) Brace #455 damaged (row A). Loading from north

First, the members are assigned an out-of-straightness equal to one tube diameter at member midspan. Then, the member is removed entirely.

The following events are identified from each analysis:

- 1) Initial yielding, λ^Y
- 2) First member failure (member buckling or tension yielding), λ^F
- 3) Ultimate collapse load, λ^{ULT}

Table 9.4-1 and Table 9.4-2 summarizes the results of the pushover analyses. Load factors for characteristic events in each loading direction are listed. These are relative load levels, referred to the characteristic loading for each direction. The residual strength ratio, RIF, is calculated for each damage condition.

Table 9.4-1 Characteristic load levels, members damaged

	Damage condition				
	#261	#363	#463	#355	#455
λ^Y					
λ^F					
λ^{ULT}					
RIF					

Table 9.4-2 Characteristic load levels, members removed

	Damage condition				
	#261	#363	#463	#355	#455
λ^Y					
λ^F					
λ^{ULT}					
RIF					

9.4.1 Program input

Control file	head-<case X>.fem	One file for each damage scenario
Structure file	stru.fem	One single file
Load file	load.fem	One single file (with <i>north</i> wave loading)
Load case 1	Dead load + live loads	Listed on the <i>stru</i> -file
Load case 2	Buoyancy	Listed on the <i>load</i> -file
Load case 3	Wave + Current + Wind loads	Listed on the <i>load</i> -file

9.4.2 Analyses

1	Perform one pushover analysis for each damage scenario, specifying an initial damage to the element: • assign the specified out-of-straightness as initial deformation • switch the local buckling formulation ON	use <i>GELIMP</i> and <i>GIMPER</i> records
2	Perform one pushover analysis for each damage scenario, removing the damaged element	use the <i>NONSTRU</i> record

9.4.3 Documentation

1. Generate P- δ plots from the final analysis of each direction
2. Take hardcopies of deformed geometry (with member utilization) at max load and at the final analysis step. (For analysis 2)
3. Generate plots of axial force vs. global displacement (N- δ plots) for critical members in the failure mechanism (buckling members, tension failure members or failing leg members). (For analysis 2)

9.4.4 General comments

Residual strength analyses are generally a repetition of the earlier pushover analyses. The only difference is that we are addressing a new structural configuration. And since the structure is damaged, we might expect a somewhat more 'difficult' behaviour. That is, a bit more checking and 'tuning' of load steps should be expected, especially if the damage provokes serious 'spring-back' behaviour. But the basic procedure is the same ...

9.5 Workshop V Pile and Soil Modelling with USFOS

9.5.1 Description of the structure

The structure object of the study is an 8-legged drilling and production jacket platform located in the Bay of Campeche, Mexico. This platform was designed to operate as a drilling platform and it was installed in 1984 in 46.64 m of water depth. The total height of the platform is 69.3 m measured from the upper deck to the bottom of the jacket.

The two-level deck is supported by eight columns arranged so that two longitudinal frames and four transverse frames integrate the structure. The upper deck located at the elevation +21.64 m above the mean sea level, supports the equipment and supplies required for drilling operations, since recently three wells were added to the structure, as well as separation tanks. Its overall dimensions are 25x45 m. The lower deck is located at the elevation +15.85 m and contains 15 christmas trees as well as production equipment.

The deck is supported by an 8-legged jacket provided with two longitudinal frames and four transverse battered (1:8) frames with horizontal framing at elevations +6.10, -6.10, -18.2, -32.4 and -46.3 m. The transverse frames are K-braced whereas the longitudinal ones are a mixture of diagonal bracing and X-bracing between central legs. Most of the structure was fabricated with mild steel, e. g. ASTM A-36 steel, whereas joint cans were constructed with high strength steel. The dimensions of the jacket at the mudline are 51X28 m.

Every leg allocates a pile of 48-inch diameter (1.22 m), which are intended to fix the structure to the sea floor. The 4 corner piles are driven into the sea floor some 95 m and 4 inner pile tips are 90 m below the mudline. Jacket legs, deck columns and piles are connected in the working point at elevation +7.62 m.

This platform is supporting 7 risers whose diameters are 30.5 cm (1), 20.3 cm (1) and 50.8 cm (5), as well as 15 conductors. The conductors are 30-inch (0.76 m) tubular elements driven 60 m below the seabed. Every leg is provided with a bumper to protect them against barge collisions. There are as well, two boat landings located at each longitudinal frame.

9.5.2 FEM model.

The platform was modelled with 1777 beam elements and 1131 nodal points. 352 different cross sections were considered and 3 different material types were utilized. 39 main tubular joints are checked during the pushover analyses, where joint cans are considered, as the case of K-joints and X-joints. The tubular joint capacity is evaluated according to the recommended practice of API RP2A (API, 1994).

The conductor guide framing is modelled in a simplified way. In the deck only primary elements of the floor system are modelled, and some equivalent elements are used instead of secondary elements. In order to model the transmission of lateral forces between piles and legs as well as between conductors and guide framing, linear dependencies are defined. Since the conductors were assumed as structural elements and therefore the soil-conductor interaction is also included in the model.

The appurtenances such as boat landings, barge bumpers, risers, and caissons were modelled as non-structural elements in the analyses.

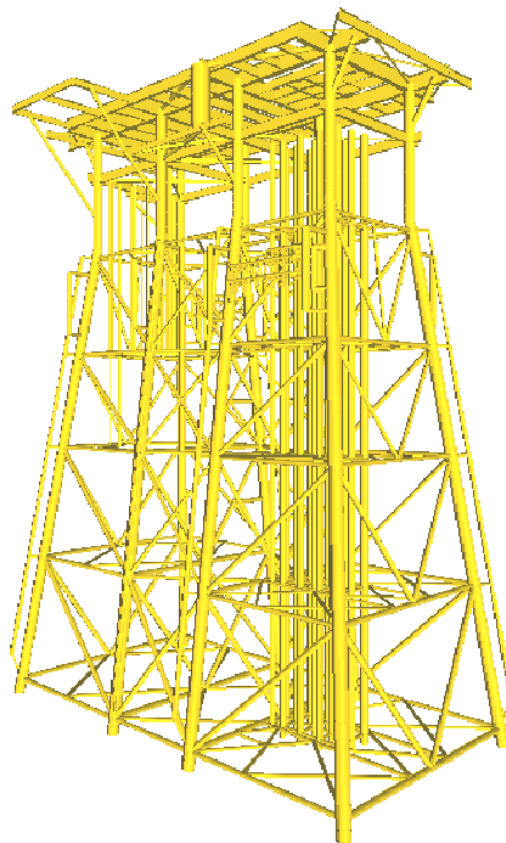


Figure 9.5-1 Drilling and production platform

9.5.3 Foundation.

The foundation of the platform consists of eight tubular steel piles, each of which is driven into the sea floor and allocated inside each jacket leg. The piles are fixed to the corresponding jacket leg and deck column at elevation +7.315 m (working point). Corner piles were driven at a depth of 98.74 m from the mudline, whereas for inner piles this depth is 90.37 m.

The piles are fabricated with six segments of different wall thickness, in a range from 2.54 cm in the pile tip to a thickness of 5.72 cm in the top. In all the segments a yield stress of 248 MPa (2530 kg/cm²) was used, except in the upper 11-meter segment of each pile in which a 345-MPa (3520 kg/cm²) yield stress steel was utilized.

The soil in the Bay of Campeche is mainly comprised of alternating layers of calcareous clays and sands. The calcareous sands are of moderate density and the clays are generally stiff and consolidated, with exception if the upper layer which is soft.

A total of ten soil layers are identified in the site and P-Y and T-Z curves for twenty-seven different elevations are given in the geotechnical study of the soil. The soil characteristics are summarized in Table 9.5-1

Soil layer	Depth * (m)	Unit weight (ton/m ³)	Friction angle (°)	Shear strength (ton/m ²)	Skin friction Compres. (ton/m ²)	Skin friction tension (ton/m ²)
1	0-9.1	1.533	-	1.62	1.78	2.66
2	12.8	1.882	30	-	7.82	6.91
3	31.1	1.79	-	19.5	9.77	9.77
4	37.2	1.962	35	-	11.93	11.65
5	53.6	1.954	-	19.5	6.84	6.84
6	58.2	1.962	35	-	13.42	12.22
7	76.8	1.738	-	14.4	5.86	6.84
8	79.5	1.962	25	-	15.73	18.12
9	89.6	1.769	-	15.7	6.84	19.62
10	105.8	1.818	-	19.5	9.79	9.77

* Vertical coordinate of the layer bottom measured from the mudline

Table 9.5-1 Soil characteristics

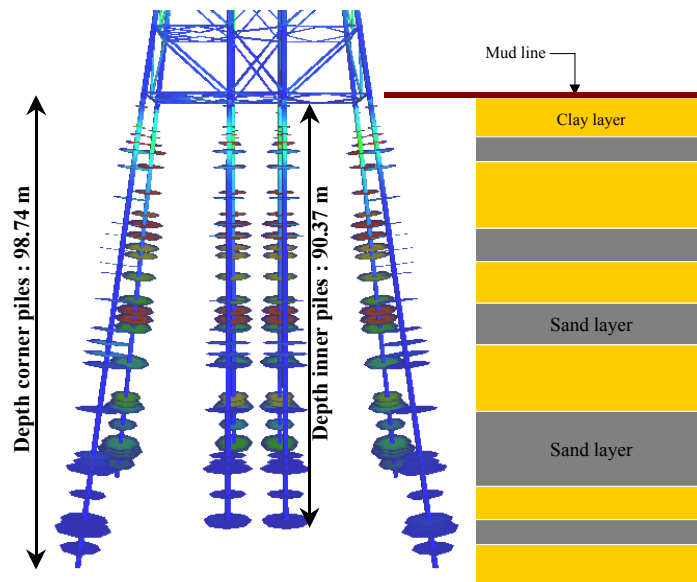


Figure 9.5-2. Foundation and soil characteristics for the platform.

9.5.4 Load modelling for reliability analysis.

The wave and wind loading on the structure is calculated according to the recommended procedure in the API-RP2A (API, 1994) by using the environmental parameters required for assessment of a fixed platform located in the Bay of Campeche.

These parameters are summarized in table 1. The wave-current loads is calculated by means of Stokes 5th order nonlinear wave theory. It is assumed that both wave and wind act simultaneously on the platform. The loading is determined for the end-on, broadside and diagonal directions.

Environmental parameter	New Design	Assessment of existing platforms			
		Elastic Range	Ultimate Limit State Exposure Category		
		All categories	Moderate	High	Very High
Return Period (years)	180	99	753	892	1485
Extreme maximum wave height (m)	16.7	15	21	21.5	23
Wave period (s)	12.06	11.46	13.52	13.70	14.24
Astronomic tide (m)	0.76	0.76	0.76	0.76	0.76
Storm tide (m)	1.04	0.98	1.20	1.22	1.27
Maximum wind velocity 10 m above MWL (m/s)					
3 s	58.77	53.30	72.60	74.22	79.09
5 s	56.79	51.51	70.16	71.73	76.43
15 s	52.54	47.65	64.91	66.36	70.71
1 min	47.18	42.79	58.28	59.58	63.49
1 hr	39.42	35.75	48.70	49.79	53.06
Current velocity (cm/s):					
0% of depth	125	111	159	163	175
50% of depth	97	83	131	135	147
95% of depth	78	70	100	102	109

Table 9.5-2. Environmental parameters for design and requalification purposes in 48.2 water depth

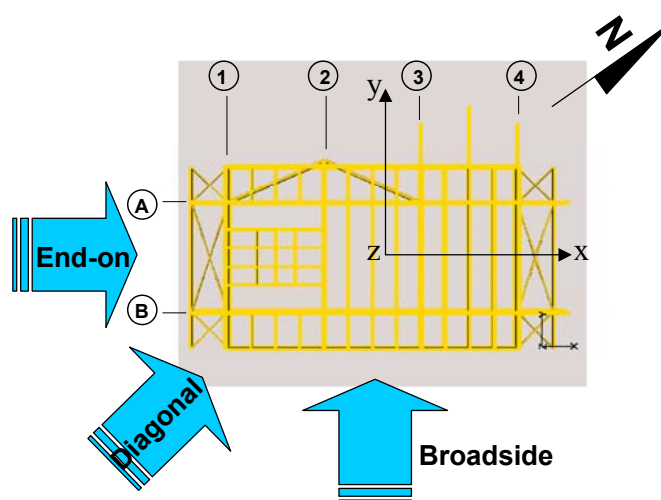


Figure 9.5-3

Loading directions considered in the analyses:
End-on (longitudinal), broadside (transverse)
and diagonal. Plan view of upper deck.

9.5.5 Lateral load on the jacket.

The lateral load effect on the jacket due to wave and current is calculated by means of the *response surface function* given by equation 5.1. Fixed wave steepness is assumed in all analyzed wave heights H by calculating the corresponding wave period as $T = 2.9569\sqrt{H}$. The other environmental parameters, e. g. maximum wind velocity, storm tide and maximum current velocity are also estimated as function of the wave height, by fitting the values given in Table 9.5-2, corresponding to 99, 180, 753, 892 and 1485 year return periods. Then, the wave forces are calculated for wave heights whose airgap is less than zero, including the wave that corresponds to zero airgap, e. g. the wave height that touches the bottom of the lower deck, H_{deck} , which was estimated to be as high as 18.65 m.

Wave heading	Blockage factor
Longitudinal	0.7
Transverse	0.8
Diagonal	0.85

Table 9.5-3. Current blockage factors used for estimation of forces in a jacket.

The API RP2A recommended procedure (API, 1997) for estimating wave-current forces is applied, and the provisions indicated in the PEP-IMP requalification criteria are considered (PEP-IMP, 1998). API's procedure recognizes the dispersion of the extreme wave by applying a wave kinematics factor of 0.85 (PEP-IMP, 1998) as well as the current blockage effect due to disturbance of the flow when passing through the structure. The blockage factors used in this study are shown in Table 9.5-3

Range of water depth from MWL (m)	Marine growth thickness (cm)
+1 to -20	6.0
-20 to -40	4.0
-40 to -80	2.0

Table 9.5-4. Marine growth thickness to be considered in requalification of fixed platforms in the BoC, (PEP-IMP, 1998)

The drag and inertia coefficients in all the elements of the jacket are set to be 1.05 and 1.2, respectively, as recommended in API RP2A approach for rough tubular members. In addition, marine growth is considered in the calculations of wave-current forces on the structure, with the thickness indicated in Table 9.5-4.

Wave-current induced forces on the jacket have been estimated with the use of the computer program WAJAC (DNV, 1982), by applying a Stokes 5th order wave theory. Base shear forces obtained for eight wave heights in longitudinal (0°), transverse (90°) and diagonal (45°) wave heading are given in Table 9.5-5

Wave height (m)	Base shear (ton)		
	Longitudinal (0°)	Transverse (90°)	Diagonal (45°)
7.09	269	328	427
10.00	541	658	790
15.00	1375	1647	1705
16.70	1752	2093	2137
18.00	2100	2494	2565
18.50	2251	2656	2741
18.60	2282	2689	2777
18.65	2297	2737	2816

Table 9.5-5. Wave-current induced loads on the jacket.

9.5.6 Program input

Two different structure models have been prepared. One is without piles (...*fixed.fem*). The other one is piled utilizing the USFOS pile and soil modelling (...*piled.fem*).

A load file utilizing the USFOS Wave input has been prepared. Wave and current data corresponding to 99 year return period and transverse loading is given initially.

Control file	head.fem	One file
Structure file	stru_fixed.fem stru_piled.fem	Two files
Load file	load.fem	One file
Load case 1	Dead load + live loads	Listed on the <i>stru</i> -file
Load case 2	Buoyancy	None
Load case 2	Wave + Current loads	Listed on the <i>load</i> -file

9.5.7 Analyses

1	Perform one pushover analysis with the given load input and the <i>fixed</i> structure.	
2	Perform one pushover analysis with the given load input and the <i>piled</i> structure.	
3	Repeat analysis 1 with joint check switched on	use CHJOINT
4	Repeat analysis 1 and 2 with different wave and current directions	Edit the load.fem file

9.5.8 Documentation

1. Generate P- δ plots from the final analysis of each direction
2. Take hardcopies of deformed geometry (with member utilization) at first member failure, max load and at the final analysis step.
3. Generate plots of axial force vs. global displacement (N- δ plots) for critical members in the failure mechanism (buckling members, tension failure members or failing leg members). The purpose of this is to document the redistribution of forces and the development of the final failure mechanism.