3 MODELLING

The USFOS analysis module is a finite element program based on an updated Lagrangian formulation.

3.1 BEAM ELEMENT

The basic structural unit used in USFOS is the two-node beam. It is used to model an entire structural member; beams as well as beam columns. Consequently, large structural systems can be modelled by means of a relatively small number of elements.

The six global degrees of freedom for each node are shown in Figure 3.1.

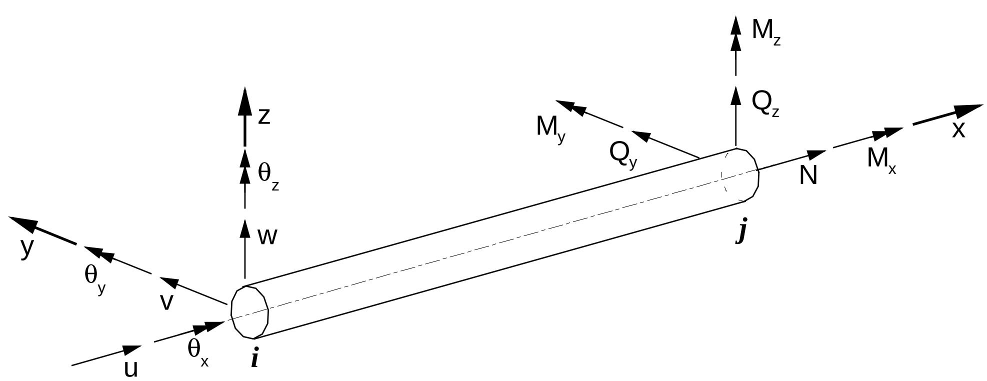


Figure 3.1Three-dimensional beam element

The element formulation of USFOS is based on the exact solution of the 4th order differential equation for a beam subjected to end forces. This yields an element shape function of trigonometric and exponential terms. For small axial forces relative to the Euler buckling load these shape functions become inaccurate, and are replaced by a 3rd degree polynomical shape function. The transition from 3rd degree polynomial to trigonometric/exponential shape function is done at a level of 0.05 of the Euler buckling load. The transition factor can optionally be specified by the user, and it is also possible to specify only one shape function to be used during the analysis (Section 6.3.C).

The two-node beam provides the following non-linear capabilities:

- **Geometric Nonlinearity** due to large lateral displacements. This yields nonlinear terms in the tangential stiffness matrix and coupling between lateral displacements and axial force.

  The influence of large nodal displacements is accounted for by updating the geometry after every displacement increment, according to an updated Lagrangian formulation.

- **Plasticity**, based on introduction of plastic hinges when the plastic interaction curve for stress resultants is exceeded. Hinges may be introduced at both ends and at midspan. The tangential equations are modified using plastic flow theory.

  In case of a plastic hinge at midspan the element is divided into two new subelements. The plastic hinge at midspan is always attributed to the first subelement. The stiffness matrix for the two subelements is assembled. The internal node is eliminated by static condensation so as to maintain the conventional 2 node beam element in subsequent analysis. This is all performed at element level without involvement of the user.

- **Elastic Plastic Column Buckling** is not a failure mode as such, but is automatically contained in the energy formulation. Since buckling is always accompanied by occurrence of plastic hinges at locations where member stress resultants exceeds the interaction surface, it does not differ from the conventional plastic failure modes.

3.2 COORDINATE SYSTEM

The global coordinate system is defined as a right-hand Cartesian system. The three global directions are denoted as X, Y and Z respectively.

A local coordinate system for the beam structural unit is defined as follows. The structural unit local x-axis is the line from the first node (i) directed towards the second node (j), as seen in Figure 3.2. The local y- and z-axis are perpendicular with respect to the local x-axis.

Input contains definition of an arbitrary vector z' in the local (x,z) plane. This vector should not coincide with the local x-axis.

If no vector z' is given, the program will generate a local coordinate system with local z-axis parallel to the global (X,Z) plane. If the local x-axis coincides with the global Z-direction, the local z-axis will be parallel to the global (X, Y) plane.

3.3 ECCENTRICITIES

Possible end offsets or rigid ends are modelled with nodal point eccentricities, as shown in Figure 3.2.

Generally, in non-linear large displacements analysis, eccentricities should be coupled to nodes rather than to structural units. According to the updated formulation the eccentricities follow the rotations of the corresponding node and are updated after each increment.

This means that the eccentricities of one structural unit are stored separately for each node, even if they are identical in the undeformed configuration.

The eccentricities do not carry distributed loads when used in combination with the beam structural unit. Hence, the eccentricities do not contribute to gravity loading and is considered being "massless".

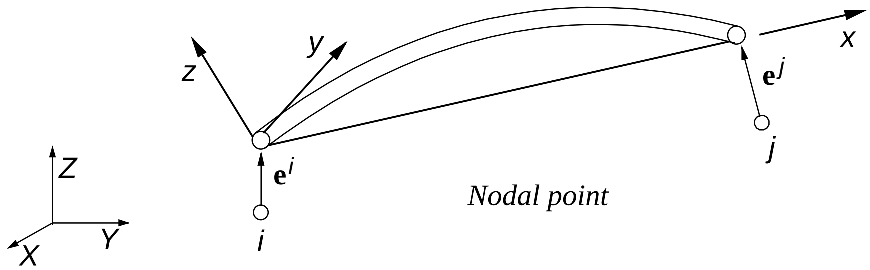


Figure 3.2Local coordinate system of beam

3.4 CROSS SECTIONS

The beam elements may be assigned one out of three cross sectional geometries, as shown in Figure 3.3. The user can also give the cross sectional parameters directly, to model an arbitrary geometry.

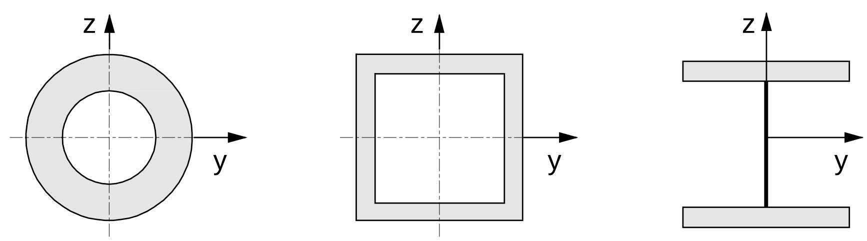


Figure 3.3Cross-sectional geometries

3.5 BOUNDARY CONDITIONS

Boundary conditions can only be specified in the global coordinate system. All degrees of freedom which are not specified as being fixed, are considered to be free. Linear springs are available, referred to local or global coordinate systems.

~~Prescribed displacements are not included.~~

3.6 LOADS

Four types of loads are available in USFOS

- *Concentrated* nodal point forces and moments.
- A linearily varying, *Distributed* load over the beam structural units.
- *Gravity* loading. The program calculates distributed loads from a given gravity vector and the volume and density of the structure.
- *Thermal* load, which accounts for degradation of yield stress and elastic modulus and thermal expansion.

For each load combination, at least one of these four load types must be given.

The mechanical loads (concentrated nodal loads and distributed element loads) are conservative loads, i.e. the direction of the loads is constant, and is not influenced by the deformation of the structure.

Element line loads may be linear between specified points along the element axis. USFOS calculates an equivalent line load with linear variation between the element ends and with the same end shear forces as the specified loads.

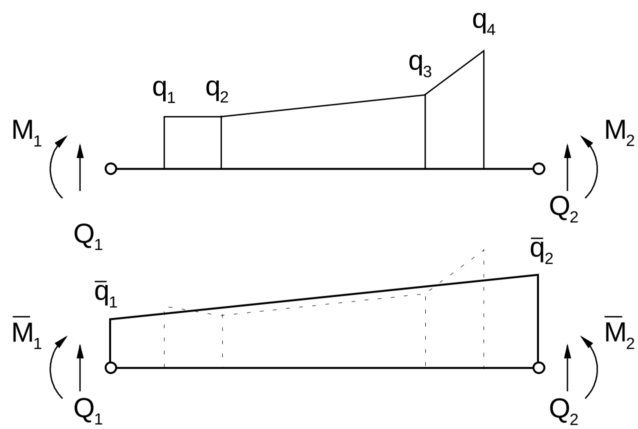


Figure 3.4Element distributed loads

3.7 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 - δ points, see Figure 3.5. Hyperelastic material behaviour (loading and unloading follows the same curve), and an elasto-plastic material behaviour with kinematic hardening are available.

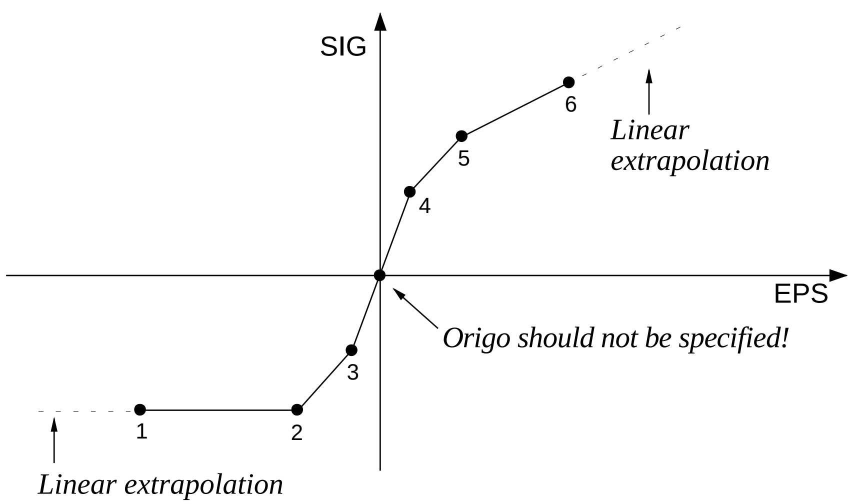


Figure 3.5Definition of spring properties by discrete points

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

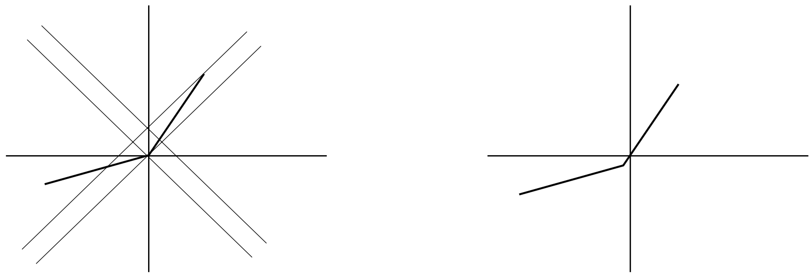


Figure 3.6Example of legal and illegal spring definition

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.

3.8 LINEAR DEPENDENCY

Linear dependency is useful in modelling structures where there, for some reason, are information available indicating that one degree of freedom (DOF) can be expressed as a linear combination of other DOFs. Such structures are for example jackets with internal piles in the legs. The pile is free to move axially within the leg, but constrained to follow its lateral displacements. Also, internal hinges can be modelled. It is a special case of the general, linear constraint equation defining a slave DOF.

Linear dependency is defined by the slave node and its master element. The actual coupling coefficients are calculated by the program, based on the location of the slave node relative to the nodes of the master element. The linear dependency feature and its possibilities are illustrated in Figure 3.7.

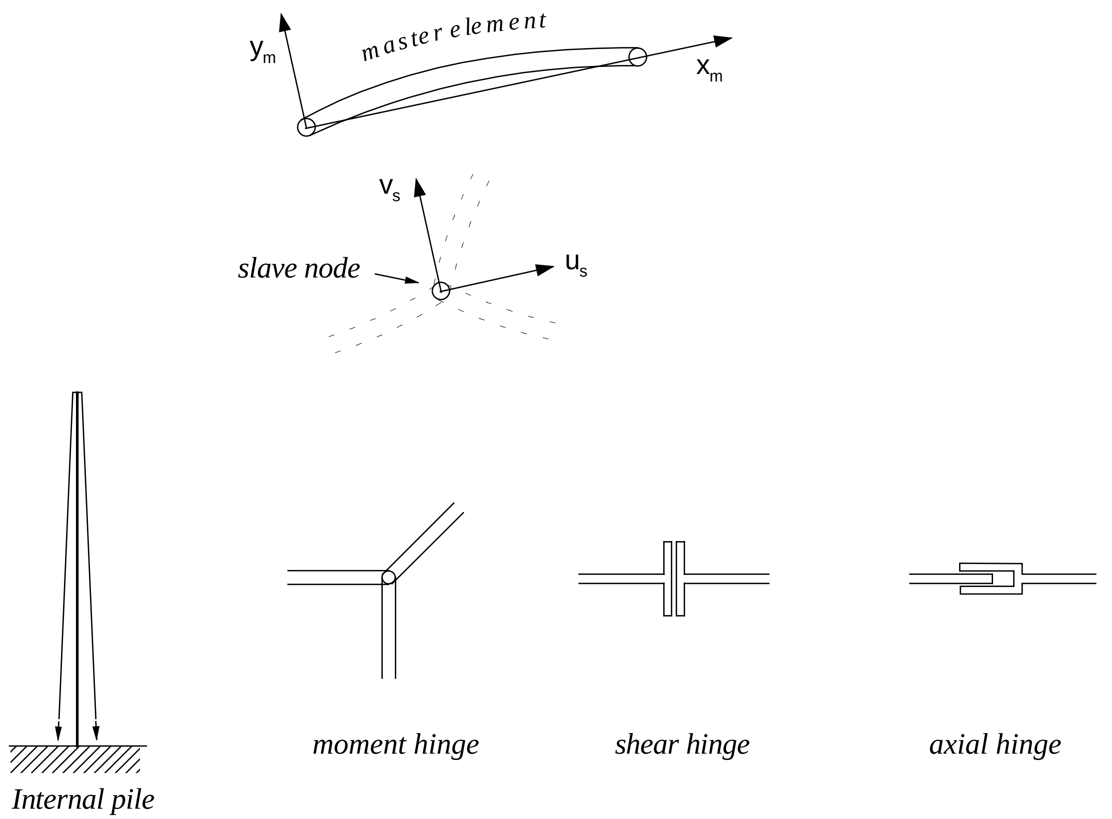


Figure 3.7Illustration of linear dependency option and its possibilities

Internal hinges can be modelled by specifying two nodes with identical coordinates at the hinge, couple the DOFs that are to be equal and let the remaining ones be free. See Figure 3.8. Then the slave DOFs will only be coupled to the master node located at the hinge.

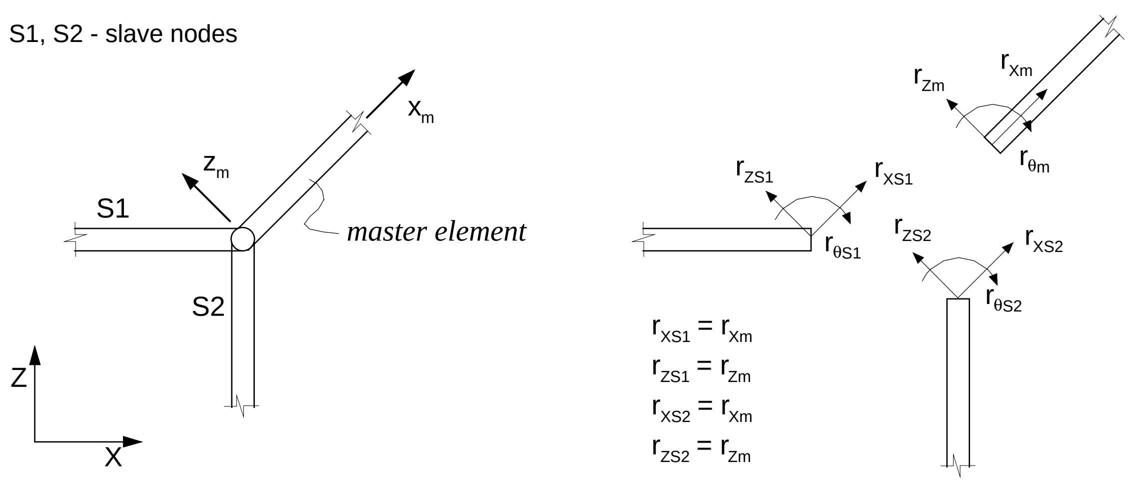


Figure 3.8Internal hinge described by doubly defined nodes and coupled degrees of freedom

The following restrictions should be observed:

- If the master element is a one-node element (spring to ground):
  The slave DOFs will be coupled to the corresponding spring dofs so that the displacements

are equal, irrespective of their relative location.

- If the master element has fixed DOFs:
  The slave DOFs are only coupled to the free master dofs, but with coupling coefficients as if all master DOFs were free. This might cause unexpected effects if the slave location is far off the master element.
- If the slave node has fixed DOFs:
  The DOFs are released and a warning is printed by the program.

3.9 PASSIVE ELEMENTS

Constructional elements may be defined as passive (or nonstructural) elements in the response analysis. Such elements do not contribute to the loadcarrying capacity of the structure. However, passive elements may be exposed to external loads.

This option may be specially useful in saving computational time (by reducing the numbers of equations to be solved) and for simulating fractured elements.

Passive element properties include:

- Element is by skipped in the structural stiffness calculation and system assembly process
- Distributed element loads enters the global load vector

Restrictions:

- At least one element meeting at a node has to be a structural elements
- Termal loads may not be associated to passive elements
- Master elements may not be considered passive

User input:

Structural elements are passfied by the *Nonstru* element list or element group command, refer Section 6.3 on the control file.

No additional input is required on the structure file.

3.10 INITIAL IMPERFECTIONS

Initial stress free deflections can be modelled for beam elements. Lateral deflection is specified by maximum offset, deflection shape and orientation. Three different imperfection shapes may be specified

- both ends rotated symmetrically
- end 1 fixed, end 2 rotated
- end 1 rotated, end 2 fixed

The direction of offset is given as angular rotation relative to local z-axis.

The specified initial imperfections affect the element elastic stiffness properties, and assume a stress-free condition.

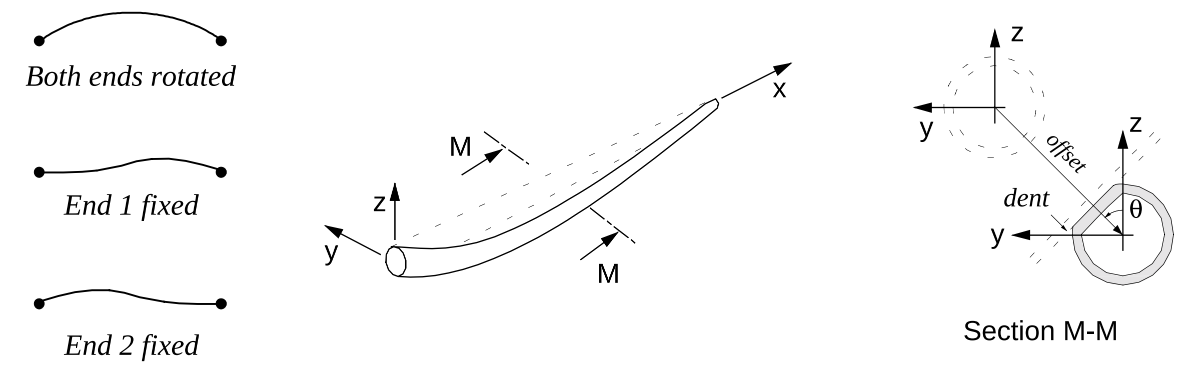


Figure 3.9Element imperfections and damages

3.11 DAMAGED TUBULAR MEMBERS WITH LOCAL BUCKLING

Behaviour of damaged tubular members may be modelled in USFOS. The formulation is designed to model damages due to impacts from supply vessels and dropped objects.

Modelling of damaged members generally includes:

- Lateral distortion of tube axis, refer Section 3.10
- Local denting/ovalization of the tube cross section

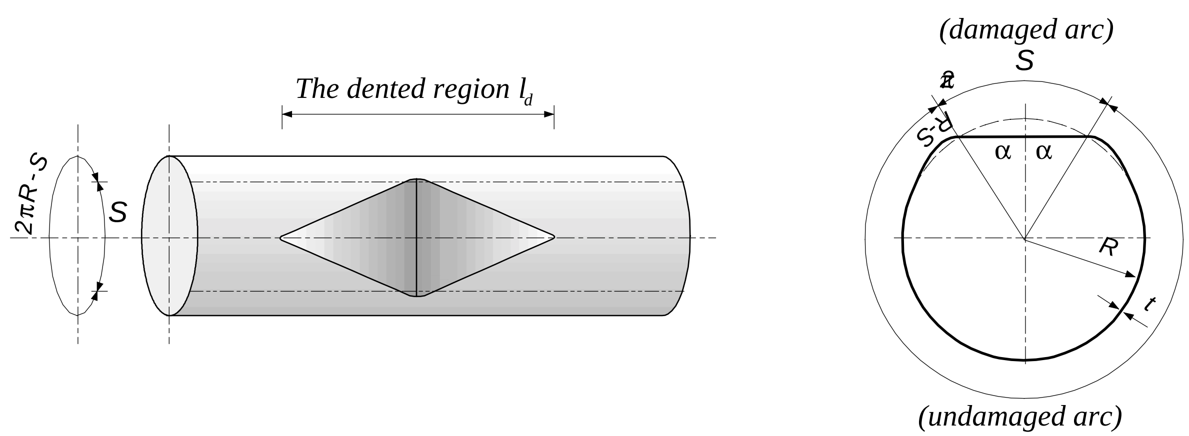


Figure 3.10Dent idealisation

The dented section is idealized as shown in Figure 3.10. The cross section consists of a dented part and an undamaged part. The load shared by the dented part is assumed to be limited by the force causing yielding at the middle of the dent. Further loading is carried by the undamaged part, alone.

The total capacity of the cross section is expressed by the plastic interaction curve between axial and bending moment, according to the dent depth and orientation. This is illustrated in Figure 3.11for a dent located on the compressive side.

In the post-collapse range the dent will grow as the load increases, especially for D/t-ratios exceeding 50. On the basis of experiments an empirical dent growth function has been established, see Figure 3.12.

The dent development and corresponding reduced cross-sectional plastic capacities, modify the elasto-plastic stiffness matrix.

Initially undamaged tubes will experience local buckling and subsequent growth of buckle in the post collapse range. This effect, which is significant for D/t > 40, is handled by the procedure described above.

Local damages are specified by the user in terms of dent depth and orientation relative to the local Z-axis as shown in Figure 3.10. The dent depth may be given individually for beam ends and midspan section. However, all dents along the element must have the same orientation and coincide with the orientation of the lateral deflection (Section 3.10).

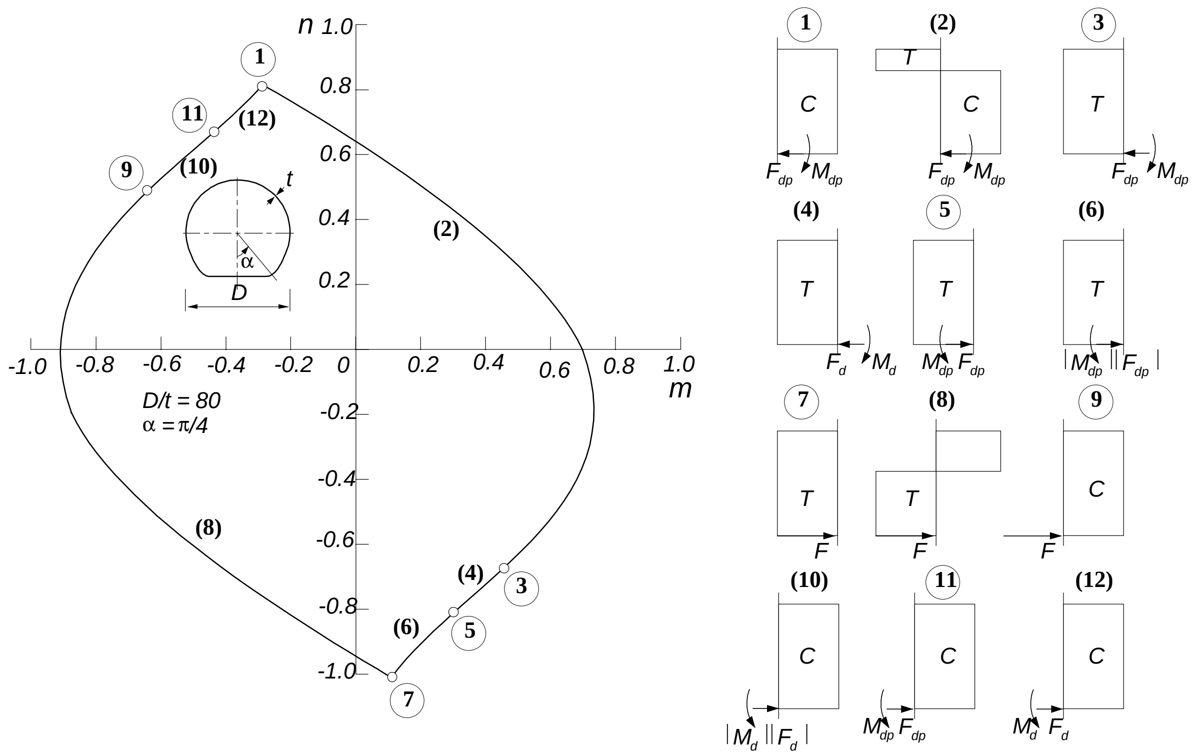


Figure 3.11Plastic interaction curve for dented section

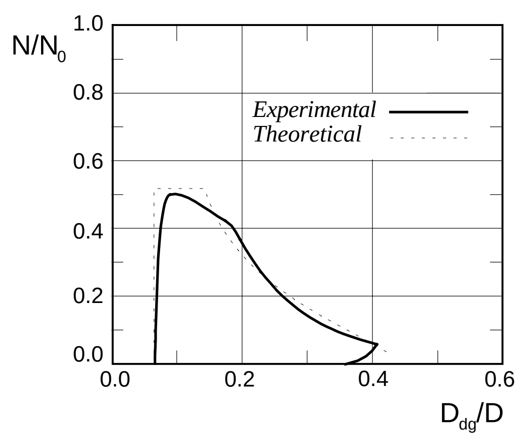


Figure 3.12Axial load versus dent growth

3.12 LOCAL BUCKLING OF RECTANGULAR SECTIONS

Local buckling and subsequently distortion of the cross sectional shape has a deterimental effect on the capacity. This phenomena may be taken into account in the collapse analysis as USFOS includes local buckling and post-buckling behaviour for rectangular sections.

Local buckling behaviour includes

- a buckling criterion
- calculation of reduced plastic capacities under local collapse
- modification of the elastoplastic stiffness matrix in accordance with the reduced cross sectional capacities

At the moment a plastic hinge is formed, the rectangular section is checked in order to decide whether local buckling occur. Buckling is forced to occur at element ends and/or midsection.

The local collapse model assumes buckling about one axis, as shown in Figure 3.13(b). The moment capacity about the buckling axis is reduced as a function of the plastic rotation as indicated in Figure 3.13 (c). The plastic moment capacity about the other axis is assumed independent of the local collapse. In a similar way the axial capacity is dependent of the hinge rotation, in fact a straight line reduction curve is adopted.

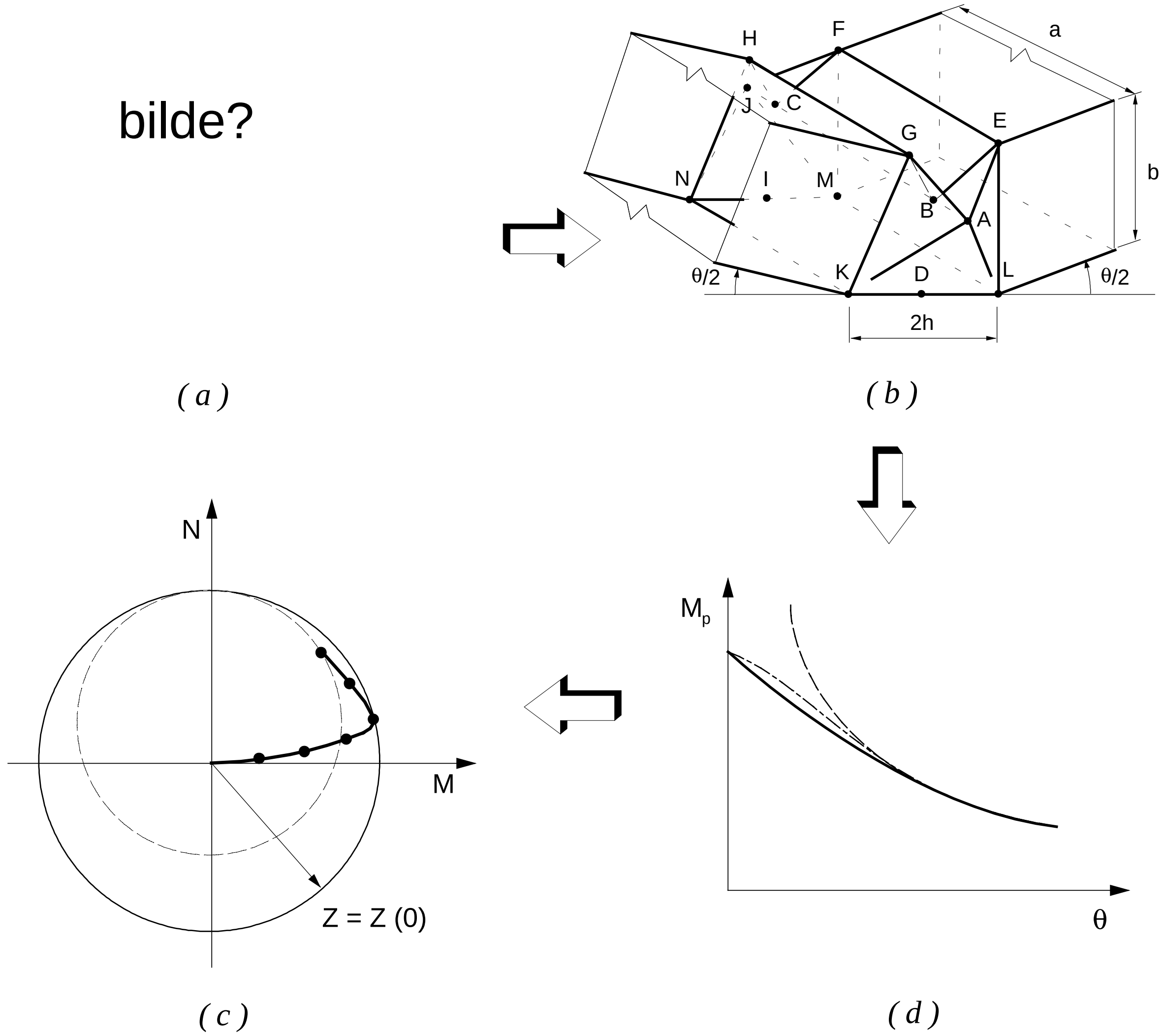


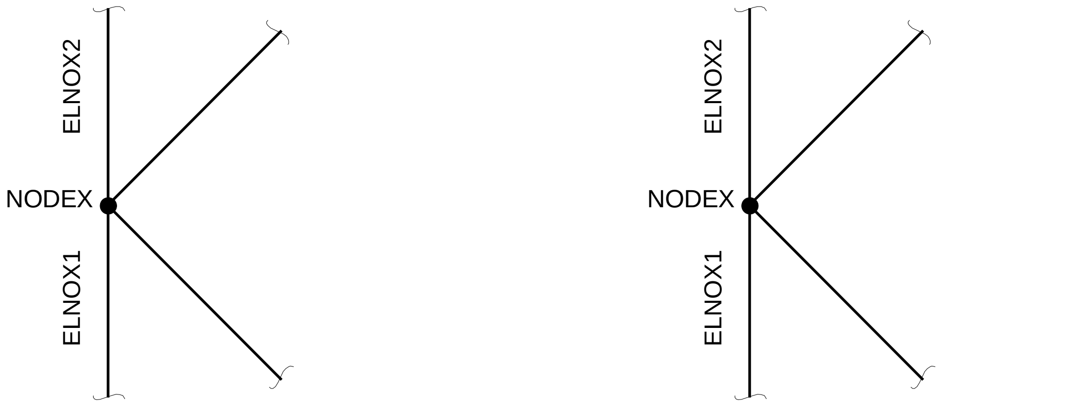
Figure 3.13Modelling of post-colapse behaviour of rectangular sections

The model is controlled by the user and may be excluded from the analysis, see Section 6.3.c.

3.13 LOCAL FLEXIBILITY OF TUBULAR JOINTS

Local flexibility can be considered for tubular joints. The user specifies the nodes where local shell effects should be included. USFOS then calculates the geometry of the tubular joints and introduces extra elements and nodal points in the finite element model.

Figure 3.14a describes a conventional joint model, and Figure 3.14b describes the model when local flexibility is included.



a) Conventional joint model *b) Conventional joint model*

Figure 3.14Joint flexibility modelling

For each brace/chord intersection, one extra node is introduced, (called "chord surface node"). The braces are then connected to the "chord center node" through the "shell property element".

Overlapping braces

For overlapping braces, the user gives the element numbers of the braces. The overlapping braces will be eccentically connected to the same "chord surface node", see Figure 3.15.

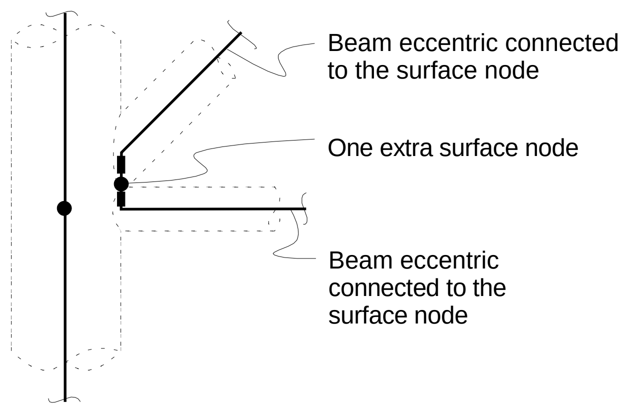


Figure 3.15Overlapping braces connected to a common "chord surface node"

Eccentricity

When the geometry of the tubular joint is calculated, any eccentricity defined by the user is taken into consideration.

The effect of the eccentricities will be included in the "shell property element", and the actual braces will be centrically connected to the "chord surface nodes", see Figure 3.16.

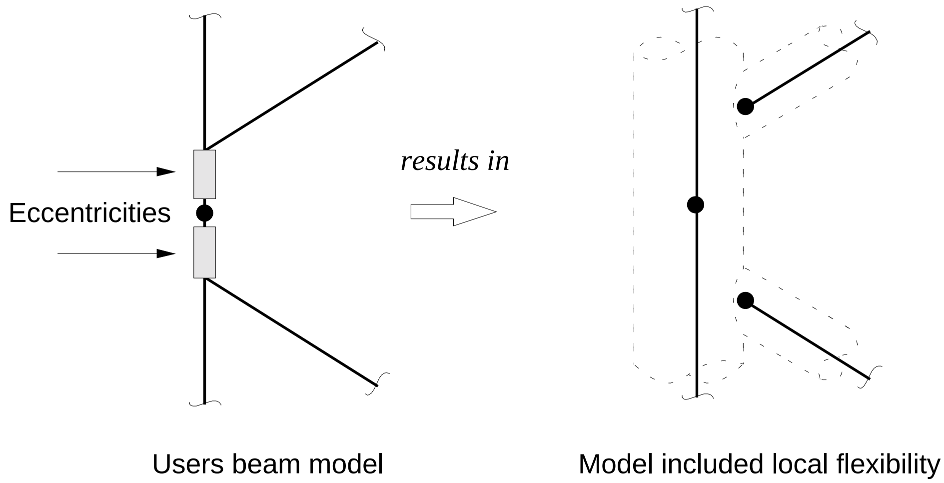


Figure 3.16Eccentricities are included in the shell property element

It is assumed that the brace's working lines hit the chord working line. The type of eccentricity described in Figure 3.17 causes error if local flexibility is introduced.

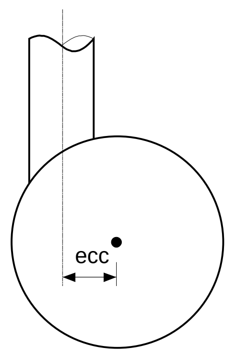


Figure 3.17Illegal eccentricity when local flexibility is introduced

3.14 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 convential 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 capacitites are calculated according to API.

Figure 3.18a describes the user defined finite element model of a tubular joint, and Figure 3.18b describes the "modified" input model.

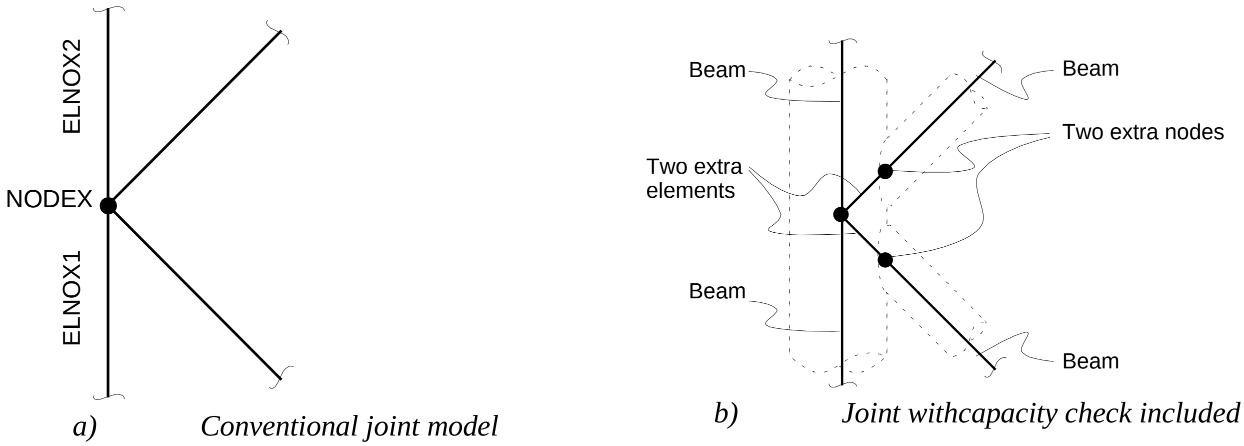


Figure 3.18Joint capacity modelling

The numbering of the extra nodes and elements are as follows, see Figure 3.19.

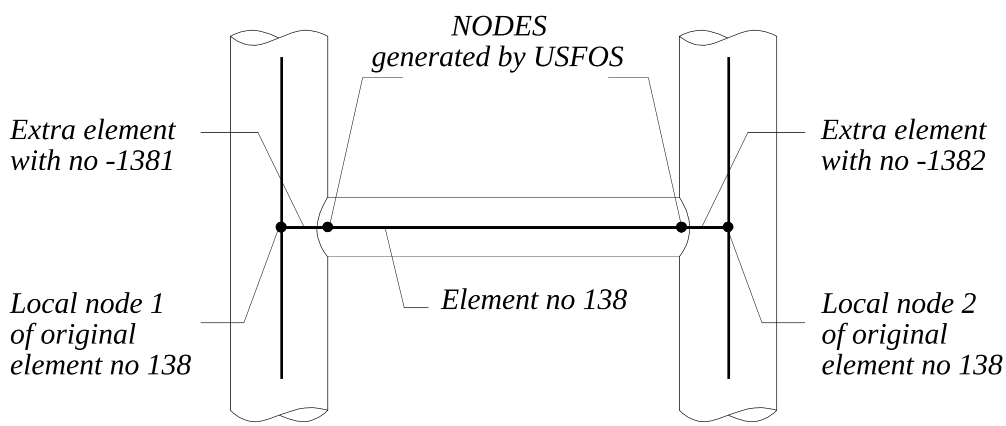


Figure 3.19Numbering of extra elements generated by USFOS

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

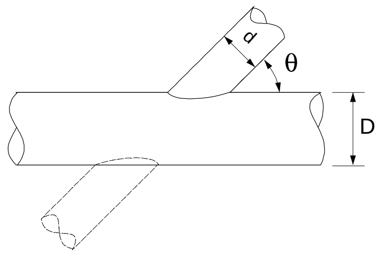
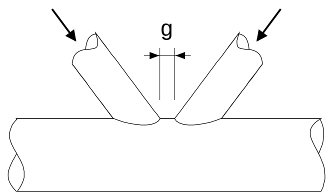
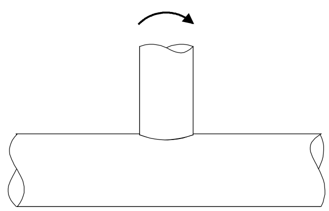
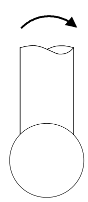
The material properties are set equal to the properties of the actual chord, but hardening is not permitted.

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, see Figure 3.20. The other cross sectional parameters are set equal to the ones for the actual brace.

Loading	Joint type	Capacity
Axial tension and compression		$P_u = (3.4 + 19\beta) \cdot \frac{\sigma_y T^2}{\sin\theta} Q_{f1}$
Axial tension and compression		$P_u = (3.4 + 19\beta) \cdot Q_g \frac{\sigma_y T^2}{\sin\theta} Q_{f1}$
In plane bending		$M_u = (3.4 + 19\beta) 0.8 \frac{\sigma_y T^2 d}{\sin\theta} Q_{f2}$
Out of plane bending		$M_u = (3.4 + 7\beta) \cdot 0.8 \cdot \frac{\sigma_y T^2 d}{\sin\theta} Q_{f3}$

D	:	Diameter of chord	$Q_{f1} = 1 - 0.030\ 0.5^2\ D/(2T)$
T	:	Thickness of chord	$Q_{f2} = 1 - 0.045\ 0.5^2\ D/(2T)$
d	:	Diameter of brace	$Q_{f3} = 1 - 0.021\ 0.5^2\ D/(2T)$
θ	:	Angle of inclination of brace	(Chord utilization of 0.5 is assumed)
g	:	Gap	
β	:	d/D	
Q_g	:	$(1.8 - 0.1\ g/T) \geq 1.0$ for $D/T \leq 40$	
	:	$(1.8 - 4.0\ g/D) \geq 1.0$ for $D/T > 40$	

Figure 3.20Joint capacity formulas (API)

Note that axial tension and compression capacity are set equal for X-joints.

With "*Joint Capacity Check*" included, the user defined brace shown in Figure 3.19. (with element no. 138) will then consist of up to 3 elements (a shortened original element + 2 stub elements).

The forces through this brace will be limited by the "weakest link" of the three elements.

In Figure 3.21 M-N interaction of the 3 elements are presented as an example. As indicated for an arbitrary force combination, the interaction curve which gives the lowest capacity is used.

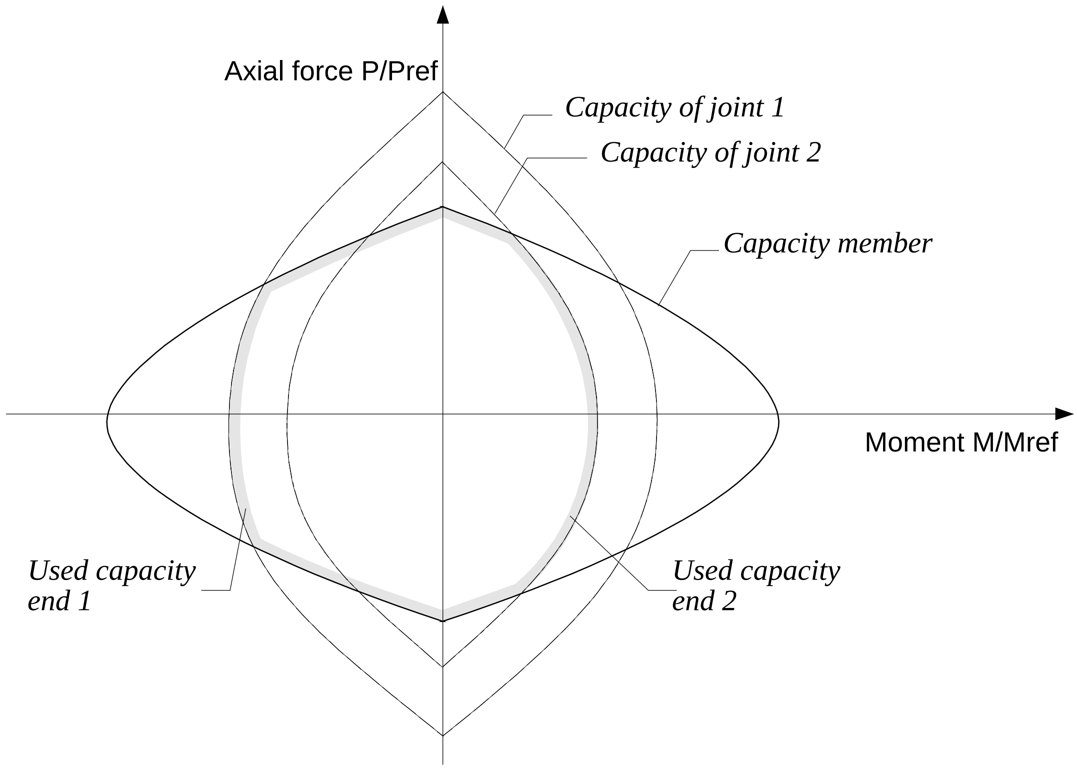
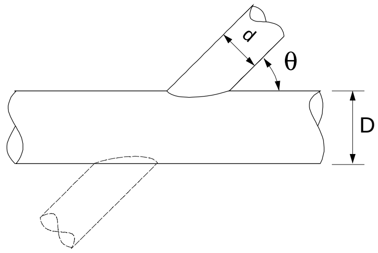
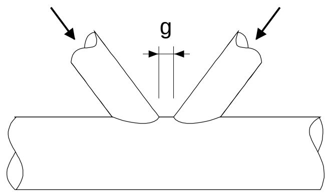
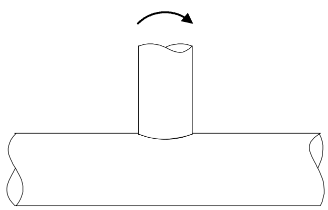
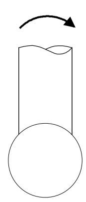


Figure 3.21Resulting capacity of brace with "JOINT CAPACITY CHECK" at both ends

In Figure 3.22 the capacity formulas according to DoE as used in USFOS are presented for the different joint types.

Loading	Joint type	Capacity
Axial tension and compression		$P_u = Q_u \cdot Q_f \cdot K_a \cdot \frac{\sigma_y T^2}{\sin\theta}$ $Q_u = (2.0 + 20\beta) \cdot \sqrt{Q_\beta}$ $Q_f = 1 - 1.63 \cdot \gamma \cdot 0.5^2 \cdot 0.030$ $K_a = (1 + 1/\sin\theta)/2$
Axial tension and compression		$P_u = Q_u \cdot Q_f \cdot K_a \cdot \frac{\sigma_y T^2}{\sin\theta}$ $Q_u = (2 + 20\beta) \cdot Q_g \sqrt{Q_\beta}$ $Q_f = 1 - 1.63 \cdot \gamma \cdot 0.5^2 \cdot 0.030$
In plane bending		$M_u = Q_u Q_F \cdot \frac{\sigma_y T^2 d}{\sin\theta}$ $Q_u = 5 \cdot \beta \cdot \sin\theta \cdot \sqrt{\gamma}$ $Q_f = 1 - 1.638 \cdot \gamma \cdot 0.5^2 \cdot 0.045$
Out of plane bending		$M_u = Q_u Q_f \cdot \frac{\sigma_y T^2 d}{\sin\theta}$ $Q_u = (1.6 + 7\beta) \cdot Q_\beta$ $Q_f = 1 - 1.638 \cdot \gamma \cdot 0.5^2 \cdot 0.021$

D	:	Diameter of chord		$Q_{f1} = 1 - 0.030\ 0.5^2\ D/(2T)$
T	:	Thickness of chord		$Q_{f2} = 1 - 0.045\ 0.5^2\ D/(2T)$
d	:	Diameter of brace		$Q_{f3} = 1 - 0.021\ 0.5^2\ D/(2T)$
θ	:	Angle of inclination of brace		(Chord utilization of 0.5 is assumed)
g	:	Gap		
γ	:	D/2T		
β	:	d/D		
Q_g	=	$1.7 - 0.9 \sqrt{g/d}, \geq 1.0$		
Q_β	=	1.0,	$\beta < 0.6$	
Q_β	=	$0.3 / ß(1-0.833ß)$,	$\beta > 0.6$	

Figure 3.22Joint capacity formulas (DoE)

3.15 ELEMENT FRACTURE

In most nonlinear analysis programs, the plastic tension strain in beam elements can be of "infinite" size. However, the forces in the elements can not exceed their plastic capacity limits, but there are no loss in loading capacity of the elements.

The "*Element Fracture*" option implies that all elements in the finite element model are checked for fracture. If the fracture criterion is violated, the loading capacity and stiffness of the element are set to zero, and the element forces are transferred to the neighbouring elements. The force redistribution is carried out by an internal loadcase.

If the structure is able to redistribute the forces in the fractured element(s), the (external) loading will continue. If not, the analysis will stop.

The fracture criterion adopted is the level 3 Crack Tip Opening Displacement (CTOD) criterion. The CTOD is calculated on the basis of the nominal strain, the corresponding stress (which may be raised due to hardening) and an assumed crack length. The value obtained is compared with the critical value.

At present, the model allows for calculation of the CTOD in the braces, **but not on the chord side of a joint**. This limitation should be kept in mind because tearing of the chord wall may be the actual failure mode.

The fracture criterion is completely governed by the magnitude of the crack length and the critical CTOD. It is the responsibility of the user to give adequate data.

The actual values depend strongly on the type of analysis and the condition of the structure. For example, typical defect sizes from manual arc welding during fabrication are 0.1 mm. However, standard inspection methods are normally not able to detect defects smaller than 3-4 mm. It is therefore reasonable to assume that cracks of such lengths are present. Even larger cracks due to gross errors can not be disregarded.

When a structure is put into operation, crack growth will occur. The crack length to assume in the CTOD assessment will then depend upon the age of the structure, the load history, the interval between inspection, the quality of inspection in terms of defect dectability etc. In fact, the defect size will be highly stochastic and the value adopted should reflect this uncertainty.

The critical CTOD value is often found to be in the range of 0.3-0.5 mm, but depends upon several factors, such as steel quality, material thickness, temperature, strain rate etc. Both large thickness, low temperature and high strain rate tend to lower the critical CTOD. For a discussion reference is made to the "Fatigue Handbook. Offshore Steel Structures", Tapir 1985.

The strain rate effect should especially be recognized when applying the criterion to collision situations.

In conclusion, no general value can be recommended for the critical CTOD value. It should be based upon tests of actual material, taken under representative conditions.

The calculation model presupposes that fracture takes place before the strain attains the ultimate strain. To comply with this the following condition should be fulfilled:

$$CTOD\text{-}crit < \pi\, \varepsilon_u \frac{\sigma_u}{\sigma_y} \cdot a \tag{3.1}$$

where ε_u - ultimate strain
σ_u - ultimate stress
σ_y - yield stress
a - defect size

Finally, it is emphasized that the fracture model is associated with considerable uncertainty. This applies to the calculation of strain as well as to the CTOD design equation. The empirical basis is obviously very limited, especially for large strains. However, compared with present practice in pushover analysis, where infinite ductility is implicitly assumed, it will always be conservative to use this option.

This is not always the case in ship collision analysis, where some kind of ductility limit should *always* be taken into account. For example, in the "Design Guidance for Offshore Steel Structures Exposed to Accidental Loads", issued by Det norske Veritas, the following constraints to the beam mode of deformations are recommended:

$$\frac{\delta}{d} = \frac{c}{2}\left[\sqrt{1} + 0.2\, \varepsilon_u \left(\frac{1}{cd}\right)^2 - 1 \right] \tag{3.2}$$

for axially fixed ends

$$\frac{\delta}{d} = \frac{1}{20}\, \varepsilon_u \left(\frac{1}{d}\right)^2 \tag{3.3}$$

for axially free ends

$c = \begin{cases} 2 \text{ rotationally fixed ends} \\ 1 \text{ for rotationally free ends} \end{cases}$
ε_u = ultimate strain
l = beam span
d = tube diameter
δ = beam deformation

The expressions have been derived from a different model and do not represent the "true" solution. However, they give indications of the amount of lateral displacements that can be tolerated and should be used to check the order of magnitude of the displacement at fracture implied by the USFOS calculation.

3.16 SHIP IMPACT ALGORITHM

The collision response of fixed offshore structures can be divided in the following deformation modes:

- Lcal deformation of the tube wall at the point of impact
- Bam deformation of the hit member
- Global deformation of the structure

In the initial stages of deformation, the response is governed by bending of the hit member, and by local denting under the load. The bending capacity of the member is reduced by the dent, and may be even further reduced if local buckling occurs at the member ends. As the beam undergoes finite deformations, the load carrying capacity may increase significantly due to development of membrane forces. The degree of membrane action depends, of course, on the axial and rotational restraints of the adjacent structure. Provided that the adjacent structure do not fail, the energy absorption is restricted either by excessive straining of the member or by joint failure.

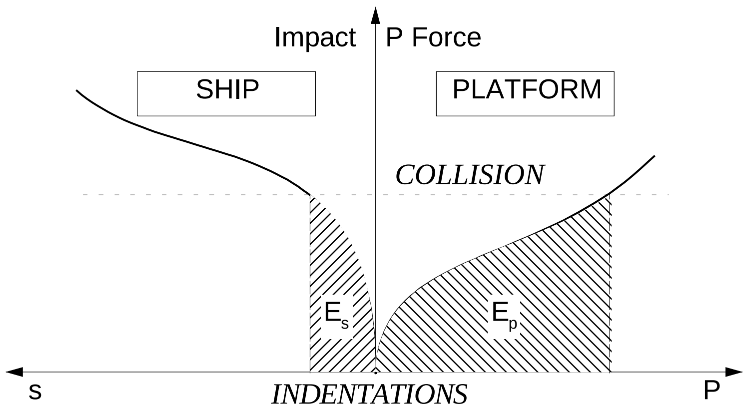


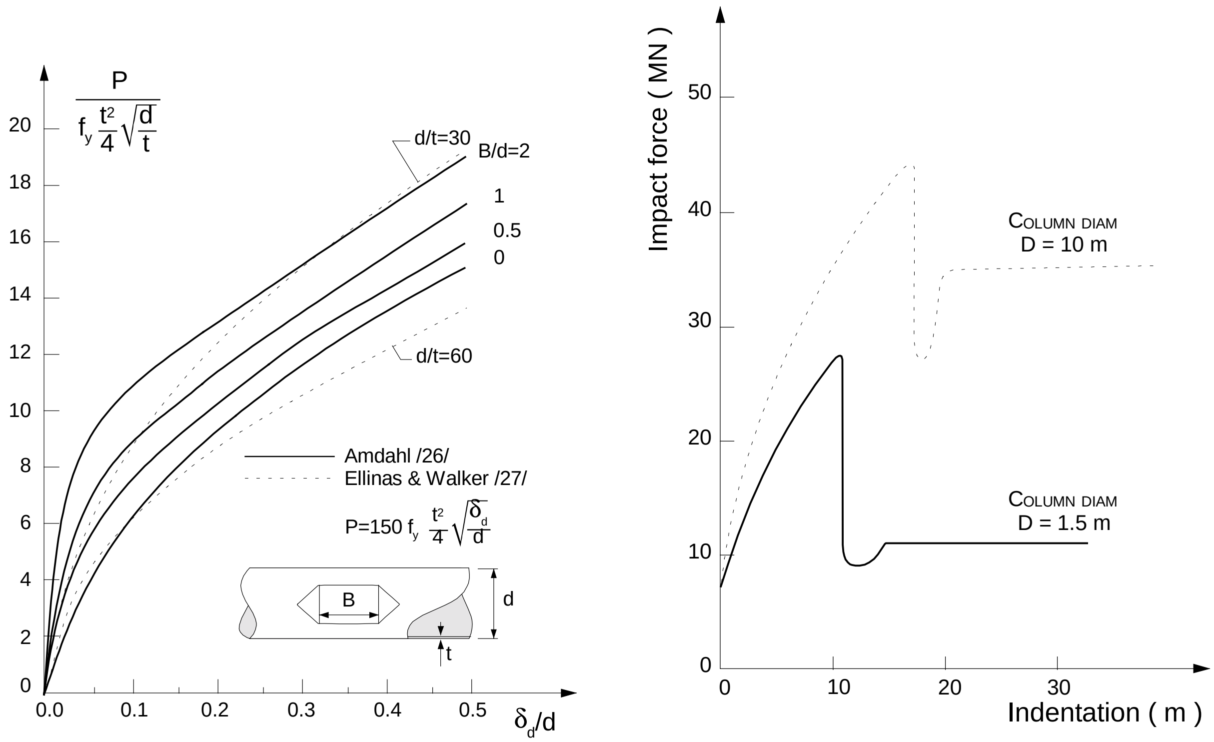
Figure 3.23Strain energy absorbtion

The impact analysis in USFOS is based on a defined impact energy, and a specified geometry of the impact (geometry of the ship and angle of impact). USFOS calculates the appropriate impact loads, and increments this load until the total impact energy has been dissipated as strain energy in the structure and the ship.

The impact force is set equal to the reference load for local indentation of the tube, or to 10 % of the mechanism load for the hit member, whichever is less.

Calculation of beam deformation and global deformation of the platform is included in the ordinary USFOS calculations. Local deformation of the tube and indentation in the ship side is implemented according to recommended curves by DnV, shown in Figure 3.24. Reduction of moment capacity in the dented member is included, as well as dent growth during impact.

The local tube wall deformations and the ship side indentation is calculated in each step. Thus, the total energy dissipation is determined, both from tube wall denting, member bending, global deformation of the platform and local indentation in the ship side.



a) *Load-deformation curves for ship indentation* b) *Load-deformation curves for tube denting*

Figure 3.24Load-deformation characteristics for ship impact

The load-deformation curve for ship indentation strictly applies to a 5000 tonnes vessel, but is not much different from the characteristics of 2500 tonnes ship. For bow and stern impact against braces it is customary to assume that the ship is infinitely stiff, with no energy absorption.

Both control of excessive member straining (fracture) and joint failure may be included in the USFOS analysis. (Section 3.14-3.15). However, the user should evaluate the input parameters for these checks carefully, to make sure they are representative for the particular structure.

3.17 DECK PLATING ELEMENT

A four node membrane element is available in USFOS.

The element has 2 degrees of freedom per node and should be used in combination with beam elements (to avoid zero stiffness terms). The local element coordinate system is described in Figure 3.25.

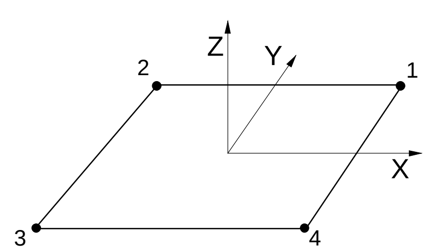


Figure 3.25Lcal coordinate system

The element connectivity and properties are defined through standarad SESAM input format (GELMNT1, GELREF1, GELTH, GECCEN).

Element surface pressure load defined through SESAM "BEUSLO" is treated as non conservative load, (the direction is defined by the current local Z-axis of the element).

3.18 STRUCTURE ANALYSIS

A simplified substructure approach is available in USFOS which is suitable for modelling of regions of a structural system with linear elastic stiffness properties,

A "premade" super element stiffness matrix is introduced to represent the substructure behaviour. This allows for a computationally efficient analysis as the number of equations to be solved in the step-by-step solution procedure may in many cases be reduced significantly.

The stiffness properties of the super-element have to be generated prior to the USFOS analysis, and no trace-back for the super-element internal nodes is available.

General Stiffless Matrix Input

The super element stiffness properties must be specified according to the General Matrix Element format defined within the SESAM system.

This element type (ELTYP=70) generally includes premade reduced stiffness matrices and corresponding nodal load vectors and may have an arbitrary number of nodes, see Figure 3.26. In the USFOS implementation the number of dofs per node is fixed to 6.

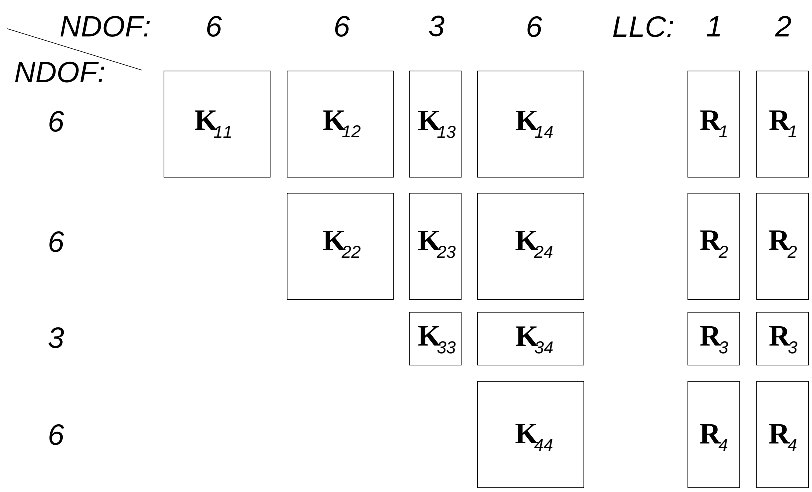


Figure 3.26Example of four node stiffness matrix and corresponding load vectors

The relevant input records for the general matrix element is shown in Figure 3.27.

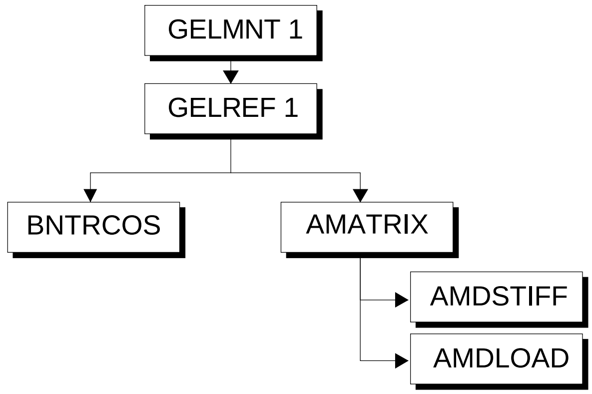


Figure 3.27Data records for general stiffness matrix element

Reduced Substructure Stiffness Matrix

The reduced stiffness matrix may be calculated for a specified structural system by USFOS as a separate task prior to the structural response analysis. The formulation is limited to the linear elastic case.

In the substructure analysis it is assumed that the unknown dofs $\mathbf{r}_f$ are subdivided into internal $\mathbf{r}_i$ and external or related DOFs $\mathbf{r}_e$:

$$r_f = \begin{bmatrix} r_i \\ r_e \end{bmatrix} \tag{3.4}$$

The assembled substructure stiffness relationship may correspondingly be written as:

$$\begin{bmatrix} K_{ii} & K_{ei} \\ K_{ie}^T & K_{ee} \end{bmatrix} \begin{bmatrix} r_i \\ r_e \end{bmatrix} = \begin{bmatrix} R_i \\ R_e \end{bmatrix} \tag{3.5}$$

Elemination of internal DOFs $\mathbf{r}_i$ (static condensation) gives:

$$\left(K_{ee} - K_{ie}^T K_{ii}^{-1} K_{ie}\right) r_e = R_e - K_{ie}^T K_{ii}^{-1} R_i \tag{3.6}$$

The superelement stiffness matrix $\mathbf{K}_s$ is then defined as:

$$K_s = K_{ee} - K_{ie}^T K_{ii}^{-1} K_{ie} \tag{3.7}$$

3.19 EXTERNAL HYDROSTATIC PRESSURE

An option is included in USFOS to account for large hydrostatic pressures on the capacity of tubular beam elements. Such a load situation may occur for structural components such as bracing members of deep sea offshore platforms in which water is sealed off.

The external pressure excerted by water introduces compressive stresses in the circumferential direction. This action reduces both the cross sectional plastic capacities as well resistance to local wall buckling.

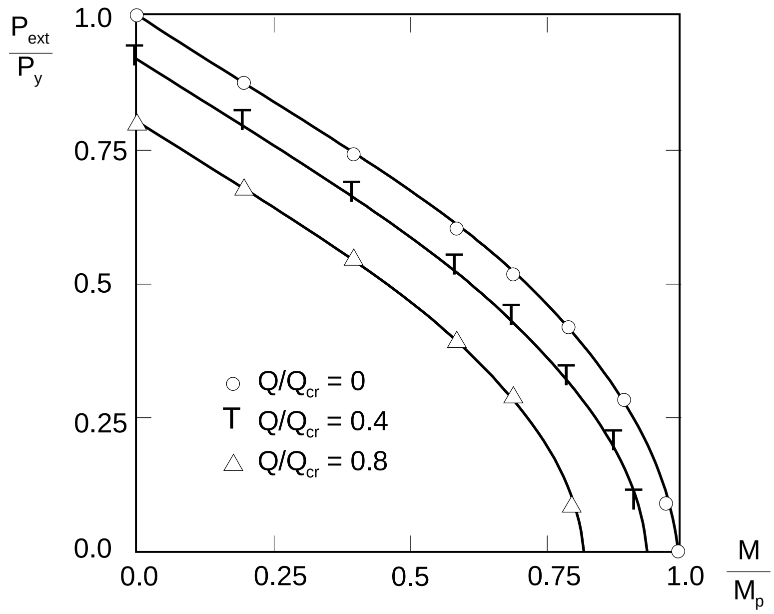


Figure 3.28Interaction curve for tube section exposed to external pressure

Furthermore, initial and subsequent ovalization of the tube section will introduce bending moments in the tube wall. Taking these factors into consideration, it may be shown that the cross-sectional capacity depends on the external pressure as shown in Figure 3.28. It is observed that with an external pressure equal to 80 % of the local elastic buckling pressure, the cross sectional capacity is reduced by approximately 20 %. It is observed that the shape of the plastic interaction surface is maintained.

In the formulation implemented in USFOS, it is chosen to use the cross section interaction curve given in Eq 3.33 in the USFOS Theory Manual also for the case with external pressure acting on the element.

The tube section axial and bending moment plastic capacities are given as a function of the external hydrostatic pressure in Figure 3.29.

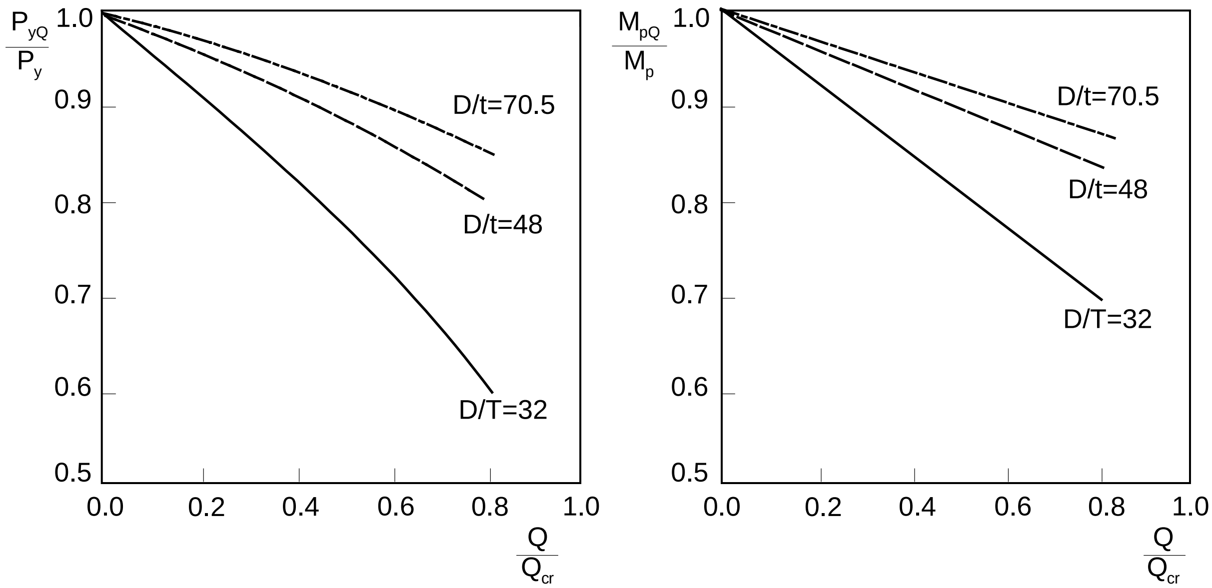


Figure 3.29Axial and bending moment plastic capacities as function of external hydrostatic pressure

The user identifies (in the input) which elements are to be exposed to hydrostatic pressure. On the basis of a specified sea-surface level, USFOS calculates the hydrostatic pressure on the relevant element as a preprocessing task.

The hydrostatic pressure also affects the behaviour of dented tubular sections due to the fact that the plastic axial and bending moment capacities, entering the dent formulation, also are calculated according to Figure 3.29.

It should, however, be noticed that the accellerating effect of the hydrostatic pressure on the local dent growth is **not** realistically modelled. This implies that USFOS may predict the local dent growth unconservatively for the case when the tube section is exposed to external hydrostatic pressure.

3.20 DYNCAMIC ANALYSIS

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 α=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:

$$
\begin{aligned}
&-\frac{1}{3} < \alpha < 0 \\
&\gamma = \frac{1}{2}(1 - 2\alpha) \\
&\beta = \frac{1}{4}(1 - \alpha)^2
\end{aligned}
\tag{3.8}
$$

where β and γ are the free parameters in the Newmarck-β method. Generally α=-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. Probably, the predictor-corrector method is favoured from an accuracy/economy point of view.

Input alternative 1:

Time-dependent loads are prescribed by the CDYNAMIC record. The rate of loading is for a typical load combination (or line in the CDYNAMIC record) given by the ratio *lfact*/Δt. Here *lfact* is the load increment factor and Δt is the time increment which is used in the numerical integration of the equation of motion. The time increment Δt may be modified from one line to the next within the CDYNAMIC record.

Figure 3.30 illustrates the specification of time dependent loads in USFOS.

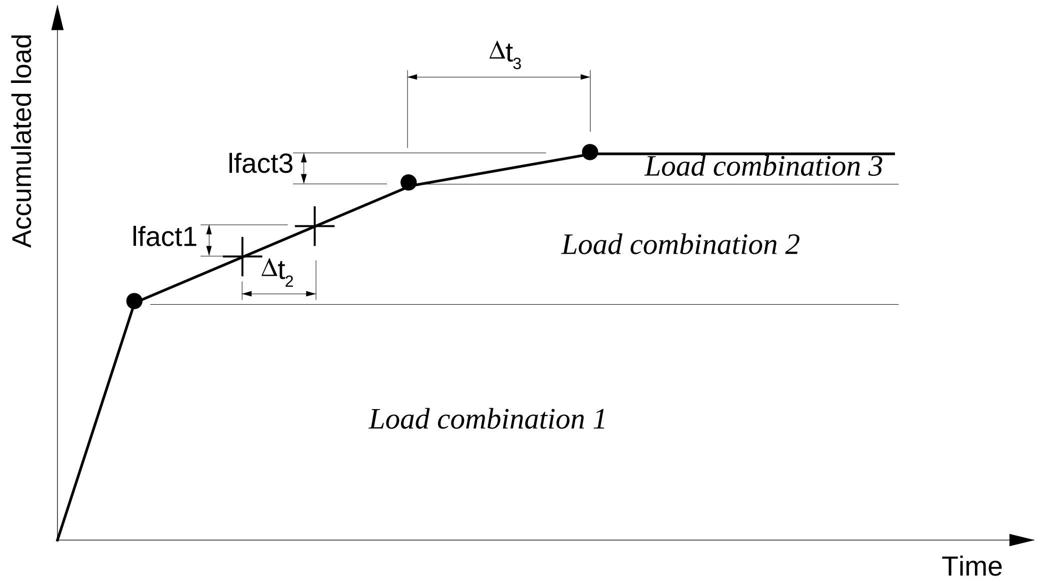


Figure 3.30Time dependent loading

Input alternative 2:

From **version 7.0** of USFOS an alternative input option for dynamic analysis is available.

According to this 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 3.31.

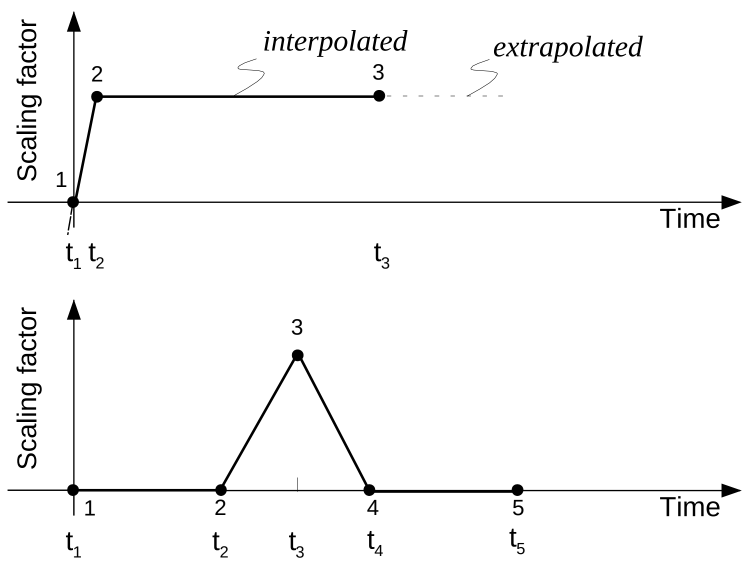


Figure 3.31Time history examples

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 3.32.
TIMEHIST : Defines a time history identified by an ID an described by discrete points.
LOADHIST : Defines a load history by connecting a loadvector to a time history.

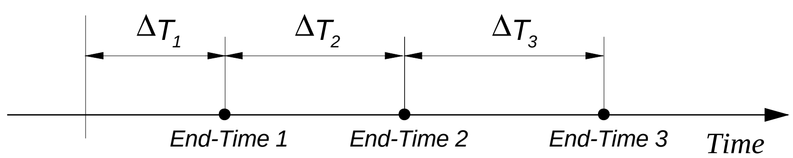


Figure 3.32Specification of time increment to be used with the different time interval