2 PROGRAM CONCEPTS

This chapter presents the basic concepts behind USFOS, and gives a short presentation of the USFOS program system.

2.1 BASIC FEATURES

USFOS is a finite element program for non-linear static and dynamic analysis of frame structures. The structure may be exposed to external loads, acceleration fields or temperature fields.

The program is specially developed for progressive collapse analysis of space frames.

- The basic philosophy behind USFOS is to use a very coarse finite element mesh, and still obtain reliable and accurate results. USFOS requires only one finite element per physical element of the structure. A structural model for linear analysis may be used directly in the USFOS non-linear analysis.
- USFOS operates on element stress resultants, i.e. forces and moments. Material non-linearities are modelled by plastic hinges at element mid span and at element ends.
- The basic element formulation of USFOS is based on the exact solution of the differential equation for a beam subjected to end forces.

In addition, non-linear spring elements are available.

Figure 2.1USFOS basic concepts

Effects of large displacements and coupling between lateral deflection and axial strain are included by using non-linear strain relations (Green strain) instead of conventional linear strain (engineering strain). This gives a very accurate representation of the element behaviour, including membrane effects and column buckling.

The tangent stiffness matrices are derived in a consistent manner from energy principles. This preserves symmetry in the equations, and allows for the use of an efficient "skyline" equation solver.

The elastic tangent stiffness matrices are calculated from closed form expressions, with no numerical integration over the element cross section or over the element length. This gives a very efficient formulation, and reduces the time consumption considerably.

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.

When plastic hinges are introduced the element tangent stiffness matrices are modified according to plastic flow theory, formulated for stress resultants.

The element forces remain on the yield surface, unless the element is unloaded and returns to the elastic state.

The load is applied incrementally. The size of the increments may be varied along the deformation path, i.e. large steps in the linear range, and smaller steps with increasingly non-linear behaviour.

If the forces of some element cross section exceed the yield surface, the load increment is automatically scaled to make the forces comply "exactly" with the yield surface.

The load increment is automatically reversed if global instability is detected.

The effect of initial deformations is included for beam elements

The effect of local buckling of rectangular sections is included. The plastic cross sectional capacity is reduced as the section shape degenerates during plastic hinge rotation.

The effects of lateral distortion, local dents and local buckling are included for tubular members. The plastic yield surfaces of the damaged sections are modified according to size and orientation of the damage. No finite element modelling of the damaged member is needed.

The effects of local flexibility of tubular joints are included. A complete shell analysis of each selected joint is included in the analysis. The shell properties of the joints are calculated by USFOS and are automatically introduced in the finite element model. No manual finite element modelling is required.

Joint capacity checks and plastic joint behaviour is implemented according to the API rules and the DoE rules. In addition, the user may specify both the capacities of each brace/chord connection and the capacity surface.

Fracture criteria are implemented according to a level 3 CTOD criterion, applicable to large deformations. In addition, it is possible for the user to specify an element to become 'fractured' after a restart.

Member rupture, and redistribution of forces from the ruptured element is fully integrated in the analysis procedure.

An integrated algorithm for ship impact analysis is implemented, accounting for

- Local deformation of the tube wall at the point of impact
- Beam deformation of the hit member
- Global deformation of the platform

Impact forces are calculated by the program, and incremented until the full impact energy is dissipated. The impact forces are unloaded, and residual forces and permanent deformations are stored for subsequent residual strength analyses.

A four noded plate element is available for modelling of in-plane deck stiffness.

Passive (non-structural) elements are available for modelling of distributed loads on components that do not contribute to the load carrying capacity of the structure. Such elements are excluded form the step-by-step solution process.

Restart of old analyses is possible at any load step. This makes it easy to control the non-linear analysis and to adjust the load specification to the non-linear characteristics of the structure.

It is possible to suppress the contribution of any USFOS element until a specified load case is reached. The element is then activated and will contribute to the global stiffness.

The effect of external hydrostatic pressure on the plastic capacities of tubular sections is accounted for.

Parts of a structural system that behaves completely elastically may be efficiently modelled by use of reduced super element stiffness matrices with a user-specified number of nodes.

Eigenvalue analysis may be performed in order to calculate elastic buckling loads and buckling modes for the static case as well as vibration frequencies and corresponding vibration modes for the dynamic case. The results may be visualised by use of the graphic presentation module XFOS.

Non-linear time-domain dynamic analysis may be performed with special reference to the dynamic ship collision problem. The formulation includes lumped or consistent structural mass, hydrodynamic added mass and structural damping. The numerical time integration, based on the HHT-alpha method, includes high frequency numerical damping.

2.2 PROGRAM OVERVIEW

The USFOS analysis system consists of three program modules, and a number of system files.

- The USFOS analysis module performs all numerical calculations, and generates two files of analysis data. One file is a print file containing general analysis results; the other file is a data file containing structure data, analysis results and restart data. This file is used for restart of analyses, and as a result database for POSTFOS. In addition, global analysis results are logged on terminal or batch-output device.
- POSTFOS is an interactive presentation module, designed to give easy access to the results of the USFOS non-linear analyses. The program is command oriented, with extensive built-in HELP functions, /8/. POSTFOS generates print files of selected analysis results, files of plot data and files of deformed geometry data.
- XFOS is an interactive system for visualisation and presentation of USFOS analysis results. The system is implemented in a standard UNIX environment and the user interface is based on the Motif graphical user interface. Three-dimensional pictures of the analysed 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.

User input to USFOS is read from symbolic files.

All control parameters for the non-linear analysis are specified 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 labelled "Structure File" and "Load file".

Figure 2.2The USFOS analysis module

Since USFOS requires only one element per physical element, a structural model developed for LINEAR analyses may be used directly in the USFOS non-linear analysis. Little extra input is needed, and it is a minor task to adapt USFOS to standard structural analysis systems. All control parameters and additional input for non-linear analysis may be specified separately, in the Analysis Control File.

USFOS is currently adapted to the SESAM'80 system. Interfaces to other program systems are developed separately.

Figure 2.3The XFOS and POSTFOS result presentation system

Table 2.1System files

System file	*Type*	*Content*
ANALYSIS CONTROL DATA	FEM	Control parameters for the USFOS non-linear analysis. From one to ten data records. Generated manually. Formatted read only.
STRUCTURE MODEL FILE	FEM	Finite element idealisation of the structure. Generated manually or by pre-processor program. Formatted read only.
LOAD FILE	FEM	Structural loads. Generated manually or by load generation program. Formatted read only.
ANALYSIS PRINT FILE	OUT	Print of analysis results: Input verification, global history output or output at each load step. Formatted write only.
ANALYSIS DATA FILE	RAF	Structure data, analysis results and restart data at each load step. Unformatted random access read and write.