FS2000

Advanced Structural Analysis
for Windows
(c) A.E.S. Ltd 1988,2025

Email:support@aes-uk.com

Table of Contents

Using the Help System

The FS2000 help system can be activated using the Help menu command or pressing the F1, when the FS2000 window has focus.

From the Help Menu Command

If the help file is activated from the menu bar Help command the Help Contents will be displayed in a separate window. The contents list can be used to negotiate through the various help topics. Links also provide the option to jump to specific topics.

Context Sensitive

The FS2000 Help system provides a context sensitive Help.

If the F1 key is pressed the help file will be activated in a context sensitive mode. In context sensitive mode the help topic displayed will be related to the menu or input form that is currently active. To make a menu command or input box active, simply highlight it.

The forward and backward buttons may be used to move within the Help file. When using context sensitive help it is sometimes useful to move backwards to the start of the current section

Search-Index

Use the search index to find topics relating to keywords

Printing Topics

The print button is useful for sending frequently accessed topics of the Help to the default printer. Command line instructions and Batch commands switches are typical topics that are likely to be printed out.

-o-

1 Getting Started

1.1 Documentation

This Help file is the primary documentation for the FS2000 core module. The optional FS2000 modules each have their own Help files.

Printable documentation for FS2000, in PDF format, is located in the Manuals folder on the CD . They will also be located in the Manuals sub-folder located in the (Program Files)/FS2000 folder.

FS2000.pdf is the documentation for the core module i.e. the FS2000 Help file in pdf format.

Section 2 describes what the FS2000 installation files (setup.exe) will do.

Documentation relating the DesNet server software is located on the CD in the DK2NetSvr folder.

-o-

1.2 Getting Started

Recommended Reading

It is suggested that Sections 3 and Sections 4 be reviewed.

Simple Model

The procedural example in Section 11 provides a basic introduction to the use of FS2000. The use of this simple illustrative example will show how to undertake the basic tasks necessary to complete an analysis.

Remember to press the F1 key to get the input instructions for each dialogue box.

Instruction Cards

The Help system contains **Instruction Cards** that demonstrate some basic operational features by displaying Instruction Cards for various analysis related tasks. These cards will contain basic instructions that the user should follow. The cards are not linked to the operation of the program, it is up to the user to complete the instructions and move onto the next card.

-o-

2 Computer Set-up and Analysis Capacity

2.1 What Set-up will do

FS2000 Directories

Apart from model directories FS2000 uses two basic directories:

- FS2000 System Directory
- FS2000 User Directory

The FS2000 System directory would normally be located in the Window's Program Files directory. This can be a read only directory.

The FS2000 User directory can be located anywhere. The set-up default is C:\FS2000. The local user must have full access to this directory. This is specified at set-up but not created.

What Set-up will do.

The Setup.exe program will:

- Install FS2000 system files in a directory of choice (C:\Program File\FS2000)
- Create a program group for the FS2000 program modules in the Windows start menu.
- Register the specified location of the FS2000 user directory (C:\FS2000)
- Install the DESKey drivers by running dk2wn32.exe.

The FS2000 directory it will have the following sub-directories.

System	System files for FS2000
Examples	Validation Examples
Props	Property Libraries (duplicates of those in the FS2000 User directory)
Manuals	Documentation in pdf format

When a local user first uses FS2000 it will detect that the FS2000 User directory does not exist. This will be created at the location specified during set-up. This can be changed using the FS_User_INI.EXE utility.

-o-

2.2 FS2000 License Control

FS2000 uses a DESKey (DK2) dongle system to control the use of the program.

On a single license FS2000 set-up, the dongle is generally fitted to the local machine running FS2000.

On a network set-up, the dongle can be used to authorise the running of FS2000 on a number of machines, subject to the concurrency user limit set by the dongle. Where FS2000 is used on a PC on network the DESKey may be fitted to any PC on that network. Any other PC, which has network access to this PC, the DESNet Server, will be able to run FS2000 using the DESKey authorisation.

All PCs running FS2000, require to have DESKey drivers loaded. The DESKey icon for the configuration utility is added to the Control Panel. The DESkey configuration utility can be used to enable local or network access.

The machine acting as the DESNet server requires the DESNet Network Server software to be loaded and running, FS2000 does not have to be installed on the machine running the DESNet Server, but can be.

-o-

2.3 Local Users - FS2000 Directories

When a new user uses FS2000 on a machine on FS2000 has been previously used the user will share the existing FS2000 User directory.

If required, each local user can have their own FS2000 User directory. These directories are created by running the **FS_User_INI.EXE** utility which is located in the FS2000\System directory (default at setup is C:\Program Files\FS2000\System).

This utility can also be used to change the registered location registry of the both the FS2000 Local User directory and the FS2000 System Directory. Note that the utility does not create the FS2000 System Directory it only registers it location.

-o-

2.4 Model Size Limits

The default maximum size model parameters are given below. These limits can be increased by modifying the FS2000's **CONFIG.INI** file. See **Resetting the Limits** below.

Analysis Modules

The **3-D Standard (Front) Solution** employs a frontal 'out of core' solution technique. Model memory allocation is dynamic with the exception of the front width. The default frontwidth is 1500 (degrees of freedom). See below for resetting this limit.
Note that the maximum ID number for linear solutions this is 9999.

3-D Non-Linear (Band) and DyNoFlex (Band) Solutions employ banded in-core profile solvers. Model memory allocation is dynamic with the exception of the number of matrix elements below the profile. The default number 1E6. See below for resetting this limit.
Note that the maximum ID number for non-linear solutions is 999.

Re-sequencing the model reduces the model memory requirements.

Load Cases Limits/Groups etc 500

This sets the maximum limits for Load Cases, Load case Combinations, Results SETs, Group SETs and Group Descriptions.

Note that the maximum ID number for non-linear solutions is 999. For linear solutions this is 9999.

Maximum Loads 5000

This sets the maximum loads that can be defined in a load case. It applies to Nodal loads, Mid-span element concentrated loads, Mid-span distributed loads, and Prescribed displacements. The limit is applied individually to each load type not collectively.

Resetting the Limits

The **CONFIG.INI** file in the FS2000 User directory can be used to increase the above limits. Note that these limits do not change the node and element limits of the solvers. This INI file is generally used to increase the Max Loads and Max Load Cases capacity of FS2000.

The maximum number of Restrained Nodes is Max Nodes / 2.

The Max Loads define the maximum number of nodal (NF) or non-uniformly distributed element loads (ED or EP) that can be defined in a load case or when load cases are combined. UDL element loads are also load case limited but not when used in combinations.

The file format is given below:

Max Loads, 5000

Max Elements, 15000

Max Nodes, 15000

Max Properties, 300

Max Sprg/Couples, 5000

Max Load Cases/Groups etc, 500

Solution Limits - Frontal Solvers

Frontwidth limit. This limit can be changed by creating the text file **FSFRONT.INI** in the FS2000 User directory. A single number entry in the file will re-define the width. It is desirable to keep this number as small as possible (speed and memory). If it is increased for specific models it is recommended that it be reduced to the default 1500 after large model use, it will run faster with a lower maximum front.

Solution Limits - Banded Solvers

Profile Matrix limit. This limit can be changed by creating the text file **FSPROF.INI** in the FS2000 User directory. A single number entry in the file will re-define the capacity. It is desirable to keep this number as small as possible (speed and memory). If it is increased for specific models it is recommended that it be reduced to the default 1E6 after large model use.

Solution Limits - Eigen Frequency-Buckling (Modal) Banded Solvers

Profile Matrix limit. This limit can be changed by creating the text file **FSPROFV.INI** in the FS2000 User directory. A single number entry in the file will re-define the capacity. It is desirable to keep this number as small as possible (speed and memory). If it is increased for specific models it is recommended that it be reduced to the default 1E6 after large model use.

-o-

2.5 Multiple Monitors

The FS2000 GUI should preferably be used in the primary Windows Display Monitor.

It can be operated in a secondary display monitor but in such a case the following type of behaviour may happen.

- Some input forms and dialogue boxes will always appear in the primary display.
- It will not be possible to re-open some definition input forms after they are initially closed (possibly).

FS2000 must be shut down and restarted to re-enable any forms that cannot be re-opened.

-o-

3 Overview of Basic Operation

3.1 New Models/Opening Models

New Models

Before a model can be generated, it is first necessary to initiate the model i.e. create the model workspace. The workspace is simply the Windows folder where the various model files used by FS2000 are located. To create the workspace for a new model, use the New Model command in the File menu. To create the workspace the user is required to select an existing Windows folder and define a new model name. The model name must be unique for the selected model folder.

It is good practice to create a folder or sub-folder model for each model.

It is recommended that models should always be created in a local drive and not a network drive. This ensures that the processing times are maximized. Always archive to a secondary back-up drive. e.g. a network drive.

Opening Models

The Open command of the File menu is used to select the current default model from existing models or the retrieve a model from an Archive file. A Standard model is a model for which a workspace currently exists. An Archive files is a single file that contains all definition data relating to the model.

Selecting Standard Model or Archive Models (MOD Files) from the List of File Types selects the appropriate model option.

- Standard models are opened by selecting files that have a XYZ file extension. XYZ file are only used to identify the model and are one of a number of necessary model binary files that should be edited by the user.
- Archive models are opened by selecting files that have a MOD file extension.

Deleting Models

Only delete models using the Delete command in the File menu. The archive file (<model>.MOD) will not be deleted when using the Delete command.

Right Hand Mouse Menu - File Form

Do not use the Right Hand Mouse Menu on XYZ files as this will corrupt the model. XYZ file are only used to identify the model and are one of a number of necessary model binary files and should not be edited or deleted by the user.

-o-

3.2 Archiving Models

Archiving merges all model definition data into a single file. Archive files have the file extension .MOD. A model is archived using the Save(Archive) command (Primary TASK:File menu).

Archive models are opened using the File menu Open command. By electing Archive Models (MOD Files) from the List of File Types, the MOD file can be selected.

All model definition data from all the optional FS2000 modules are included in the archive file. Results data is not included in the archive file.

The main uses of the archive file is for:

- Providing a single model back-up file
- Copying models to other users
- Reducing disc space

If an archive file exists in the model folder it will NOT be deleted when the model is deleted.

To archive a model the user is required to select an existing folder, usually a backup (network) drive for the archive file. Archive files can be saved under a different name to that of the model but when an Archive file is opened the original model name will be adopted.

Right Hand Mouse Menu - File Form

Since the MOD file is a single file the Right Hand Mouse Menu can be used to manipulate MOD files.

User Defined Data

If a user creates files for special purposes e.g. files containing model definition commands then these file will always be archived if the file has the same name as the model and the extension starts with the letter u(U).

Files with two extensions will not be archived. i.e.

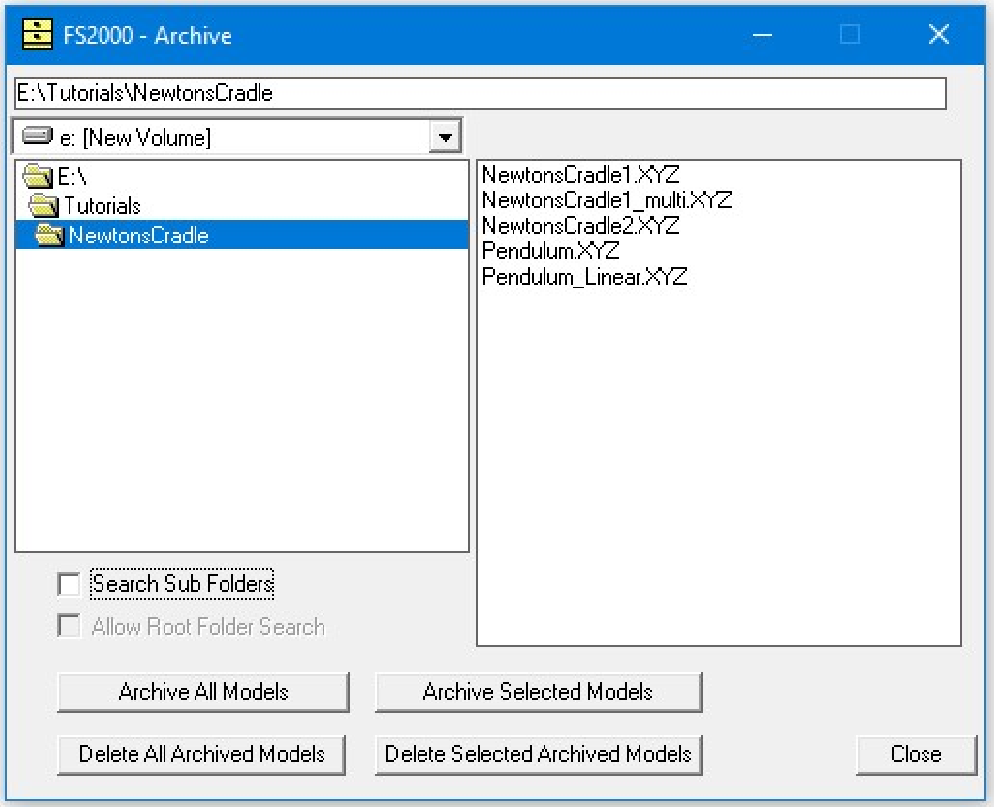
<Filename>.UM_Test will be archived

<Filename>.UM.Test will NOT be archived

Archive Utility

The Archive Utility is used to clean up model folders where multiple models exist.

If can be started from the GUI File menu or the Window's FS2000 Start menu.



The list box on the RHS display all active models in the selected folder and optionally including its sub folders.

Archive All Models - To archive all models in the selected folder.
Archive Selected Models - To archive selected models in the selected folder.
Delete All Archived Models - To delete all models in the selected folder that have at some time been archived i.e. a MOD file exists.
Delete Selected Archived Models - To delete selected models in the selected folder that have at some time been archived i.e. a MOD file exists.

The **Search Sub Folders** is active will display all active models in all sub folders of the selected folder.

If the **Allow Root Folder Search** is active then all folder in that drive will be searched. This will not search the C: drive because of its size (number of folders). If a drive has too many folders it will search for some time before the limit (32000) is reached at which point it will crash.

Note that the current model can be archived by this utility but not deleted.

If the model archive files (MOD files) are backed up to a network drive the it is safe to delete the model folders (assuming they only contain FS2000 models) because the folder structure can be recreated when the archive model (MOD file) is opened in FS2000.

-o-

3.3 Basic Stages of Analysis

The basic stages of any analysis are:

- **Model Definition**
- **Load Definition**
- **Analysis - Solution of Load Cases**
- **Post- Processing**
- **Results Output**

In FS2000's GUI the basic stages are called TASKS and a menu is used for TASK selection. When a TASK is selected, menu commands become available which are dependent upon the specific task.

Model Definition-TASK

In this TASK, the basic model geometry is defined. The GUI has some very efficient routines for creating model geometry and properties and is the approach that would always be used.

Note that model geometry may also be defined by either/or a combination of the following methods:

- Automatic generation using the specific model type generators.
- Command Line Definition Instructions.
- Importing a CAD DXF file (stick drawing of the structure for beam type elements).
- Spread Sheet Data - Node and element lists can be conveniently converted to command line instructions. Often used when converting other program structural geometry to FS2000 format.

Load Definition-TASK

In this TASK, the loading on the model is defined. Loading is categorized and saved as separate load cases. Load cases are identified by a reference number, an ID number and are also assigned a descriptive title.

Model load data may be defined or modified by using the text editor. If new load cases are created in a text editor they must then be opened and saved in the Load Definition TASK (registers the load case).

Analysis-TASK

This is the stage where the load cases are submitted for solution. This stage produces ***Raw Result Cases*****.** The ID number of the Raw Result Cases matches the Load Case ID number. Raw results must be post-processed

Post-Processing

The raw result files produced at the Analysis stage cannot be accessed for output until they are converted to ***Processed Results Cases***. The post-processing stage is also used to combine raw result cases to produce factored result cases. For finite element solutions the post-processing stage is used for stress averaging and stress output coordinate selection.

If a single Raw Results Case is processed, then ID number of the Processed Result Case will match the Load Case ID number. If a Load Case Combination is processed, then the Processed Result Case matches the Load Case ID number.

Output/Results -TASK

This stage is where the Processed Result Cases are output. Results output can take the form of text listings or graphics plots. Both forms of output can be interactively interrogated or printed out to form analysis reports.

-o-

3.4 Interactive Processing/Batch Processing

The Analysis, Post-Processing and the Results Output stages employ program modules that may be activated by Direct user Action or by the use of Batch Mode command line instructions.

DIRECT USER INTERACTIVE OPERATION

In this mode of operation all program activities follow direct user action using the menu commands or input forms within the FS2000 environment. For simple models with few load cases it may seem to be more convenient to operate the program modules using this mode. But this is often not the case. Most jobs for various reasons require repeat runs and batch mode is ideal for this. Interactive mode should only be used to 'prototype' the batch mode commands.

BATCH MODE COMMAND LINE OPERATION

For large models or models with numerous load cases or load case combinations it is often more convenient and almost essential to operate the program modules using Batch Mode command line instructions. Command line instructions can be incorporated into batch list files to enable the user to set up multiple module processing that eliminates user intervention.

FS2000's Batch module interprets the batch lists and initiates each process accordingly. The Batch module is a separate Windows application and may be used to run different models to the model loaded in FS2000.

With batch mode operation the whole analysis process can be repeated with a few mouse clicks - an essential feature for the design of models with a large numbers of load cases.

Batch mode operation is simply a method of recording and replaying interactive actions. When using FS2000 interactively it will be seen that some input boxes have a Batch button. The boxes with batch buttons indicate processes that may be also initiated using command line process switches. When the Batch button is clicked then the current options are converted to command line switches and the command line appended to the last batch file that was opened in the Batch Process Control. The batch file may be run at any time using Batch Process Control module.

It is recommended that the user should become familiar with batch operation by using it in conjunction with interactive operation at the very earliest stages of using FS2000.

-o-

4 General Operational Features

4.1 Units

Any system of consistent units may be used as the stiffness solutions do not use any unit dependent constants. A unit description may be entered during model creation. This description entry is purely for reference purposes and will be echoed on any input or output data.

If the model uses the property library files or other optional modules such as design code checker or load generation modules it is **essential** that specific units be used.

Two basic unit systems can be used in FS2000, one is the **SI System** and the other is the **USA System.** When new models are created an option is available to select the required unit system. The unit requirements for the two systems are defined below.

The UDEF command define the model type. US unit model have a UDEF=1 designation.

SI System

Length	metres	(m)
Rotation	Radian	
Force	Newtons	(N)
Moment	Newton-m	(Nm)
Mass	kg	(kg)
Pressure	N/m2	(Pa)
Velocty	m/s	
Acceleration	m/s2	(1g=9.81m/s2)
Density	kg/m3	(typically 7850 for steel)
Stress	N/m2	(typically for steel E=205E9 Grade 50 Yield=345E6)

For the stress output 1 Pa = 1 N/mm2 = 1E6 N/m2

If the above S.I. units are used to define the model and then the user may select ‘Engineer’s Units’ (default) for the result format. The formatting gives the output in the following units.

Formatted Load Definition data

Force - kN

Moment - kNm

Distributed Loads - kN/m

Formatted Results

Deflections – mm

Forces – kN Moment -kNm

Stresses - N/mm^2

USA System

Length	inches	(ins)
Rotation	Radian	
Force	pound	(Lbs)
Moment	pound-inches	(Lb-ins)
Mass	pound	(Lbs) Note: Dyn Mass = Lbs/386 (W/g)

Pressure	psi	(psi)
Velocity	ins/s	
Acceleration	ins/s2	(1g=386 ins/s2)
Density	Lbs/ins3	(typically 0.283 for steel) W/ins3
Stress	Lbs/ins2	(psi) (typically for steel E=30E6 Grade 50 Yield=50E3

If the above USA units are used to define the model then the user may select 'Engineer's Units' (default) for the output format. The formatting gives the output in the following units.

Formatted Load Definition data

Force - kips

Moment - kip-ins

Distributed Loads - Lbs/ins

Formatted Results

Deflections – ins

Forces - kips Moment – kip-ft

Stresses - ksi

-o-

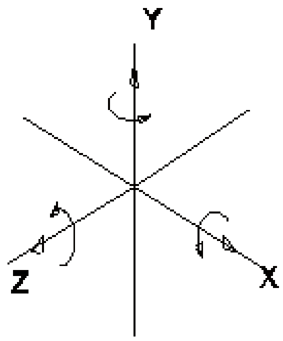
4.2 Sign Convention

Sign convention is based on the right-handed orthogonal system. The right hand corkscrew rule is adopted for moments.

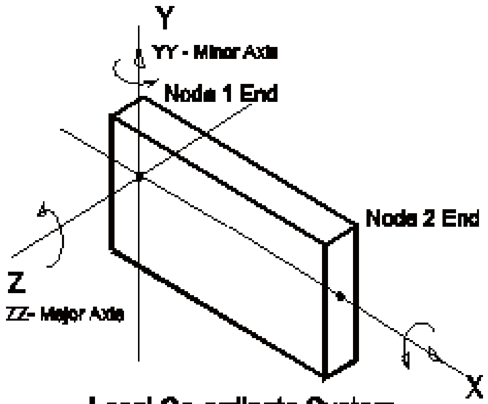
The exception to that shown is the output of beam element forces. In this case the sign convention of the start node forces and moments have their sign changed. This is done so that the sign of the force indicates the condition of the member i.e. +ve axial load always signifies tension regardless of element end.

For solids the local element co-ordinate system is dependent upon the element type.

It is recommended that the 'y' axis be used as the vertical axis. This is simply because the default graphics view assumes this orientaion. If the FSWave or FSWind are to be employed it is essential that the 'y' global axis is vertical.



Global Co-ordinate System



Local Co-ordinate System

Shear Force Diagrams.

When defining elements in straight line (beam elements) it is good practice to ensure the elements are orientated in sequence i.e. Node 1 of an element is connected to Node 2 of the previous element. It makes no difference to the analysis results in any respect but it does ensure that the shear diagram on line elements does not give the impression that shear is changing between elements.

-o-

4.3 Co-ordinate Systems

There are three basic types of co-ordinate systems used by FS2000 these are:

- Global
- User Defined Local
- Local Element

Global	Nodal Definition Nodal Loads Restraints & Prescribed Displacements Element Loading
User Defined Local	Nodal Definition (Node Coordinate Systems) Finite Element Stress Results (Optional)
Local Element	Load Definition Beam Force & Stress Results Couple Force Resullts Finite Element Stress Results (Optional)

Global and user defined co-ordinate systems are described in the following section.

Local Element Co-ordinate Systems

Element co-ordinates refer to the local system of an element. All elements have their own co-ordinate system and they are dependent upon the element type. All element systems are right-handed orthogonal systems.

Solid Elements

Shell elements usually have their X axis directed form Node 1 (I Node) to Node 2 (J Node) and the Z axis normal to the shell surface. Solid elements are usually parallel to the global Cartesian axis. For solids refer to the specific element description for element co-ordinate description. Stresses from solid and shell elements are usually referenced to a co-ordinate system.

Beam (Line elements) & Couples

The local element system is used for:

- Element Loading Definition (optional)
- Element Forces and Stresses - Always

The local system is a function of element orientation, defined by the node co-ordinates and in the case of 3-D analysis, the Local Rotation angle.

The main rules applicable to the local system are:

The x direction is always +ve from Node 1 to Node 2

The local x-y direction is always parallel with the Global Cartesian Y axis for zero element rotation (ROT=0). Exception is when local x axis is parallel to the global y axis.

Positive element rotation is Clockwise looking from Node 1 to Node 2

Care should be taken when the element (local) x axis is parallel with the Global Y axis. In this instance the local z axis points towards global z for zero rotation.

The figures that show the Co-ordinate System/Local Rotation illustrate the relationship between Global and local element for local rotation angles of zero.

During model definition input data is sometimes required to be referenced to a local element system. This done by referencing the specific input e.g. offset definition, to the element label of an existing element. The local system is than taken to be identical to that of the element selected.

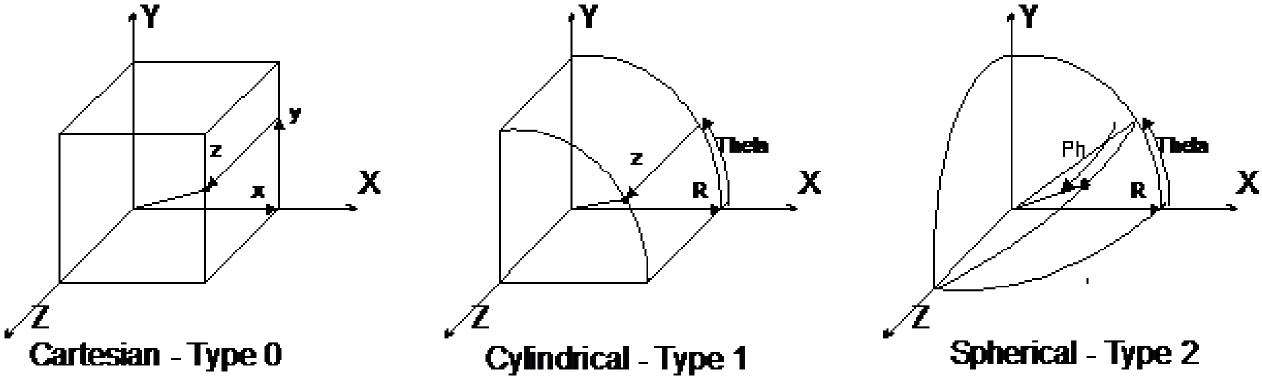
For beam and pipe elements see Global and Local Loading Figures

-o-

4.3.1 Local & Global Co-ordinate Systems

Global Co-ordinate systems

The global systems may be regarded as absolute frames of reference since they cannot be moved or rotated. The three predefined global co-ordinate systems are Cartesian (System No. 0), Cylindrical (System No. 1) and Spherical (System No. 2). Each have a common global origin. The Cartesian system is the default system.



Local Co-ordinate systems

Local co-ordinate system are used for node definition and for the output of FE stresses.

Local co-ordinate systems are co-ordinate systems created by the user, As many as 15 local systems (No 6 to 20) can be created. Each is assigned one of three co-ordinate types.

Type 0 is Cartesian,

Type 1 is Cylindrical

Type 2 is Spherical.

Type 3 is Conical (uses R at Z=0 and a cone angle to define a surface where x and z are interdependent)

They are used for node definition and for the output of FE stresses.

The origin of local systems can be defined at any point in space and be defined with any orientation. They are similar to those shown above with the exception that the X, Y & Z axis are locally orientated.

The local co-ordinate system can be based on the definition of an origin location and three rotations or the definition of 3 nodes.

When a local system is created it is assigned an ID number.

Coordinate system are created using the Coordinate System input form.

Active System

Once a system becomes active the x, y and z directions refer to the axis of the system. Thus if the current system is a Type 1 (Cylindrical) system these directions represent r, theta (degrees) and z. When a node is defined it is assigned to the currently active system.

Generation and copy routines similarly use the active co-ordinate system. If the current system is a Type 1 (Cylindrical) then copy increments in the Y direction (degrees) will from an arc with the x co-ordinate being the radius.

Copying -Unless the Global Cartesian is active the Active co-ordinate system must be the same as the node being copied node.

Generation between nodes and on elements. The node generated will be defined in the current active co-ordinate system. If other than the Global Cartesian co-ordinate is active the definition must be in that system

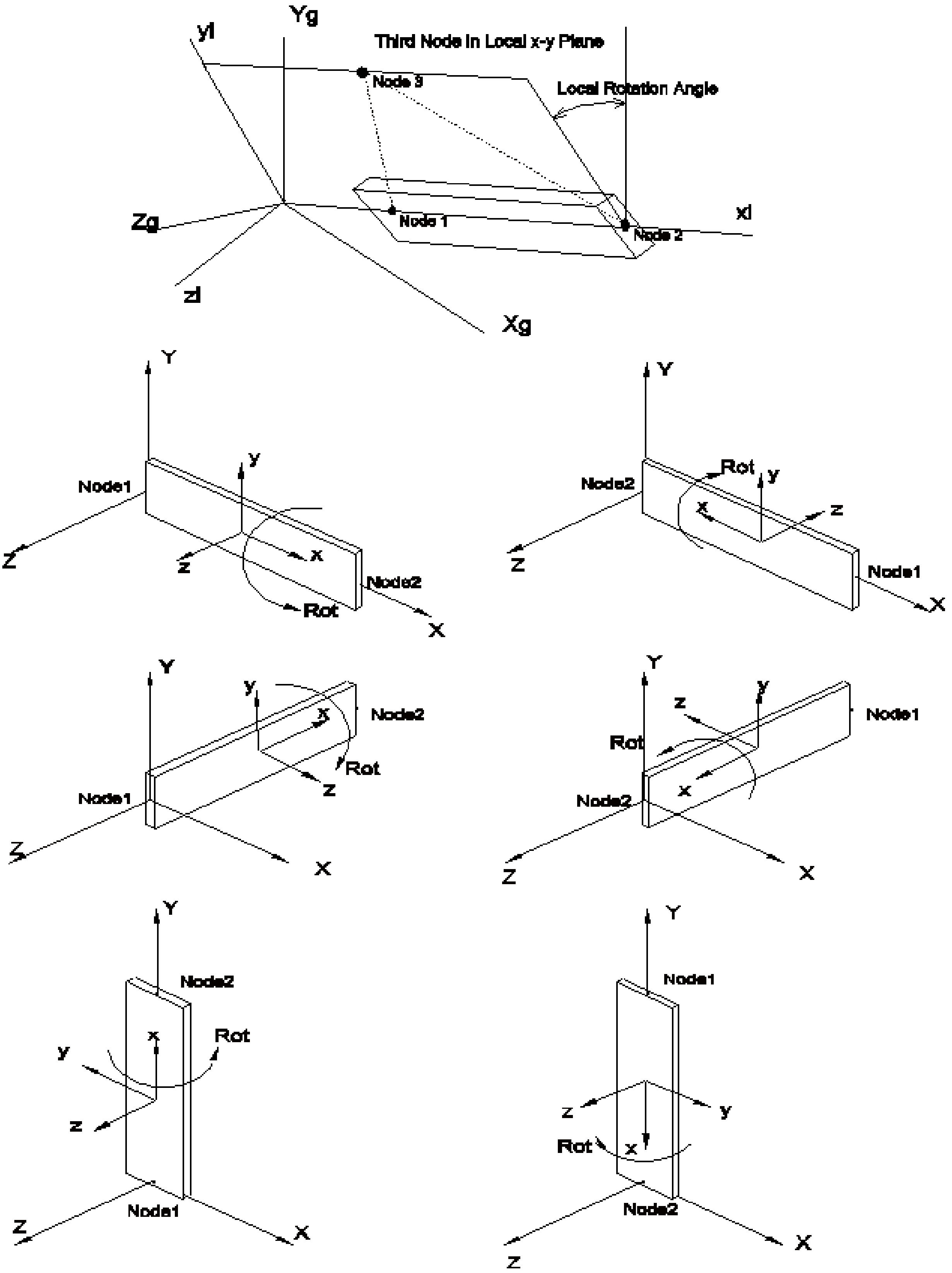
Translation The translation parameters must be in the same co-ordinate system as the subject node.

Surfaces

By using an appropriate coordinate systems and specifying a constant value for a co-ordinate a surface may be defined. The use of relative co-ordinate definition provides a very convenient method for generating nodes on surfaces. Generation and copy routines will similarly create surfaces.

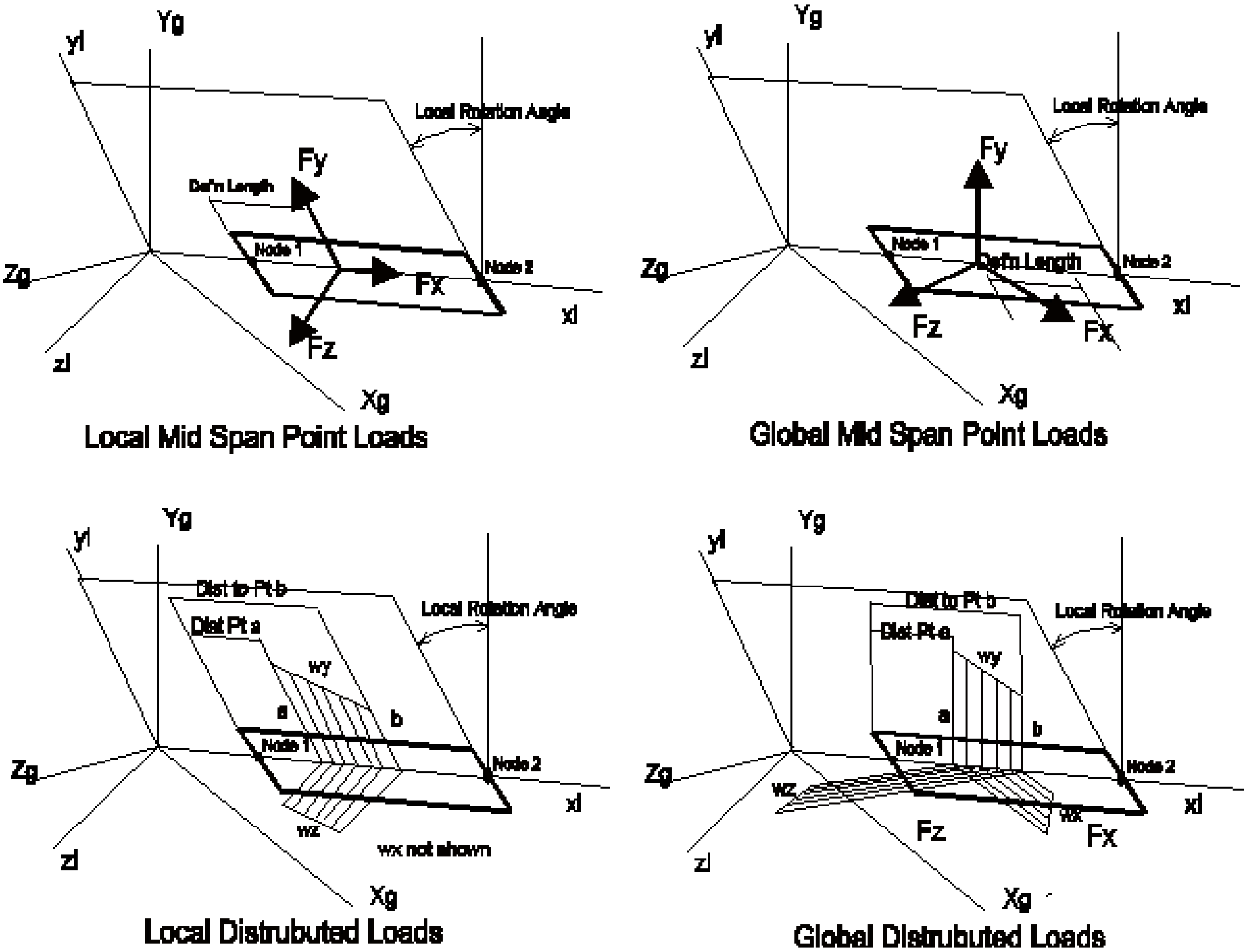
-o-

4.3.2 Beam Element Co-ordinate System/Local Rotation



-o-

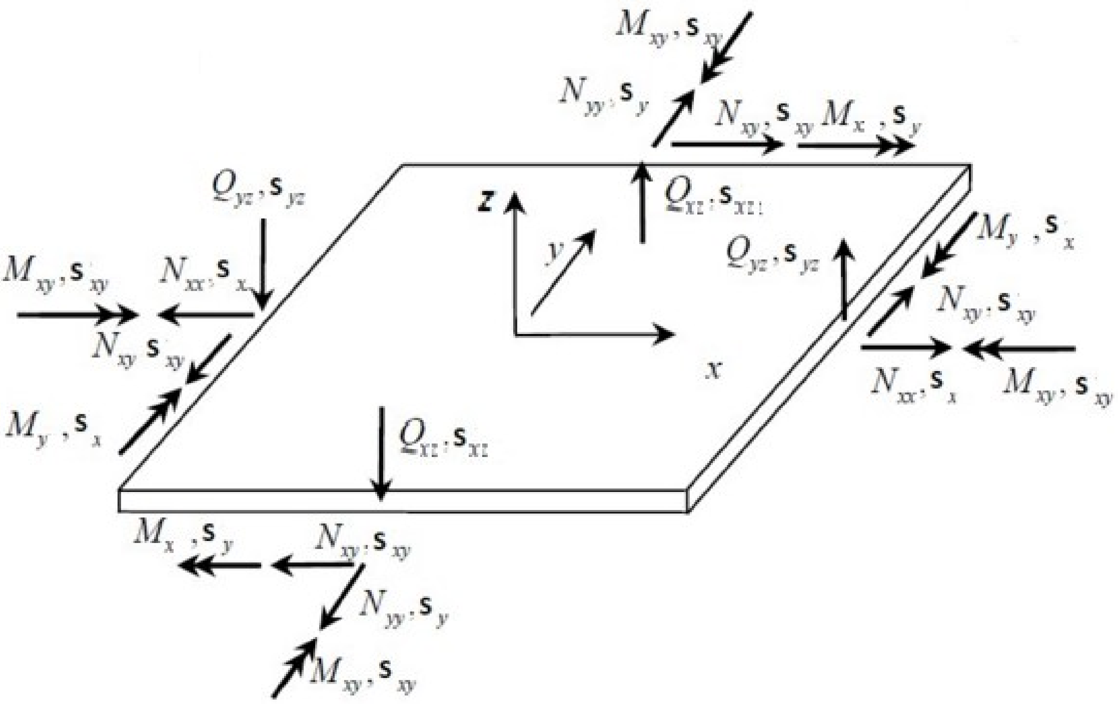
4.3.3 Beam Element Loading Co-ordinate Systems



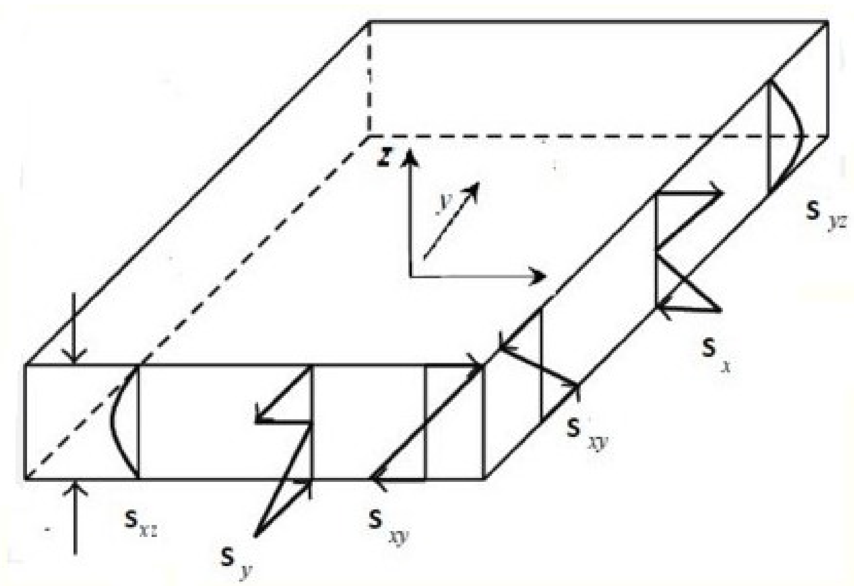
-o-

4.3.4 Shell and Solid 3-D Elements

Shell Element Output Conventions (Local)



Bending and In-plane Actions



Stresses due to Bending Actions (Sx, Sy and Sxy due to in-plane actions are omitted from Figure)

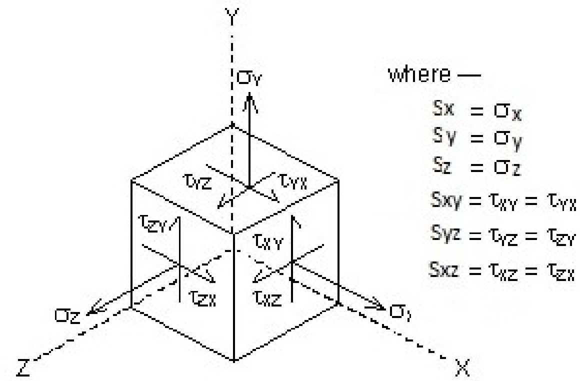
+ve Mx or My will produce a +ve stress (Sy or Sx) on the top surface
+ve Mxy the twisting moment will produce a +ve shear stress (Sxy) on the top surface
+ve Nx or Ny the direct normal loads will produce +ve normal stress (Sx or Sy)
+ve Nxy the in-plane shear load will produce +ve shear stress (Sxy)
+ve Qyz or Qxz the transverse shear loads will produce a +ve shear stress (Syz or Sxz)

When plotting shell forces the stress options in the FE-Plots menu equate to the following force actions:

Sx Direct	Mx (Sy Direct)
Sy Direct	My (Sx Direct)
Sz Direct	Mxy
Sxy Shear	Nx
Szy Shear	Ny
Sxz Shear	Nxy
S1 Principle	Qyz
S2 Principle	Qxz
S3 Principle	Wood-Armer Mx*
Stress Intensity	Wood-Armer My*

Solid 2-D and 3-D Elements Conventions

2-D elements are defined in the X-Y plane only.



Shell Stresses -Output

Extrapolation of Stress to Node Locations

The stresses in parametric elements are evaluated at the integration(Gauss) points. They could be evaluated to the node locations but for various reasons they are more accurate when evaluated at the Gauss points. From an engineering aspect the stresses at the node locations are of more interest To obtain nodal stresses the Gauss point stresses are extrapolated to the node locations assuming a bilinear linear variation.

This extrapolation is not undertaken with material non-linear elements apart from the Type51-4Node Shell and in this element extrapolation is on only undertaken following solution convergence.

Coordinate System

By default all 2-D and 3-D solid elements output their stresses in the global coordinate system.

By default all shell elements output their stresses in their local coordinate system.

The stresses can be converted to any coordinate system. This conversion is undertaken when a result case is post-proceesed. GROUPS - The coordinate system to be used is identified by the element's group attribute which equates to the Coordinate System ID number.

If an irregular mesh is used the component stresses of shell elements will be often be difficult to interpret if the only default system is used. Von-Mises being scaler are will always be easily interpreted.

Averaged Element Stresses

If the the finite element solution was exact the nodal stresses evaluated at a given element node would be identical to that of an adjacent connected element. In reality for various reason there will be a variation and there will be stress discontinuities across the mesh. These discontinuities will become apparent when the a stress contour is displayed.

These discontinuities can be removed by employing a commonly used technique of averaging the stresses at a common node. This stress smoothing my appear to be a somewhat of a cheat but it does in fact, in most cases, produce a more realistic solution. However, if the difference between unaveraged and averaged stresses is large it would indicate a lack of mesh convergence i.e. the mesh is too coarse for the stress variation.

The optional stress averaging process is under taken when the result case is post processed. The following points should be considered when averaging is being employed.

- When dealing with 2-D and 3-D solids this averaging process can be generally applied globally across the model without any difficulty.
- In models with geometric and material discontinuities this process when applied globally may not be valid. This is especially true of when shell elements are used. Elements meeting at a corner or element with different wall thickness are examples of case where stress averaging should not be applied. The element's group attribute is used to identify which elements can be average with each other.
- Shell stresses are evaluated in the element's own co-ordinate systems which may be different for each element. In such cases the user has to select which elements are to be included in any averaging process and ensure that output co-ordinate systems are compatible. The element's group attribute is used to identify an element's output coordinate system.

GROUPS - The element's group attribute is used identify the elements between which everaging is required.

MESH CONVERENCE A good measure of the suitability of a mesh is to compare contour stress plots of the unaveraged and the averaged stresses. If there is a large variation then it is an indication that the mesh is too coarse.

Wood-Armer Shell Moments

The Wood-Armer shell moments are evaluated using the following.

Top

$$M_x^* = M_x + |M_{xy}|$$
$$M_y^* = M_y + |M_{xy}|$$
$$If\ \ M_x^* < 0 : \quad M_x^* = 0$$
$$M_y^* = M_y + \left|\frac{M_{xy}^2}{M_x}\right|$$
$$If\ M_y^* < 0 : \quad M_y^* = 0$$
$$M_x^* = M_x + \left|\frac{M_{xy}^2}{M_y}\right|$$

Bottom

$$M_x^* = M_x - |M_{xy}|$$
$$M_y^* = M_y - |M_{xy}|$$
$$If\ \ M_x^* > 0 : \quad M_x^* = 0$$
$$M_y^* = M_y - \left|\frac{M_{xy}^2}{M_x}\right|$$
$$If\ \ M_y^* > 0 : \quad M_y^* = 0$$
$$M_x^* = M_x - \left|\frac{M_{xy}^2}{M_y}\right|$$

Coordinate system transforms during post-processing can be used to align then moment directions to any desired skew direction.

-o-

4.4 Nodes and Element Numbering

The numbering system adopted in a model has no effect on the solution efficiency of FS2000 provided that the internal solution re-numbering modules are used prior to the analysis run. The renumbering modules (Wavefront and Bandwidth) are available in the Analysis TASK.

If node and element labels are to be used for identification purposes then minimise the label ranges used for identification i.e. minimise the gaps in the numbering sequence. If a model had only two element E1 and E30000 then this would be very inefficient because of the number of unnecessary looping that would be undertaken in the various processing operations.

When creating models interactively it is recommended that the user should concentrate on the visual/spatial aspects of the model and not on a node and element numbering sequence. There are few advantages in adopting a numbering based on node and element labels simply to identify spatial location within a model. Use Groups for this purpose. Groups are a powerful secondary identification system for nodes and elements that provide far more flexibility than labels.

The exception for adopting numbering sequences occurs when using certain generation routines. In this instance it may be necessary to adopt numbering patterns. However, most interactive routines enable nodes or elements to be selected by visual means or group identification in addition to label identification.

The Node and Element definition menus provide routines to eliminate or reduce the size of the gaps in node and elements numbering.

-o-

4.5 Load & Results Case Combinations

Load case combinations are lists of load cases or result cases each with a corresponding load factor. They are created from list forms which are accessed using commands in the Menu:Comb menu. When creating combinations, load factors are defined for each load case. This enables partial load factors to be used when combining results in compliance with certain limit state design codes.

Load Case Combinations have four areas of application.

- **Load Case Definition** - Combining Load Cases in Load Definition TASK.
  Useful for improving housekeeping of complex loading by allowing sub-division of loads into component load case. ie combinations are used to combine the component load cases into one single case for analysis. Can be done interactively or in Batch mode.

- **Pre-Processing** - Combining Load Cases at time of analysis.

  Pre-processing creates a new load case at the time of analysis based on the load case and load factors listed in a Load Case Combination. It enables different load case results to be combined into a single load case i.e. merges load cases prior to analysis. Load cases that contain nodal mass definition should not be included because the mass may be unwantedly combined with acceleration.

  It can be used for reducing the analysis solution times for multiple load cases, improving housekeeping of complex loading by allowing sub-division of loads into component load cases or for solving multiple load cases when using non-linear analysis where all the loads have to be applied simultaneously.

  Load factors are utilised when combining the cases.

  To pre-process a combination at Analysis Solution the combination number is preceded by a P(p). The Load Case formed and the results of the solution will assigned the same number as the combination. Note that load cases in the combination cannot have the same number as that of the combination.

  The merged load case will not contain any load definition commands but its description will identify that it is a pre-merged combination and show the combination number used the create the result.

  It is also possible to combine load case combinations in a pre-process solution. If the combination number is preceded by a P(p) then the cases in the combination will be interpreted as combinations and not load cases. They will be combined into one case using the load factors in all combinations. When combining combinations it is advised to include this in the combination description otherwise there is no way of distinguishing it from a load case pre-combination. Note that load cases in any of the combination cannot have the same number as that of the combination.

  Pre-processing should be always be used when the **3-D Standard Solver** is being used to undertake non-linear analysis (P-Delta, tension only etc.) and preferably when prescribed displacements are present

  An alternative to Pre-processing is Dynamic Load Case merging.

- **Analysis Solution** - Submitting multiple load cases for analysis solution.

  - **3-D Standard Solver** Combinations can be used to submit more than one load case for solution. This is a very efficient solution method since the model stiffness matrix is only assembled once, for the first load case. Subsequent load cases are back substituted. This requires a fraction of the solution time for full assembly. To submit a combination at Analysis Solution the combination number is preceded by a C(c). If prescribed displacements are being used there are restrictions with this method.

  - **3-D** NL**/ Pile Analysis** This solver is a static incremental solver where a load combination can be used to define the sequence in which the loads are applied. The analysis process produces a single result case with the same number as the combination. It is similar to a pre-combined combination but the loads are applied sequentially.

  - **FS-DyNoFlex** This solver is a dynamic incremental solver where a load combination is used to define the sequence in which the loads are applied. It is similar to the previous but the load case combination may also be used to reference load time curves. (see FS-DyNoFlex documentation)

A more complete description of how load cases and load combination are processed during solution is given in Section 6.2.

- **Post- Processing** - Combining Load Case Results after solution analysis.

  This is the most common use of load case combination. This capability enables load case results to be combined (and factored) in the Post-processor following analysis solution, a common feature of LFRD. This especially useful when a small number loaded cases are used in larger number different combinations and would be used in preference to Pre-Processing. It is also possible to combine load case combinations when post-processing (see Post-Processing batch commands).

  Post-processing using the the principle of superposition and is therefore only theoretically applicable to linear solutions. However, it is possible to post-process non-linear solutions but it is not possible to combine linear and non-linear solutions. All result cases to be combined must be of the same solution type. The exception is that seismic response solutions (only multiple mode) can be combined with linear and non-linear solutions.

The PostPro menu command in the Analysis & Results TASK makes the Post-Processing dialogue box visible

-o-

4.6 Results SETS

A results SET is simply a list of Processed Results cases. SETs are used by some of the results modules to process multiple cases. They are created from list forms which are made visible using commands in the Menu:Comb menu.

The main uses of a result SETS are:

- Creating multiple case output files using Standard Multiple-Out
- Plotting URs for multiple cases from Standard Individual-Out or Design Module Checks
- Creating sorted UR lists form standard stress output and design output

-o-

4.7 Groups

Groups are a powerful secondary number system for node and element identification. The program uses group selection as an option in virtually all aspects of model manipulation involving the selection of nodes and elements for both model definition selective results output.

Groups provide a far more flexible method of identification than labels. This secondary identification function is especially useful when using interactive-graphical methods to create models. Groups may be created/used in all analysis TASKS using the Group Menu.

Node or element groups can be saved to named Group SETS enabling multiple SETs to be created for different functions. These functions include

- Model generation
- Definition data identification
- Model spatial identification
- Load definition
- Results Interrogation selective plots etc.
- Results output identification

Output listing of both definition data and results data can be sorted by group either wholly or partially or be restricted to selected groups.

The **Group**, the **Group SET** and the **Active Group** are the fundamentals when using groups

Group

An element or a node may be assigned to a Group identified by a number. Each Group can be assigned a descriptive title. A node or element can only be assigned to one group i.e. there is only one memory location per node or element for the group at any one time.

Group SETS

When node or elements have been assigned to Groups the groupings may be saved as a named Group SET. Any number of Group SETS may be saved and recovered at any time. This enables Groups of nodes or elements to be formed for any number of reasons.

The most common use of groups on large model would be for identification. In this instance the nodes and elements would be assigned to suitably named groups and saved as Group SET called say 'Primary Identification Group'. When formatting the definition data or producing results output this set could be used to form the output by sorting by group or by simply adding the group attribute to the label.

e.g. Node 10 would be replaced by Node 10-G30 where G30 where G30 refers to nodes at some particular location of the model.

Active Group

The basis for creating a Group SET is to define the Active Groups, one for nodes and one for elements. The current active groups are shown in the in the top toolbar (left of the Activity Status box). The Active Groups dialogue box will also appear if these buttons are clicked. The left mouse button or 'A' hot key can also be used to define the active group.

If a group is made active then all nodes and elements will be assigned to the Active Group as they are defined.

The Group Menu may also be used to assign, modify etc existing node and elements group attributes using the current Active Group.

It is often more convenient to assign groups in the latter stages of model creation using the Group Menu.

-o-

4.8 Output - Formatted Definition & Results Data

This section describes how the definition data and output text results can be processed and printed to form report output. Saving and printing graphical View Files which provide a permanent graphical record of model features are described later in the Saving and Opening Views section.

The formatted files produced by the output modules are continuous text files without page breaks. Data headers only occur once for each section. This is done so that if required, data can be easily cut and pasted between Windows applications. The file naming convention is described in the Formatted Files Created in FS2000 section.

Pagination and multiple page headers are only included when the file is being printed from FS2000. Paginated files are output files that include page breaks, page numbers and page headers, exactly what the printer produces. The Report Collator has the capability to paginate and merge output into text file using a print to file option.

Selective Output

The volume of output generated is dependent upon the size of the model the type of analysis and the number of load cases and load case combinations. On small models it is very often practical to output all the data but even on moderate sized model it may be more efficient to restrict the output. Output may be restricted by Group attribute i.e. only nodes or elements assigned to specific groups are output. Stress and design output can also be restricted to elements whose UR value is above a defined limit.

Selective Output and Sub-Cases

The various modules that produce the formatted output data provide the user with a high degree of flexibility as to what should to be included in the output and how it should be categorised. Sub-cases are way of separating and identifying different output options for the same result case.

There most common use is for result data. A typical example of this would be to create two text output files for the same result case. One could be the end connection loads for a specific connection type in the model, this could have the sub-case name 'Conn_A' and the other could be the deflections of specific nodes with the sub-case name 'Deflections'.

Sub-Cases may also be applied to formatted definition data. A common use for this would be to separate model data from load data.

-o-

4.8.1 Definition Data Formatting

The module that creates the formatted input data files is activated from the Data menu using the Format Definition Data command. This is a global menu.

Batch run command name: **IN**

Document Files (DC Files)

If text files <modelname>.DC1 or <modelname>.DC2 exist they will be incorporated into the MTM definition file when the file is created. The purpose of the these files is to enable the user to incorporate additional descriptive text into the definition data output. The files may be of any length but they must be text files (not Word Processor documents).

DC files are archived with the model.

DC files are not incorporated in MTM Sub-Cases unless the Sub-Case is called 'Rep'.

DC1 File The main use of this would be to provide a generic front cover to the analysis report. This cover would contain model details (customers, QA boxes etc) that cannot be included in the standard model description data.

DC2 File The main use of this would be to provide the capability to include introductions, descriptions of the model, analysis methods and results summaries etc. Such a document would normally be created in a word processor (using proportional fonts to ensure correct formatting e.g. courier) but if the word processor document is saved as a text DC2 file it may be incorporated with the model and archived with it. This approach will always ensure that the descriptive documentation will always be available with the model for any user at future date as it will be incorporated into the archive MOD file.

If files Default.DC1 or Default.DC2 exist in the FS2000 directory, they will be regarded as generic files and will be included into all models when the model is first opened in FS2000. If the files are deleted from the FS2000 directory they will not be incorporated.

Formatting Load Cases

When load cases are formatted summaries listings or listing of all loading within a load case can be output.

When load case summaries are formatted a summation of loading will also be produced. This is very useful for undertaking QA load checks.

However for this summation to be produced the load case must be processed for solution. Load cases that have not being processed will have the summation populated with ******* characters. This is a requirement to ensure that such data is applicable to the current state of the model.

Note: Loading from **FE-Solid** elements are not included in summaries only loading on line elements are included.

```
*** Load Case Summary  ***

Case No    Description                                                        Fx         Fy         Fz

 1         Self weight                                                          0.00     -519.38
0.00

 2         Internal Fluid density                                            ****       ****       ****

 5         Valve and Actuator / Gearbox weight                                 0.00     -111.63
0.00

 49        WAV Env:SWL                                                         5.24      100.35
-0.26

 100       WAV Env:001yr_000deg_Tmax_Crest                                   205.99      127.35
-11.55
```

When the solution uses **Pre-Combined Cases** or when **Dynamic Load Case Merging is** being employed the component load cases are not individually processed. In such situations the data used for the load summation can be created without the requirement to undertake a solution using the LOADA command in a Batch file.

Note that the LOADA command does not include finite (solid) element loading contributions.

> The command LOADA 5 would process L5 and the case summation would appear in subsequent formatted load case listings.

> The command LOADA C500 will similarly process all load cases listed in combination C500.

Note that there is no requirement to eliminate the ****** in a non run load case, it will not effect the results.

-o-

4.8.2 Results Data

The modules that create the formatted Results Data are activated in the Output/Results Task using the StdOut menu commands. These are identified below by their menu command name. Also given below is the Batch run command name

StdOut Menu - Individual Results Output
This module will create a formatted text file showing the displacements, forces, stresses etc. for individual results cases. It also creates stress ratio (Von-Mises) data used for plots and UR sorts.

Batch run command name: **OUT6**

StdOut Menu - Multiple Results Output
This module will create a formatted text file showing the displacements, forces, stresses etc. for more than one result case. The results cases to be processed are defined in a Results SET.

Batch run command name: **MOUT6**

StdOut Menu - Sorted Unity Ratios (URs)
This module will process the stress ratio files created in the Individual Results Output or Design modules. The module creates formatted text file showing the URs values. The listing can be sorted by element label or UR value. Individual or more commonly, multiple cases (using Results SETs) may be processed.

Batch run command name: **URSORT**

Optional Design Modules
These design models produce summary output or detailed output which is listed by label and/or group. They also produce unity ratio (UR) data which can be processed in the Sorted Unity Ratios (URs) to produce output sorted by load level.

Reaction Summation

This is normally a summation of the loads applied to the model in the loading configuration but in some circumstances this my not be the case. The following describes these exceptions.

- FE-Solid element loading contribution is always excluded in the formatted load case data summation.
- Solver Used

  When a LINEAR (Std 3-D Analysis) solution is undertaken the StdOut module will list all restraint reaction forces and ground couple and the Reaction Summation represents the sum of these effects. This will agree with the load summation shown in the formatted load definition data (note that solid elements contribution is always excluded in the formatted load case data summation).

  When a NON-LINEAR result case is being processed the summation may differ from the applied loading because the summation is a summation of the loading applied only to degrees of freedom. This means that the reaction summation will be less, if any element load effects are applied directly to restrained freedoms.

  This will occur in the following cases:

  - When element loading is applied to restrained freedom e.g. element loading is applied to an element with restrained nodes.
  - When nodal loads are applied to restrained freedoms.
- If the model includes piled foundations (foundation springs not included).
- If the model includes Type 7 or Type 8 beam elements (internal beam ground springs are not included).

-o-

4.8.3 Viewing/Printing Output Data

The View/Print Report Data command of the Data menu provides a file view utility that enables the formatted output files to be viewed and printed. This utility is also be used to submit the data to the printer.

This view file utility can also be started by clicking the View button in the forms which are used to create/format the result data.

The Report Collator described in the next section can also be used for viewing and printing data.

-o-

4.8.4 Collating/Printing Reports

The Report Collation–Data Selection utility described in Section 8, provides a far more convenient method of printing output in cases where a number of cases are require to be printed. This utility is used to create a **Print List** of the Definition and Existing Results Data.

The print lists can be used to submit output to the printer for batch printing or for creating merged output files using a **Print to File** Option. The Print to File option will create paginated output files, this can only be done using the Report Collator. Paginated files are output files that include page breaks, page numbers and page headers, exactly what the printer produces.

If output is to be printed using a **PDF** writer then the Report Collator can be used to merge all text output into one pdf file.

Individual files from the print list can be selected for viewing or printing.

This print list effectively forms an index for analysis report text and can be edited to make it more meaningful to the specific analysis.

Selection window lists enable all current Results Case to be selected and added to the print list. The print list can be saved (<Model>.BPL). If a print list files exists it will always be loaded. Multiple lists may be created by using a secondary list option.

If the use of the Report collater is combined with Batch analysis processing the entire process from analysis to report printing becomes a simple and repeatable operation.

-o-

4.8.5 Selective Output

The volume of output generated is dependent upon the size of the model, the type of analysis and the number of load cases and load case combinations.
On small models it is very often practical to output all the data but even on moderate sized model it may be more efficient to restrict the output.
Output may be restricted by Group attribute i.e. only nodes or elements assigned to specific groups are output. Stress and design output can also be restricted to elements whose UR value is above a defined limit.

Sub-Cases

The various modules that produce the formatted output data provide the user with a high degree of flexibility as to what is to be included in the output. Sub-Cases are provided so that multiple output files may be created for the same results case. Sub-Cases may also be applied to the definition data.

The data required to be output depends very much upon the purpose of the analysis. The following illustrates what is considered a minimum amount for typical design analysis involving a number of load cases.

Definition Data

All definition data	Output using a an identification Group SET

Standard Results

Multiple Results Output	Restricted to Reactions and High Stress Ratios Stresses
Multiple Unity Ratio	Von-Mises restricted to URs > .6

Design Results

Summary Output	Restricted to critical members
Multiple Unity Ratio	Member UR restricted to URs >. 4

Graphics Plots showing model connectivity

Graphics Plots showing Von-Mises and Member UR plots.

-o-

4.8.6 Sub-Cases

The various modules that produce the formatted output data provide the user with a high degree of flexibility as to what is to be included in the output. Often it advantages to have more than one output based on the same data.

Sub-Cases are provided so that multiple output files may be created for the same results case. Sub-Cases may also be applied when producing formatted definition data.

An example of this would be the requirement to have two lists, one showing reactions for all results cases in a Result SET and the other showing only the maximum and minimum reactions for the same Result SET. Since multiple output is identified only by the SET number, an additional form of identification is required, these are Sub-cases.

Another example would be where there is a requirement to undertaken both structural and pipework design checks in a model for the same result cases. Since both checks produce member design output identified by the result case number, the latter would overwrite the first. The use of Sub-Cases will prevent this.

When data is selected to be viewed or printed a sub-menu will appear to select sub-case for cases that have Sub-Cases.

The file extension is used to identify sub-cases.
modelname.<Structure>.O7 would be used for a standard results output data, e.g. there could be two output data file one, ***modelname.<Gropup1>.O7*** and ***modelname.<Gropup2>.O7***

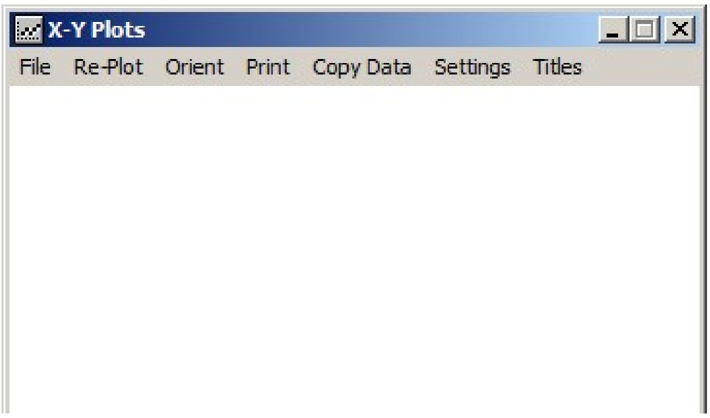
-o-

4.8.7 Graph Plotting

The FS2000 Graph plotting utility is started from the Run Graph command (File menu). The plotting utility is a separate Windows application and more than one instance may be activated.
This program is used to plot various types of results.

- Data Plot
- Stress Linearisation
- Time History - Solution Monitor
- Time History - Displacement ,forces and stresses

When the utility is started the following will become visible.



The **File** command menu is used to select the data to be plotted - see below.
Whenever time history data is opened for plotting a scratch file called **<modelname>.~PLOTSCR** is created. If when the data is opened and if the X min and X max are set with the Disable Auto Scale is active then the data in the scratch file will be limited to time interval defined by the time range.

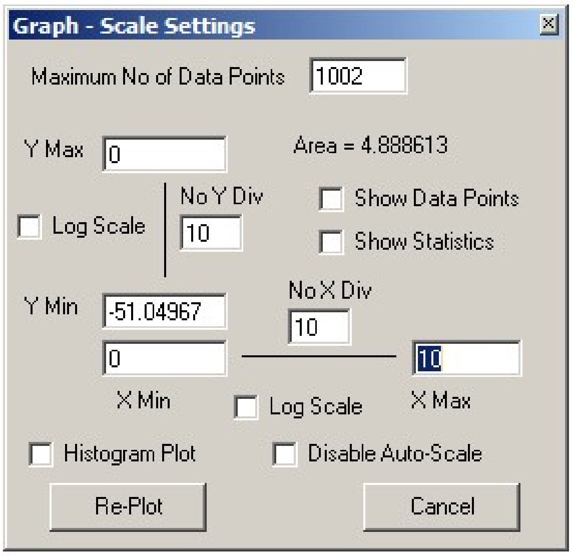
The **Orient** command changes the orientation between portrait and landscape.

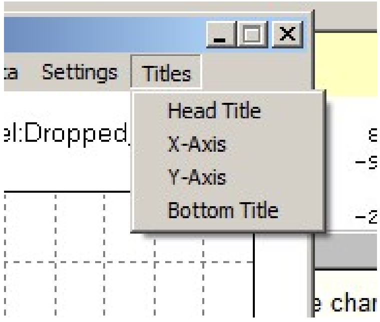
The **Print** command print to the Default Windows printer

The **Data Copy** command will copy the most recently plotted data to the Windows clipboard. The data can then be pasted into a text editor or spreadsheet e.g. Excel. The data is in the format described below.

The **Settings** and **Titles** commands are used to control the appearance for the graph. The X-Y limits enable easy zooming of any portion of the plot.

The Area = xxx , is the area under the graph.





The **File** command menu has then following commands:

- Data Plot
- Stress Linearisation
- Time History - Solution Monitor
- Time History - Displacement ,forces and stresses

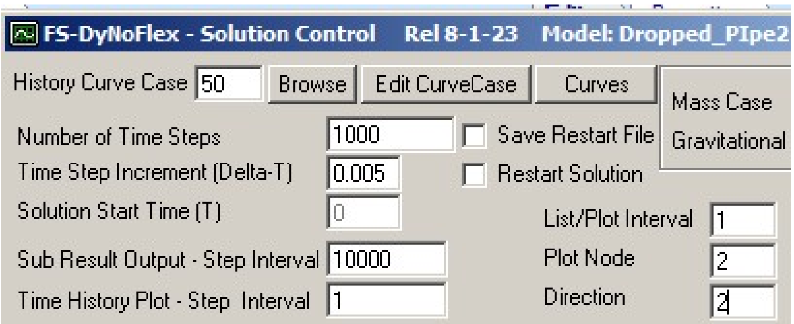
The **Data Plot** option is used to plot simple x-y plots. The file to be plotted can be any name but it must comply with the following ASCII text file format. A simple example is also shown.
The x and y data may be delimited by commas or spaces not TABS.

Format	Plot File Example
Graph Title	DYNOFLEX ANALYSIS
X Axis Title	TIME
Y Axis Title	DISPLACEMENT
Bottom Line Description	TIME HISTORY
x1, y1	9.9999998E-03 -9.809994
x2, y2	2.0000000E-02 -9.809982
-	2.9999999E-02 -9.809971
-	3.9999999E-02 -9.809959
xN, yN	5.0000001E-02 -9.809948
	5.9999999E-02 -9.809937
	7.0000000E-02 -9.809925
	7.9999998E-02 -9.809914
	9.0000004E-02 -9.809902
	0.1000000 -9.809891
	0.1100000 -9.809879
	0.1200000 -9.809868
	0.1300000 -9.809856

The **Stress Linearisation** option is used to plot through thickness stress plots in solid element meshes and produce linearised stress design data. The type of data is commonly used in various pressure vessel design codes. This is described in Section 4.8.7.1.

Time History - Solution Monitoring

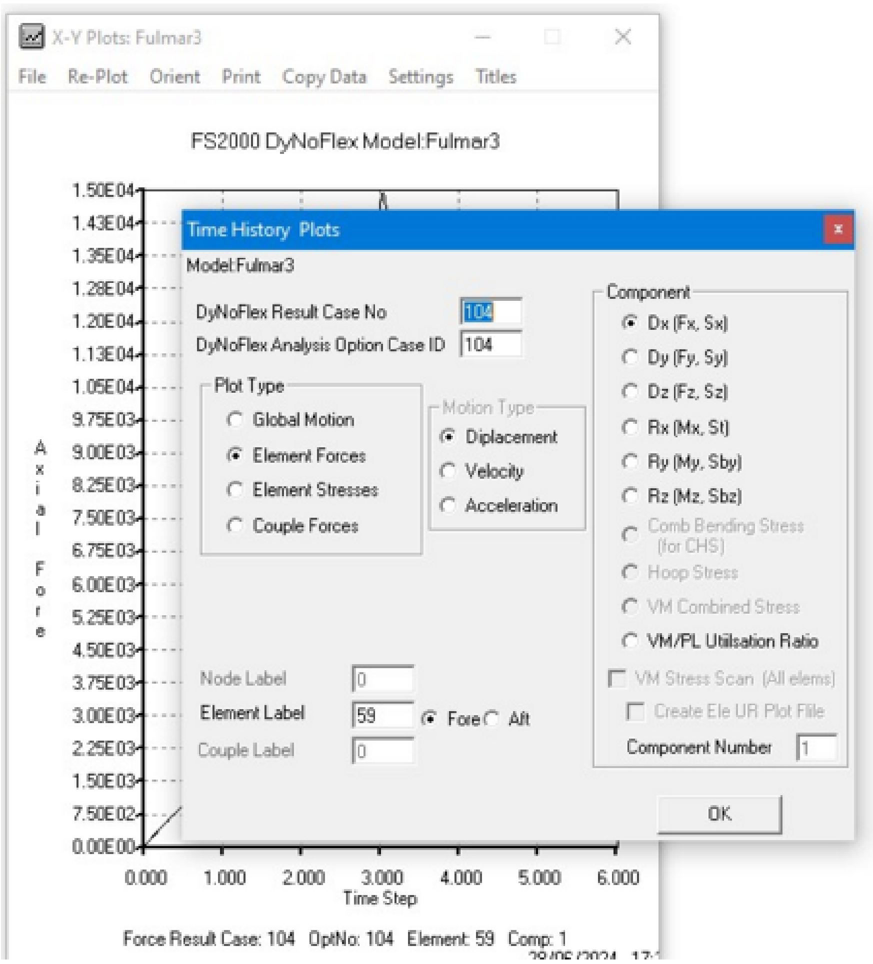
This command plots the displacements of the specified degree of freedom in a DyNoFlex solution. The **List/Plot Interval, Plot Node** and **Direction** (1-6) in the DyNoFlex options form are used to specify the freedom to be monitored. This date can be plotted as the solution progresses. When the command is selected only the solution Case No has to be specified.



The **Time History** option is used to create plot data for displacement, motion, force, stress or couple force response as a function of time from a DyNoFlex solution.

The plot interval has to be set in the DyNoFlex solution option's **Time History Plot Interval** setting. A setting of 1 will plot all time steps. Note that Type 0 element and Type 0 couples (linear elements) do not produce element force or stress plot data.

The Time History plot option form shown below is used to select the data to be plotted. It can also be used to scan a time history for stress maximums.



The **VM/PL Utilisation Ratio** will plot the plastic UR value assuming linear interaction $F/F_p + M_x/M_{px} + M_y/M_{py} = 1$

The **VM Stress Scan (All elements)** option will scan all time steps for all elements and plot the maximum (fore and aft) VM Combined Stress or VM Utilisation Ratio as a function of element label.

Setting the plot type to Histogram gives the best presentation.

If **Stress Scan (All elements)** option is active then the **Create Ele UR Plot File** option can be used create a Von-Mises UR plot file that can be used to plot the maximum UR values on the model using the DyNoFlex Case number.

-o-

4.8.7.1 Stress Linearsation

The **Stress Linearisation** option is used to plot through thickness stress plots in solid element meshes and produce linearised stress design data. The type of data is commonly used in various pressure vessel design codes.

The data file used to evaluate linearised stresses would normally be created and plotted from the FE-Stress Results Inspection form. This form creates a data file called **<modelname>.~LPROF**.

Any data file can be processed providing in complied with the following file format.

The first line defined the number of stress entries. Commas or spaces can be used for data delimiters.

El No, NodeNo, x-coord, y-coord, z-coord, Sx, Sy, Sz, Sxy, Syz, Sxz, 0, CaseNo

```
11
30  41     0.0000  -2.0000   0.0000    -45.86     -0.10     -13.79     -0.06       0.00
0.00   0    1
30  304    0.0000  -1.9000   0.0000    -36.54      0.37     -10.85     -0.88       0.00
0.00   0    1
30  46     0.0000  -1.8000   0.0000    -27.31      0.90      -7.92     -1.67       0.00
0.00   0    1
35  324    0.0000  -1.7000   0.0000    -19.15      1.98      -5.15     -2.12       0.00
0.00   0    1
35  51     0.0000  -1.6000   0.0000    -10.99      3.05      -2.38     -2.59       0.00
0.00   0    1
40  344    0.0000  -1.5000   0.0000     -2.58      4.12       0.46     -2.79       0.00
0.00   0    1
40  56     0.0000  -1.4000   0.0000      5.77      5.22       3.30     -3.01       0.00
0.00   0    1
45  364    0.0000  -1.3000   0.0000     16.03      5.55       6.47     -2.82       0.00
0.00   0    1
45  61     0.0000  -1.2000   0.0000     26.06      6.04       9.63     -2.69       0.00
0.00   0    1
50  384    0.0000  -1.1000   0.0000     42.09      3.54      13.69     -1.51       0.00
0.00   0    1
50  66     0.0000  -1.0000   0.0000     58.51      0.80      17.79     -0.25       0.00
0.00   0    1
```

When the data file is processed the following processed data will appear. The data shown is written to a file called **<modelname).BLINEAR**

```
Model:StressLinearisation_8N

LINEARISED STRESSES

Surface 1 at Node 41
Surface 2 at Node 66
Section width  1

Stresses at Surface 1

          Sx          Sy          Sz          Sxy          Syz          Szx
Peak    -45.860     -0.100     -13.790     -0.060     0.000     0.000
Linear  -47.824      0.746     -14.122     -1.554     0.000     0.000

Derived Stresses (Linearised Components) at Surface  1
Principle  Stress  S1 = 0.7955595
Principle  Stress  S2 =-14.1221
Principle  Stress  S3 =-47.87411
Stress Intensity      = 48.66967
Von Mises Stress      = 43.18838

Mean (Membrane) accross section

          Sx          Sy          Sz          Sxy          Syz          Szx
        -0.029     3.112     0.925     -2.024     0.000     0.000
```

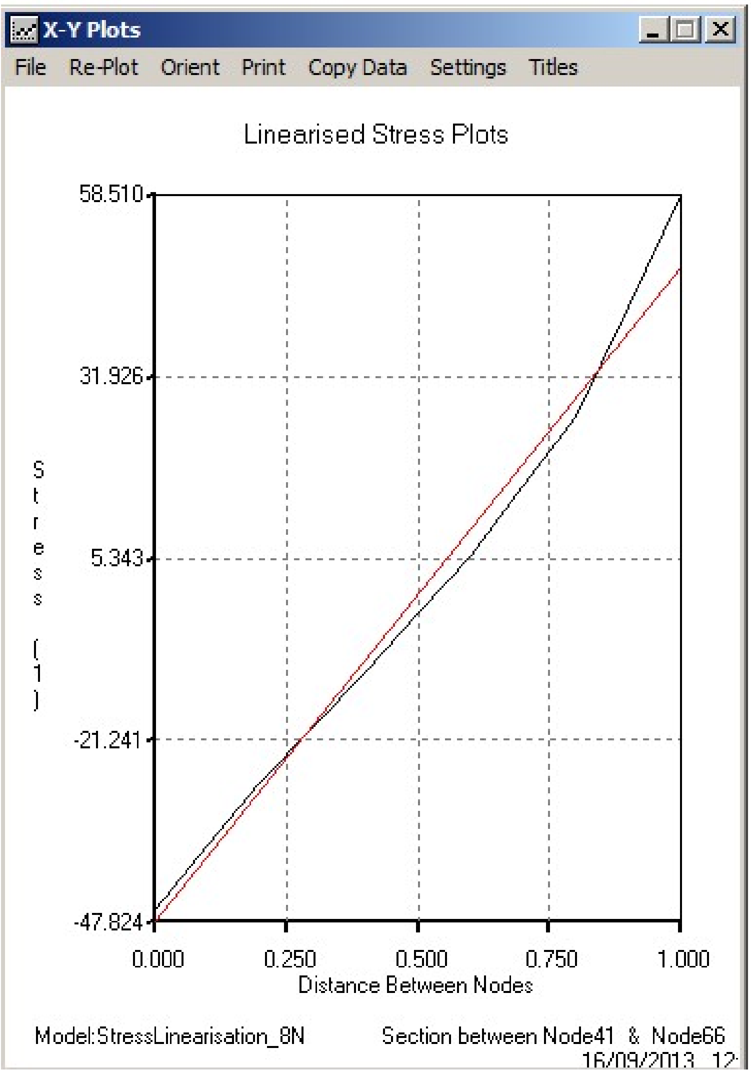
```
Derived Stresses at Mean Section
Principle  Stress  S1 = 4.102852
Principle  Stress  S2 = 0.925
Principle  Stress  S3 =-1.020352
Stress Intensity      = 5.123204
Von Mises Stress      = 4.479417

Stresses at Surface 2

          Sx         Sy          Sz          Sxy         Syz        Szx
Peak     58.510    0.800     17.790    -0.250     0.000     0.000
Linear   47.765    5.478     15.972    -2.493     0.000     0.000

Derived Stresses (Linearised Components) at Surface  2
Principle  Stress  S1 = 47.91193
Principle  Stress  S2 = 15.9721
Principle  Stress  S3 = 5.331618
Stress Intensity      = 42.58031
Von Mises Stress      = 38.38265
```

The **Re-Plot** command is used to plot the principle linearised directions stresses i.e the actual stress and the linearised stress distributions across the section.



-o-

4.9 Formatted Files Created in FS2000

The following lists the type of text output files that can be created in FS2000 (Standard Modules). The file extensions used by FS2000 are shown below. Some of the more specialised modules e.g. pile analysis, will produce additional files, these are described in the documentation for the respective module.

The asterisk represents the Case number.

If Sub-Cases are used the file extension used below is preceded with the sub case name e.g. **.<Sub-Case>.MTM** or **.<Sub-Case>.M7**

Standard Output		
Definition Data	**.MTM**	Definition Data
Individual Results Output	**.O***	Standard Output- Single Cases
Multiple Results Output	**.M***	Standard Output - Multiple Cases
Individual Unity Ratio	**.7***	Von Mises Stress Ratio - Single Case
Multiple Unity Ratio	**.T***	Von Mises Stress Ratio - Multiple Cases
Member Design Output - Structural and Pipework		
Summary Output	**.I***	Summary Output
Full Output	**.S***	Actual/Allowable Output
Individual Unity Ratio	**.8***	Member UR - Single Case
Multiple Unity Ratio	**.9***	Member UR - Multiple Case
Tubulat Joint Design Output		
Summary Output	**.K***	Summary Output
Full Output	**.J***	Actual/Allowable Output
Individual Unity Ratio	**.{***	Member UR - Single Case
Multiple Unity Ratio	**.}***	Member UR - Multiple Case

-o-

4.9B Saving and Opening Views

Views from any active Viewport may be saved so that they may be recovered at any time later in any Viewport. The facility gives the user the ability to save important views for inclusion in reports at a later time. Labels, dimensions, force diagrams may be included in views.

When Views are saved they are not saved as pictorial views they are saved as a set of view attributes i.e. view settings. When views are recovered the saved settings are used to re-draw the model with the previous view setting but with the model in its current state.

View Files are archived with the model.

Updating All Settings

By default, when a view is recovered not all attributes will be reset. If all attributes are required to be reset then the **Open View-Update All Settings** menu option must be made active. This will restore All attributes. Labels and visibility setting (e.g. show restraints) are the type of attribute associated with this option switch.

Task Dependency

Saved Views are TASK dependent. When a view is saved it will, depending upon the TASK active when it was saved, restore the TASK and its status at the time of saving. This includes the restoration of load cases and results cases. The facility gives the user the ability to save important results for inclusion in reports at a later time.

If you do not wish to restore the TASK only save views from the general view setting TASKs shown below with a *.

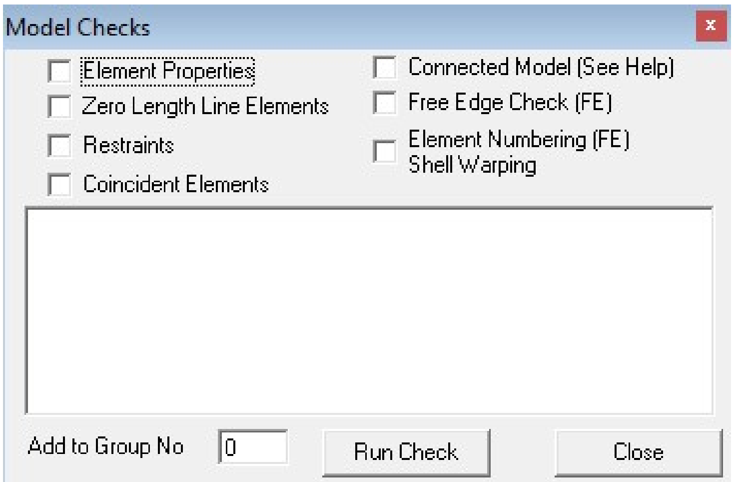
The following summarises TASK dependent features when saving views.

	TASK	Behaviour
*	Primary	General view settings
*	Model Definition	General view settings
	Load Definition	Task recovery, Load Cases and Load Case view settings
	Design Parameters	Task Recovery, Design Parameter plots and labels
*	Analysis	General view settings
	Output/Results	Task recovery, Results Case, Plots e.g. UR plots

-o-

4.10 Model Checking

The model checking routines provide additional checks to the basic model checks, which are carried out when the model is saved.



The **Add to Group** is used for the identification of erroneous nodes or elements. If a value greater than zero is entered then erroneous nodes or elements will be added to that group.

Element Properties

This option will check all elements and report if any property codes assigned to the element have zero properties defined. Only property codes with all properties defined as zero will be identified.

It will also check for elements whose property codes have not been defined. These will be graphically highlighted and added to the Error Group.

Zero Length Elements

This option will check for zero length beam and pipe elements.

Restraints

This option will check for the presence of basic restraints. It will not check if the model is correctly restrained. A model may not run because of lack of restraint even though it passes this check.

Coincident Elements

This option will check if are coincident (duplicated). In the case of solid elements the check will report coincident centres which may not necessarily be coincident elements. Coincident elements will be graphically highlighted and added to the Error Group.

Connected Model

This option will check to see if all elements of the model are interconnected ie the model is not in two or more basic chunks. To run this check it is first necessary to run the Bandwidth Optimiser (Analysis:Menu:Reseq:Bandwidth Optim) and create the solution renumber file. If this is not done the check will report all elements are correctly connected. The nodes of non-connected elements will be graphically highlighted and added to the Error Group.

Free Edge Check (FE)

This option will draw the free edges of finite element mesh regions. Lines that are visible within boundies of the mesh indicate cracks in the mesh i.e. non-connected elements. These are caused by the presence of coincident nodes. These can be removed with the use of the Merge(Nodes) command (Model Definition:FESolids:Merge(Nodes)).

Dissimilar elements type will also be identified this check.

Element Numbering

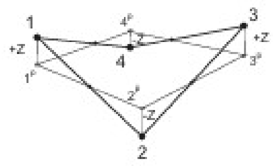
This option will check to see if there are any errors in the individual element numbering sequences. It includes checking that 2- solid elements are numbered anti-clockwise and that 3-D solids have their MNOP face in the local positive direction relative to the IJKL face. Both of these can be fixed using the Reverse Normals command (FESolids menu). Elements failing the test will be graphically highlighted and added to the Error Group.

Shell Element Warping

Shell elements in FS2000 are formed fro the superposition of a flat plate and flat membrane elements. It all of the nodes do not lay on a common flat plane the element is warped.

Warped elements should be avoided as what may be considered a small offset can greatly effect the accuracy of the element.

The maximum warping factor for all 4 node shell elements will be evaluated. The thickness Warping Factor in FS2000 is defined as WP = 2Z/t.



If the Warping Factor is greater than 0.1 the element can be added to a group.

-o-

4.11 Model Conversions

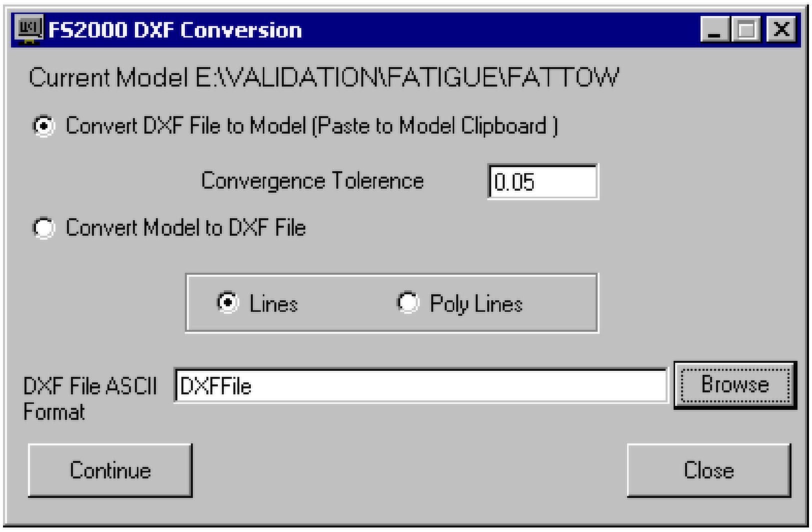
-o-

4.11.1 DXF Utility

A model conversion utility can be used to convert DXF files to FS2000 models and FS2000 models to DXF files .Only ASCII format DXF files can be converted.

The utility is started fusing the DXF-Conv command in the FS2000 program group in the Windows Start Menu.

When the utility is started, the form shown below will become visible.



Only beam type elements can be converted to DXF lines.

Each line or segment of a polyline in a DXF file will be converted to an element. If the vertices of a line are coincident or within the set tolerance of the vertices of another line then they will be connected to a common node at the location of the vertices.

When converting DXF files always ensure that Lines or Polylines do not overlay each other.

When a DXF file is converted it will be copied to the FS2000 clipboard. The **Paste** command (Model Definition\File menu) is used to add the DXF model definition from the clipboard to the current model.

The clipboard file, which is located in the FS2000 folder, is called buffer.mdl.

-o-

4.11.2 Importing GID Meshes

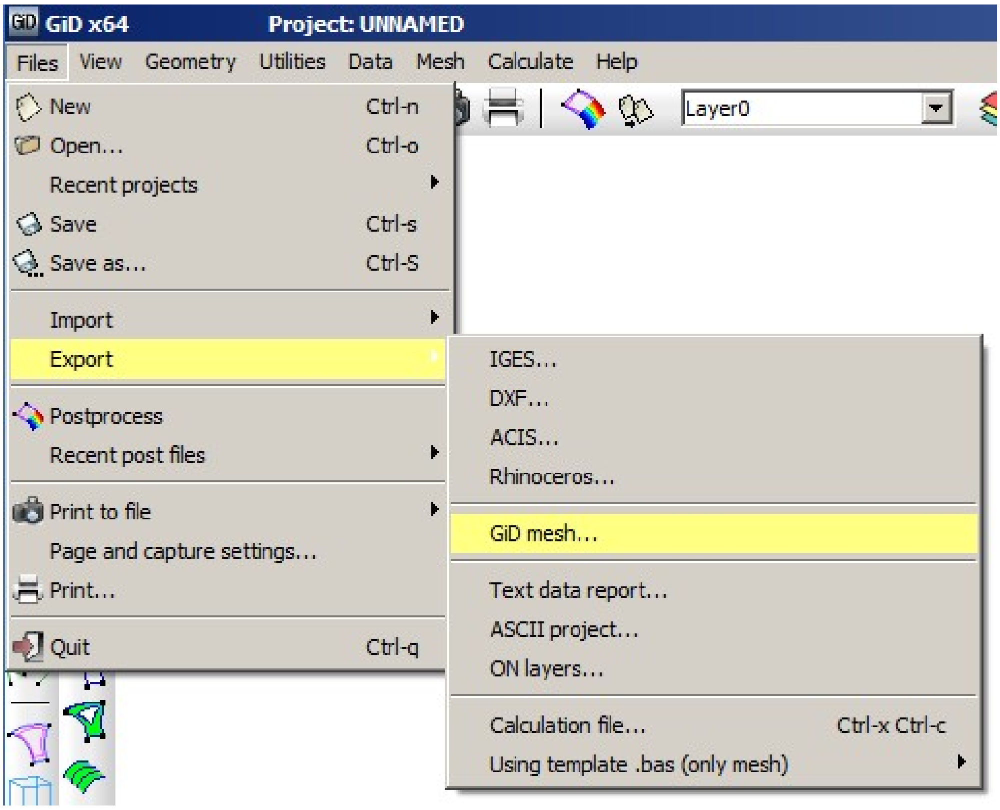
GID is a pre and post post processor for finite element analysis.

GiD allows the generation of large meshes in a fast and efficient manner for surfaces and volumes, using meshers based on different techniques.

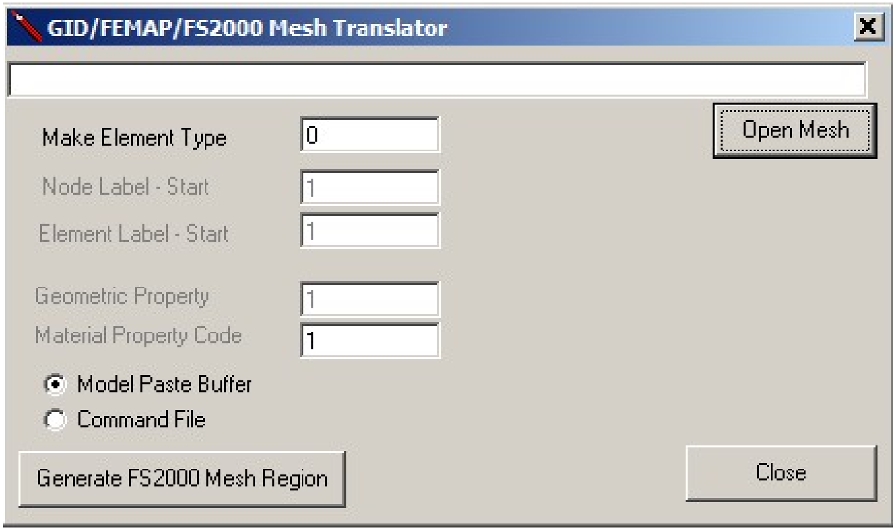
It is is relatively inexpensive and is highly recommended. The program has a free evaluation version which although limited on mesh size, can be very useful.

An interface program can be used that will convert a GID mesh to a FS2000 model.

From the file menu in GIF the Export/GID mesj.. commend can use to create a *.MSH mesh file.



The GID converter can be started for the Window's Start Menu FS2000/Utiliies/GID Mesh Conversion or starting FS2000/System/FSMeshCon.exe



The converted mesh can copied to the FS2000 paste buffer and then pasted into the model using the Paste Command or it can be copied to a <model>.U* file.

-o-

4.11.3 STAAD Model Conversion

Conversion Procedure

1. Open a new model in FS2000 – Do not define anything.
2. Start the STAAD- FS2000 converter program (FS2000\System\STAAD_Con.exe).
3. Select and open a STAAD definition file (.STD)
4. Click the Continue button.
5. Go to the FS2000's Model Generation TASK.
6. From the File menu select the Paste command.
7. Save the imported Model.
8. Exit the Model Definition TASK and then Save (Archive) the Model.
9. Open the Model from Archive format (this registers the load case etc with the model).

A log file is created (<name>.SRD.log) in the STAAD file folder. This may be useful to identify any errors associated with the conversion process. It does not show unrecognised STAAD commands only errors encountered interpreting recognised STAAD commands.

The converter should create an almost duplicate copy of the STAAD (where there are FS2000 compatible features) with the following exceptions.

- All elements will be assigned to Material Property Code 1
- Apart from pipe properties, the Geometric Property code table will only be populated with STAAD property descriptions. These entries represent, in order, the assignments from the MEMBER PROPERTIES command. All properties have to be re-entered using the FS2000 property utilities. It may be beneficial to edit the descriptions in the STAAD file to make the interpreted description more meaningful.
- The REPEAT LOAD command will be converted to a Load Case Combination. The ID number of the combination will be the same as that of the STAAD load case that contained the REPEAT COMMAND. If the STAAD load case contained other load definition commands then a load case of the same number will also be created. Note that STAAD load case combinations are numbered sequentially from 1 onwards and they will overwrite REPEAT combinations which use the same ID number. Use the Start Combination ID No to avoid this.
- Spring Restraints will be fixed.
- STAAD Groups (START GROUP DEFINITION command) will be saved as FS2000 groups. There will be a Group SET for each STAAD Group; the Group SET description will be that of the STAAD group. In each SET there will be one group and that will be the same group number as the Group Set number.

- UNITS – In most case these should be interpreted correctly and the model will be identified as a SI System or USA System FS2000 model. There could be a conflict if the unit definition in the STAAD files changes e.g. loads are defined in a different system to the model geometry.

Node & Element Labeling

It is possible that the STAAD model will have huge gaps in its numbering sequence. FS2000 is more efficient if such gaps are minimized. Theses can be removed or minimised by re-numbering the model after it is opened from Archive or at any future time.

-o-

4.11.4 Exporting to ANSYS

A FS2000 to ANSYS model converter (FS-ANSYS.EXE) is available upon request for AES.

This converter will create an ANSYS model based on the basic model geometry. No loading data is converted.

-o-

4.12 ETable Utility

The ETable is a utility that creates element plot data and output listings from solution results for specific situations that cannot be processed using the more generic post-processing routines.
The utility can only be run in batch mode.

The plot routine will plot the data values contained in the ETableX file at the fore and aft end of beam elements.
The X plot parameter corresponds to the MODE command line switch shown below.
The X plot selection is set in the Line Plot Settings form.

The current in built routines process the following result types (Modes)

1. ETABLE1 Maximum elasto-plastic stress and total strains for Plastic Pipe elements
2. ETABLE2 The maximum stress point stress ranges contained in .Y files obtained from the fatigue module FATIG2
3. ETABLE3 Evaluates the geometric curvature of string of beam elements based on the displaced nodal locations.
4. ETABLE4 Maximum elasto-plastic stress and accumulative(equivalent) plastic strains for Plastic Pipe elements
5. ETABLE5 Strain components and accumulative(equivalent) plastic strains for solid elements

The command format for these are described below.

ETABLE C1/C2/C3/C4/

C1 is the operation Mode. The functions of the other switches vary according to the mode of operation.

MODE = 1 Stress/Total Strain Processing (Type 6(6, 7 or 11) plastic pipe (see also MODE = 4)

The output in this mode is the elasto-plastic stress and the corresponding total axial strain ($\Delta\varepsilon$p + $\Delta\varepsilon$ e).

The output from this mode of operation is a binary ETableX plot file i.e. an ETable1 plot file and an output listing showing stresses and strains.

C1= 1 This mode is used to create an element plot table that can be used to plot the maximum plastic stress and total (elastic + plastic) strains for a Type 6 pipe element. It also creates a tabular listing which shows the stresses and strains at the pipe element integration points (see C4 parameter).

C2 is the DyNoFlex result case number

C3 is strain output limit for the result listing - %strain

C4 If C4=A then output strain listing will be appended to the StdOut individual output file e.g. **<modelname>.O"C2"**. If this omitted the listing will be written to **<modelname>.STST.O"C2".** Note that when using the A option, the ETABLE command should only be used once after the OUT6 command or it will append the strain data more than once.

Note that the strains that are plotted will be total strain or accumulative plastic strain depending upon which Mode was last executed i.e. Mode1 for total or Mode 4 for accumulative.

Stress data for specific elements can be listed at the end of an interact solution.

Plotting Stresses and Strains

The maximum pipe element axial stress and total axial strain data can be plotted of the model using the Table Values command (Output/Results TASK:Plots menu). Note that this command does not auto scale so use the scaling scroll buttons (diagram usually too high = scroll down). Also the strain output is in % strain regardless of the labeling (MPa).

The default is to plot maximum positive strains and stresses. Minimum negative strains and stresses can be plotted by changing the Line Plot option from Both to Major Axis (reselect from plot menu).

Note that the strains that are plotted will be total strain or accumulative plastic strain depending upon which Mode was last executed i.e. Mode1 for total or Mode 4 for accumulative.

MODE = 4 Stress/Accumulative Strain Processing (Type 6(7 or 11) plastic pipe

The output in this mode is the elasto-plastic stress and the corresponding Accumulative(Equivalent) plastic strain. If there is no plastic cycling this is the effective plastic strain.

The Accumulative(Equivalent) plastic strain includes the effects of hoop and radial strain (due to pressure) whereas the strains from MODE 1 is only axial strain will likely be different.

This is essentially the same as MODE 1 but produces the output file **<modelname>.STSTA.O"C2.**

Note that the strains that are plotted will be total strain or accumulative plastic strain depending upon which Mode was last executed i.e. Mode1 for total or Mode 4 for accumulative.

Stress data for specific elements can be listed at the end of an interact solution.

MODE = 2 Y File (FATIG2) Maximum Stress Point Stress Ranges.

The output from this mode of operation is a text ETableX plot file i.e. an ETable2 plot file.

C1= 2 With this mode an ETable Plot file will be created <modelname>.~ETAB.2.'RC' ('RC' is defined using C4).

C2 is the FATIG2 Y stress range file number

C3 is stress type Enter 1 or 2 for Chord or Brace Stress (/Direct Stress or Signed VM stress)

C4 is the ETAB file number of the i.e. 'RC'. To plot ETable X values, they must be associated with a standard post-processed result case. This parameter is used to make the association

MODE = 3 Curvature of a line of Beam Elements.

This mode will evaluate the curvature distribution in a string of beam elements based on the displaced nodal location. The main use of this routine would be to evaluate curvature when spar elements are used to model catenary type action.

The beam string is identified by using groups in a Group SET. The following restrictions apply:

- All elements in the group must be connected in line.
- All elements must all be of the same length.

The output from this mode of operation is a text ETableX plot file i.e. an ETable3 plot file.

C1= 3 With this mode an ETable Plot file will be created <modelname>.~ETAB.3.'RC' ('RC' is defined using C2).

C2 is the Result Case Number

C3 is the Group SET used to identify the elements by group attribute

C4 is the Group Number attribute

C5 is first or last node label at the ends of the beam string

MODE = 5 Accumulative Strain Processing for Solid Elements (Type30, 40 and 70)

This produces the output file **<modelname>.STSTS.O"C2.**

The output in this mode are the Gauss point:
Elastic and Total component strains and accumulative (VM equivalent) plastic strain.
Component stresses and equivalent VM stress.

C1 is the operation Mode

C2 is the DyNoFlex result case number

C3 is %strain output limit for the result listing - %strain

C4 If C4=A then output strain listing will be appended to the StdOut individual output file e.g. **<modelname>.O"C2"**. If this omitted the listing will be written to **<modelname>.STST.O"C2".** Note that when using the A option, the ETABLE command should only be used once after the OUT6 command or it will append the strain data more than once.

-o-

4.12.1 ETableX Format

The ETableX command can be use to plot both FS2000 generated ETableX plot data or data from user defined ETableX plot files.

The plot routine will plot the data values contained in the ETableX file at the fore and aft end of beam elements.

An ETableX file is identified by the file specification **<modelname>.~ETAB.'X'.'RC'**

X is the ETable mode ID
RC is the result case association number

Note that .~ are scratch files and will not be archived and will be deleted when the model is deleted. Copy to **<modelname>.UETAB.'X'.'RC'** if archiving is a requirement.

The X parameter is set in the Line Plot Settings form.

A ETableX file is a text file with the following format.

Ele_No, Fore_Ens_Data, Aft_End_Data

A typical file extract is shown below.

```
1,1.769E+07,1.966E+07
2,2.446E+07,6.344E+07
3,8542000,1.067E+07
4,1.272E+07,4.905E+07
20,2.446E+07,6.344E+07
```

-o-

4.13 Frame & Mesh Generation

This Frame Wizard utility is used to generate basic model geometry of some common structural configurations using only basic outline definition parameters. The utility has its own Help file.

The Frame Wizard is started from a command in the **TASK(Model Definition)** menu.

When used it enables the resulting node and element definition to be simply pasted into a new model i.e. a model with no definition. This model can then be saved.

If the generated model segment is required to included in an existing model i.e. copied into another model then Copy Sub-Model command in the model definition TASK can be used for this purpose.

The utility can generate commonly use beam element configurations:

- Basic Frame
- Roof Trusses
- Bridge Trusses
- Towers

It also has a utility that will generate basic pipe sections using quad shell elements. The utility can generate the following basic pipe related shapes:

- Pipe Bend (Elbow)
- Straight pipe
- Conical pipe

.

Meshing Templates

A sub folder in the Examples folder contains some basic models that can be copied or sub-meshed to form disk and annular meshes.

-o-

5 Fundamental Definitions (Element types etc)

5.1 Nodes

Nodes are identified by their label. Their only attributes are their co-ordinates and the optional Group attribute.

Nodes in their simplest interpretation may be considered as points that are used to define the ends of elements. However, in structural analysis they do play a far greater role than that.

Element properties establish the stiffness relationship between connecting nodes. The matrix solution yields the displacements of nodes. From these displacements, the forces in the elements of the structure are evaluated.

Each node has degrees of freedom associated with it. The number of degrees of freedom depends upon the analysis module used. In 2-D planar analysis there are three degrees of freedom: x and y translations and a z rotation. In 3-D analysis there are six degrees of freedom: three translations and three rotations.

For a model to be analysed successfully it is **ABSOLUTELY ESSENTIAL** that every node within the structure (within a structure means attached to an element) is not free to move in any of its degrees of freedom unless it is attached to an element which is part of a restrained system of elements or is itself retrained. (See Restrained Nodes and Soft spring Option).

The physical interpretation of this can be gained by considering the following analogies. In the case of 3-D analysis, the analogy would be to consider the node as a spherical ball that cannot be allowed to rotate about its own axes or translate in any of the three translational degrees of freedom. In the case of 2-D analysis, one considers the node as a cylindrical bearing that cannot be allowed to rotate about its own axis or translate in either of the two translational degrees of freedom.

Nodes need not just be used to define the ends of straight spans. Any number of nodes may be positioned along the mid-span of an element. Deflection are only evaluated at nodes

The accuracy of a linear beam element solution does not increase by increasing the number of nodes on a span, beam element formulations are exact. The accuracy of a finite element solution does depend on the number of nodes ie the element mesh density.

Free nodes not attached to the structure are always saved. Free nodes are ignored i.e. are not visible (apart from in the model definition TASK) nor listed in any output.

-o-

5.2 Element Types

The following summarises the basic type of elements that are available for analysis. The different element type can be freely used in the same model.

Beam Elements

Beam elements are used to represent skeletal structures or line type structures an are the most commonly used and most versatile element. Beam element properties can varied to model a large variety of physical configurations. Moment releases, offsets, tapered properties etc. can be applied to beam elements. The element **Type** attribute is used to define non-linear beam element types.

Solid Elements (Finite Elements)

A range of shell and solid finite elements are available to model plated or solid structures.

Pipe Elements

A pipe element is a type of beam element but has the properties of pipe and can be subjected to internal/external pressure loading. Data in the Geometric Property Table (Outside diameter and wall thickness) identifies a pipe element.

Curved Beam Element (Pipe Bend)

The element can be used to model curbed beams or pipe bends (elbows). It is defined in similar way to a beam elements but it has a radius of curvature as an additional defined property. It uses a third node (at the centre of curvature or tangent intersection point) to define its orientation

Rigid Link

A rigid link is a special form of beam element. It will transmit forces and moments between nodes but will not deflect. There are restrictions regarding the use of rigid links.

Couple Elements

A Couple element is a general node to node or node to ground spring element. It directly couples selective degrees of freedom of a node to another node or node to ground using a defined stiffness. Gap elements and other non-linear connection element are types of couple elements.

Gap Element

This is a non-linear couple element (Type 10, 11 or 12) used to model make and break contact configurations with and without friction. By applying interference (-ve gap) it can also be used to model pre-load. This is a common type of couple element a has a specific property form.

-o-

5.2.1 Beam Element Types

This section described the beam elements available in FS2000. The geometric and material properties of the elements are defined by reference to the property tables.

Type 0	Linear beam
Type 2	Linear bend element (3rd node - tangent intersection)
Type 3	Linear bend element (3rd node - centre of curvature)
Type 6	General Non-lineal beams
Type 7	Non-linear beam on distributed ground couples
Type 8	Non-linear beam on distributed ground non-linear springs
Type 15	Spar (Catenary) element with large displacement capability
Type 16	Linear beam with large displacement capability

-o-

5.2.1.1 Beam Elements

This section describes the elements that are most commonly used to model framed (skeletal) structures.

Basic Element Description

A beam element is the term used to describe structural sections and piping. All beam elements require to be connected to two nodes within the structure. When a beam element is connected to a node the element is connected to all the degrees of freedom applying to that node. When a second element is connected to the same node, the degrees of freedom are similarly linked and the resulting effect is that the two elements are rigidly connected. The node is still free to move but takes on the combined stiffness of each of the elements in each of the degrees of freedom of the node.

When an element is defined, the user assigns property code numbers which identify the properties of that beam from a property tables. This simplifies input. Consider a structure consisting of 100 elements, but comprising only three different types of elements. By using secondary code reference only the definition of three section types is required.

The property codes values refer to entries in model dependent property tables. Two types of property tables are used for beam elements, these are:

- The Geometric Property Table, this is used to define the geometric properties, i.e. the areas and I values of a beam.
- The Material Property Table, this is used to define the material properties, i.e. Young's Modulus, Poisson's Ratio, etc.

The following sections describe basic beam element types and their associated features.

Element Types

Curved Beams (Pipe Bends)

Local Orientation

Moment Releases

Offsets

Tapered Beams

Rigid Links

Tension Only/Compression Only

Gap Elements

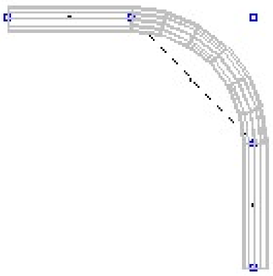
-o-

5.2.1.2 Curved Beams (Pipe Bends)

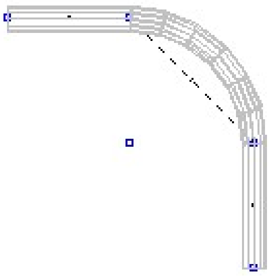
This is an elastic circular beam type element defined by its end nodes and a radius of curvature. The theta angle of the bend can vary between 0 and 90 degrees.

A Type 2 or Type 3 beam element identifies this type of element. The only difference between the two types is the node used for orientation.

The orientation of the bend is defined by a third node that is located at the center of curvature of the bend for a Type 3 bend or the intersection of the tangents for a Type 2 bend.



Type 2



Type 3

Properties are defined in the same manner as would be applied to straight pipe. Piping stress and flexibility factors can also be applied to the element.

Internal pressure, thermal, gravitational loading and UDLs (element & property code defined) can be applied to the element. Note that element and property code UDL will be applied as lumped nodal loads based on the chord length.

-o-

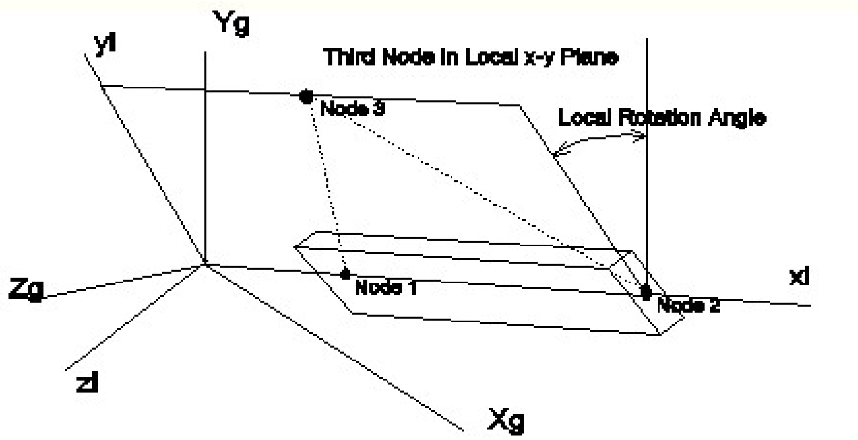
5.2.1.3 Local Orientation

The end nodes of an element define it global angular orientation. In 3-D analysis it also necessary to define the local rotation angle of line elements.

The local rotation angle of line element may be defined by:

- Direct entry of the rotation angle (ROT or Beta angle)
- Definition of a third node

When models are saved local rotation angles are re-evaluated based on the third node unless the third node is zero i.e. the third node will always take precedence over any defined rotation angle.



The convention used to define local element rotation is shown in more details in Section 4.3.2.

For intermediate angles other than those in the principle planes the general rule is that the local Y axis always points in the direction of global Y for zero element rotation (Local Rot = 0).

Local Rotation can be defined in the Element Input box(by rotation angle definition or third node) or graphically by picking the third node.

Local element rotation can be shown graphically. In a non virtual views they are represented by the equivalent of the webs of an I beam. The yellow edges indicate the fore end and the direction of the local Y axis.

-o-

5.2.1.4 Moment Releases

Element end moment releases can be applied to element ends by defining release codes (0 to 3). These codes may be defined in the Element Input box when the element is being defined or using the Moment Release command after the element has been created. The latter is usually more convenient.

Release Codes

RelZ is used to identify the Major axis release in the Element Input box.

RelY is used to identify the Minor axis release in the Element Input box. (not used in 2D analysis)

The code convention used for end moment releases are:

0	Both ends fixed (default)	
1	Node 1 (fore) end released	Node 2 (aft) end fixed
2	Node 1 (fore) end fixed	Node 2 (aft) end released
3	Both ends released	

Translational & Torsional Releases

Translational releases can be modelled by specifying a "stiff" spring/couple between coincident nodes. Alternatively, for torsional releases the torsional constant J may be made zero in the main element or in a dummy element connected to it.

-o-

5.2.1.5 Offsets

Rigid beam offsets permit the definition of a beam in which only part of its length is elastic and/or in which the end of the beam does no intersect the actual node location. They are used to include the effect of joint eccentricities without incurring the use of additional joint nodes.

Note that offset may also be applied to shell elements.

The effects of beam offsets could be modeled using rigid links or stiff elements but these would require additional nodes.

When beam offsets are defined only the elastic portion of the beam is used for load definition, self weight generation and code checking.

Offsets are defined using the Offset Definition box. They are copied or deleted using the Beam Element Definition form.

If the local rotation of an offset element is defined using a third node and the offsets use global definition, then the local rotation angle will be evaluated using the orientation of the offset position of the element ends.

If the local rotation of an offset element is defined by the 3rd node method and local offsets use local definition then the element will first align and then the offsets will be applied

Offset co-ordinates are defined using either the global axis or the local axis of a selected reference element.

There are some restrictions/effects to be considered when using the subject element as the reference element. These are

> If the element is rotated (local rotation) the element will rotate (sweep) about the original element axis rather than rotate about its own offset axis.
>
> If the element is parallel to the Y axis the elements will be rotated through 90 deg unless the offsets at both ends of the beam are the same.

Large Displacement Analysis

Offset should not be use with non-linear elements when the large displacement option is active eg Type 6 as they will be ignored during the solution phase.

Viewing Offsets

Beam element offsets can be made visible using the options in the Model Display Switch form. They are always visible when the Virtual view is active.

-o-

5.2.1.6 Rigid Links

Rigid links are special elements that transmit forces and moments but which do not themselves deform. The displacements and forces at each end of rigid link are not independent therefore one node attached to the link may be eliminated in the solution phase.

Rigid link effects are implemented by applying an offset to any element connected to the fore end of the rigid. This imposes restrictions in the use of rigid links (see below). Because of these restrictions if it may be preferable to use a stiff massless beam or a stiff Type 0 couple.

Rigid links are specified by defining the Geometric Property code (Geom box in Element Definition box) of the element as -1. Rigid links can be identified in a model by using Groups and the Add Elem by Attribute command.

Since the displacements and forces at each end of rigid link are not independent one node attached to the link may be eliminated.

Some limitations/restrictions on rigid links are listed below.

- The fore end of rigid link cannot be connected to the fore end of another rigid link
- The fore end of a rigid link cannot be restrained
- The fore end of a rigid link cannot be connected to a solid element or a non-linear couple element
- Nodal forces or prescribed displacements cannot be applied to the fore node
- Element loading cannot be applied to a rigid link
- Offsets cannot be applied to a rigid link
- Releases do not apply to a rigid link
- Rigid Links will be ignored if the DyNoFlex large displacement option is active

Other Ways to model Rigid Links

- Using stiff conventional beam element.
- Using a Type 0 couple

-o-

5.2.1.7 Tapered Beams

Tapered beam elements defining using property codes for each end of the beam (Beam Element Definition box).

The Geom code is used to define the geometry of the fore end i.e. Node 1 end of the beam and the Taper code is used to define the aft end i.e. Node 1end of the beam. If a zero value for this code is used (default) then a parallel beam using the Geom code properties is assumed.

The Taper code also defined the type of taper. A **Type A** is used if the Taper Code is entered as a positive value and a **Type B** is used if the Taper Code is input as a negative value.

Taper Type A assumes a linear variation in c.s.a. and Iy (minor axis) and a cubic variation in Iz and torsional constant.

a = a(x)

Iy = Iy(x)

Iz = Iz(x3)

J = J(x3)

This is exact except for the torsional value for rectangular beams that vary in only one dimension. The user must establish the element y axis parallel to the side which varies in size.

Taper Type B assumes a quadratic variation in c.s.a. and a fourth order variation in Iy, Iz, and torsional constant.

a = a(x2)

Iy = Iy(x4)

Iz = Iz(x4)

J = J(x4)

This exact for square and circular sections. A small error will be introduced for rectangular beams that vary in both axis.

For the purpose of load generation all tapered beam are assumed to have a linearly varying self weight. (slightly conservative)

-o-

5.2.1.7 Tension Only/Compression Only

Tension Only or Compression Only elements are identified and specified by the Geometric Property code using the Non Linear Geometric Properties input box.

When running the 3-D Standard solver all beam elements, apart from bend elements are considered to be Type 0 and can be designated as 'tension only' or 'compression only' elements. They may be applied to beam and pipe elements.

The Tension/Compression Only option must be activated when using the 3-D Standard solver.

-o-

Type 0 Linear Beam

This is the default linear beam element for beam and pipe analysis.

It has the following non-linear options when run with the 3-D Standard linear solver. Treated as a linear beam in non-linear solvers.

- Stress stiffening/ P- Delta
- Tension/Compression Only

Note: A Type 0 Beam is treated as a linear beam in the non-linear solvers.

Stress stiffening/ P-Delta

In structural analysis P-Delta analysis is the general term used to describe analysis that takes into account the effects of lateral deflections and axial force (geometrical stiffening).

Two formulations are available,

- Beam Stability Functions (Structural Analysis, R.C. Coates et al) - Most accurate for buckling but excludes shear deformation.
- Beam Geometric Stiffness Matrix (Theory of Matrix Analysis, J.S. Przemieniecki)

If the Y Direction Shear is specified as zero Beam Stability Functions will be implemented otherwise Beam Geometric Stiffness will be implemented.

In the program option for this effect is termed **Stress Stiffening (P-Delta)**.

P-Δ or P-δ? P-Δ is the global structure displacement effects(sway) and P-δ is the local member (span) effects. In terms of program implementation there is no difference between the two and depending upon mesh density both effects will be included. At least one mid-span node is required for P-δ to be included. If double curvature exists in the span 3 nodes will increase the accuracy.

Tension Only/Compression Only

The **Non-Linear** button in the Geometric Input box of FS2000's GUI is used to define this type of bi-linear element.

Restrictions

Solution Option	Property Recognition	Comments
3-D Standard	All	Tension/Compression Only or P-Delta Effects
3-D Non-linear	All	Linear
Eigen Frequency	All	Linear + P-Delta
Eigen Buckling	All	Linear + P-Delta
Dynamic Response	All	Linear
DyNoFlex	All	Linear

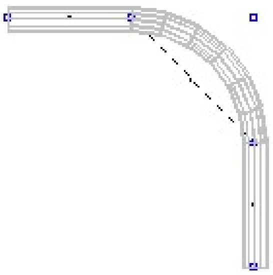
-o-

Type 2 Bend Element

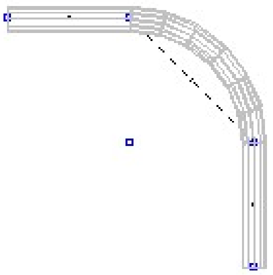
This is a circular elastic beam type element defined by its end nodes and a radius of curvature. The theta angle of the bend can vary between 0 and 90 degrees.

The orientation of the Type 2 bend is defined by a third node that is located at the intersection of the tangent lines.

When used for pipework analysis the element may be assigned with flexibility and stress intensification factors.



Type 2



Type 3

Properties are defined in the same manner as would be applied to straight beam or pipe elements. Piping stress and flexibility factors can also be applied to pipe elements.

Internal pressure, thermal, gravitational loading and UDLs (element & property code defined) can be applied to the element. Note that element and property code UDL will be applied as lumped nodal loads based on the chord length.

Restrictions

Not to be used for DyNoFlex large displacement solutions if undergoing significant solid body displacement.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	Linear
Non-linear	All	Linear
Eigen Frequency	All	Straight Linear + PDelta
Eigen Buckling	All	Straight Linear + PDelta
Dynamic Response	All	Linear
DyNo Flex	All	Linear

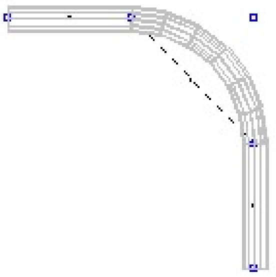
-o-

Type 3 Bend Element

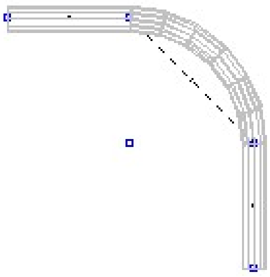
This is circular linear elastic beam type element defined by its end nodes and a radius of curvature. The theta angle of the bend can vary between 0 and 90 degrees.

The orientation of the Type 3 bend is defined by a third node that is located at the center of curvature of the bend.

When used for pipework analysis the element may be assigned with flexibility and stress intensification factors.



Type 2



Type 3

Properties are defined in the same manner as would be applied to straight beam or pipe elements. Piping stress and flexibility factors can also be applied to pipe elements.

Internal pressure, thermal, gravitational loading and UDLs (element & property code defined) can be applied to the element. Note that element and property code UDL will be applied as lumped nodal loads based on the chord length.

Restrictions

Not to be used for DyNoFlex large displacement solutions if undergoing significant solid body displacement.

Solution Option	Property Recognition	Comments
3-D Standard	All	Linear
Non-linear	All	Linear
Eigen Frequency	All	Straight Linear + PDelta
Eigen Buckling	All	Straight Linear + PDelta
Dynamic Response	All	Linear
DyNoFlex	All	Linear

-o-

Type 6 General Non-linear Beam

This is a general non-linear beam element used for modeling straight beam or straight pipe. It has the following non-linear features.

- Stress Stiffening/ P-Delta.
- Large Displacement. (DyNoFlex solver only).
- Tension-Compression only (3D-Standard solver only).
- Beam (ideal beam) plasticity and Stress/Strain for pipe elements.
- Moment-Curvature stiffness definition.
- Soil- pile interaction (Refer to FS-Pile documentation) - Pile elements have to be specifies as Type 6 elements.

Stress stiffening/ P-Delta

In structural analysis P-Delta analysis is the general term used to describe analysis that takes into account the effects of lateral deflections and axial force (geometrical stiffening).

Two formulations are available,

- Beam Stability Functions (Ref:Structural Analysis, R.C. Coates et al) - Most accurate for buckling but excludes shear deformation.
- Beam Geometric Stiffness Matrix (Ref:Theory of Matrix Analysis, J.S. Przemieniecki).

The latter is the most commonly used formulation used by structural analysis programs but the former will produce more accurate bending moment results in tension controlled structures.

If the Z Direction Shear is specified as zero then Beam Stability Functions will be implemented otherwise Beam Geometric Stiffness will be implemented. Note that Type 16 beams always use Stability Functions.

In the program options the term for this effect is **Stress Stiffening (P-Delta)**.

P-Δ or P-δ? P-Δ is the global structure displacement effects(sway) and P-δ is the local member (span) effects. In terms of program implementation there is no difference between the two and depending upon mesh density both effects be included. At least one mid-span node is required for P-δ to be included. If double curvature exists in the span at least 3 nodes should be used.

Large Displacement

When the deflections are large it is possible to account for the change in stiffness by updating the global stiffness matrix based on the shape of the deformed structure. The program uses a co-rotational (convected coordinate system) approach to large displacement(Rankine & Brogan) in which the non-vectorial nature of rotations is accounted for. In this approach the displacement that cause stresses are separated from those due to rigid body motion. The implicit assumption in this method is that whilst rotations can be large (rigid body) the rotations that cause stress must be small. Rotations are limited to 180 degrees.

Tension-Compression Only

Geom Type	
Type 10	Compression only
Type 11	Tension only

Plasticity (Material Model)

Material non-linear behaviour in beams can be accounted for by:

- Ideal Plasticity behaviour (Frame Plasticity)
- Defined moment curvature relationships

Material non-linear behaviour in pipe elements can be accounted for by:

- Ideal Plasticity behaviour (Frame Plasticity)

- Defined moment curvature relationships
- Defined Stress/Strain curves

Frame Plasticity uses a fictitious force approach in which excessive moments above the plastic limits are re-distributed to connected elements as a moment at the node undergoing plasticity (equilibrating shear loads are also applied to the element undergoing plasticity). This approach permits unloading down the yielded surface. The method is element length dependent in that it assumes lumped plasticity at the span ends and segmenting a span provides a more concentrated effect because the equilibrating shear loads closer to the hinge point. However, element length effects only the stiffness characteristics, limits loads and energy absorption are not effected by length effects. This change in stiffness would not be that apparent in frame type structure where the end beams are connected to one or more other beams. Generally in frames this method is most efficient (sol'n convergence to collapse-number of iterations) when the elements are continuous between the points of shear loading and points of shear support. If in doubt, do a length sensitivity check.

The **Non-Linear** button in the Geometric Input box of FS2000 is used to define the material model i.e. **GEOM TYPE** and the geometric plastic section properties. When selecting properties from element libraries the plastic modulus values will be entered automatically.

Ideal beam plasticity (frame plasticity) is where the moment and axial resistance at the nodes of the beam are restricted to the plastic limits (F_p , M_{px} & M_{py}). Types 1-4 employ ideal beam plasticity and the plastic limit are evaluated using the following limits and the interaction formula shown below.

F_p = Yield*Area
M_{px} = Yield*PlasticSz
M_{py} = Yield*PlasticSy
T_p = 0.577*Yield*PlasticTorsion

Note that the Plastic Torsional modulus may required to be additionally factored (assumes Von-Mises - 0.577) to account for a different failure theory e.g. 0.5 for max shear strength.

Moment curvature can be applied to all beam and pipe elements.

Plasticity based on a defined stress-strain curve which a Von-Mises yield function can only be applied to pipe elements. This uses a thin wall model so there may be slight differences from standard stress output which uses the moment and thick wall properties for stress evaluations.

The **GEOM TYPE** designation is used to define the type of non-linear plastic behavior. The interaction surfaces (Type 1 to 4) are for frame plasticity.

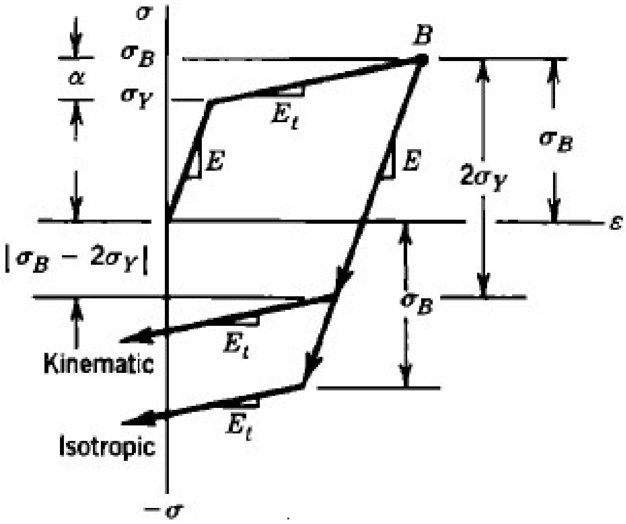
Type 1	No Interaction	$F/F_p = 1;\ M_x/M_{px} = 1;\ M_y/M_{py} = 1$
Type 2	Linear Interaction	$F/F_p + M_x/M_{px} + M_y/M_{py} = 1$
Type 3	Tube Interaction	$(M_x/Mr_{px})^2 + (M_y/Mr_{py})^2 = 1$
Type 4	Beam Interaction (I Sections)	$(M_x/Mr_{px})^2 + M_y/Mr_{py} = 1$
Type 6	Stress-Strain Curve (Pipe)	Non-linear elastic
Type 7	Stress-Strain Curve (Pipe)	Kinematic memory model
Type 8	Moment Curvature Beam	Non-linear elastic
Type 9	Moment Curvature Beam	Bi-linear kinematic memory model
Type 10	Moment Curvature Beam	Multi-Linear kinematic memory model
Type 11	Stress-Strain Curve (Pipe)	Isotropic memory model

In the above for I Beam Interaction (Ref:Limit States Design of Steelwork, David Nethercot):

$Mr_{px} = (1 - 2.5n^2)M_{px}$ for n < 0.2
$Mr_{px} = 1.125(1 - n)M_{px}$ for n > 0.2
$Mr_{py} = (1 - 0.5n^2)M_{py}$ for n <.447
$Mr_{py} = 1.125(1 - n^2)M_{py}$ for n >.447
where $n = F/F_p$

In the above for Tube Interaction:(Ref:Steelwork Design Guide to BS5950, SCI)

$Mr_p = M_{px} \cos(n\pi/2)$ where $n = F/F_p$



Plasticity Based on a Defined Stress Strain Curve

If pipe elements are defined as being Geom Type 6, 7 or 11 element types, then plastic behavior of the element can be defined using a stress strain curve.

Type 7 and 11 elements use a Von Mises failure yield criteria using only axial, bending and hoop interaction (shear stresses are not included).

(References.Computational Methods for Plasticity, EA de Souza, D Peric, DRJ Owen,2000).

Geom Type 6 is an elastic non-linear formulation defined by a piecewise stress-strain curve. The stress-strain curve is defined in the RC constants table. The Table ID is required to be the same ID as that used for the element's Material Property code. Use the Geom Type 7 for cyclic loading. Although end cap and the Poission effect of internal pressure are included, the onset of plasticity is based only on axial effects i.e. hoop stress interaction is neglected. Use Type 7 or 11 if hoop is present.

Geom Type 7 is a kinematic Von-Mises plasticity model which uses a piecewise stress-strain curve (RC constants table) or a Ramberg-Osgood type relationship to define the stress-strain relationship. This kinematic cyclic model has preserves the curve shape so that the Bauschinger effect on cycling loading is included. The Table ID is required to be the same ID as that used for the element's Material Property code. Hoop stress effects on plasticity are included (hoop stress must be less than yield). Shear stress effects on the yield surface are neglected are neglected. The element uses numerical integration at 12 points on the circumference to evaluate the plastic stress-strain behavior. The output from this model includes accumulative strain.(Ref: Computational Inelasticity / J.C. Simo, T.J.R. Hughes, Springer, 1998.)

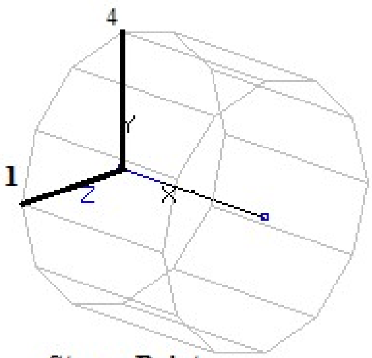
Geom Type 11 is a isotropic Von-Mises plasticity model which uses a piecewise stress-strain curve (RC constants table) or a Ramberg-Osgood type relationship to define the stress-strain relationship. The Table ID is required to be the same ID as that used for the element's Material Property code. Hoop stress effects on plasticity are included (hoop stress must be less than yield). Shear stress effects on the yield surface are neglected are neglected. . The element uses numerical integration at 12 points on the circumference to evaluate the plastic stress-strain behavior. The output from this model includes accumulative strain..(Ref: Computational Inelasticity / J.C. Simo, T.J.R. Hughes, Springer, 1998

In Type 6(7) and 6(11) elements the hoop stress and radial stresses are included when evaluating the effective stress and corresponding yield function. This enables the NR solution to adjust the axial components to bring the stress points to the yield surface and enables the evaluation of axial, hoop and radial plastic strains required to evaluate the effective plastic strain . Note that the hoop stress must be always be below yield otherwise the solution will fail.

The ETable utility is used to extract the plastic stresses and strains from the solution. This utility also enables the maximum element stresses and strains to be plotted using the using the Table Value command (Output TASK/Plots menu). The elasto-plastic stresses (in the output) are the axial (longitudinal) pipe components. Both total axial strain and accumulative effective plastic strains can be obtained. The accumulative strains include the plastic strain components in the hoop and radial directions.

Initial Plastic State - The Pipe Moment Curvature utility described below can be used to define the initial plastic state (strains in Type 6(7) & 6(11) elements). It does this by creating a special restart file that contains the necessary material model plasticity parameters. This would be used in cases where a defined curvature has initially yielded the section.

Element Length Considerations - The method is element length dependent. This element assumes a linear stiffness distribution between stress points at each end of the beam. In plastic bending the length of the plastic zone is short therefore ensure that long beam elements do not span into or across plastic zones. This prevents the linear sections becoming too soft and the plastic zone being too stiff. If in doubt, do a length sensitivity check. Generally good results will be obtained in most cases for element lengths less than 2D and can be more if the moment variation is low.



Stress Point Convention

Moment Curvature

If elements are defined as being Geom Type 8 or Geom Type 9 elements, then the bending stiffness behavior of the element is defined using a moment curvature relationship which are evaluated using cubic shape functions. The bending stiffness behavior can be linear or non-linear.

Note that Non-linear **Material** solution option is required to be active for the moment curvature option (See Type16 elements for linear moment curvature element that does not require the Non-linear **Material** solution option to be active).

This moment curvature element is also well suited to the analysis of **Flexible Pipe** that has a relatively very low bending stiffness properties and where the displaced shape is predominantly due to tensile stiffening action. The moment is based on curvature (numerical integration) is evaluated using cubic displacement shape functions. This approach lessens the inaccuracy which can sometimes be present when explicit beam element stiffness coefficients and nodal displacements are used to evaluate the moment in tension controlled configurations (where shorter elements would be required increase accuracy).

The non-linear characteristics are define using RC curves. If the RC table entry does not contains any data i.e. does not exist or has zero entries the z axis then EI values will be used to define a linear moment curvature relationship. Similarly, if Sz is specified as zero the property code IE values will be used i.e. becomes a linear beam.

Geom Type 8 - The bending stiffness of this element is defined using a multi-linear moment curvature relationship. The moment curvature relationship is a non-linear elastic model. The RC constants table is used to specify the moment curvature relationship. The RC table entries for the two axis are referenced using the Plastic Sz and Sy where the value of Sz and Sy are the RC table property codes. If Sz = Sy then the resultant moment will be used to obtain the moment limits based on the RC(Sz) curve.

Geom Type 9 - This is the similar to the Type 8 but the Type 9 employs a kinematic bi-linear memory model. First two points in the RC curve define the bi-linear curve. Note that there is no interaction between the two axis.

Geom Type 10 - This is the similar to the Type 8 but the Type 10 employs a kinematic multi-linear memory model. Note that there is no interaction between the two axis.

Element Length Considerations - The method is element length dependent. Avoid long beam elements that span across points of contraflexure - split the span. If used in plastic bending - the length of high curvature is short therefore ensure that long beam elements don't span into or across plastic zones. Similar to but less sensitive than stress-strain plasticity. If in doubt, do a length sensitivity check.

RC Constants Table

The RC constants table is used to define the non-linear curve. Each entry in the table defines a piecewise curve. Up to 7 data point can be used to define the curve.

The curve always starts at the origin. The RC-X1 and RC-Y1 are used to define the first point, which cannot be at the origin. **The first point must produce a positive slope segment**. Other segments of the curve may be positive or negative. Segments of the curve cannot be vertical (infinite stiffness).

Pipe-Moment-Curvature Utility

The Pipe Moment Curvature utility can be used to evaluate the moment curvature relationship for a specific pipe type (single or multi-layer). This utility will create the RC table entry for a single or multi-layered pipe with defined non-linear material properties.
It can also be used to evaluate the moment curvature relationship following a cyclic yielding e.g pipe reeling.
The RC table entries for stress-strain properties can be generated based on definition by the following types:

- Linear with constant yield (Ideal plasticity)
- Bi-Linear
- Ramberg-Osgood

It can also be used to create a DyNoFlex restart file that contains the initial plastic strains following a plastic curvature cycle e.g. pipe reeling.

The utility can be started from the Windows Start Menu or directly (FS2000\System\PipeMom_Curve.exe). If has its own Help file.

Special Features

The **ESTR** **load definition** command can be used with this element. These commands can also be used with all non-linear beam type elements (eg Types 7, 8, 15 & 16)

Restrictions

Tension/compression only not available with this element when using non-linear solvers (Use Type 15 or connect through a couple).

Solution Option	Property Recognition	Comments
3-D Standard	All	Interpreted as a Type 0 beam type
3-D Non-linear	All	Non-linear - no large displacement option
Eigen Frequency	Mass included	Interpreted as a Type 0 beam type
Eigen Buckling	All	Interpreted as a Type 0 beam type
Dynamic Response	Mass included	Modes from Frequency Sol'n
DyNoFlex	All	Non-linear (Interpreted as a Type 0 beam type for the linear soln option)

-o-

Type 7 NL beam on distributed ground couples

This element is a general non-linear beam element (Type 6), which incorporates a ground support in the form of ground couples (Type 0, Type 10 or Type 12 Couple) which have their stiffness properties defined in terms of unit length. The advantages of this element is that additional node couples do not need to be defined. The element will behave in a similar manner to a beam supported on nodal ground couple at each end of the element and the stiffness of the foundation is defined in terms of unit length.

The element is extremely useful for modeling beams or pipes supported on a frictional or frictionless ground surface e.g pipeline on seabed.

The properties of the ground couple are defined by the CO property of the element. The value of the CO attribute is used to reference an entry in the IC Constants table. For this element the IC table is data is interpreted as follows.

ICO Is used to define the orientation of the support.

If a zero value is specified then this will result in the local axis of the couple aligning to the local axis of the beam and with the local gap x-axis pointing towards the direction of the global y axis (similar to a beam with the local orientation defined as zero). This type a alignment is very useful when using Gap couples (Type 10) to model ground support since no additional ordination definition is required and the ground vertical direction will always be normal to the beam x axis in a direcrion towards glogal y.

A non-zero value is used to reference a coordinate system. The support couple local coordinate system will align to the defined coordinate system.

IC1 This is used to define the properties of the couple by reference to an entry in the Couple Elements Property Table. The properties are interpreted in the same manner as would be done for a standard (node connected) couple element. The support may be defined as a gap element (Type 10 or 12 Couple) or a standard linear spring (Type 0 Couple). Normal damping cannot be defined using the couple properties for this type of element.

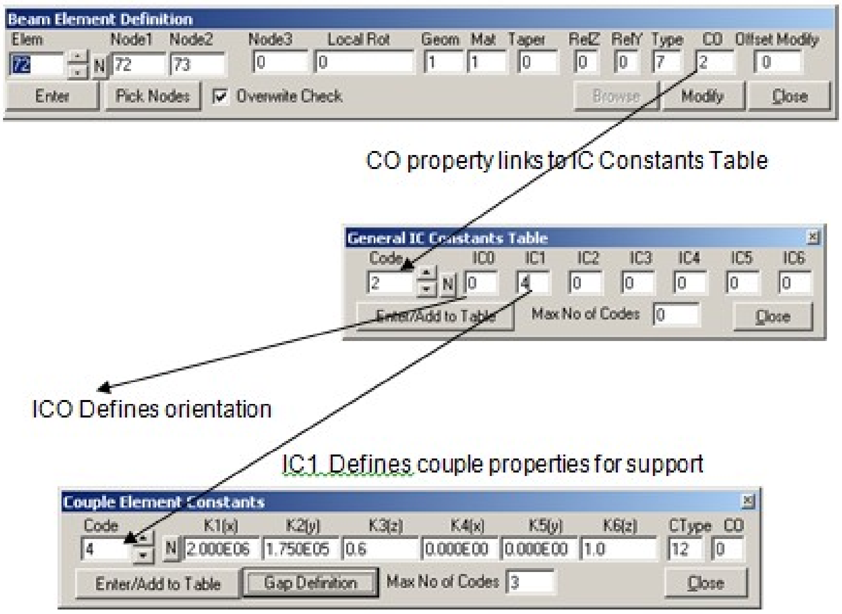
Special Features

The **ESTR** and **SEFO** **load definition** commands can be used with this element. These commands can also be used with all non-linear beam type elements (eg Types 7, 8, 15 & 16)

Restrictions

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	Interpreted as a Type 0 beam type. No ground support
3-D Non-linear	All	Non-linear - no large displacement option
Eigen Frequency & Buck'g	Mass included	Interpreted as a Type 0 beam type. No ground support
Dynamic Response	Mass included	Modes from Frequency Sol'n
DyNoFlex	All	Non-linear only - Ignored in a Linear Solution

The example below illustrates how properties are referenced. In this example **Elem 72** using **Couple Property Code 4** to define the vertical support.


Beam Element Definition
CO property links to IC Constants Table
General IC Constants Table
ICO Defines orientation
IC1 Defines couple properties for support
Couple Element Constants

-o-

Type 8 NL beam on distributed non-linear springs

This element is a general non-linear beam element (Type 6), which incorporates a ground support in the form of elastic non-linear translational springs which have their stiffness properties defined in terms of unit length. The element will behave in a similar manner to a beam supported on nodal non-linear ground springs (Type 4 Couple). The advantages of this element is that additional node couples do not need to be defined and the stiffness of the foundation is defined in terms unit length.

The non-linear properties of the spring may be defined differently for each local **x**, **y** or **z** direction and differently depending upon whether the movement is positive or negative.

The element is extremely useful for modeling buried beam/pipe elements. For this type of application the non-linear soils stiffness can be specified to suit the direction of movement. Note that this is an elastic non-linear spring which my not be suitable for cyclic loading. If cyclic loading is of importance then Type 7 couples should be considered.

The properties of the ground couple are defined by the CO property of the element. The value of the CO attribute is used to reference an entry in the IC Constants table. For this element the IC table is data is interpreted as follows.

ICO Is used to define the orientation of the support.

If a zero value is specified then this will result in the local x-axis of the non-linear support springs aligning with the local x-axis of the beam. The local y the direction of the support springs will point in the direction of the global y axis (similar to a beam with the local orientation defined as zero). This type of alignment is very useful for the modelling of buried pipelines.

A non-zero value is used to reference a coordinate system. The support spring will align to the defined coordinte system.

IC1 to IC6 These are used to defined the non-linear elastic stiffness properties for each of the local translational spring directions. It does this by referencing an entry in the RC Contants Table. The directions referenced by the IC constants are given below. If a zero value is specified the no stiffness will be applied in that direction

IC1 + X Direction Stiffness

IC2 + Y Direction Stiffness

IC3 + Z Direction Stiffness

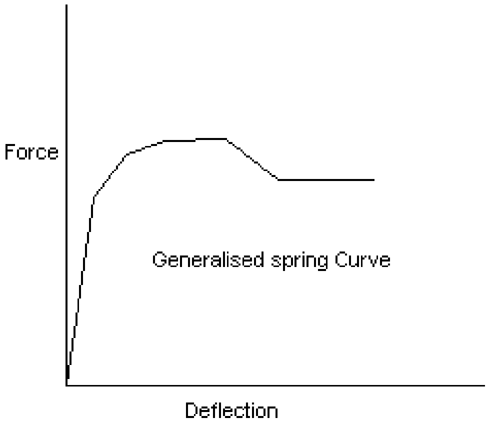
IC4 - X Direction Stiffness

IC5 - Y Direction Stiffness

IC6 - Z Direction Stiffness

RC Constants

In this element the RC constants table is used to define a non-linear spring curve ie a force/defection curve. Each entry in the table defines a piecewise curve. Up to 7 data point can be used to define the curve. The curve always starts at the origin. The RC-X1 and RC-Y1 are used to define the first point, which cannot be at the origin. **The first point must produce a positive slope segment**. Other segments of the curve may be positive or negative. Segments of the curve cannot be vertical (infinite stiffness). Also avoid using very stiff segments ie near vertical.



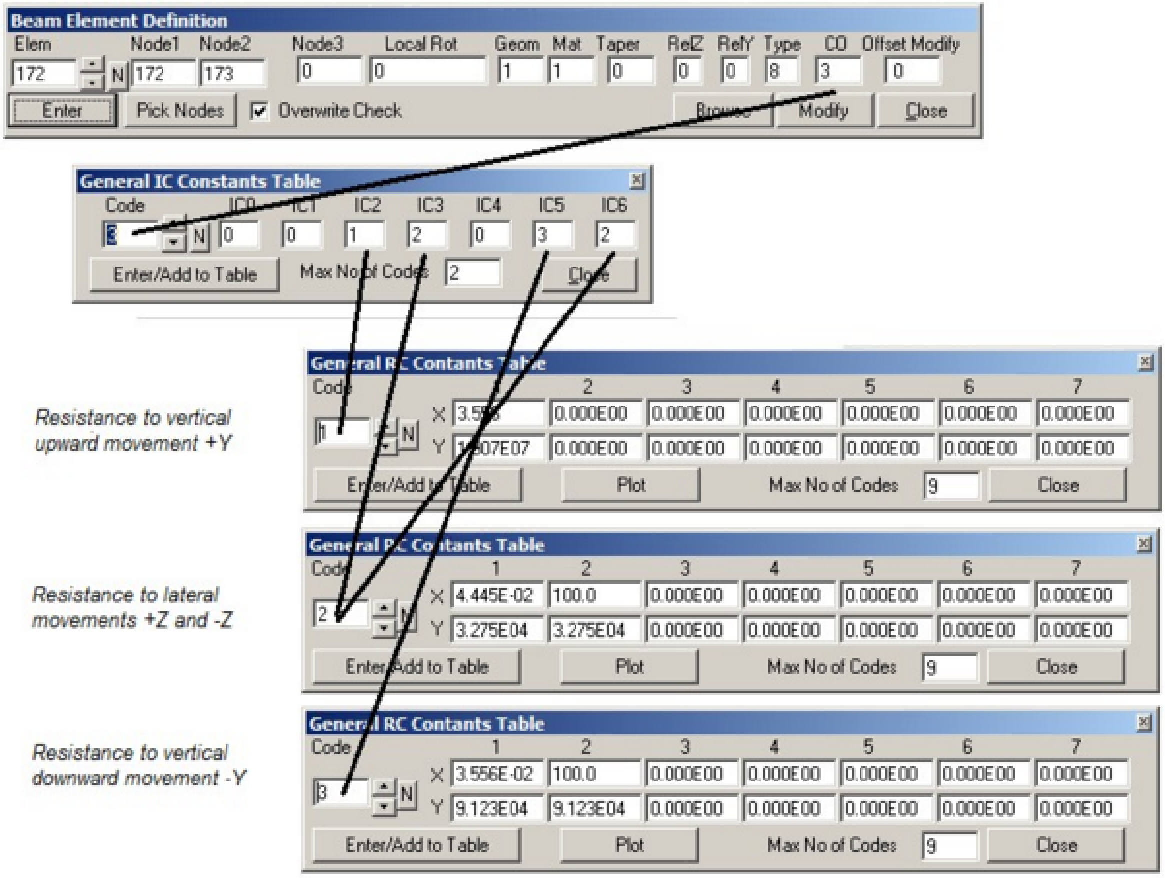
Special Features

The **ESTR** and **SEFO** load definition commands can be used with this element. These commands can also be used with all non-linear beam type elements (eg Types 7, 8, 15 & 16)

Restrictions

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	Interpreted as a Type 0 beam type. No ground support
3-D Non-linear	All	Non-linear - no large displacement option
Eigen Frequency & Buck'g	Mass included	Interpreted as a Type 0 beam type. No ground support
Dynamic Response	Mass included	Modes from Frequency Sol'n
DyNoFlex	All	Non-linear only - Ignored in a Linear Solution

The example below illustrates how properties are referenced. In this example **Elem 172** uses **RC1(+Y), RC2(+Z & -Z) and RC3(-Y)** to define the directional support springs.


Beam Element Definition
Elem Node1 Node2 Node3 Local Rot Geom Mat Taper RelZ RelY Type CO Offset Modify
Enter
Pick Nodes
Overwrite Check
Browse
Modify
Close
General IC Constants Table
Code IC0 IC1 IC2 IC3 IC4 IC5 IC6
Enter/Add to Table
Max No of Codes
Close
Resistance to vertical upward movement +Y
General RC Contants Table
Enter/Add to Table
Plot
Max No of Codes
Close
Resistance to lateral movements +Z and -Z
General RC Contants Table
Enter/Add to Table
Plot
Max No of Codes
Close
Resistance to vertical downward movement -Y
General RC Contants Table
Enter/Add to Table
Plot
Max No of Codes
Close

-o-

Type 15 Spar (Catenary) Element

This is a large displacement spar element. It has no flexural stiffness and can only sustain axial loads. It has the following features:

- Large Displacement
- Stress Stiffening (P-Delta)
- Tension/Compression Only
- Plasticity - Stress/Strain Curve Defintion
- Damping

This element requires to be run using the non-linear solution option. Note that the damping force is combined with the stiffness force on output.

When using this element in 3-D analysis it may be necessary to use the Soft Spring solution option if the elements are connected to other elements that do not provide sufficient restraint the end nodes.

Damping

Damping is defined by the element CO attribute. CO/RC X1 is used to define damping (Force/velocity) in the normal x direction. The value of the CO constant identifies the code number in the RC constants table. The damping magnitude is the X-1 entry. The units are in Force/Velocity per unit length of element e.g. N/(m/s)/m in the axial direction of the element. The length is based on the undeformed geometry.

The damping contribution can be applied using a damping matrix or using a restoring force approach. If the Def Element Damping is active in the DyNoFlex solution options a damping matrix will be employed. The damping matrix is the recommended approach. The damping magnitude is applied equally to both ends of element.

If the X-1 entry is defined as a negative value the the element damping will be interpreted as stiffness proportional damping (βK) and absolute value of the X-1 entry will represent β. This requires the **Def Element Damping** option to be active in the DyNoFlex solution options.

Tension Only/Compression Only

The **Non-Linear** button in the Geometric Input box of FS2000 is used to define this type of bi-linear element.

Geom Type 10 Compression only

Geom Type 11 Tension only

Plasticity Based on a Defined Stress Strain Curve

The plastic behavior of the element can be defined using a stress strain curve.

Geom Type 21 is an elastic non-linear formulation defined by a piecewise stress-strain curve. The stress-strain curve is defined in the RC constants table. The Table ID is required to be the same ID as that used for the element's Material Property code.

Geom Type 22 is a kinematic plasticity model which which uses piecewise stress-strain curve (RC constants table) or a Ramberg-Osgood type relationship to define the stress-strain relationship. The Table ID is required to be the same ID as that used for the element's Material Property code.

Geom Type 23 is an isotropic plasticity model which uses piecewise stress-strain curve (RC constants table) to define the stress-strain relationship. The Table ID is required to be the same ID as that used for the element's Material Property code.

Geom Type 24 is an isotropic plasticity model which uses the a two point bi-linear stress-strain curve (RC constants table) or a to define the stress-strain relationship. The Table ID is required to be the same ID as that used for the element's Material Property code. This is the same as Geom Type 24 but will ignore any more than 2 points in the curve definition (legacy option).

Special Features

The **ESTR** (defined strain) and **SEFO** (defined load) **load definition** commands can be used with this element.

Restrictions

Only recognised as spar element by the 3-D Non-Linear & DyNoFlex analysis modules. (Type 0 Element in 3-D Standard module - Make the I value >>>0 to behave as a spar))

NOT to be used for a linear solutions in DyNoFlex, always use the non-linear solution option when using this element.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	Interpreted as a Type 0 beam type (Dist loads lumped at nodes)
3-D Non-linear	All	Non-linear - no large displacement option
Eigen Frequency & Buckl'g	Mass included	Linear + P-Delta
Dynamic Response	Mass included	Modes from Frequency Sol'n
DyNoFlex	All	Only use with Non-linear option active. Can be used with 2 & 3 DOF solution options

-o-

Type 16 Elastic Large Displacement Beam

Large displacement elastic beam with or without stress stiffening. It has the following non-linear features

- Large Displacement
- Stress Stiffening/P-Delta (Always uses Stability Functions)
- Tension/Compression only

See Type 6 element description for additional information.

Moment Curvature Formulation

If elements are defined as being Geom Type 8 elements, then the bending stiffness behavior of the element is defined using a moment curvature relationship which are evaluated using cubic shape functions.

This moment curvature element is also well suited to the analysis of flexible pipe with relatively very low bending stiffness properties where the displaced shape is predominantly due to tensile stiffening action. The moment is based on curvature (numerical integration) is evaluated using cubic displacement shape functions. This approach lessens the inaccuracy which can sometimes be present when explicit beam element stiffness coefficients and nodal displacements are used to evaluate the moment in tension controlled configurations (where shorter elements would be required increase accuracy).

This element is also recommended for Eigen flexural buckling solutions.

Restrictions

Large displacement features are only recognised by the DyNoFlex analysis module.

Considered to be Type 0 in other the analysis modules ie only P-Delta and Tension/compression only features available.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	Interpreted as a Type 0 beam type
3-D Non-linear	All	Non-linear - no large displacement option
Eigen Frequency & Buckl'g	Mass included	Interpreted as a Type 0 or Type 16 beam type
Dynamic Response	Mass included	Modes from Frequency Sol'n
DyNoFlex	All	Non-linear only - Ignored in a Linear Solution

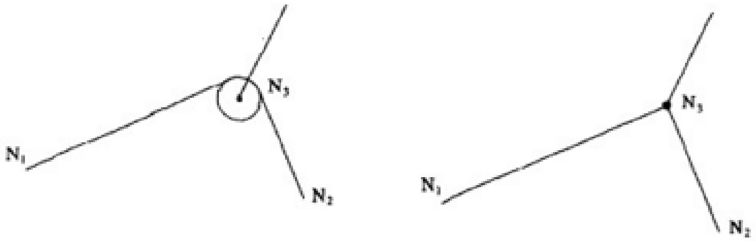
-o-

Type 17 Pulley Element

This is a 3 node element that can be used to represent the action of a simple pulley. By its nature this element introduces a mechanism into the model and can only be solved using a dynamic non-linear solution using the DyNoFlex solver. Solutions with this element involve rigid body motions and therefore require a Large Displacement solution.

The following basic assumptions are made:

- The cable is flexible and can only carry tensile load.
- The pulley is frictionless and the tension is each leg of the pulley is equal.
- The variational effect of the weight/mass of the pulley cable is neglected.



The element is defined as a Type 17 beam element. Node 1 (N1) and Node 2 (N2) are defined as normal beam would be.

The third node of the element, N3, is defined by the element's CO property. This element does NOT use the element's Node 3 property.

The Geometric and Material properties are defined to represent the cable properties as would be done with a Type 15 Spar element.

If the pulley cable has defined mass it will be lumped at N1 & N1 based on the initial geometry and remain constant during the solution.

The pulley will always be represented (visually) as a 2 node beam element between Node 1 & Node 2. The axial load on the pulley will be shown accordingly i.e. between N1 and N2.

Element Damping

It is very unlikely that a solution is possible without providing damping to the pulley cable. This can be achieved local to the pulley by using two translational damping elements between N1 to N3 and N2 to N3. Type 15 Spar elements or Type 3 Couple elements can be used for this purpose. Type 15 Spar elements are more convenient because the cable will appear visible on deflection plots and thus better indicate the final orientation of the pulley arrangement. Note that these damping elements must have zero stiffness to prevent them from carrying static load.

This type of element will generally be used in a hanging wire type scenario (mechanism). As such it will usually require global damping to be applied using the Global Viscous Damping solution property to reduce post transient oscillations. Any global damping will also be applied to the pulley and may eliminate the requirement for local damping described above.

The most cases the magnitude of the damping is purely artificial and does not represent any physical reality. It should be a high enough such that the solution is damped in a practical solution period.

Time Step

The solution will likely require a very small time step.

Restrictions

Only recognised as pulley element by the 3-D Non-Linear & DyNoFlex analysis modules. Not to be used in other solvers.

NOT to be used for a linear solutions in DyNoFlex. Always use the non-linear solution with the P-Delta and Large Displacement options active with this element.

If N1 or N2 are restrained, the restraint force will be incorrectly reported. Use node to ground couples to provide the restraint or incorporate a secondary stiff element to avoid this anomaly.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	Interpreted as a Type 0 beam type between N1 and N2
3-D Non-linear	All	Will not converge
Eigen Frequency & Buckl'g	All	Interpreted as a Type 0 beam type between N1 and N2
Dynamic Response	All	Interpreted as a Type 0 beam type between N1 and N2
DyNoFlex	All	Only use with Non-linear & Large Displacement options active

The formulation for this element is based on the following:
Aufaure M , A finite element of cable passing through a pulley.Computer & Structures Vol46(No5):807–812, 1993.

-o-

5.2.2 Finite Elements -Solid

This section described the finite elements available in FS2000. The geometric and material properties of finite elements are defined by reference to the property tables.

The wall thickness of shell (plate) elements is defined using the AREA property in the Geometric Property table.

Type 30	2-D Plane Solid
Type 40	Axisymmetric Solid
Type 50	Thin Shell (4Node) *Recommended element
Type 51	Thick/Thin Shell
Type 52	Thick/Thin Shell (4Node) *Recommended element
Type 53	Thin Shell - Large Displacement
Type 60	Membrane
Type 70	Hex 3-D Solid
Type 71	Tetra 3-D Solid

-o-

Type 30 2-D Plane Solid (3, 4, 6 & 8 Node)

This is a two dimensional isoparametric element in the X-Y plane for the analysis of plane stress or plane strain models. The element has only two translational degrees of freedom per node.

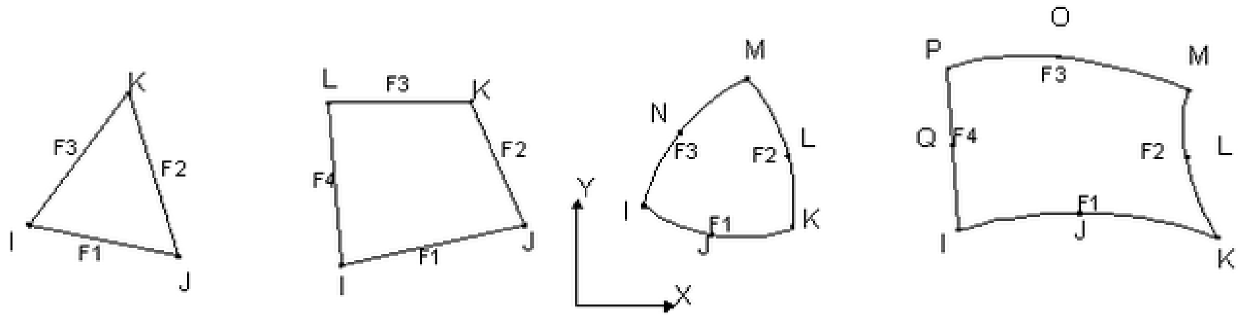
Plane stress conditions exist when the dimension in the z direction is small compared to the x and y direction such that the stress in the z direction can be taken to be zero.

Plane strain conditions exist when the dimension in the z direction is large compared to the x and y direction such that the strain in the z direction is zero (stress in z direction).

If a wall thickness (t) is defined for the element the Plane Stress formulation will be used (no stress in z direction).

The high order version of the element (6 & 8 nodes) can tolerate irregular shapes without much loss of accuracy and can model curved boundaries.

The element has Plastic capability for use in DyNoFlex non-linear solutions. The material model employs a Von-Mises isotropic memory model. The stress-strain relationship can be based on a bi-linear curve defined by a plastic tangent modulus, a piecewise stress-strain curve or a Ramberg-Osgood type relationship.



Node Numbering

Nodes MUST be number counterclockwise.

Element Coordinate System

The element always adopts the global coordinate system.

Properties

Module of Elasticity, Poisson's Ratio, Density, Coefficient of Thermal Expansion, t(optional - constant across element)

Solution/Solution Options

Note that solutions options should be consistent in all connected elements.

The solution options for 4 node element are:

0 Extra Shape Functions (QM6) - Recommended

1 Reduced Integration (2 x 2 Gauss for bending; 1 x 1 for shear)

2 Full Integration (2 x 2 Gauss for all terms)

The 8 Node Element uses:

2 x 2 for bending terms

2 x 2 for shear terms

Loading

Distributed Face Loads

Normal & Tangential In-plane Edge Loads

Thermal Expansion - Nodal Temp Defn

Gravitational

Nodal loads require to be input per unit of depth

Face loads are applied in force/unit length units. Edge loads are force/unit length units. Positive normal face loads go into the element. Positive tangential edge loads go in the same direction to the node numbering sequence. Loads can be defined differently for each edge node. If only one node load is defined it will be applied to all edge nodes. If mid-side node loads are defined a parabolic distribution will be applied.

Results

Stress components, principle stresses, stress intensity and von-Mises stress are output at all nodes in the global coordinate system.

Stresses are extrapolated from the Gauss points to the corner nodes. Mid side node stresses are the averaged corner nodes.

Restrictions/Recommendations

The element must be numbered in an anti-clockwise direction

The 3 node triangular option (constant strain element) is not recommended other than a filler element in non-critical areas of the mesh.

The 8 node option recommended for thermal strain solutions if there is a significant temperature gradient across an element. This is because the 8 node element (parabolic) has the same order of mechanical strain as thermal strain i.e. linear strain. The 4 node can in some cases produce unrealistic stresses for course meshes.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	Linear, Restraints - The model must be restrained globally as though it were a 3-D solid.
Non-linear	None	Use DyNoFlex
Eigen Frequency & Buckl'g	None	Neglected
Dynamic Response	None	Negelcted
DyNo Flex	All - No Dyn Mass	See Section 6.2.6.3
Heat Transfer		See Section 6.2.7

(References. Finite Elements in Plasticity, Owen & Hinton, 1980: Computational Methods for Plasticity, EA de Souza, D Peric, DRJ Owen,2000).

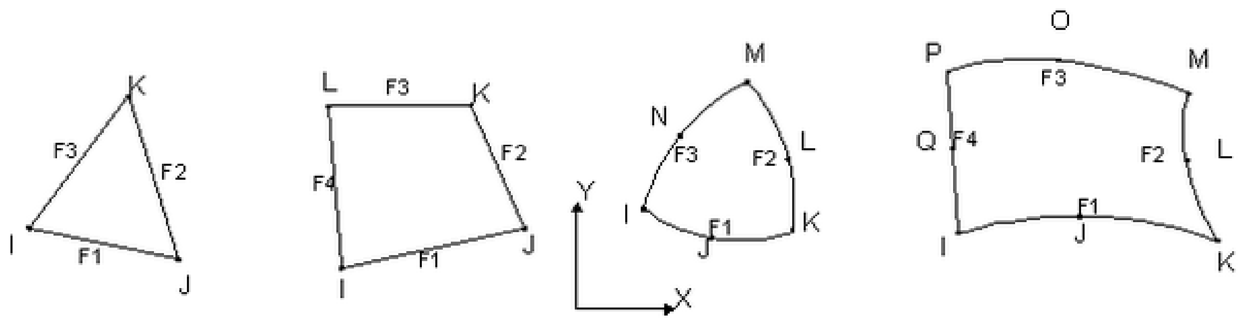
-o-

Type 40 Axisymetric Solid

This is a two dimensional element in the X-Y plane for the analysis of solids of revolution. The element is essentially a 2-D plane strain stress formulated for solids of revolution. The description of the element is the same as the 2-D plane Solid element apart from the following exceptions.

The global y-axis must be the axis of symmetry and the structure should be modeled in the +X quadrant.

The element has Plastic capability for use in DyNoFlex non-linear solutions. The material model employs a Von-Mises isotropic memory model. The stress-strain relationship can be based on a bi-linear curve defined by a plastic tangent modulus, a piecewise stress-strain curve or a Ramberg-Osgood type relationship.



Element Coordinate System

The X axis represents the radial coordinate and the Y axis represents the axial coordinate. Only a global system is used.

Solution/Solution Options

Note that solutions options should be consistent in all connected elements.

The solution option for 4 node element is:

0 Extra Shape Functions (QM6) - Recommended

1 Reduced Integration (2 x 2 Gauss for bending; 1 x 1 for shear)

2 Full Integration (2 x 2 Gauss for all terms)

The 8 Node Element uses:

2 x 2 for bending terms

2 x 2 for shear terms

Loading

Normal Edge Load

Tangential In-plane Edge Loads

Centrifugal Acceleration - X direction acceleration represents rotational velocity (rad/s)

Thermal Expansion - Nodal Temp Defn

Normal edge loads are applied in force/unit area units. Tangential edge loads are in force/unit length. Positive normal loads go into the element. Positive in-plane tangential edge loads go in the same direction to the node numbering sequence. Loads can be defined differently for each edge node. If only one node load is defined it will be applied to all edge nodes. If mid-side node loads are defined a parabolic distribution will be applied.

Nodal forces require to be applied on a 360 degree basis. A line load of 1 N/m would require to be applied as a line load of 1 x $2.\pi.r$ N

Results

Stress components, principle stresses, stress intensity and von Mises stress are output at all nodes in the global coordinate system. The stress axis are:

X Radial

Y Axial

Z Hoop (Tangential)

XY In-plane shear stress

Stresses are extrapolated from the Gauss points to the corner nodes. Mid side node stresses are the averaged corner nodes.

Restrictions/Recommendations

The element must be numbered in an anti-clockwise direction

The 3 node triangular option (constant strain element) is not recommended other than a filler element in non-critical areas of the mesh.

The 8 node option recommended for thermal strain solutions if there is a significant temperature gradient across an element. This is because the 8 node element (parabolic) has the same order of mechanical strain as thermal strain i.e. linear strain. The 4 node can in some cases produce unrealistic stresses for course meshes.

A singularity will exist when a radius of zero (x dimension) is defined. In the program this is replaced by a small value but the resulting stresses in at r=0 may stilll be less than correct.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	Linear, Restraints - The model must be restrained globally as though it were a 3-D solid.
Non-linear	None	Use DyNoFlex
Eigen Frequency & Buckl'g	None	Neglected
Dynamic Response	None	Negelcted
DyNo Flex	All - No Dyn Mass	See Section 6.2.6.3
Heat Transfer		See Section 6.2.7

(References. Finite Elements in Plasticity, Owen & Hinton, 1980: Computational Methods for Plasticity, EA de Souza, D Peric, DRJ Owen,2000).

-o-

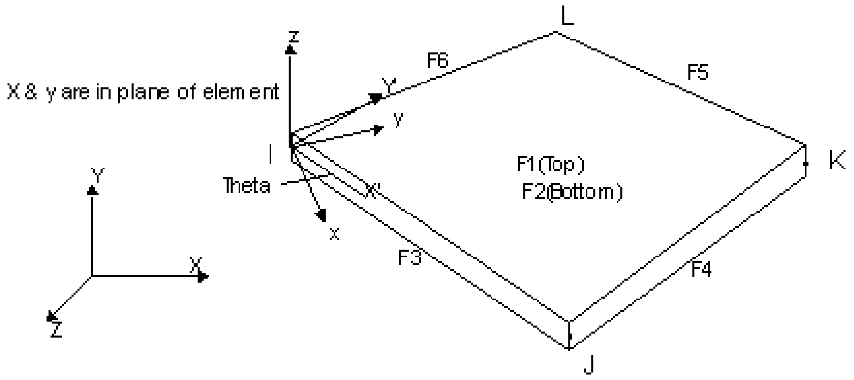
Type 50 Thin Shell (3 and 4 Nodes)

This is a three dimensional flat linear elastic thin shell element with membrane and bending capabilities. The bending formulation is based on **Kirchhoff** plate theory in which the effects of transverse shear deformations effects are neglected making it very suitable to model thin shell structures. It is the recommended element for general plate and shell analysis. The term 'Thin Shell' does not mean that this is not suitable for thick plate application, quiet the contrary, it only means that transverse plate shear deformations are excluded and transverse shear stresses Syz & Sxz are not evaluated.

The element has six degrees of freedom (three translations and three rotations) per node. The stress convention is shown in Section 4.

The bending formulation is based on the DK2 element. For the quadrilateral element the bending uses four overlaid DK2 elements.

In-plane formulations options are given below.



Elastic Foundation

The plate can be be supported on an elastic (Winkler) foundation normal to the local x-y plane. The foundation pressure is directly related to the displacement in the the local z direction.The foundation stiffness is defined by the Winkler foundation modulus k (N/m^3 in SI units). The element CO property indirectly defines the value of k by referencing an entry in the RC Table in which RC-X1 defines the magnitude of k. Not that Type 53 can provide a compression only support.

Node Numbering

Nodes may be numbered clockwise or counter-clockwise. The direction defines the element coordinate system.

Element Coordinate System

The element x axis goes from the first node to the second node. The y axis lies in a plane defined by the three nodes and is perpendicular to the x axis. The z axis goes toward the third node. A local rotation angle can be used to re-define the direction of the x axis in the plane of the element. There is no requirement or advantage to align the element coordinate systems because during post-processing the shell stresses can be output(aligned) using any type of coordinate system.

Offsets

A rigid offset can be define in the local z direction . It is applied equally to all element nodes. Shell offsets can be made visible using the options in the Model Display Switch form. Element loading is applies directly to the node locations and ignores any defined offset.

Properties

t(constant wall thickness), Module of Elasticity, Poisson's Ratio, Density, Thermal Coefficient of Expansion

Solution/Solution Options

Note that solutions options should be consistent in all connected elements.

The solution options for in-plane element actions are:

0 Extra Shape Functions (QM6) - Applicable to 4 node quad

1 Reduced Integration (2 x 2 Gauss for bending 1 x 1 for shear) - Applicable to 4 node quad

2 Full Integration (2 x 2 Gauss for all terms) - Applicable to 4 node quad

3 RDOF (Allman-Rotational degrees of freedom) - Applicable to 3 & 4 node elements

The 6 node element uses: 3 Gauss for bending and 3 for shear.

For out-plane bending actions.

DK2 formulation.

Geometric Stiffening

The effect of in-plane edge loading and plate bending action may be included by activating the P-Delta option.

Loading

Distributed Face Loads

Normal Edge, Tangential In-plane Edge & Tangential Out-Plane Edge Loads

Thermal Expansion - Nodal Temp Defn

Gravitational

Face loads are applied in force/unit area units. Edge loads are in force/unit length. Positive normal face loads go into the element. Positive in-plane tangential edge loads go in the same direction to the node numbering sequence. Positive out-plane tangential edge loads go into the element. Loads can be defined differently for each edge node. If only one node load is defined it will be applied to all edge nodes. If mid-side node loads are defined a parabolic distribution will be applied.

Normal face loads on faces 1 and 2 can be defined differently at the 4 corner node. If only one node is defined it will be applied to all nodes. If mid-side nodes are present the applied load will be the mean of the corner nodes.

Results

Stress components, principle stresses, stress intensity and von Mises stress are output at all nodes in the element coordinate system. Stress output is evaluated for top surface, mid surface and bottom surface.The stresses may be transformed to any coordinate when the case in post-processed.Note that any transform coordinate system used in post-processing must have the z axis in the parallel with the transverse plate direction if element forces are being evaluated

Stresses are extrapolated from the Gauss points to the corner nodes.

Element force and moments listings are included when result cases are processed by StdOut or MultiOut. This output is written to a Sub case with <model>.ShellF.O'n" for single cases or <model>.ShellF.M'n" for multi cases. Wood-Armer moments are included in the element force listings.

Restrictions/Recommendations

Unless the RDOF solution option is active the triangular element is not recommended other than a filler element in non-critical areas of the mesh. This is because the the in-plane action will be overly stiff.

For double curvature shells the triangular element (with RDOF) can also be overly stiff.

The 4 node RDOF solution can suffer from in-plane zero energy modes. This is more likely in flat square meshes that do not have a rotational in-plane restraint. This is unlikely to occur with mesh patterns and restraints used used in practice.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	P-Delta included.
Non-linear	All	Only 4 Node QM6 or RDOF Options - Thermal strain excluded - P-Delta Effect excluded.

Eigen Frequency & Buckl'g	All - Dyn mass included	Only 4 Node QM6, 3 Node and 3 & 4 Node RDOF Options. P-Delta included.
Dynamic Response	Non	Response effects from freq sol'n included but no stress evaluation
DyNo Flex	All - Dyn mass included	Only 4 Node. QM6 or RDOF Options. Thermal strain excluded. P-Delta Effect excluded.

-o-

Type 51 Thick/Thin Shell

This is a three dimensional flat linear elastic thick/thin shell isoparametric element with membrane and bending capabilities. The bending formulation is based on **Mindlin** plate theory in which the effects of transverse shear deformations are included.

For general 4 node plate analysis the Type 50 is recommended because it will produce a converged solution with a less refined mesh

If **Mindlin** plate theory is required and a 4 node element is being used, the Type 52 is recommended rather than this element type.

The bending of the 8 node element is based on Hughes Heterosis Element. This is effectively a 9 node element that uses internal degrees of freedom for the 9'th node.This 8 node element is the recommended configuration for this type of element.

The element has six degrees of freedom (three translations and three rotations) per node (8, 4 or 6, 3 Node element). The 3 & 6 node triangular element is not recommended apart from fill in, in non-critical areas.

The 4 node element may require more elements than a Type 50 element for the same degree of accuracy in plate bending applications.The 4 node element assumes a linear rotational displacement variation - constant strain for bending. This results in constant bending across the element

Concentrated nodal loading loading should be avoided with the 4 node element, always try to apply the loading in a consistent manner over a small area. This is because zero energy modes may be activated. This will be evident in the solution.

The element stress convention is shown in Section 4.

Plasticity (Non-Linear Solution Only)

The 4 node element has Plastic, P-Delta and Large Displacement capability for use in DyNoFlex non-linear solutions - Solution Options 0 or 1 only. The P-Delta and Large displacement formulations are the same as those for the Type 53 Thin shell. This is not a large strain element and the implicit assumption is that whilst rotations can be large (rigid body), the rotations that cause stress must be small. When using this 4 node element mesh size sensitivity checks should be undertaken.

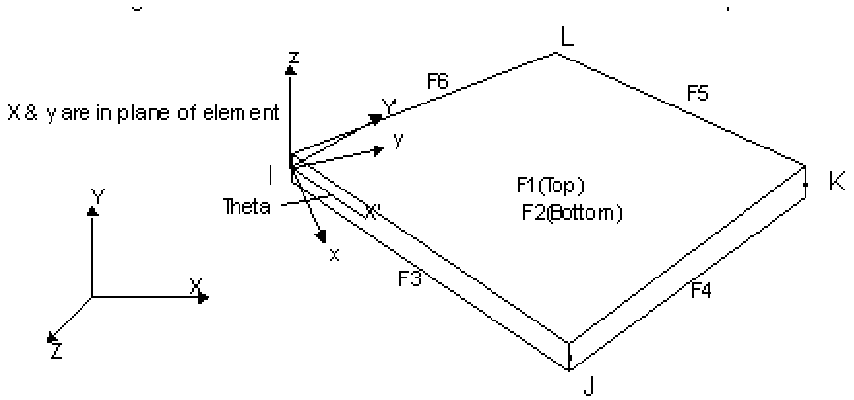
The plastic formulations uses a layered which accounts for the spread of plasticity across the section (2 to 12 layers- even number reqd Default=12). The stress-strain relationship can be based on a bi-linear curve defined by a plastic tangent modulus, a piecewise stress-strain curve or a Ramberg-Osgood type relationship see 6.2.6.3. Transverse shear stresses are not used in the evaluation of the yield function.

The stress output are the stresses at the Gauss points at the mid point in the outer layers extrapolated to the node locations.

Reported mid-plate stresses are actually the mean of the outer mid-plane stresses.

Reported top and bottom surface stresses are the mid plane stress in the outer layers. This will reduce the stresses obtained to that from a linear solution (91% for 12 layers under pure bending).

(References: Finite Elements in Plasticity, Owen & Hinton, 1980, Computational Methods for Plasticity, EA de Souza, D Peric, DRJ Owen,2000).



Elastic Foundation

The plate can be be supported on an elastic (Winkler) foundation normal to the local x-y plane. The foundation pressure is directly related to the displacement in the the local z direction.The foundation stiffness is defined by the Winkler foundation modulus k (N/m^3 in SI units). The element CO property indirectly defines the value of k by referencing an entry in the RC Table in which RC-X1 defines the magnitude of k. If k is defined as a -ve value than the foundation will only apply compressive support (non-linear solution only).

Node Numbering

Nodes may be numbered clockwise or counterclockwise. The direction defines the element coordinate system.

Element Coordinate System

The element x axis goes from the first node to the second node. The y axis lies in a plane defined by the three nodes and is perpendicular to the x axis. The y axis goes toward the third node. A local rotation angle can be used to re-define the direction of the x axis in the plane of the element. There is no requirement or advantage to align the element coordinate systems because during post-processing the shell stresses can be output using any type of coordinate system.

Offsets

A rigid offset can be define in the local z direction . It is applied equally to all element nodes. Shell offsets can be made visible using the options in the Model Display Switch form. Element loading is applies directly to the node locations and ignores any defined offset.

Linear Properties

t(constant wall thickness), Module of Elasticity, Poisson's Ratio, Density, Thermal Coeff of Expansion.

Non-Linear Properties

Tangent Modulus Et, Yield Criteria and Number of Layers (Default = 12). See 6.2.6

Solution/Solution Options

Note that solutions options should be consistent in all connected elements.

Always use Option 0 for 8 node or 6 node element.

The solution options for in-plane element actions are:

0 Extra Shape Functions (QM6) - Applicable to 4 node quad

1 Reduced Integration (2 x 2 Gauss for bending 1 x 1 for shear) - Applicable to 4 node quad

2 Full Integration (2 x 2 Gauss for all terms) - Applicable to 4 node quad.

3 RDOF (Allman-Rotational degrees of freedom) - Applicable to 3 & 4 node elements

The 4 node element uses: 2 x 2 Gauss for bending and 1 x 1 for shear.

The 8 Node Element uses:2 x 2 for bending, in-plane and shear terms.

The 6 node element uses: 3 Gauss for bending and 3 for shear.

For out-plane bending.

The 8 node uses Hughes Heterosis Element. This is effectively a 9 node element that uses internal degrees of freedom for the 9'th node (3 x 3 Gauss for bending and 2 x 2 for shear).

The 4 node element uses: 2 x 2 Gauss for bending and 1 x 1 for shear.

The 6 node element uses: 3 Gauss for bending and 3 for shear.

Loading

Distributed Face Loads

Normal Edge, Tangential In-plane Edge & Tangential Out-Plane Edge Loads

Thermal Expansion - Nodal Temp Defn

Gravitational

Face loads are applied in force/unit area units. Edge loads are in force/unit length. Positive normal face loads go into the element. Positive normal face loads go into the element. Positive in-plane tangential edge loads go in the same direction to the node numbering sequence. Positive out-plane tangential edge loads go into the element. Loads can be defined differently for each edge node. If only one node load is defined it will be applied to all edge nodes. If mid-side node loads are defined a parabolic distribution will be applied.

Normal face loads on faces 1 and 2 can be defined differently at the 4 corner node. If only one node is defined it will be applied to all nodes. If mid-side nodes are present the applied load will be the mean of the corner nodes.

Results

Stress components, principle stresses, stress intensity and von Mises stress are output at all nodes in the element coordinate system. Stress output is evaluated for top, mid and bottom surfaces. The stresses may be transformed to any coordinate when the case in post-processed. Note that any transform coordinate system used in post-processing must have the z axis in the parallel with the transverse plate direction if element forces are being evaluated.

Element force and moment listings are included when result cases are processed by StdOut or MultiOut. This output is written to a Sub case with <model>.ShellF.O'n" for single cases or <model>.ShellF.M'n" for multi cases. Wood-Armer moments are included in the element force listings.

Restrictions/Recommendations

When using 8 node elements to model shells with curvature ensure that the mid-side nodes are flat ie do not produce a warped element.

The triangular element is not recommended other than a filler element in non-critical areas of the mesh.

The 4 node element uses reduced shear (single integration pt.) for the bending terms and this can result in zero energy modes being activated. This can happen when concentrated loading or restraints are applied to a flat mesh. Note the the Type 52 does not suffer from this effect.

The 4 node RDOF solution can suffer from in-plane zero energy modes. This is more likely in flat square meshes that do not have a rotational in-plane restraint. This is unlikely to occur with mesh patterns and restraints used used in practice.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	Linear no restrictions
Non-linear	Non	4 Node element only, QM6 or Red Int. Excludes Thermal Expansion
Eigen Frequency & Buckl'g	None	Neglected
Dynamic Response	Non	Neglected
DyNo Flex	Non	4 Node with QM6 element only, . Excludes Thermal Expansion

-o-

Type 52 Thick/Thin Shell (4 Node)

This is a three dimensional flat linear elastic thick/thin shell element with membrane and bending capabilities. The bending formulation is based on **Mindlin** plate theory in which the effects of transverse shear deformations are included.

The bending formulation is based on the MITC4 element (*Ref. Finite Element Prodedures, Bathe,K-J*). This formulation is very robust and can be used for both thick and thin plates and does not suffer from shear locking or zero energy modes).

The 4 node element has six degrees of freedom (three translations and three rotations) per node.

The 4 node element assumes a linear rotational displacement variation - constant strain for bending. This results in constant bending across the element. In the MITC4 element the shear can vary.

The element stress convention is shown in Section 4.

Plasticity (Non-Linear Solution Only)

The 4 node element has Plastic, P-Delta and Large Displacement capability for use in DyNoFlex non-linear solutions. The P-Delta and Large displacement formulations are the same as those for the Type 53 Thin shell. This is not a large strain element and the implicit assumption is that whilst rotations can be large (rigid body), the rotations that cause stress must be small. When using this 4 node element mesh size sensitivity checks should be undertaken.

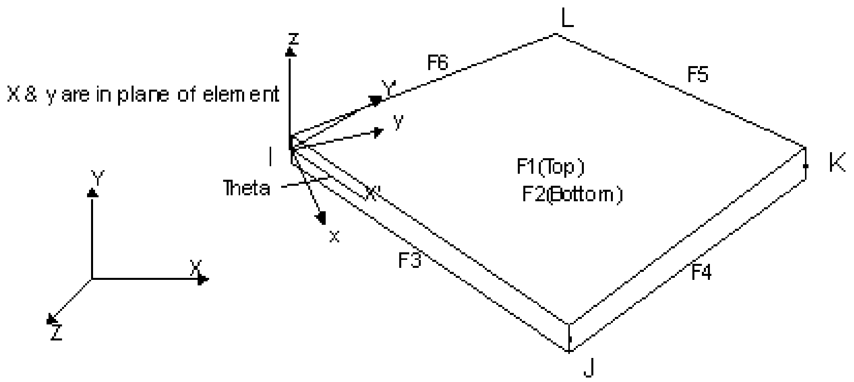
The plastic formulations uses a layered which accounts for the spread of plasticity across the section (2 to 12 layers- even number reqd Default=12). The stress-strain relationship can be based on a bi-linear curve defined by a plastic tangent modulus, a piecewise stress-strain curve or a Ramberg-Osgood type relationship see 6.2.6.3. Transverse shear stresses are not used in the evaluation of the yield function.

The stress output are the stresses at the Gauss points at the mid point in the outer layers extrapolated to the node locations.

For layered option the reported mid-plate stresses are actually the mean of the outer mid-plane stresses. Reported top and bottom surface stresses are the mid plane stress in the outer layers. This will reduce the stresses obtained to that from a linear solution (91% for 12 layers under pure bending).

Note that a non-plastic DyNoFlex solution option does not employ a layered formulation.

(References: Finite Elements in Plasticity, Owen & Hinton, 1980, Computational Methods for Plasticity, EA de Souza, D Peric, DRJ Owen,2000).



Elastic Foundation

The plate can be be supported on an elastic (Winkler) foundation normal to the local x-y plane. The foundation pressure is directly related to the displacement in the the local z direction.The foundation stiffness is defined by the Winkler foundation modulus k (N/m^3 in SI units). The element CO property indirectly defines the value of k by referencing an entry in the RC Table in which RC-X1 defines the magnitude of k. If k is defined as a -ve value than the foundation will only apply compressive support (non-linear solution only).

Node Numbering

Nodes may be numbered clockwise or counterclockwise. The direction defines the element coordinate system.

Element Coordinate System

The element x axis goes from the first node to the second node. The y axis lies in a plane defined by the three nodes and is perpendicular to the x axis. The y axis goes toward the third node. A local rotation angle can be used to re-define the direction of the x axis in the plane of the element. There is no requirement or advantage to align the element coordinate systems because during post-processing the shell stresses can be output using any type of coordinate system.

Offsets

A rigid offset can be define in the local z direction . It is applied equally to all element nodes. Shell offsets can be made visible using the options in the Model Display Switch form. Element loading is applies directly to the node locations and ignores any defined offset.

Linear Properties

t(constant wall thickness), Module of Elasticity, Poisson's Ratio, Density, Thermal Coeff of Expansion.

Non-Linear Properties

Tangent Modulus Et, Yield Criteria and Number of Layers (Default = 12). See 6.2.6

Solution/Solution Options

Note that solutions options should be consistent in all connected elements.

The solution options for in-plane element actions are:

0 Extra Shape Functions (QM6) - Always used in DyNoFlex plastic solutions.

1 Reduced Integration (2 x 2 Gauss for bending 1 x 1 for shear). Not used in DyNoFlex.

2 Full Integration (2 x 2 Gauss for all terms). Not used in DyNoFlex.

3 RDOF (Allman-Rotational degrees of freedom).

For out-plane bending.

The element uses: 2 x 2 Gauss for bending and shear (MITC4).

Loading

Distributed Face Loads

Normal Edge, Tangential In-plane Edge & Tangential Out-Plane Edge Loads

Thermal Expansion - Nodal Temp Defn

Gravitational

Face loads are applied in force/unit area units. Edge loads are in force/unit length. Positive normal face loads go into the element. Positive normal face loads go into the element. Positive in-plane tangential edge loads go in the same direction to the node numbering sequence. Positive out-plane tangential edge loads go into the element. Loads can be defined differently for each edge node. If only one node load is defined it will be applied to all edge nodes. If mid-side node loads are defined a parabolic distribution will be applied.

Normal face loads on faces 1 and 2 can be defined differently at the 4 corner node. If only one node is defined it will be applied to all nodes. If mid-side nodes are present the applied load will be the mean of the corner nodes.

Results

Stress components, principle stresses, stress intensity and von Mises stress are output at all nodes in the element coordinate system. Stress output is evaluated for top, mid and bottom surfaces. The stresses may be transformed to any coordinate when the case in post-processed. Note that any transform coordinate system used in post-processing must have the z axis in the parallel with the transverse plate direction if element forces are being evaluated.

Element force and moment listings are included when result cases are processed by StdOut or MultiOut. This output is written to a Sub case with <model>.ShellF.O'n" for single cases or <model>.ShellF.M'n" for multi cases. Wood-Armer moments are included in the element force listings.

Restrictions/Recommendations

Only a 4 node element.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	Linear - no restrictions
Non-linear	Non	Excludes Thermal Expansion
Eigen Frequency & Buckl'g	None	Neglected
Dynamic Response	Non	Neglected
DyNo Flex	Non	Only QM6 for plasticity. QM6 or RDOF for non-plastic Excludes Thermal Expansion

-o-

Type 53 Thin Shell (3 and 4 Nodes)

This is identical to the Type 50 Thin shell element but with the addition of P-Delta and Large Displacement capability for use in DyNoFlex non-linear solutions.

The plate bending **P-Delta** formulation for this element is based on the following:
A simplified geometric stiffness in stability analysis of thin walled structures by the finite element method, Ivo Senjanović1, Nikola Vladimir1 and Dae-Seung Cho2,Inter J Nav Archit Oc Engng (2012) 4:313~321

Large Displacement

When the deflections are large it is possible to account for the change in stiffness by updating the global stiffness matrix based on the shape of the deformed structure. The program uses a co-rotational (convected coordinate system) approach to large displacement(Rankine & Brogan) in which the non-vectorial nature of rotations is accounted for. In this approach the displacement that cause stresses are separated from those due to rigid body motion. The implicit assumption in this method is that whilst rotations can be large (rigid body) the rotations that cause stress must be small. Rigid body rotations are generally limited to 180 degrees.

The element definition rotation theta-angle (**Local Rotation**) must ALWAYS be zero for Large Displacement solutions.

Elastic Foundation

The plate can be be supported on an elastic (Winkler) foundation normal to the local x-y plane. The foundation pressure is directly related to the displacement in the the local z direction.The foundation stiffness is defined by the Winkler foundation modulus k (N/m^3 in SI units). The element CO property indirectly defines the value of k by referencing an entry in the RC Table in which RC-X1 defines the magnitude of k. If k is defined as a -ve value than the foundation will only apply compressive support (non-linear solution only).

Restrictions/Recommendations

Element defined loading does not follow the element orientation in a large displacement solution .i.e. face and edge loads are applied as global loading based on the undeformed geometry.

Offsets - These should not be used in Large Displacement solutions.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	P-Delta
3-D Non-linear	All	Only 4 Node QM6 or RDOF Options - P-Delta. Thermal strain excluded.
Eigen Frequency & Buckl'g	All - Dyn mass included	Only 4 Node QM6, 3 Node and 3 & 4 Node RDOF Options. P-Delta included.
Dynamic Response	Non	Response effect from freq sol'n included but no stress evaluation.
DyNo Flex	All - Dyn mass included	Only 4 Node QM6 or RDOF Options - P-Delta- Large Disp. Thermal strain excluded.

-o-

Type 60 Membrane Shell

This is a three dimensional flat linear elastic shell isoparametric element with only membrane capabilities. The element has three degrees of freedom (three translations and three rotations) per node. Only the two local X and Y translational freedoms are active. To ensure stability a small stiffness (1E-15) is applied to the non-active degrees of freedom.

Node Numbering

Nodes may be numbered clockwise or counterclockwise. The direction defines the element coordinate system.

Element Coordinate System

The element x axis goes from the first node to the second node. The y axis lies in a plane defined by the three nodes and is perpendicular to the x axis. The y axis goes toward the third node. A local rotation angle can be used to re-define the direction of the x axis in the plane of the element.

Properties

t(contant wall thickness), Module of Elasticity, Poisson's Ratio, Density, Coefficient of Thermal Expansion

Solution/Solution Options

Note that solutions options should be consistent in all connected elements.

The solution options for element are (first 3 are for quad elements only):

0 Extra Shape Functions (QM6)

1 Reduced Integration (2 x 2 Gauss for bending 1 x 1 for shear)

2 Full Integration (2 x 2 Gauss for all terms)

3 RDOF (Allman-Rotational degrees of freedom) (3 & 4 node)

Loading

Distributed Face Loads

Normal Edge, Tangential In-plane Edge & Tangential Out-Plane Edge Loads

Thermal Expansion - Nodal Temp Defn

Gravitational

Face loads are applied in force/unit area units. Edge loads are in force/unit length. Positive normal face loads go into the element. Positive in-plane tangential edge loads go in the same direction to the node numbering sequence. Positive out-plane tangential edge loads go into the element. Loads can be defined differently for each edge node. If only one node load is defined it will be applied to all edge nodes. If mid-side node loads are defined a parabolic distribution will be applied.

Normal face loads on faces 1 and 2 can be defined differently at the 4 corner node. If only one node is defined it will be applied to all nodes. If mid-side nodes are present the applied load will be the mean of the corner nodes.

Warning - Unless other elements are able to provide restraint, loading should not be applied in other than the active degree of freedom of the element i.e. Tangential Edge In-plane.

Results

Stress components, principle stresses, stress intensity and von Mises stress are output at all nodes in the element coordinate system

Restrictions/Recommendations

When using 8 node elements to model shells with curvature ensure that the mid-side nodes are flat ie do not produce a warped element.

Unless the RDOF solution option is active the triangular element is not recommended other than a filler element in non-critical areas of the mesh.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	Linear no restrictions
Non-linear	All	Only 4 Node QM6, 3 Node and 3 & 4 Node RDOF Options - Linear
Eigen Frequency & Buckl'g	All - Dyn mass included	Only 4 Node QM6, 3 Node and 3 & 4 Node RDOF Options. Linear
Dynamic Response	Non	Response effect from freq sol'n included but no stress evaluation
DyNoFlex	All - Dyn mass included	Only 4 Node QM6, 3 Node and 3 & 4 Node RDOF Options - Linear

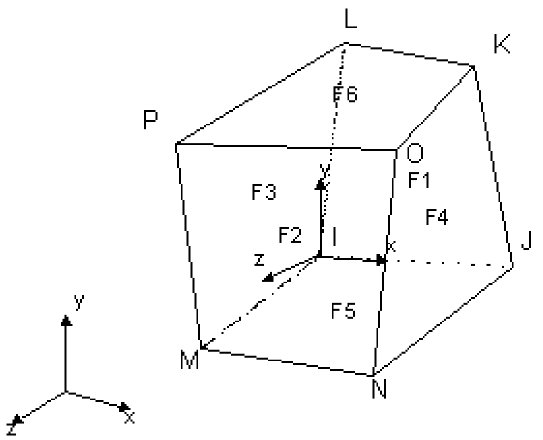
-o-

Type 70 Hex 3-D Solid

This is an 4 node three dimensional isoparamtric solid element. The element has only three translational degrees of freedom per node. There are no moment conditions. To ensure stability a small stiffness (1E-15) is applied to the non-active rotational degrees of freedom.

The element has Plastic capability for use in DyNoFlex non-linear solutions.The material model employs a Von-Mises isotropic memory model. The stress-strain relationship can be based on a bi-linear curve defined by a plastic tangent modulus, a piecewise stress-strain curve or a Ramberg-Osgood type relationship.

(References. Finite Elements in Plasticity, Owen & Hinton, 1980).



Node Numbering

The nodes of Face 1 (IJKL) and Face 2 (MNOP) MUST be numbered counterclockwise when looking down the local z axis. Face 2 must be in the direction of the local positive z axis relative to Face 1

Element Coordinate System

The element x-axis goes from the first node to the second node. The y-axis lies in a plane defined by the three nodes and is perpendicular to the x-axis. The y-axis goes toward the third node. The z-axis completes a right hand cartesian system

A local rotation angle can be used to re-define the direction of the x-axis in the plane of the element.

There is no requirement or advantage to align the element coordinate systems because during post-processing the element stresses can be output using any type of coordinate system.

Properties

Module of Elasticity, Poisson's Ratio, Density, Coefficient of Thermal Expansion

Solution/Solution Options

Note that solutions options should be consistent in all connected elements.

The solution options for element are:

0 Extra Shape Functions (QM6) - Recommended

1 Reduced Integration (2 x 2 Gauss for bending 1 x 1 for shear)

2 Full Integration (2 x 2 Gauss for all terms)

Loading

Normal Pressure Loads

Thermal Expansion - Nodal Temp Defn

Gravitational

Positive face pressures go into the element. Face pressures can be defined differently at the 4 corner node. If only one node is defined it will be applied to all nodes.

Results

Stress components, principle stresses, stress intensity and Von-Mises stress are output at all nodes in the global coordinate system.

Stresses at the nodes are the Gauss point stresses.

Restrictions/Recommendations

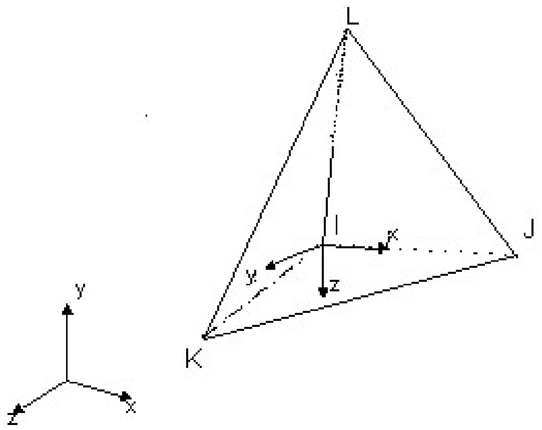
The solution option with the extra shape functions (QM6) is always recommended. The option will give a solution comparable to a 20 node solid element for near cube proportions.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	All	Linear, Restraints - The model must be restrained globally as though it were a 3-D solid.
Non-linear	Non	Use DyNoFlex
Eigen Frequency & Buckl'g	None	Neglected
Dynamic Response	Non	Negelcted
DyNo Flex	All - No Dyn Mass	See Section 6.2.6.3
Heat Transfer		See Section 6.2.7

-o-

Type 71 Tetra 3-D Solid

This is an 4 node or 10 node three dimensional isoparametric solid element. The element has only three translational degrees of freedom per node. There are no moment conditions. To ensure stability a small stiffness (1E-15) is applied to the non-active rotational degrees of freedom.



Node Numbering

The nodes of the 10 node element are between IJ, JK, KL, IL, JL and KL. Face 1 is (IJK) Face 2 iss (IJL) and Face 3 (IKL).

Element Coordinate System

The element x-axis goes from the first node to the second node. The y-axis lies in a plane defined by the three nodes and is perpendicular to the x-axis. The y-axis goes toward the third node. The z-axis completes a right hand cartesian system

A local rotation angle can be used to re-define the direction of the x-axis in the plane of the element. There is no requirement or advantage to align the element coordinate systems because during post-processing the stresses can be output using any type of coordinate system.

Properties

Module of Elasticity, Poisson's Ratio, Density, Coefficient of Thermal Expansion

Solution/Solution Options

Single Option

Loading

Normal Pressure Loads

Thermal Expansion - Nodal Temp Defn

Gravitational

Positive face pressures go into the element. Face pressures can be defined differently at the 4 corner node. If only one node is defined it will be applied to all nodes.

Results

Stress components, principle stresses, stress intensity and Von-Mises stress are output at all nodes in the global coordinate system.

Restrictions/Recommendations

Solution Option	Property	Comments

	Recognition	
3-D Standard	All	Linear no restrictions
Non-linear	Non	Neglected
Eigen Frequency & Buckl'g	None	Neglected
Dynamic Response	Non	Neglected
DyNo Flex	Non	Neglected

-o-

5.2.3 Couple Elements

Couples are springs that connects specific nodal degrees of freedom to other nodal freedoms or to a ground restraint.

Couple nodes should always coexist at the same location apart for Type 0 or Type 7.

The effect of shear offset is included in Type 0 and Type 7 couples providing their orientation is defined by node location in a similar manner to beam elements. These types of element are useful and avoid the restrictions associated with rigid links.

Couples are created in the GUI using the Couple Definition input form. There are different types of couple elements, the couple type (CType) and the its properties are defined by the Couple Constants Table

The linear couple element (CType 0) is the most commonly used type and can be effective used to model hinges, slideways, guides and other type of mechanisms by applying rigid coupling action between nodes using a high defined spring stiffness.

Care should be exercised when using stiff couples because very high stiffness relative to the other stiffness of the structure can produce numerical inaccuracies (ill conditioning) in the matrix solution on very large models. A stiffness of typically 1000 x the stiffness of the stiffest connected element should be ideal. A good guide for rigid couples is to equate them to the axial stiffness of a typical beam element of the structure, i.e. spring stiffness=1000 x AE/L. For a typical structure using S.I. units 1E9 should be suitable for most models. If in doubt use a soft spring and increase the stiffness if the results show too much relative couple deflection.

Orientation

The local coordinate system of couple elements are defined by one of the following methods:

- Node location, in a similar manner to a local beam element coordinate system
- Reference to a beam element, aligns to the referenced element.
- Reference to a coordinate system, aligns to the referenced coordinate system

Reference to a beam element or coordinate system, if defined, takes precedence over node to node orientation.

Unlike the connectivity for beam elements the nodes connecting spring elements are usually coincident, in such a case the orientation of the spring may be defined by reference to an existing beam element or to a coordinate system. If neither of these are defined the orientation will be aligned to the global system.

Restraining Models using Ground Couples

Node to ground Couples may be used to provide restraint for model in the form of ground springs. Models may be solely restrained using ground spring thus eliminating the use of conventional restraints. Ground spring restraints can be easily orientated in any direction unlike conventional restraints that are always in the global cartesian system. When using the non-linear solvers prescribed displacement can be imposed using the CDISP load case command

*************** Warning - Spatial Effects *************************

Couple nodes should always coexist at the same location apart for Type 0 or Type 7 which can account for shear offset.

Shear offset (Type 0 and Type 7 couples) will not be accounted for if the orientation of the couple is defined using a reference element or by reference to a coordinate system, i.e nodes are required to be spatially coincident.

-o-

Couple Element Types

The following summarises the different type of couple elements in FS2000.

Type 0	Linear couple
Type 1	Linear Damper (node to ground - global coordinate system
Type 3	Translational & Torsional Spring/Damper
Type 4	Non-linear translational spring
Type 5	Journal Contact Element
Type 6	Journal Contact Element
Type 7	Non-linear Large Displacement
Type 8	Impact-Energy Monitor
Type 10	Compression Gap with friction
Type 11	Tension Gap
Type 12	Compression Gap with friction - as 10 but with conservative friction
Type 14	Node to Ground Surface Contact (2-D in 3-D space)
Type 15	Compression/Friction Gap
Type 16	Node to Ground Surface Contact
Type 20	Vessel Element (RAO Definition)

-o-

Gap Elements

A gap element is a special form of couple element and can be used to represent two surfaces which may maintain or break physical contact and may slide relative to each other. Frictional shears force effects between sliding surfaces can be accounted for.

It is identified (CType 10, 11, 12 & 15) and specified by the Couple Constants Table. To simplify definition, the Gap Definition Button can be used to make the Gap Definition input box visible.

Only the **3D NL/Pile Analysis** and **FS-DyNoFlex** modules are capable handling friction and specified gap clearance/interference. The **3-D Standard** module will only interpret gap elements as simple make or break contact elements.

Gaps may be define as Compression Gaps or Tensile Gaps (Hooks). Friction cannot be applied to Tensile Gap elements

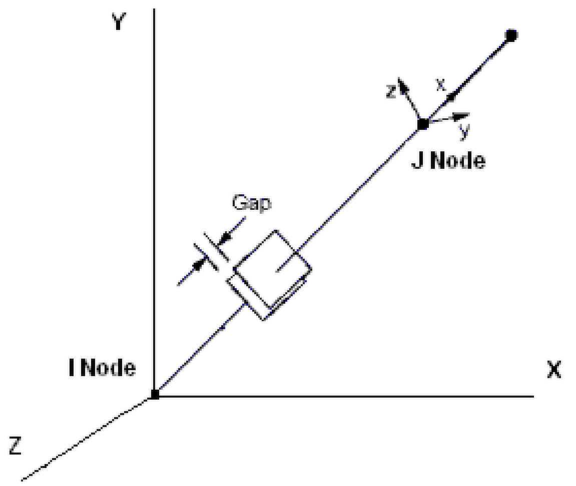
Friction effects are accounted for by equivalent forces, which resist in the direction of relative displacement. The magnitude of the resisting force is limited by the product of the gap's normal force and the coefficient of friction (Coulomb friction).

If Gap Size is specified as a negative value the element will produce forces in an unloaded structure equivalent to an interference fit of the gap magnitude. If the gap is defined with a positive value, the gap will require close that amount before it makes contact.

The Gap Stiffness-Normal is the stiffness of the gap when it is closed. The normal stiffness should be based upon the stiffness of the surfaces in contact. Typically a value of 1E9 would be a stiff gap for most structures created in S.I units. A good 'feel' for relative stiffness can be obtained by equating it to the stiffness of the connecting element i.e. AE/L for the case of a beam connected in the axial direction.

The Gap Stiffness-Tangential is the sticking stiffness that is active when a gap is sliding and friction id defined. The stiffness defines mobilisation distance for the limiting friction to become fully effective.

The convergence of gap elements is assisted by using soft stiffness since this reduces the tendanct to 'bounce'. Hence always use as soft a stiffness as can be tolerated by the physical modelling of the structure.



Orientation of gap elements

The sign convention for Gap elements is that positive local displacement will open a compression gap and negative displacement will close it. Positive local displacement in this context is the relative displacement of the I node relative to the J node i.e. the I node moving towards the J node. Tensile gaps (hooks) act in a converse manner.

- Node to Node Gaps: When these are referenced to an element or a coordinate system for their orientation then the numbering of the gap is not arbitrary, the numbering must relate to the direction of the reference system.

- Node to Ground Gaps: When these are referenced to an element or a coordinate system for their orientation then the positive direction should point away from the node such that in the cases of a compression gap the gap will opens for positive node displacement.

-o-

CType 0 Linear Spring/Couple

This Type 0 couple is a standard linear elastic couple element . The element can be Node to Node or Node to Ground.

The local element local co-ordinate system is define by either:

- Node Location
- Reference to a beam element's orientation
- Reference to a Coordinate System

For standard linear couples the constants K1 to K6 are used to define the stiffness of the couple in terms the element local co-ordinate system. i.e.

K1 x translation

K2 y translation

K3 z translation

K4 x rotation

K5 y rotation

K6 z rotation

The effect of shear offset is included if the couple is defined as a node to node couple and the orientation of the couple is define by the location of the nodes in a similar way to conventional beam elements.

Shear offset will not be accounted for if the orientation of the couple is defined using a reference element or by reference to a coordinate system.

Restrictions

None

Solution Option	Property Recognition	Comments
3-D Standard	All	Linear
Non-linear	All	Linear
Frequency	All	Linear
DyNo Flex	All	Linear

-o-

CType 1 Linear Damper - Node to ground

This is a Type 1 Couple element. It is a concentrated nodal damper and is applied to the global degrees of freedom of the node to which it is applied. It can be used in linear and non-linear analysis.

K1-Linear Damping Force/Velocity in the X direction

K2-Linear Damping Force/Velocity in the Y direction

-

-

K6 -Linear Damping Force/Velocity in the rotational Z direction

Restrictions

Only recognised by the DyNoFlex analysis module.

Acts as a CType 0 couple in 3-D Standard analysis module.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	As spring constants	Linear
Non-linear	Ignored	Linear
Frequency	As spring constants	Linear
DyNo Flex	All	Linear

-o-

CType 3 Translational & Torsional Spring/Damper

Linear and rotational damping in this element is applied in the local X axis of the element.

Element can be Node to Node or Node to Ground

Large displacement - If the couple is referenced to a Type-6 beam element then the couple local coordinate system will move with that of the referenced element. If a reference element is not defined then local coordinate system will be defined by the current position of the defining nodes or in the case of a Node to Ground, the global coordinate system.

Properties

K1 - Linear spring

K2 - Linear damping Force/Velocity

K3 - Damping proportional to velocity   (C=K2+K3*Abs(Vel)) Vel=Velocity in the local x direction

K4 - Torsional spring

K5 - Torsional damper Torque/Ang Velocity

The damping contribution can be applied using a damping matrix or using a restoring force approach. If the Def Element Damping is active in the DyNoFlex solution options a damping matrix will be employed. The damping matrix is the recommended approach. The damping magnitude is applied to equally to both ends of element.

Special Features

The **CFACT** command is used factor the stiffness (all components) of a the element. It main use to to effectively add or remove couple elements (birth & death) during a time history solution. This should only be used with linear elastic

Restrictions

Only recognised by the **DyNoFlex** analysis module. Although in principle this a linear element (with K3=0), this element requires to be run using a **Non-linear** solution option.

Acts as a CType 0 couple in 3-D Standard analysis module.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	As spring constants	Linear
Non-linear	Damping Ignored	Static Linear
Frequency	As spring constants	Linear
DyNo Flex	All	Nonlinear solution only

-o-

CType 4 Non-linear Translational Couple

Non-linear elastic Spring (Node to Node or Node to Ground)

This element is a general non-linear elastic translational spring element. The non-linear properties of the spring may be defined differently for each local direction and differently depending upon whether the movement is positive or negative.

Large displacement - If the couple is referenced to a Type-6 beam element then the couple local coordinate system will move with that of the referenced element. If a reference element is not defined then local coordinate system will be defined by the current position of the defining nodes or in the case of a Node to Ground, the global coordinate system.

The properties of the spring are defined in a similar manner a standard spring/couple by referencing the K1 to K6 values in the Couple Ele Constants table. For this element each of the K1 to K6 values are used to reference an entry in the RC Constants table. The **RC constants** are then used to define the non-liner spring curve. For this element the I table is data is interpreted as follows.

K1 to K6 These are used to defined the non-linear elastic stiffness properties for each of the local translational spring directions. It does this by referencing an entry in the RC Constants Table. The directions referenced by the K constants are given below. If a zero value is specified the no stiffness will be applied in that direction

K1 + X Direction Stiffness

K2 + Y Direction Stiffness

K3 + Z Direction Stiffness

K4 - X Direction Stiffness

K5 - Y Direction Stiffness

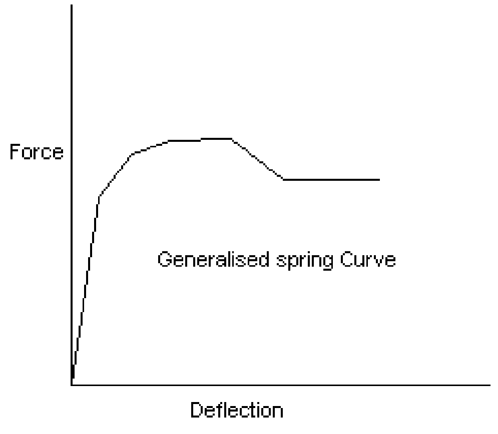
K6 - Z Direction Stiffness

RC Constants

In this element the RC constants table is used to define a non-linear spring curve i.e. a force/defection curve.

Each entry in the table defines a piecewise curve. Up to 7 data point can be used to define the curve.

The curve always starts at the origin. The RC-X1 and RC-Y1 are used to define the first point, which cannot be at the origin. **The first point must produce a positive slope segment**. Other segments of the curve may be positive or negative. Segments of the curve cannot be vertical (infinite stiffness). Also avoid using very stiff segments ie near vertical.



Special Features

The **CFACT** load defintion command can be used with this element.

The **CFACT** command is used factor the stiffness (all components) of a the element. It main use to to effectively add or remove couple elements (birth & death) during a time history solution.

Restrictions

Only recognised by the 3-D NL & DyNoFlex analysis modules.

Acts as a CType 0 couple in 3-D Standard analysis module.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	As spring constants	Linear
Non-linear	Damping Ignored	Non-linear
Frequency	As spring constants	Linear
DyNo Flex	All	Non-linear solution only

-o-

CType 5 Rectangular Contact Couple

Compression gap element. The gap surrounding the node is a rectangular enclosure and can be defined differently in the principle directions.

It can be 'node to node' or 'node to ground'.

The gap direction is always orientated in the local y and z directions of the element.

Large displacement - If the couple is referenced to a Type6 beam element then the couple local coordinate system will move with that of the referenced element.

Note that when referencing a coordinate system or beam element for orientation the numbering of the couple also dictates the direction. If the I-J nodes of of the element are switched the couple direction will rotate 180 degrees about the x axis. The I node of the couple when moving towards the J node should be in the same direction as the +y axis of the reference coordinates.

The stiffness property defines the properties of the contact.

K1 - Normal Stiffness - local y and z direction

K2 - Tangential Stiffness (zero defaults to K1/100) -local x direction

K3 - Coefficient of Friction - Local x from contact in the local y direction

K4 - Coefficient of Friction - Local x from contact in the local z direction

K5 - Gap

K6 - Initial condition 1=Close 0=Open

In K1 is defined as a negative integer value then the normal stiffness of the gap will be defined by a non-linear spring. The absolute value of K1 is then used to reference an entry in the RC Constants table that defines the non-linear relationship of the normal stiffness.

If K5, the gap is specified > 0 then gap in all directions will be the same.

If K5, the gap is specified as 1E12 then the gap distance will all be the same and be set by the distance between the defining nodes.

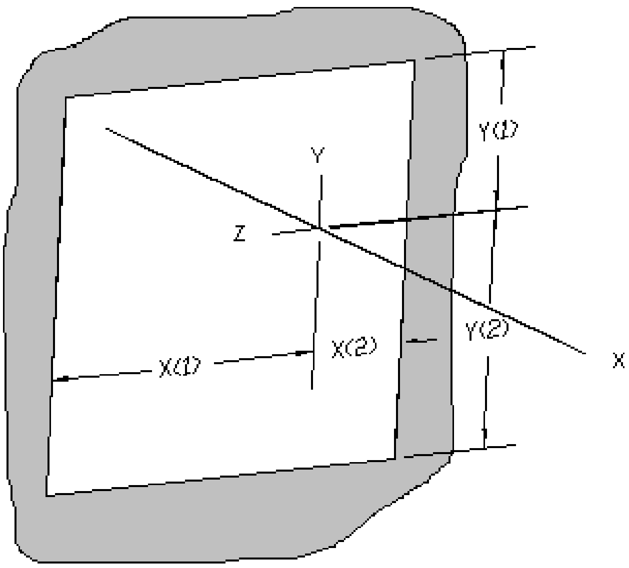
If K5, the gap is specified < 0 then the gaps for each direction will be set by reference to the RC(Constants) Table, with the value being the entry in the table. The table entries define the following:

x(1) = +z local direction gap

x(2) = -z local direction gap

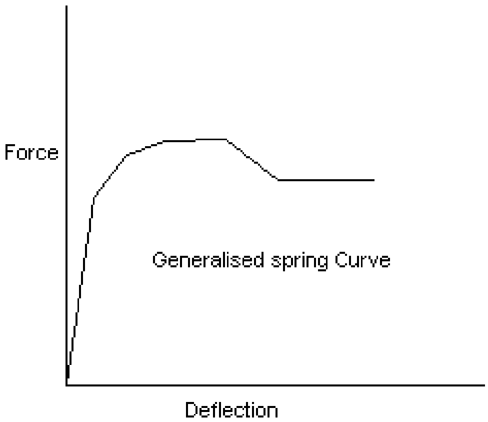
y(1) = +y local direction gap

Y(2) = -y local direction gap



RC Constants

In this element the RC constants table is used to define a non-linear spring curve i.e. a force/defection curve. Each entry in the table defines a piecewise curve. Up to 7 data point can be used to define the curve. The curve always starts at the origin. The RC-X1 and RC-Y1 are used to define the first point, which cannot be at the origin. **The first point must produce a positive slope segment**. Other segments of the curve may be positive or negative. Segments of the curve cannot be vertical (infinite stiffness). Also avoid using very stiff segments ie near vertical.



Restrictions

Full features only recognised by the 3-D NL & DyNoFlex analysis modules.

Acts as a CType 0 couple in 3-D Standard analysis module.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	As spring constants	Linear
Non-linear	All	Non-linear
Frequency	As spring constants	Linear
DyNo Flex	All	Non-linear solution only

-o-

CType 6 Cylindrical Contact Couple

A Compression gap element. The gap surrounding the node is a cylindrical enclosure.

The gap direction is always orientated in the local y and z directions of the element. It can be 'node to node' or 'node to ground'.

The stiffness property defines the properties of the contact.

K1 - Normal Stiffness - local y and z direction

K2 - Tangential Stiffness (zero defaults to K1/100) -local x direction

K3 - Coefficient of Friction - Local x from contact in the local y & z (radial) direction

K4 - 0

K5 - Gap

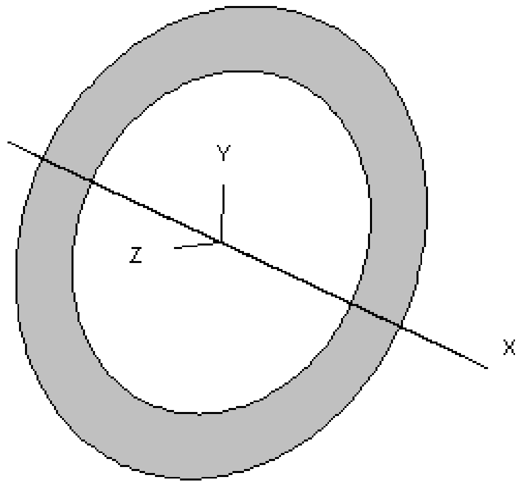
K6 - Initial condition 1=Close 0=Open

In K1 is defined as a negative integer value then the normal stiffness of the gap will be defined by a non-linear spring. The absolute value of K1 is then used to reference an entry in the RC Constants table that defines the non-linear relationship of the normal stiffness.

If K5, the gap is specified > 0 then gap in all directions will be the same.

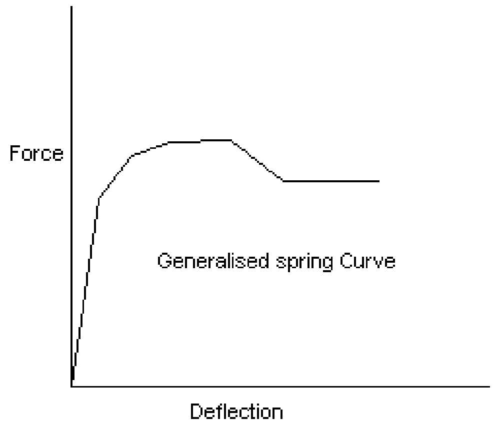
If K5, the gap is specified as 1E12 then the gap distance will all be the same and be set by the distance between the defining nodes.

Note that the CType 5 couple converges more readily.



RC Constants

In this element the RC constants table is used to define a non-linear spring curve i.e. a force/defection curve. Each entry in the table defines a piecewise curve. Up to 7 data point can be used to define the curve.   The curve always starts at the origin. The RC-X1 and RC-Y1 are used to define the first point, which cannot be at the origin.   **The first point must produce a positive slope segment**. Other segments of the curve may be positive or negative. Segments of the curve cannot be vertical (infinite stiffness). Also avoid using very stiff segments ie near vertical.



Restrictions

Full features only recognised by the 3-D NL & DyNoFlex analysis modules.

Acts as a CType 0 couple in 3-D Standard analysis module.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	As spring constants	Linear
Non-linear	All	Non-linear
Frequency	As spring constants	Linear
DyNo Flex	All	Non-linear solution only

-o-

CType 7 General Non-linear Large Displacement Couple

This is similar to is a standard linear couple (spring) element but with the following two additional features:

- Large displacement - If the couple is referenced to a Type6 beam element then the couple local coordinate system will move with that of the referenced element.
- Non-linear springs. Elastic or Inelastic non-linear springs can be defined in the 6 degrees of freedom

For couples the constants K1 to K6 are used to define the stiffness of the couple in terms the element local co-ordinate system. i.e.

K1 x translation

K2 y translation

K3 z translation

K4 x rotation

K5 y rotation

K6 z rotation

The effect of shear offset is included if the couple is defined as a node to node couple and the orientation of the couple is define by the location of the nodes in a similar way to conventional beam elements.

Shear offset will not be accounted for if the orientation of the couple is defined using a reference element or by reference to a coordinate system.

If any of the K's are defined as a negative integer value then the stiffness will be defined by a spring curve. The absolute value of K is used to reference an entry in the RC Constants Table. The entry in the RC constants table define the non-linear spring stiffness.

The CO couple property specifies element's non-linear model:

CO = 0 : Non-linear elastic spring. Unloads down the RC curve.

CO = 1 :Isotropic bi-linear memory model.

CO = 2 : Kinematic bi-linear memory mode. Same as CO = 4 (Legacy)

CO = 3 : Isotropic memory model with constant force limit (can be slower to converge on pipeline applications - recommend using CO = 1 with Et=0).

CO = 4 : Kinematic multi-linear memory mode (initial curve shape is preserved during cyclic loading).

CO = 5 : Kinematic multi-linear memory mode (slope is continued during cyclic loading).

RC Constants Table

The RC constants table is used to define a non-linear spring curve i.e. a force/deflection curve.

Each entry in the table defines a piecewise curve. Up to 7 data point can be used to define the curve.

The curve always starts at the origin. The RC-X1 and RC-Y1 are used to define the first point, which cannot be at the origin. **The first point must produce a positive slope segment**. Other segments of the curve may be positive or negative (CO = 0 or CO = 4). Segments of the curve cannot be vertical (infinite stiffness). In the solution the final slope is maintained if the deflection extends beyond the curve definition.

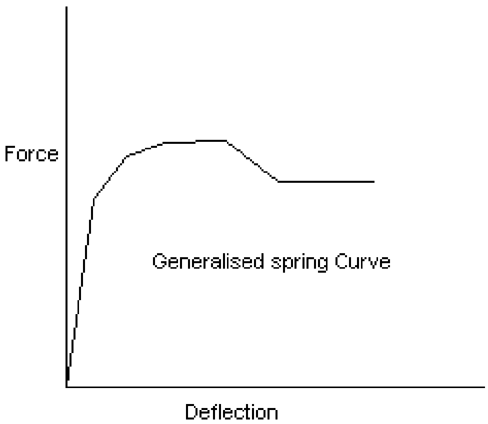
CO = 0 : Spring force follows a generalised RC curve (elastic non-linear).

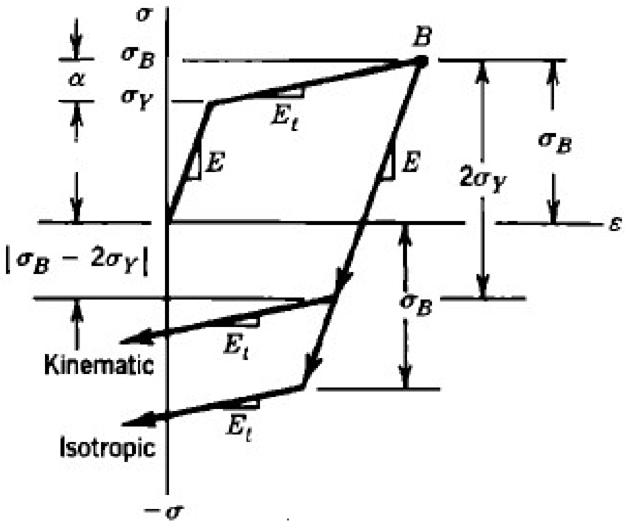
CO = 1 : First two points define a bi-linear curve (Et can be zero).

CO = 2 : First two points define a bi-linear curve (Et can be zero).

CO = 3 : First point define the stiffness and the force limit (Et=0).

CO = 4 & 5 : Up to 7 points define a multi-linear curve.





Special Features

The **CDISP** and **CFACT** load defintion commands can be used with this element. The **CDISP** command is used to define relative movement between coupled degrees of freedom.

The **CFACT** command is used factor the stiffness (all components) of a the element. It main use to to effectively add or remove couple elements (birth & death) during a time history solution. This should only be used with linear elastic

These command should only be used with a C0 = 0 type element, i.e. linear or non-linear elastic.

Restrictions

Full features only recognised by the 3-D NL & DyNoFlex analysis modules.

Offset shear cannot be included in large displacement solutions.

Acts as a CType 0 couple in 3-D Standard analysis module.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	As spring constants	Linear
Non-linear	All	Non-linear
Frequency	As spring constants	Linear
DyNo Flex	All	Non-linear solution only

-o-

CType 8 Pipe Impact-Energy Dent Monitor

This Node to Node couple element is primarily for the modeling of tubular beams that undergo local plastic denting due to the action of lateral point loading during beam plasticity solutions (DyNoFlex solution required). It can also be applied in a more general sense for local energy monitoring.

The basis of operation is that the strain energy in the couple is monitored during an incremental solution ($E=\sum F\Delta\delta$) and the total energy obtained. Accordingly this element requires an incremental solution using DyNoFlex.

If the couple is attached to a node connecting two Type 6 pipe beam elements then the pipe denting effects of any loading applied to that J node of the couple will be assessed. The plasticity type (GeomType) of these Type 6 elements must be must be less than 6 i.e. frame plasticity element types only. This loading on the connecting elements is obtained from the couple force components normal to the elements.The varying dent characteristics are evaluated under the incremental loading and if the loading is sufficiently high to cause local plastic denting the reduction of plastic moment capacity of the beam section will be taken into account in the plastic solution.

The solution **Time History Plot-Step Interval** option should be set to 1 if the DNV dent model is to be used. This is only a requirement because the axial loading in the dented element is obtained from the plot data from the previous time step.

Dent properties - depth, plastic moment etc and total energy absorption can be plotted using FS-Graph.

The element can also be used to obtain energy plots from any incremental solution. If the couple is attached to the J node of any beam element then the energy applied to that node is evaluated and can be plotted using FS-Graph. See **Energy Monitoring** below.

Dynamic Solutions - Whist the element can be used in dynamic solutions it cannot used for energy monitoring in dynamic solution because energy may be accounted for more than once due to the cyclic loading in the couple. Additional consideration of the resulting impulsive contact force which determines the dent depth and resulting reduction in plastic moment capacity is also required, this can be difficult when mass contact is involved.

Tube Denting

Two dent models are available:

- ELLINAS C.P. and WALKER A.C., Damage on Offshore Tubular Bracing Members. Proceedings of the IABSE colloquium on ship collision with bridges and offshore structures. Preliminary report, Copenhagen 1983.
- DNVGL-RP-C202 Structural design against accidental loads

The pipe element **CO Property** is used to identify the label of the CType 8 couple. The impacted pipe elements must be Type 6 (GeomType 1-4) beam elements.

The CType 8 Couple element **Reference Element** must be the element whose **J node** is attached to the couple, element **EJ**.

The CType 8 couple properties defined by its property code are used to identify the impacted elements and their properties. The properties used are:

K1 - * Pipe OD

K2 - * Pipe Wall Thickness

K3 - * Pipe Material Yield Strength

K4 - B - Dent length - If this value is non-zero then the DNV dent model will be used.

K5 - Design Axial Load - If this value is specified the effect of compressive element loading will be included (DNV dent model only)

K6 - ** Stiffness of the Couple element in all degrees of freedom

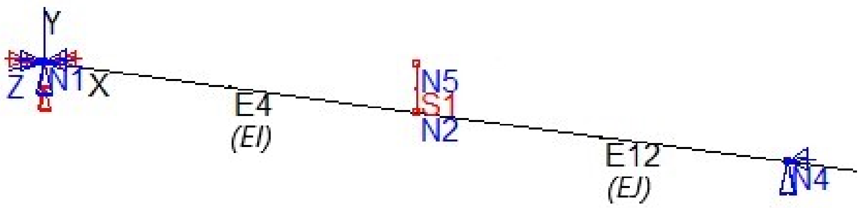
CO - The element Label of the I node connected pipe element, element EI. If C0 = 0 local denting will be ignored.

* Note that K1 to K3 will always be entered when the model is saved using the pipe properties of **EI**.

** If K6 is zero a value of 1E12 will be entered when the model is saved

The following illustrates the definition required for impact at Node 2 i.e. the node connecting pipe elements E4(**EI**) & E12 **EJ**). The nodes of S1(Dent Couple) can be coincident.

The impact load or displacement is applied to N5, the J node of Couple S1.



Elem		Connectivity				Length	Geom	Mat	Release			CO	Bend
No	Type	Node-1	Node-2	Node-3	Rot			Codes	Code	Z	Y	Code	Radius
4	6	1	2	0	0.000	1.0000	1	0	1	0	0	1	0.000
12	6	2	4	0	0.000	1.0000	1	0	1	0	0	1	0.000

Cpl Elem	Connectivity			Ref.Ele.	Coord Sys	PropCode
No	Node-1	Node-2	Rot			
1	2	5	0.000	4	0	2

*** SPRING/COUPLE CONSTANTS TABLE

Code	Type	K1	K2	K3	K4	K5	K6	CO
2	8	1.000E-01	5.000E-03	3.450E+08	0.000E+00	0.000E+00	1.000E+12	12

Dent properties - depth, plastic moment etc and total energy absorption can be plotted using FS-Graph. The components of a Type 8 couple show the following:

Dx - Dent depth.

Dy - Reduced Plastic moment at the dent. (For Ellinas & Walker - If d/D>0.5 Mpd=0.02Mp)

Dz - Energy absorbed by the dent (always uses Ellinas & Walker correlation).

Rx- Energy absorbed by frame displacement.

Ry - Total energy absorbed.

Rz - Denting force

Element Length Considerations. When using this element it is desirable although not essential that the pipe element should span between the point of loading and the point of support. If the span is segmented there is tendency for softening of the plastic resistance. Hence it is not recommended to use very short elements at each side of the dent node.

Energy Monitoring

This element can also be used in cases where the energy absorbed at a load point on a structure is required to be evaluated. The beam elements do not require to be pipe elements and any number of element can be be connected to the couple. The CO couple should be set to zero for this scenario i.e no connecting EI element. Ensure the connecting elements are frame plasticity types (0 > GeomType < 6).

FS-Graph - Ry couple component will plot the Total energy absorbed and the Rz couple component the resultant force applied to the J node. Note that the resulting force is an absolute value.

Dent Models

Ellinas & Walker Dent Model

If dent> 0.5D Mpd = 0.02Mp

$$m_p := \frac{\sigma_{ay} \cdot t^2}{4}$$

$$P_d := k \cdot m_p \cdot \frac{(dent)^{0.5}}{D_m^{\ 0.5}}$$

$$\sigma_{pd} := \left[\left[\frac{16}{9} \cdot \frac{(dent)^2}{D_m^{\ 2}} + \frac{t^2}{D_m^{\ 2}} \right]^{0.5} - \frac{4}{3} \cdot \frac{dent}{D_m} \right] \cdot \left(\frac{D_m}{t} \cdot \sigma_{xcL} \right)$$

$$\beta(d_d) := \left(1 - \frac{\sigma_{pd}(d_d)}{\sigma_{xcL}} \right) \cdot \left(\frac{d_d}{D_m} \right)^{0.5}$$

$$M_{pd}(d_d) := \left(\cos(\beta(d_d)) - \beta(d_d) \right) \cdot M_p$$

DNV Dent Model

If wd > 0.9D wd = 0.9D

$$\frac{R}{R_c} = kc_1 \left(\frac{w_d}{D} \right)^{c_2} \quad \text{for: } \frac{w_d}{D} \leq 0{,}5 \qquad (3.8)$$

$$R_c = f_y \frac{t^2}{4} \sqrt{\frac{D}{t}}$$

$$c_1 = 22 + 1{,}2 \frac{B}{D}$$

$$c_2 = \frac{1{,}925}{3{,}5 + \frac{B}{D}}$$

$$k = \begin{cases} 1 & \text{if } \frac{N_{Sd}}{N_{Rd}} \leq 0{,}2 \\ 1 - 2\left(\frac{N_{Sd}}{N_{Rd}} - 0{,}2 \right) & \text{if } 0{,}2 < \frac{N_{Sd}}{N_{Rd}} < 0{,}6 \\ 0 & \text{if } \frac{N_{Sd}}{N_{Rd}} \geq 0.6 \end{cases}$$

$$\frac{M_{red}}{M_p} = \cos\left(\frac{\theta}{2}\right) - \frac{1}{2}\sin\theta \qquad (3.18)$$

$$M_p = f_y D^2 t$$

$$\theta = \mathrm{acos}\left(1 - \frac{2w_d}{D} \right)$$

where:

N_{Sd} = design axial compressive f
N_{Rd} = design axial compressive r
B = width of contact area
w_d = dent depth
D = diameter of tubular memb
t = thickness of tubular memb

Restrictions

Full features only recognised by the 3-D NL & DyNoFlex analysis modules.

Acts as a CType 0 couple in 3-D Standard analysis module using K6 in all freedoms.

Solution Option	Property Recognition	Comments
3-D Standard	K6 applied to all Ks	Linear
Non-linear	Type 8 Specific	Non-linear
Frequency	As spring constants	Linear - Probable mechanism due to zero K values
DyNo Flex	Type 8 Specific	Non-linear solution only

-o-

CType 10 Compression/Frictional Gap

Compression gap element. The gap direction is always orientated in the local x direction of the element. It can be 'node to node' or 'node to ground'.

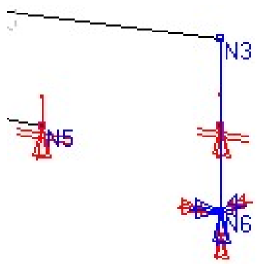
Large displacement - If the couple is referenced to a Type 6 beam element then the couple local coordinate system will move with that of the referenced element.

The element has 3 basic conditions: open; closed & sticking; closed & sliding. When the element is closed & sticking it is connected by a spring stiffness, this is gap sliding stiffness. I governs the distance the gap will slide before the frictional force (μN) is fully mobilised. When the gap is sliding the gap stiffness effect is replaced by a force that represents the sliding resistance. From a solution convergence aspect it is often beneficial to have a soft spring but the relative stiffness of the overall problem (physical configuration) must also be considered. The distance moved before the contact slides is governed by the tangential stiffness and is often termed the mobilization distance.

Using Reference Elements or Coordinate Systems for Couple Orientation

When using **reference** orientations with contact couples the following behavior should be noted:

- The connectivity direction of the reference element dictates the contact direction because it defines the local couple X direction.
- The X direction of the coordinate system dictates the contact direction because it defines the local couple X direction.
- With a 'node to node' couple, the node connectivity also dictates the direction (so always run the connectivity in the same direction as the reference X direction). The gap closes when the I Node moves towards the J Node i.e. the distance between the nodes becomes increasingly more negative as the I node moves in the +ve X direction relative to the J node.
- The force direction convention of a 'node to ground' is the reverse of that of a 'node to node (consistent with nodal restraint convention).

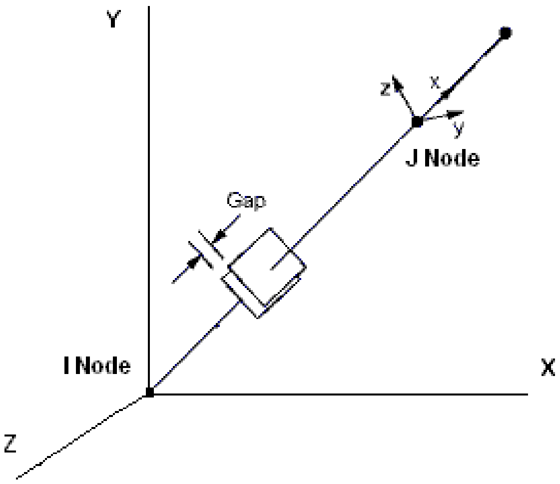


The orientation of the gap can be seen by the visible orientation symbols

Node to Node - N3 moves towards N6 the gap closes.

Node to Ground - N5 moves down the gap closes

Note that If the I Node and J node are switched the I node has to move in the -ve local X direction for the distance between the I and J becomes increasingly less i.e. the contact direction is reversed.



The stiffness property defines the properties of the gap.

K1 - Normal Stiffness

K2 - Tangential Stiffness (zero defaults to K1/100)

K3 - Coefficient of Friction - Local y

K4 - Coefficient of Friction - Local z

K5 - Gap - Local x direction

K6 - Initial condition 1=Close 0=Open

CO/RC X1 - Tangential Stiffness - Local z

CO/RC X2 - Normal Damping

If K1 is defined as a negative integer value then the normal stiffness of the gap will be defined by a non-linear spring. The absolute value of K1 is used to reference an entry in the RC Constants table that defines the non-linear relationship of the normal stiffness.

K2 is the Tangential (Sliding Stiffness). Applied in both Local y and Local z directions unless the Local z direction is specified using CO/RC X1.

K3 & K4 define the coefficients of friction (μ) in the local directions. This is used to evaluate the sliding resistance - μN.

If K4 is zero then the sliding resistance will oppose the direction of the resultant displacement- Coulumb Friction.
If K4 is non zero then the resistances in the local directions will be independent and be based on the value μ defined by K3 & K4.
If K4 is specified as a negative integer value then the lateral friction in the z direction can be defined as a Function of Gap z displacement.The absolute value of K4 is used to reference an entry in the RC Constants table that defines the non-linear relationship of the lateral friction.
If K4 is negative and K3 is also negative then the friction will be the same in both axis. Note that K3 should never <0 unless K4<0..

If K5, the gap is specified < 0 then the element will induce an interference by the value specified.

If K5, the gap is specified as 1E12 then the gap distance will be set by the initial distance between the defining nodes (see also Type 15). Useful for defining uneven profiles since the gap is not dependent upon the gap property.

CO/RC X1 is used to define the Tangential sliding stiffness in the local z direction. Note that CO/RC X1 is the value of X1 in the RC table, where the table code number is defined by the CO couple property.

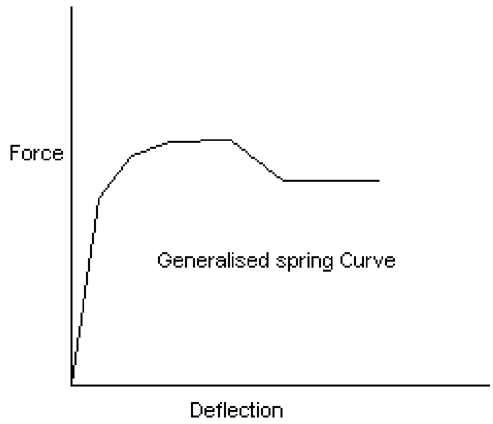
CO/RC X2 is used to define damping (Force/velocity) in the normal x direction. Note that CO/RC X2 is the value of X2 in the RC table, where the table code number is defined by the CO couple property. Damping is removed when the gap is open. This is somewhat an unrealistic criteria (physical condition).

Large Displacement

If the orientation of the couple is referenced to Type 6 Beam element then the couple will rotate with the beam axis.

RC Constants Table

In this element the RC constants table is used to define a non-linear spring curve e.g. a force/deflection curve or the friction as a function of deflection . Each entry in the table defines a piecewise curve. Up to 7 data point can be used to define the curve.   The curve always starts at the origin. The RC-X1 and RC-Y1 are used to define the first point, which cannot be at the origin when used for stiffness definition.   Fore stiffness definition t**he first point must produce a positive slope segment**. Other segments of the curve may be positive or negative. Segments of the curve cannot be vertical (infinite stiffness).



Restrictions

Full features only recognised by the 3-D NL & DyNoFlex analysis modules

Acts as a contact (compression) only element in the 3-D Standard analysis module.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	Contact	Make of break contact, no gaps and no friction. K1 to K6 are interpreted as stiffness values.
Non-linear	All	Non-linear
Frequency	As spring constants	Linear
DyNo Flex	All	Non-linear solution only Require FullNR

-o-

CType 11 Tension Gap

This is a tensile contact element. The gap direction is always orientated in the local x direction of the element. It can be only used as a 'node to node' .

- Large displacement - If the couple is referenced to a Type 6 beam element then the couple local coordinate system will move with that of the referenced element.

The stiffness property defines the properties of the gap.

K1 - Normal Stiffness

K5 - Gap

K6 - Initial condition 1=Close 0=Open

If K5, the gap is specified < 0 then the element will induce an interference by the value specified.

If K5, the gap is specified as 1E12 then the gap distance will be set by the distance between the defining nodes.

Large Displacement

If the orientation of the couple is referenced to Type 6 Beam element then the couple will rotate with the beam axis.

Restrictions

Gap features only recognised by the 3-D NL & DyNoFlex analysis modules

Acts as a contact (tension) only element in the 3-D Standard analysis module.

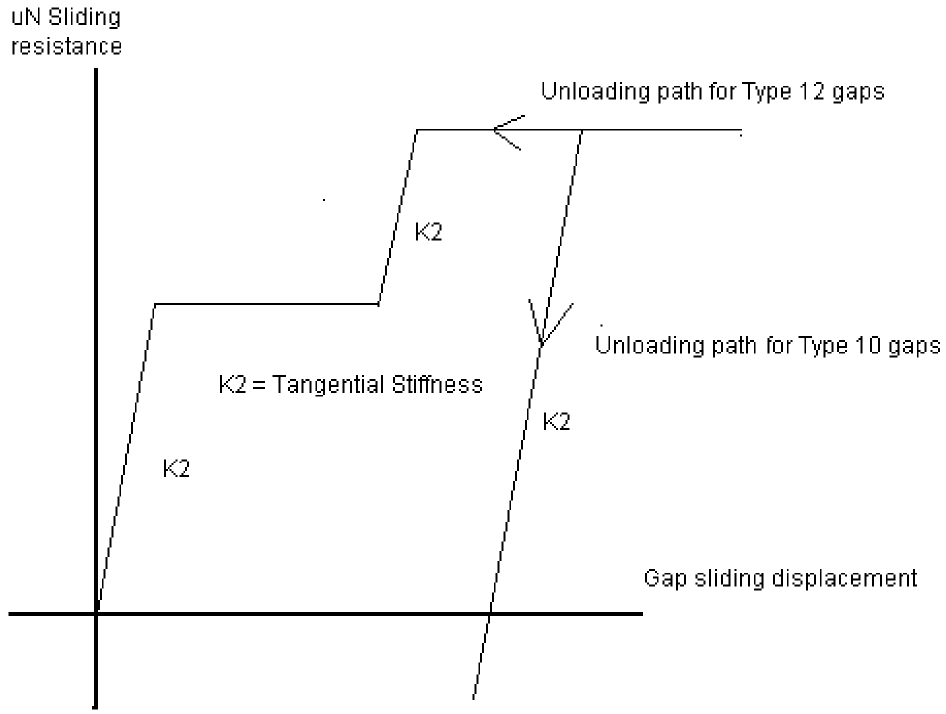
Solution Option	**Property Recognition**	**Comments**
3-D Standard	Conatct	Make of break contact, no gaps and no friction. K1 to K6 are interpreted as stiffness values.
Non-linear	All	Non-linear
Frequency	As spring constants	Linear
DyNo Flex	All	Non-linear solution only Require FullNR

-o-

CType 12 Compression/Frictional Gap (Conservative)

Compression gap element. Conservative friction assumes no energy is lost and the gap is can unload down the same path. In all other respects it is the same as a CType 10 Gap element. The following illustrates the difference between the Type 10 and Type 12 elements.

The Type 12 element generally converges faster than a Type 10 and recommended if unloading is not a requirement.



-o-

CType 14 Node to Ground Surface Contact (2/3-D)

This is a node to ground friction gap element where the ground is represented by a surface. The element is defined as a spring to ground couple. The couple type is defined by the element stiffness couple property code which defines the surface properties.

The element is essentially a 2-D element which can be define in 3-D space. The element nodes only recognises two degrees of freedom i.e. translations in the local x and y directions

A node can be connected to more that one surface.

The element supports Large Displacements. The position of couple node relative to the surface define the gap distance.

The surface can be straight or a circular surface.

> A straight surface is defined by reference to a grounded beam element. The solid surface is on the negative local y axis side. This beam element follows the standard beam orientation rules e.g. a local rotation of 180 degrees will switch the surface direction. The end nodes define the extent of the surface.
>
> A circular surface uses the Ref Element property of the couple to reference the label of a node that defines the centre of the circle. Angles Theta1 and Theta2 define the end points of the surface. The sign of the radius dictates whether the surface is internal or external.

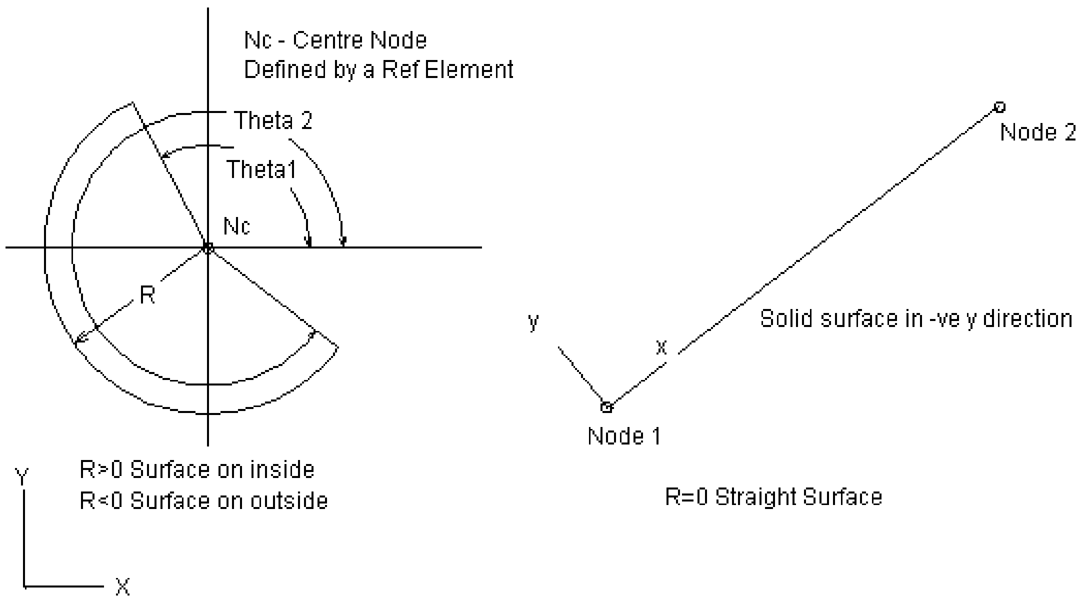
The stiffness property of th couple defines the properties of the surface as below.

K1 - Normal Stiffness

K2 - Tangential Stiffness (zero defaults to K1/100)

K3 - Coefficient of Friction (Conservative friction model)

K4 - Theta 1 (Degrees)	Zero for straight surfaces
K5 - Radius	Zero for straight surfaces
K6 - Theta 2 (Degrees)	Zero for straight surfaces



Special Considerations

For non large displacement solutions the more easily define contact elements are likely to be more suitable eg Type 10 or Type 15.

The element can only be used to model node to ground behavior. The nodes used to define the surfaces i.e. the center node and end nodes of the reference beam must by fixed in all degrees of freedom. They can however be moved in the x and y directions using prescribed displacements.

An additional requirement is that the nodes used to define the surfaces **MUST** be connected to the rest of the model. This can be done with dummy elements with zero stiffness properties. It is good practice to use the reference elements as the dummy elements and match the element label with the couple property code. When connecting these dummy elements, connect them in a manner that minimises the matrix bandwidth.

Convergence difficulties can be experienced with these elements. Use of a low contact stiffness may aid convergence.

Restrictions

Full features only recognised by the 3-D NL & DyNoFlex analysis modules

Acts as a CType 0 couple in 3-D Standard analysis module.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	As spring constants	Linear
Non-linear	All	Non-linear
Frequency	As spring constants	Linear
DyNo Flex	All	Non-linear solution only

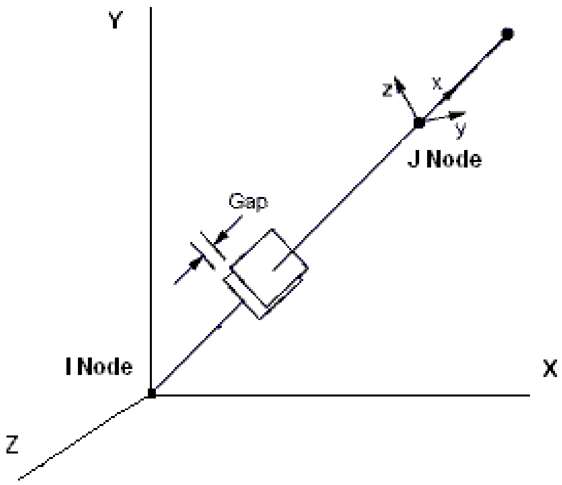
-o-

CType 15 Compression/Frictional Gap

Compression gap element. The gap direction is always orientated in the local x direction of the element. It's main use is for Large displacements.

This element is very similar to a Type10 or Type 12 couple element. The main feature of this element is that the gap will always be equal to the varying distance between the nodes plus any gap set by K5 (usually set to zero) and contact is made only when the nodes become together in the local x direction. This implies that is should always be orientated using a reference element and thereby provide a flat contact surface normal to the reference element's local x axis.

Large displacement - If the couple is referenced to a Type 6 beam element then the couple local coordinate system will move with that of the referenced element otherwise the orientation is always as the orientation of the undeformed geometry . In most cases for large displacement the element would be connected to the end of a beam element which is also used to define the X axis of the element.



The stiffness property defines the properties of the gap.

K1 - Normal Stiffness

K2 - Tangential Stiffness (zero defaults to K1/100)

K3 - Coefficient of Friction - Local y

K4 - Coefficient of Friction - Local z

K5 - Gap - Local x direction

K6 - Friction model 0 - Non-conservative Non Zero - Conservative

CO/RC X1 - Tangential Stiffness - Local z

CO/RC X2 - Normal Damping

If K1 is defined as a negative integer value then the normal stiffness of the gap will be defined by a non-linear spring. The absolute value of K1 is then used to reference an entry in the RC Constants table that defines the non-linear relationship of the normal stiffness.

K2 is the Tangential (Sliding Stiffness). The element has 3 basic conditions: open; closed & sticking; closed & sliding. When the element is closed & sticking it is connected by a spring stiffness, this is gap sliding stiffness. I governs the distance the gap will slide before the frictional force (μN) is fully mobilised. When the gap is sliding the gap stiffness effect is replaced by a force that represents the sliding resistance. From a solution convergence aspect it is beneficial to have a soft spring but the relative stiffness of the overall problem (physical configuration) must also be considered.

K3 & K4 define the coefficients of friction (μ) in the local directions. This is used to evaluate the sliding resistance - μN. If K4 is zero then the sliding resistance will oppose the direction of the resultant displacement. If K4 is non zero then the resistances in the local directions will be independent and be based on the value μ defined by K3 & K4.

If K5, the gap is specified < 0 then the element will induce an interference by the value specified. Not normally used

CO/RC X1 is used to define the tangential sliding stiffness in the local z direction. This this is not defined the value defined for K2 will be used. Note that CO/RC X1 is the value of X1 in the RC table, where the table code number is defined by the CO couple property.

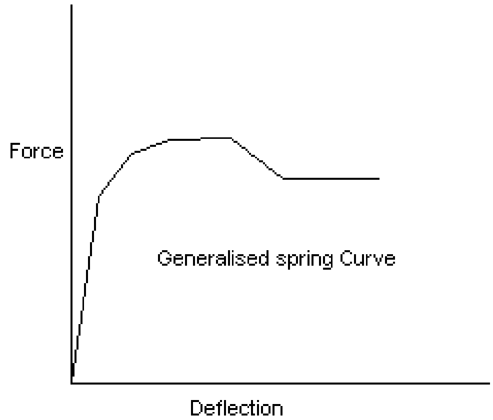
CO/RC X2 is used to define damping (Force/velocity) in the normal x direction. Note that CO/RC X2 is the value of X2 in the RC table, where the table code number is defined by the CO couple property.

Large Displacement

If the orientation of the couple is referenced to Type 6 Beam element then the couple will rotate with the beam axis.

RC Constants Table

In this element the RC constants table is used to define a non-linear spring curve ie a force/defection curve. Each entry in the table defines a piecewise curve. Up to 7 data point can be used to define the curve. The curve always starts at the origin. The RC-X1 and RC-Y1 are used to define the first point, which cannot be at the origin. **The first point must produce a positive slope segment**. Other segments of the curve may be positive or negative. Segments of the curve cannot be vertical (infinite stiffness).



Restrictions

Full features only recognised by the 3-D NL & DyNoFlex analysis modules

Acts as a contact (compression) only element in the 3-D Standard analysis module

Solution Option	**Property Recognition**	**Comments**
3-D Standard	As spring constants	Linear
Non-linear	All	Non-linear
Frequency	As spring constants	Linear
DyNo Flex	All	Non-linear solution only

-o-

CType 16 Surface Contact - Node to Surface

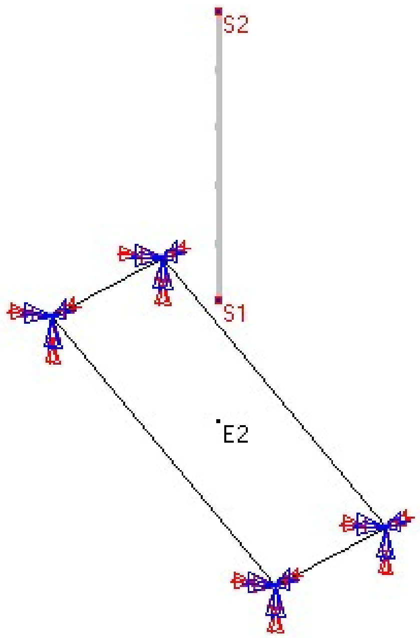
This is a node to ground surface contact element. Gaps and frictional sliding resistance can be defined. The node is connected to a Type 16 Couple and the surface is defined by a shell element.

The couple element is defined as ground couple and the couple type is defined by the element stiffness couple property code. The couples's Ref Ele identifies the target surface shell element.

The element properties are similar to those for Type 10 or Type 12 Compression Gaps. The K5 property can be used to select the reversible or non-reversible friction model i.e. Type 10 or Type 12 behavior.

The target surface is defined by a 3 or 4 node shell element and is identified by the couple's **Reference Element** property. The orientation of the gap is always normal to the shell surface. Only the node locations of the element are used. The shell elements can be any element type in the range 50-69. A Type 61 would be the efficient because no stiffness properties are evaluated for this null designation.

The nodes of the surface defining shell element **MUST** be restrained in all degrees of freedom to ensure that the surface is a ground surface. The surface element must also be connected to the structure by a low stiffness (zero)0 dummy element to ensure that the total element assembly is a connected structure i.e. the surface element is recognised as being part of the model.



Node to Ground
Ele Connectivity Ref.Ele. Coord
Sys Prop Code
Node-1 Node-2 Rot
1 1 1 0.000 2 0
1
2 2 2 0.000 2 0
1

COUPLE CONSTANTS
Prop Type CO K1 K2 K3
K4 K5 K6
1 16 0 1.000E+07 0.000E+00 2.000E-01
0.000E+00 0.000E+00 0.000E+00

Element E2 is a fully restrained Type 60 shell, must be fully restrained .

The stiffness property defines the properties of the gap.

K1 - Normal Stiffness

K2 - Tangential Stiffness (zero defaults to K1/100)

K3 - Coefficient of Friction - Local y

K4 - Coefficient of Friction - Local z

K5 - Gap (additional) - Local x direction or Friction Model Selection (see below)

K6 - Initial condition 1=Close 0=Open

If K1 is defined as a negative integer value then the normal stiffness of the gap will be defined by a non-linear spring. The absolute value of K1 is then used to reference an entry in the RC Constants table that defines the non-linear relationship of the normal stiffness.

K2 is the Tangential (Sliding Stiffness). The element has 3 basic conditions: open; closed & sticking; closed & sliding. When the element is closed & sticking it is connected by a spring stiffness, this is gap sliding stiffness. I governs the distance the gap will slide before the frictional force (μN) is fully mobilised. When the gap is sliding the gap stiffness effect is replaced by a force that represents the sliding resistance. From a solution convergence aspect it is beneficial to have a soft spring but the relative stiffness of the overall problem (physical configuration) must also be considered.

K3 & K4 define the coefficients of friction (μ) in the local directions. This is used to evaluate the sliding resistance - μN. If K4 is zero then the sliding resistance will oppose the direction of the resultant displacement. If K4 is non zero then the resistances in the local directions will be independent and be based on the value μ defined by K3 & K4.

If K5, the gap is specified < 0 then the element will induce an interference by the value specified (added to gap between surface and node). This would normally always be zero in this type of element i.e. the Gap is set by the distance between the surface and the target node.

If K5 is set to 1E12 the friction model will assume non-conservative friction and the Gap will be that when K5 is set to zero.

Large Displacement

The element can be used in both small and large displacement solutions.

Restrictions

Full features only recognised by the 3-D NL & DyNoFlex analysis modules

Do not include in linear analysis i.e Standard 3-D module.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	Non	Do not use. Reference element (shell) invalid.
Non-linear	All	Non-linear
Frequency	As spring constants	Do not use
DyNo Flex	All	Non-linear solution only

-o-

CType 20 Vessel Element (RAO)

This element is used to define vessel motions in waves using RAO.

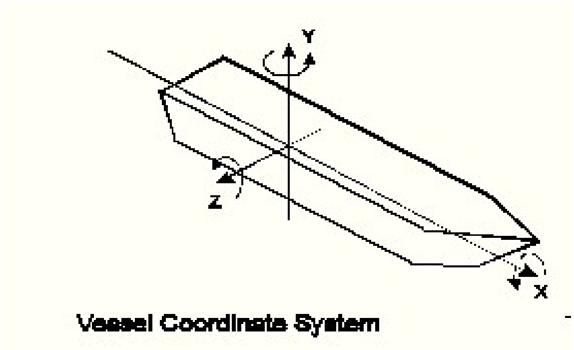
Vessel type motions can be incorporated into a time history analysis using RAOs (Response amplitude operators). The RAO prescribed motions are applied to a mode using Type 20 couples. A Type 20 could be regarded as a Vessel element. The local orientation of the couple defines the orientation of the vessel using the standard FS2000 coordinate systems and should be aligned relative to the global system.

Type 20 elements may be defined as Node to Ground couples or Node to Node Couples.

If displacements or motions of a vessel are required to be defined in addition to those defined by the RAOs i.e. a vessel moving off station, then the couple would be defined as a Node to Node couple so that the additional displacement can be defined using prescribed displacements of the connected node. Degrees of freedom of this connected node should be fully restrained in freedoms where there is additional displacement.

The Type 20 couple has 6 degrees of freedom and couple stiffness have to be defined in each of the degrees of freedom to which RAOs are to be defined. If a non zero stiffness is defines for a direction for which there is no corresponding RAO, then the stiffness will act as a ground spring in that degree of freedom. If an RAO is define in a direction where the spring is zero, then there will be no displacement in that degree of freedom.

The 6 local degrees of freedom of the couple correspond to the motion defined by the 6 directions in the RAO data file. The right-hand corkscrew rule is applied and the Y axis (Heave) is vertical.
Only if the local vessel coordinate system is in alignment with the global will the RAO Wave direction be the same as the FS-Wave angle in the UWData file.



K1	x translation	**A1, P1**	**Surge**
K2	y translation	**A2, P2**	**Heave**
K3	z translation	**A2, P3**	**Sway**
K4	x rotation	**A3, P4**	**Roll**
K5	y rotation	**A5, P5**	**Yaw**
K6	z rotation	**A6, P6**	**Pitch**

Phase Angles - A positive phase angle lags the wave.

Wave Direction - This is the same convention used by FS-Wave. If the Vessel axis aligns with the global axis then:

- A wave approach angle of 0 degrees corresponds to wave traveling in the positive X direction i.e. following sea in the above.
- A wave approach angle of 90 degrees corresponds to a wave traveling in the negative Z direction i.e. starboard beam sea in the above.

RAO data can be presented using different conventions if the source RAO data does not follow the above convention the source data will require to be modified. Identification parameter D1 to D4 enable most RAO data to be converted to FS2000 convention without too much manipulation.

Direction Identifiers **D1, D2**, and **D3** are used to identify which columns of the source RAO data apply to which direction.
e.g.
1,2,3 Implies Surge(1), Heave(2), Sway(3)
1,3,2 Implies Surge(1), Sway(3), Heave(2)

Direction Identifier **D4** is used to switch the sign of the RAO phase angles.
e.g. 1 if positive phase lags the wave, -1 if positive phase leads the wave

The magnitude of the couple stiffness used in the Type 20 element does have an effect on the resulting motion. If the stiffness is to small then motion may not be achieved and if the stiffness is too high, then convergence problems may be experienced. Avoid using high stiffness values (1E20). Stiffness values 10000 times the stiffness of connected structure will provide sufficient accuracy. A value of 1E10 or 1E12 is recommended for most systems.

RAO motions are requires to be defined in all 6 degrees of freedom in the local coordinate system of the vessel. If motions are not require to be defined in a particular degree of freedom then the magnitude of the amplitude must be specified as zero in that degree of freedom.

In time history load case up to 4 different RAOs can be assigned any number of vessels. The RAO is assigned to the vessel using the couple's CO property. The CO number relates to the position in the UWData file. The RAO list in the UWData file is a list that contains the file IDs of the RAO data files. RAO data files are identified by their model file name extension.

The section of the data in the <modelname>.UWData'N' that defines the type of wave train is shown below. In this case two vessels are present in the model.

```
2              No of RAO Files
URAO1_45        Model file extension
URAO2_45        Model file extension
```

The file name extensions of RAO data files are arbitrary but always start them with a U. A U extension identifies a user defined file that will always be archived. The following convention is recommended for RAO file extensions.
Consider the example **<modelname>.URA02_45** The 02 represents the **CO** constant (Vessel 02) and the **45** represents the wave direction relative to Vessel 02.

Linear interpolation is used to match the RAO frequency data points with specific wave frequencies.

RAOs are defined by a text file (space or comma delimited) using the following format. The frequencies must be in ascending order e.g. Freq2 > Freq1 etc. The description is not used by the program and is there for user reference.

RType, D1, D2, D3, D4, Description
Number of frequency points (NF)
Freq1, A1, P1, A2 , P2, A3 , P3, A4, P4, A5 , P5, A6 , P6
Freq2, A1, P1, A2 , P2, A3 , P3, A4, P4, A5 , P5, A6 , P6
...
..
Freq(NF), A1, P1, A2 , P2, A3 , P3, A4, P4, A5 , P5, A6 , P6

Where:

RType If **RType**=0 rotational amplitudes are degrees/m wave height . If **RType**=1 rotational amplitudes are degrees/maximum wave slope(degrees)

D1 to D3 Identifications (see above)

D4 can be used to change the phase convention.

Description This is for user comments and not used by the program.

Feq(NF) is in radians/s

A1 to **A3** are translational amplitudes in m/m wave height

A4 to **A6** are rotational amplitudes in degrees/m wave height or degrees/MWS (MWS=Maximum wave slope in degrees=180H/L)

P1 to **P6** are phases in degrees. Positive phase lags the wave D4 = 1. Positive phase lead the wave D4 = -1

A simple example below shows a unit RAO in which y translation follows the wave surface elevation.

```
0,1,2,3,1,Wave Surface
2
0,0,0,1,0,0,0,0,0,0,0,0,0
100,0,0,1,0,0,0,0,0,0,0,0,0
```

Special Features

The **CFACT** command is used factor the stiffness (all components) of a the element. It main use to to effectively add or remove couple elements (birth & death) during a time history solution.

Restrictions

Full features only recognised by the DyNoFlex analysis module.

Acts as a CType 0 couple in 3-D Standard analysis module.

Solution Option	**Property Recognition**	**Comments**
3-D Standard	As spring constants	Linear
Non-linear	Neglected	
Frequency	As spring constants	Linear
DyNo Flex	All	Non-linear solution only

-o-

5.3 Restraints & Prescribed Displacements

Once the basic geometry of the model has been created, i.e. all the nodes and elements have been defined, it is necessary to restrain the model, i.e. apply the appropriate support boundary conditions for the analysis. It is difficult to define rules for restraints as the restraints are very much dependent upon the particular structure, but the following rule must be complied with.

The model in the global sense must be restrained in all the degrees of freedom for that particular analysis.

The actual loading applied to the structure is totally independent of the restraints required for a successful solution. A common fault amongst inexperienced users is to forget to restrain the model in directions in which there are no applied loads. A typical example of this is the 3-D analysis of a straight beam with loads only in one plane, e.g. y loads. In such a case it is still necessary to apply restraints in both the z translation and x rotation.

Restraints are applied using the Restraints definition box.

Using Ground Springs to Restrain a Model

Models may be restrained using Node to Ground Couples. This provides a very convenient method of applying restraints to a model that are not orientated to the global axis, since Ground Springs can be referenced to a coordinate system or the orientation of a beam element. By using relatively stiff springs the support can be made effectively rigid.

Using Prescribed Displacements to Restrain a Model

The model may be restrained solely by using prescribed displacements and thereby provide a restraint set which is load case dependent. It should be noted that to restrain a node with zero displacement requires that a very small displacement be prescribed, i.e. 1E-12. This is necessary since the program convention is that a zero prescribed displacement signifies a freedom. Prescribed displacements can be applied to restrained freedoms.

WARNING 3D Standard Solution

When running multiple load cases using the **3D Standard** Solution, load cases containing prescribed displacements must not be included with other load cases unless they also are required to have the same prescribed displacements. The reason for this is that prescribed displacements define the stiffness matrix of the model. Since the matrix is not reformed when running multiple-load cases it will not therefore take into account possible different restraint conditions ie the prescribed displacement will be applied to all cases in the same run.

When running multiple load cases using the **3D Standard** Solution with prescribed displacements the following restrictions should be considered.

- The first load case in the combination must contain the prescribed displacements which will be applied to all load cases in the combination.
- Unless the prescribed displacements are identical in each load case no prescribed displacements should exist in any other load case other than the first.

It is always safer to run cases with prescribed displacements using a separate solution. If the load cases are required to be combined then submit the solution as a pre-combined case ie Pre-Processing. Using this approach the prescribed displacements will be applied to all load cases in the solution, and must be contained in only one of the load cases.

Pre-Processing

Pre-Processing is the preferred method when prescribed displacements are required to be applied to other load cases. This ensures the principle of superposition is valid. This is a typical requirement when using prescribed displacements to restrain a model i.e. make the restraints load case dependent. Using Pre-processing ensures that all load cases are subject to the same displacements i.e.where the prescribed displacements will be, and must be, contained in only one of the load cases in the combination.

Post-Processing

Results cases with and without prescribed displacements may be combined in the Post-Processor. Care must be taken using the approach so it does significantly invalidate it the principle of superposition. To ensure superposition validity only load cases with the same prescribed displacements should be combined.

WARNING Non-linear Incremental Analysis

When using the **3-D NL** or the **FS-DyNoFlex** incremental solvers it is a requirement that ALL prescribed displacements in a time history solution must be defined in the first load case listed in the load case combination used to define the load history.

The magnitude of the displacement can vary between load cases in the combination but they can only be applied to the same degrees of freedom identified in the first load case. It is not essential to include PDs in all load cases, in older versions of FS2000 this was a requirement.

When using time curves it may be convenient not to list the load cases in chronological order so as to avoid defining PD in all load cases.

Restraints and Prescribed Displacement Reactions

There are sometimes slight differences in format between the output obtained for restraint and prescribed displacement reactions depending upon whether the solution was undertaken using the linear solver or a non-linear solver. These differences have no effect of the displaced solution and the resulting loads and stresses would be identical.

If Nodal Forces are applied in a restrained direction (same DOF)

Linear: *It will be included in the restraint list even though it results in no element loading. The Nodal Forces will be included in the restraint summation.*
Non-Linear : *It will not be included in the restraint list. The Nodal Forces will not be included in the restraint summation and will therefore be different from the input load summation.*

If Prescribed Displacements added to a Restrained Node (different DOF)

Linear *PDs will be included in the Restraint Reaction List and vica-versa.*
Non-Linear *PDs will only be shown in the PD List*

Non-Linear Solution with Solid Elements

If the model contains shell elements (Type 50 or 53) then the defined element load contribution e.g. gravity to a restraint connected to that element will not be included in the restraint force output listing (small effect for practical meshes). This only effects direct loads not moments.

If the model contains solid elements, the reactions will not be evaluated or listed when using a non-linear solver. If reaction magnitudes are a solution requirement then the model should be use Ground Couples to provide the model restraint. These will be listed and can be plotted in the same manner as restraints.

-o-

5.4 Property Tables

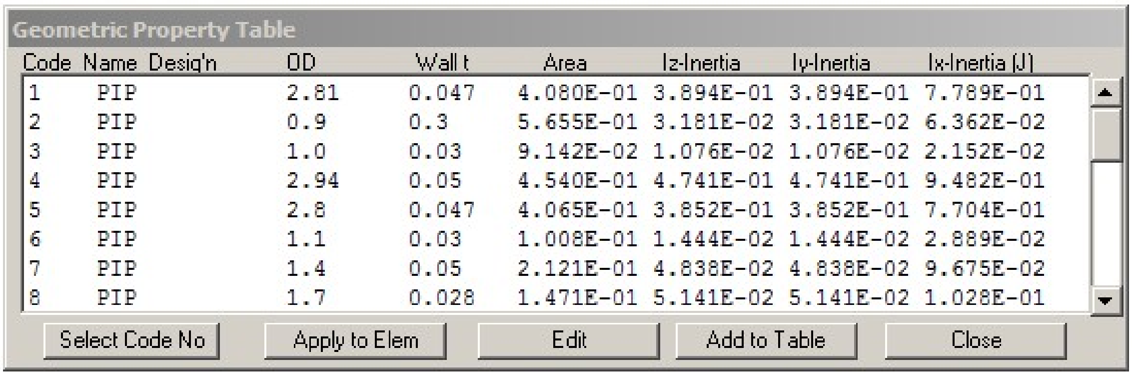
-o-

5.4.1 Geometric Property Tables

The geometric property code attribute (Geom & Taper in Element Defn box) of an element are used to identify the appropriate properties of an element by referencing it to the Geometric Property Table.

Property data is entered in the geometric property table using the Geometric Properties box.

The property Table list can also be view from the Data menu. This list only shows the main properties. It is very useful for assigning properties to elements using the mouse pick (**Apply to Element** button).



Geometric Property Table

Code	Name	Desig'n	OD	Wall t	Area	Iz-Inertia	Iy-Inertia	Ix-Inertia (J)
1	PIP		2.81	0.047	4.080E-01	3.894E-01	3.894E-01	7.789E-01
2	PIP		0.9	0.3	5.655E-01	3.181E-02	3.181E-02	6.362E-02
3	PIP		1.0	0.03	9.142E-02	1.076E-02	1.076E-02	2.152E-02
4	PIP		2.94	0.05	4.540E-01	4.741E-01	4.741E-01	9.482E-01
5	PIP		2.8	0.047	4.065E-01	3.852E-01	3.852E-01	7.704E-01
6	PIP		1.1	0.03	1.008E-01	1.444E-02	1.444E-02	2.889E-02
7	PIP		1.4	0.05	2.121E-01	4.838E-02	4.838E-02	9.675E-02
8	PIP		1.7	0.028	1.471E-01	5.141E-02	5.141E-02	1.028E-01

Stiffness Data

The properties used in the stiffness analysis to obtain deflections (and ultimately forces) are:

A Section area

Iz 2nd Moment of Area about the local element z axis

Iy 2nd Moment of Area about the local element y axis

J Torsional constant for the section

Ay Shear area (always optional)

Az Shear area (always optional)

It is not necessary to always define all of these properties, it depends on the type of analysis and the modeling requirements.

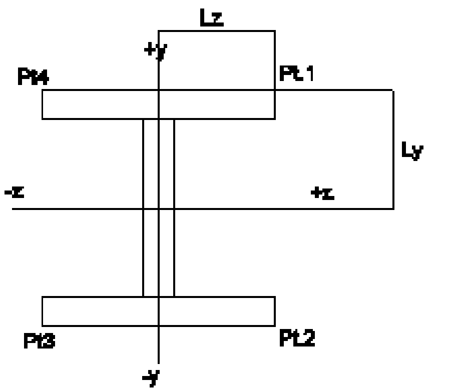
Zero entries are permitted providing mechanisms are not formed within the structure.

If mechanisms are not structurally significant use the Soft Spring analysis option to eliminate them numerically.

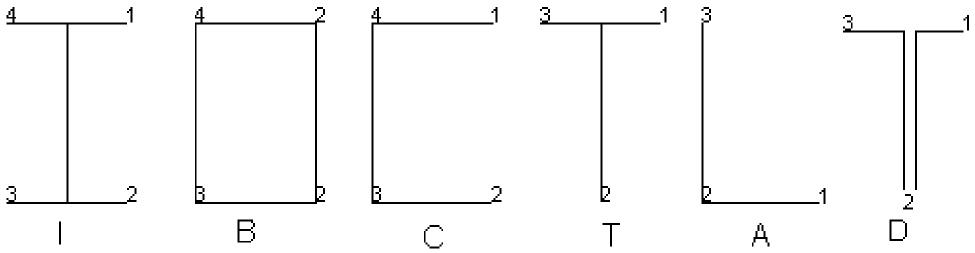
The Shear areas are used to evaluate both shear stiffness and shear stress.

Stress Data (Points)

To evaluate stresses in beams, stress points are require to be defined. Stresses can be evaluated at up to 4 stress points.



The conventions used for library sections for stress points and are:



Axial Stress

Sa = Fx/A

Pressurised Pipe

Hoop Stress Sh = Δp.Do/2t where Δp = Pi - Po For Type 6(7) & 6(11) DyNoFlex Plasticity Solutions Sh = Δp.(Do-t)/2t

Fx = True Wall Force

True Wall Force = Effective Axial Force + Pressure End Cap where the Pressure End Cap = Pi.AI - Po.Ao

Axial Force output in FS2000 shows the Effective Axial Force

Note: For a free ended pressurised pipe the effective axial force is zero and the true wall force is the end cap force.

Pressure (Poisson) strain is based on εp = (1 - 2.μ)(Pi.Ai - Po.Ao) / (EAs) This formulation is equally applicable to thick or thin pipe sections.

Hoop Stress and the associated VM stress displays use by default Sh = Δp.Do/2t. This can be changed to Sh = Δp.(Do-t)/2t (Line Plot Setting). This only affects GUI display, plots all other output relating to hoop stress always uses Sh = Δp.Do/2t.

Corrosion allowance and mill tolerance are only employed in the design checker FS-Pipe.

Shear Stresses

The average shear stresses in the Z and Y direction are calculated based on the appropriate shear areas (Az and Ay) defined by the user. These areas are also used in the analysis to include the effect of shear beam stiffness.

Ave Y Shear Stress Sy = Fy/Ay

Ave Z Shear Stress Sz = Fz/Az

Torsional Shear Stresses

Torsional shear stresses are evaluated by defining the torsional sectional modulus (Zt).

Torsional. Shear Stress Tx = Mx/Zt

Bending Stresses

Bending stresses may be evaluated at up to 4 locations across the section. This is done by specifying up to 4 sets of values for Ly and Lz to define the locations. The locations are termed **Stress Points**. The values Ly and Lz are the physical dimensions from the section N/A. to the point on the cross section where the stress is to be calculated.

The location of the stress points above are as defined in the property libraries

If Lz or Ly are equal to zero then bending stresses will not be printed.

Bending Stress Sbz = Mz.Ly/Iz

Bending Stress Sby = My.Lz/Iy

Bending Stresses in **Angles** - This simple approach to bending in angles (geometric axis) is only applicable to cases where the angle is restrained laterally along its length. Note that the design code checkers assume angles to be un-restrained (conservative).

Combined Stresses

Combined stresses are evaluated at the Lz and Ly locations by the following:

Combined Stress (Von-Mises) = [(Sbz+Sby+Sa)**2 + 3(Sy**2 + Sz**2 + Tx**2)]**.5

For Pipe Sections

Bending Stress Sb = (Sbz**2+Sby**2)**.5

Longitudinal Sl = Sb + Sa

Hoop Stress Sh = ∆p.Do/2t where ∆p = Pi - Po

Shear Stress T = Tx + (Sy**2 + Sz**2)**.5

Combined Stress(Von-Mises) Se = [Sl**2 - SlSh + Sh**2 + 3T**2]**.5

The Von-Mises stress evaluation is based on the assumption that maximum bending stress and maximum direct shear occur at the same point. This will produce slightly conservative VM stress values. The Beam Stress Inspection form can be used to show the the stresses for the actual shear stress distribution. This is accomplished by evaluating the stresses at 48 circumferential stress points and noting the maximum.

Property Evaluation Utility

For non standard sections the Geometric Property Utility and be used to evaluate the geometric properties. The structural properties are evaluated using standard formulas for the 2nd moment of area etc.

The following are used for the torsion & shear properties for which there are no definitive formula.

For rectangular sections the effective shear area = 0.75bd

For sections such a I beams and boxes the effective shear area = $D.t_w$

Where D is the depth of the section or height of the web for built-up sections

For open sections i.e. I beams

J = Sum of bt**3/3 for all section elements

Zt = J/t where t is the larger of web or flange thickness.

For box sections

J = 2.T.t.(B-t)**2.(D-T)**2 / (B.t + D.T - T**2 - t**2)

Zt = 2At where t is the smaller of web or flange thickness and

A =the mean enclosed area

For rectangular sections

J = a.b**3(16/3-3.36b/a(1-(b/a)**4/12))

Zt = 8a**2b**2/(3a+1.8b)

Where a = longer side/2 and b = shorter side/2

For pipes

J = 2*I

Zt = 2J/D

Eff Shear Area = A/2

Plastic Properties

Plastic Bending Modulus S = t(D**2 -2Dt + 4t**2/3)

Plastic Torsional Modulus TMOD = 2(D**3 - d**3)/3

-o-

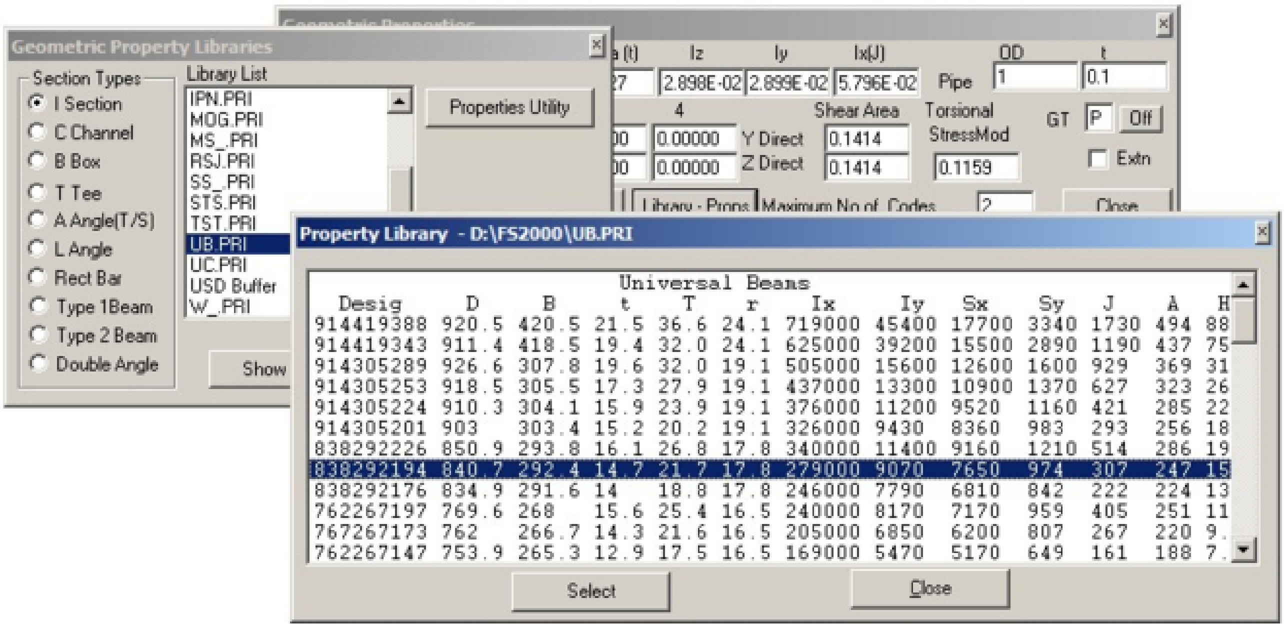
5.4.1.1 Geometric Property Libraries

Element section properties may be entered directly into the Geometric Property Tables using the Geometric Properties box or may be added by selecting sections from entries from pre-defined Property Libraries.

The Properties Library files are located in the FS2000 user folder. They are (distribution duplicates) also duplicated in the FS2000/Properties folder.

Model Dependent Library file may also be used. These are normally used when using the Geometric Property Generation Utility to create element properties.

Entries to Properties Libraries and Model Dependent Libraries and be made using the Geometric Pr



Units – Two types of section libraries are used, SI-Unit libraries and USA-Unit libraries. The library file formats and the units of the libraries are consistent with standard structural property tables available in structural handbooks.

To use the tables it is essential that the model be created in either the **SI-Unit** system or the **USA-Unit** system. A library unit conversion is used by the program to enable access to both types of section library, regardless of the model unit system.

There are nine different property file formats for each of the following section types. The last character of the file extension of the library file identifies the section types.

I	I beams and H columns
C	Channels
B	Box sections
T	Tee section
A	Angle sections (no bending stiffness i.e. pure strut/tie)
L	Angle section
R	Rectangular bar
D	Double Angle
1	Type 1 Beam section (unsym I beam)
2	Type 2 Beam section (unsym I beam)

The following **SI-Unit** property table files are used for standard British Rolled Sections and are supplied with the program.

Property File	Type	Description
UB.PRI	I	Universal beams
UC.PRI	I	Universal columns
RSJ.PRI	I	Joist sections
PFC.PRC	C	Channel sections (RSC also)
RHS.PRB	B	Rectangular hollow sections
SHS.PRB	B	Square hollow section
RST.PRT	T	Tee sections
TUB.PRT	T	Tee section cut from UBs
TUC.PRT	T	Tee sections cut from UCs
RSA.PRA	A	Angles (partially complete)
EA.PRL	L	Equal angles
UEA.PRL	L	Unequal angles
DEA.PRD	D	Double equal angle
DUA.PRD	D	Double unequal angle

The following property **SI-Unit** table files are used for standard European sections and are supplied with the program.

Property File	Type	Description
HEA.PRI	I	I Sections
UEB.PRI	I	I Sections
HEM.PRI	I	I Sections
IPE.PRI	I	I Sections
IPN.PRI	I	I Sections

The following **USA-Unit** property table files are used for standard USA sections and are supplied with the program. The underscore (_) identifies USA-Unit property tables.

WS_.PRI	I	W Shapes
HP_.PRI	I	HP Shapes
MS_.PRI	I	M Shapes
SS_.PRI	I	S Shapes
CS_.PRB	C	Standard Channels
MC_.PRB	C	Miscellaneous Channels
HS_.PRB	B	Rectangular tubing
WT_.PRT	T	W Tee sections
MT_.PRT	T	M Tee sections
ST_.PRT	T	S Tee sections
AA_.PRA	A	Angles (Tie/strut)
AS_.PRL	L	Angles
DE_.PRD	D	Double Angles
DU_.PRD	D	Double Angles

Creating and Editing Property Libraries

There is no limit to the number of property library files the user may create it is only necessary to ensure that when named, they are identified to the appropriate group by the file extension. Property files must exist in the FS2000 directory. Exceptions to this are model-related tables (see below).

It is recommended that new property libraries be always created using the Geometric Property Generation Utility. This may not be possible if standard structural sections are being used and properties are require to be exact i.e. fillet radii and tapered flanges effects included.

When creating new files manually it is better to copy an existing table of a similar type and then edit it to requirements i.e. delete all existing entries but one and then add the new entries below that. The format fields of the single entry may be used as a template for the new entries.

If section properties are to be added to the files ensure that the appropriate file type is used. e.g. in the case of bearing piles (I sections) the data could be added to either of the first three files above since the section type is similar in each case.

The files contain more data than is required for stiffness analysis since the file format is also used for the Design Code Checkers. Data not required for the stiffness analysis includes t, T, r, Sx, Sy, H. If these are not entered then the quantity must be replaced by zeros to ensure that the file is correctly read.

The filename of property files must be a 2 or 3 character name. The section designation of table entries is a numeric only designation with up to 9 characters. Within the program the file name and the designation identify all property code data originating from table files. e.g. UB 914419388 identifies a 914 x 419 388 kg Universal Beam.

If the designation is preceded by - ve sign, i.e. it is a negative number, the section will be treated as a welded built-up section in the Design Code Checkers.

Model Dependent Property Libraries

Often it will be found convenient to create custom property libraries that are related to specific models. The main advantage of this is that it enables the library to be archived with the model and eliminates the need to maintain large mixed model related libraries.

Unlike standard libraries the model related libraries must reside in the model directory and possess the model files name. The file extension is still used to identify the section type.

When model dependent table entries are used they can be identified by the following ID:

MDI	I sections
MDC	Channel sections
MDB	Box sections
MDT	Tee sections
MDA	Angle sections (no bending stiffness i.e. pure strut/tie)
MDL	Angle section
MDD	Double Angle section
MDR	Rectangular bar
MD1	Type 1 beam section
MD2	Type 2 beam section

Use the Geometric Property Generation Utility to create Model Dependent Libraries.

-o-

5.4.1.2 File Formats

Property libraries are plane text files (ASCII). Each section entry must be contained in one line and spaces are used to separate data fields.

The unit type of the library is defined at the beginning of the first line. INCH is used to signify an USA-UNIT type library as shown below. If this is not present the library is a SI-Unit library.

```
INCH   ASTM  W Shapes
Desig D   B   t   T    r     Ix     Iy     Sx    Sy    J    A     H
44335  44  15.9  1.03  1.77  1.18  31100  1200  1620  236  74.7  98.5  535000
```

The file formats for each of the library types is shown below (one entry only). Only British SI-Unit libraries are shows, US-Unit library formats are identical but the units are in inches.

Angle Principle Axis

The Ix and Iy for Type L angles section can be either geometric or principle axis I values. All standard libraries use the geometric axis which is the normal convention for property tables. If the principle axis use it means that the major axis aligns with the major principle axis. The program will set the stress points value accordingly. Note that this does affect the design codes checks which always use principle axis regardless of the I value used in the library.

UB.PRI (Type I)

Universal Beams

Desig D B t T r Ix Iy Sx Sy J A H

914419388 920.5 420.5 21.5 36.6 24.1 719000 45400 17700 3340 1730 494 88.7

RSC.PRC (Type C)

Rolled Steel Channels

Desig D B t T r Ix Iy Sx Sy J A H Cy

432102 431.8 101.6 12.2 16.8 15.2 21400 629 1210 153 61.0 83.5 .217 2.32

RHS.PRB (Type B)

Rectangular Hollow Sections

Desig D B T Ix Iy Sx Sy J A

502525 50 25 2.5 10.6 3.44 5.41 3.26 8.41 3.47

SHS.PRB (Type B)

Square Hollow Sections

Desig D B T Ix Iy Sx Sy J A

202 20 20 2 .759 .759 .951 .951 1.22 1.42

RST.PRT (Type T)

Rolled Structural Tees

Desig D B t T r Ix Iy Sx Sy J A H Cy

419457194 460.2 420.5 21.5 36.6 24.1 44100 22700 2190 1670 856 247 0 10.3

RSA.PRA (Type A)

Angles (No bending stiffness)

DESIG D B T RXX RYY RUU RVV A

50506 50 50 6 1.5 1.5 1.89 .968 5.69

UEA.PRL (Type L)

Angles

Desig D B t T A Cx Cy Ix Iy Ruu Rvv Tan(Ang)

20015018 200 150 18 18 60.1 3.86 6.34 2390 1155 6.97 3.22 0.549

DEA.PRD (Type D)

Double Angles
Desig D B t T s Ix Iy Sx Sy J A H Cy

10010012 100.0 100.0 12.0 12.0 10.0 414 939.9 0 0. 21.66 45.4 0.0 2.90

BAR.PRR (Type R)

RSect Type Library Entry (mm ; cm3 ; cm4)

Desig D B Ix Iy Sx Sy J A

100025 1000.0 25.0 2.083E05 1.302E02 6.250E03 1.563E02 5.126E02 2.500E02

BMS.PR1 (Type 1 Beam) or BMS.PR2 (Type 2 Beam)

1 Section Library Entry (mm ; cm3 ; cm4)

Desig D B Bb t T Tb Ix Iy Sx Sy J A H Cx Cy

20032 200 200 100 6 10 10.0 2780 750.3 291.9 126.6 11.30 40.80 0 10 7.67

-o-

5.4.1.3 Plane Area Properties

The **Element Data (Full list box)** command in the **Data** menu makes the Element Properties box visible.

If the Element Query is used to list Element and Couple data and this list box is visible it will be populated with more detailed element data. Loading on the beam element will also be listed if in the Load Definition TASK.

Plane Area Property Generation

If a triangular mesh exists and the **Mesh Properties** button is clicked the mesh properties will appear in the box.

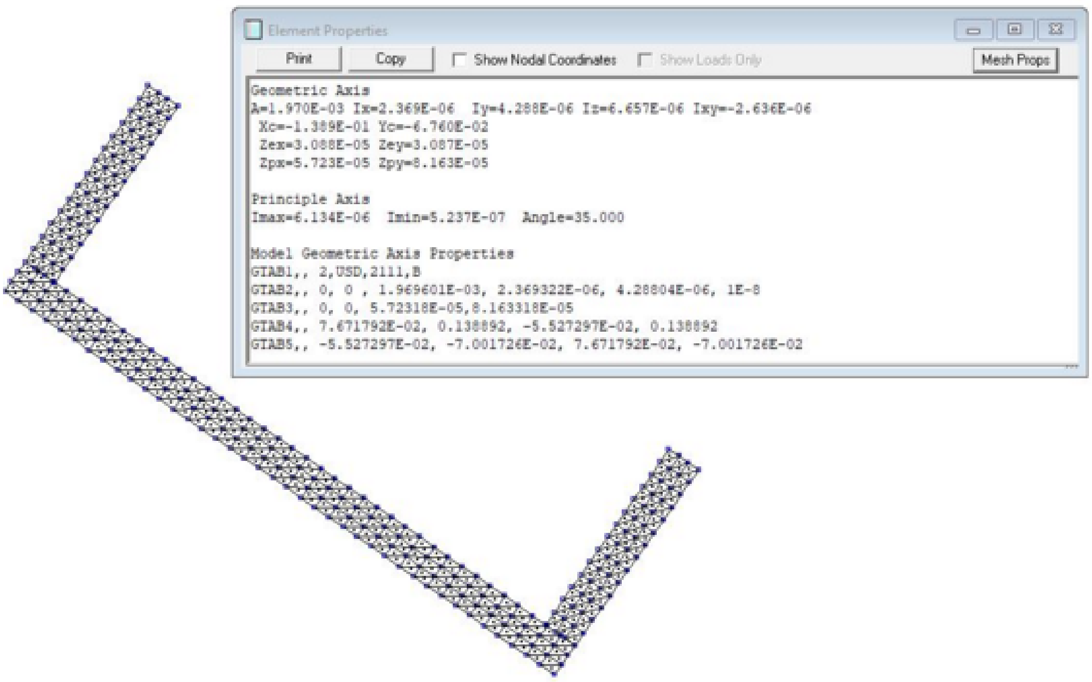
The structural properties of meshed plane areas can be evaluated from plane mesh shapes. The meshes or elements within a mesh do not have to be connected.

The plastic properties (Zpx & Zpy) are dependent upon the mesh density . To evaluate the center of area, a requirement for plastic properties, the area of each element is lumped at its centroid and will therefore will require a fine mesh for acceptable accuracy.

Elastic triangular properties are exact and the only requirement is that the mesh represent the boundary.

- The mesh must be crated in the X-Y global plane
- The elements must all be 3 node triangles.
- There no requirement for the areas for elements to be connected - it's not a FE mesh.
- The elements must be not overlayed otherwise the overlayed area will have a doubling effect.

If the graphic screen is printed the properties will be shown on the plot along with the mesh. The data in the form cannot be printed using the Print button, use the Copy button to transfer the data..



The GTAB commands enables the properties to be copied directly to the Geometric Property Table by copying and pasting the display to the Interpret Command input.

The Iz is the second polar moment of area. As in the case of open sections this is not the torsional constant Iz(J) used to evaluate torsional twist. 1E-8 is entered for Iz(J) in the GTAB2 command.

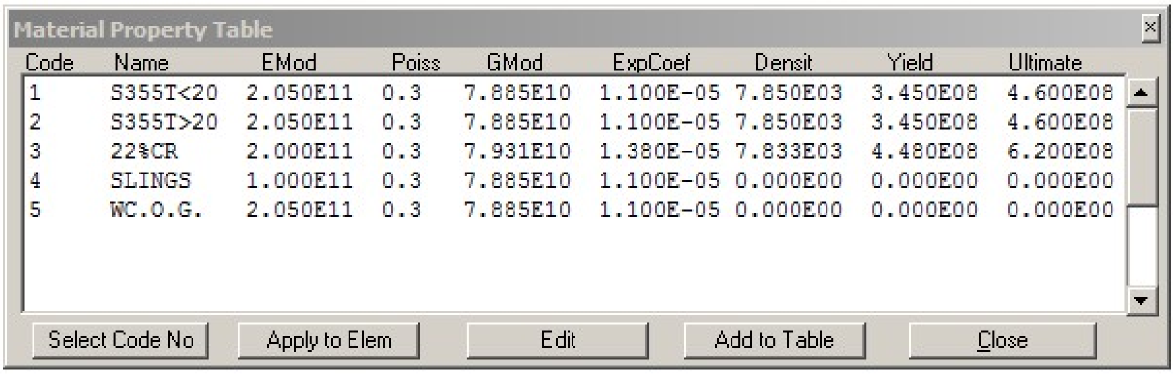
Note that the stress point data (GTAB4 and GTAB5 commands) assume the sections to be doubly symmetric and are based on the maximums distance from the neutral axis.

-o-

5.4.2 Material Property Tables

The material property (Mat in the Element Defn box) code attribute of an element is used to identify the appropriate material properties of an element by referencing it to an entry in the Material Property Table. Data is entered into the table using the Material Properties Input box.

A property Table list can also be view from the Data menu. This list only shows the main properties. It is very useful for assigning properties to elements.



Material Property Table

Code	Name	EMod	Poiss	GMod	ExpCoef	Densit	Yield	Ultimate
1	S355T<20	2.050E11	0.3	7.885E10	1.100E-05	7.850E03	3.450E08	4.600E08
2	S355T>20	2.050E11	0.3	7.885E10	1.100E-05	7.850E03	3.450E08	4.600E08
3	22%CR	2.000E11	0.3	7.931E10	1.380E-05	7.833E03	4.480E08	6.200E08
4	SLINGS	1.000E11	0.3	7.885E10	1.100E-05	0.000E00	0.000E00	0.000E00
5	WC.O.G.	2.050E11	0.3	7.885E10	1.100E-05	0.000E00	0.000E00	0.000E00

The Standard Material Property Table is a list of the following properties each with a numeric reference code.

E	Modulus of Elasticity
G	Modulus of Rigidity
Poiss	Poissons Ratio
Den	Material Density
Coef. Exp	Coefficient of Linear Expansion
Yield Stress	Yield value of Material
Ult Stress	Ultimate tensile strength

The Extended Material Property Table is list of properties used in pipework analysis or other analysis which calls for the use of extended properties (thermal dependant). The properties available in the extended table are:

Ultimate Tensile Strength

Cold Allowable Stress

The following properties may be defined as a function of temperature dependent. Up to 15 points may be used to define the property curve.

Temp Coef Exp(T) E(T) Allowable Stress(T)

The following summarises how the extended properties are used by the program:

ExpCoef(T) - used to evaluate thermal strain based on the load case element temperature - This should be defined as mean value .e.g. B31.3 Table C-3

E(T) - not used. Model stiffness is based on cold E modulus i.e. **E**

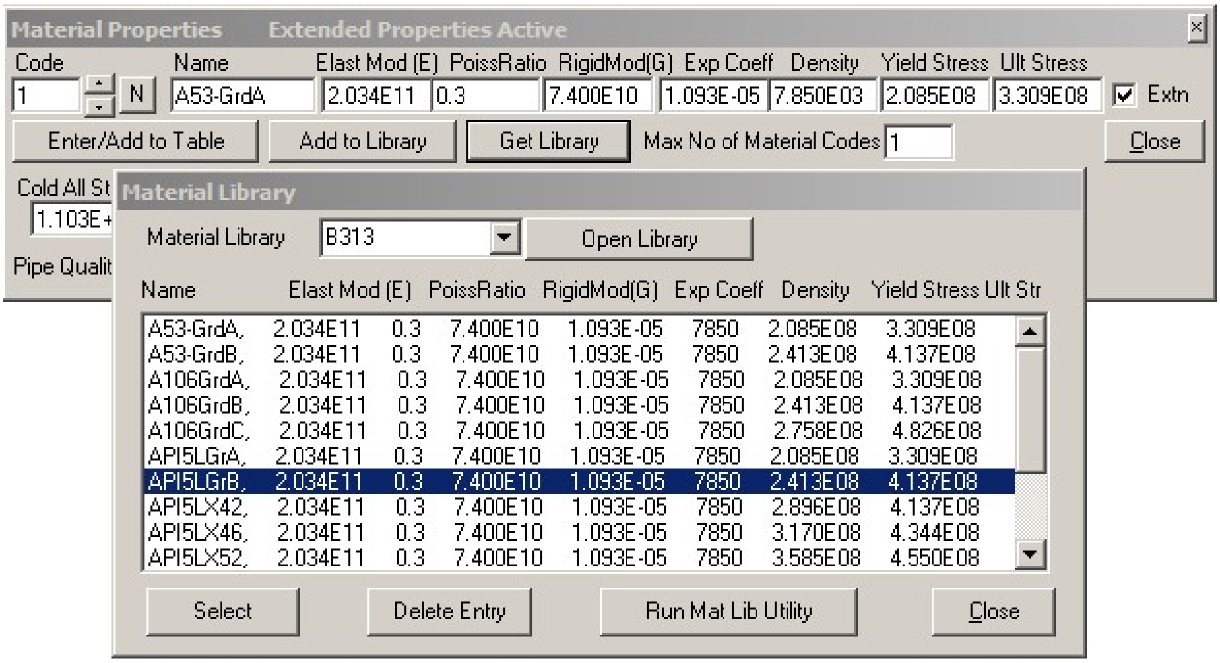
AllowStress(T) - is only used by FS-Pipe for piping design checks

-o-

5.4.2.1 Material Property Library

Standard Structural Material Libraries

Material property data can be saved and retrieved from the material property library files.



These files are text files with the file extension .PRM. These files must reside in the FS2000 directory.

The default material library provides with FS2000 is named MATERIAL.PRM

The default material library may be changed by simply entering a new name in the name box and then opening it. A pull down list may be used to select a library from existing libraries.

It is recommended that data to be added to the libraries should be added using the Material Property Table's **Add to Lib** button. The Add to Lib button copies the properties currently shown in the boxes to the library.

This library list box has the following control buttons

Select	This will load the selected entry into the Material Property Table
Delete Entry	This will remove an entry from the current library
Run Mat Lib Utility	This will start the Extended Material Library Utility - See below

Extended Material Libraries

Additional properties can be retrieved from the extended material property libraries. These additional properties are thermal and pipework related properties

These libraries are text files with the file extension .PRE. These files must reside also in the FS2000 directory.

When properties are retrieved from the standard structural table they will also, if they exist be retrieved by name association from the extended library. This means that properties in the extended library must also exist in a standard library of the same name for them to be retrieved. List order between standard and extended libraries is not important since they are linked by name association only.

The **Add to Lib** button in Material Properties Form will add the extended data properties corresponding to the property table code number **DIRECTLY** to the library. Hence the data must have first been entered into the model material table before it can be added to the library.

When extended data is added to the library it is not shown in the library list. This list shows only standard structural properties.

To delete entries from the Extended library use a text editor and delete between the dotted lines.

Material Library Utility

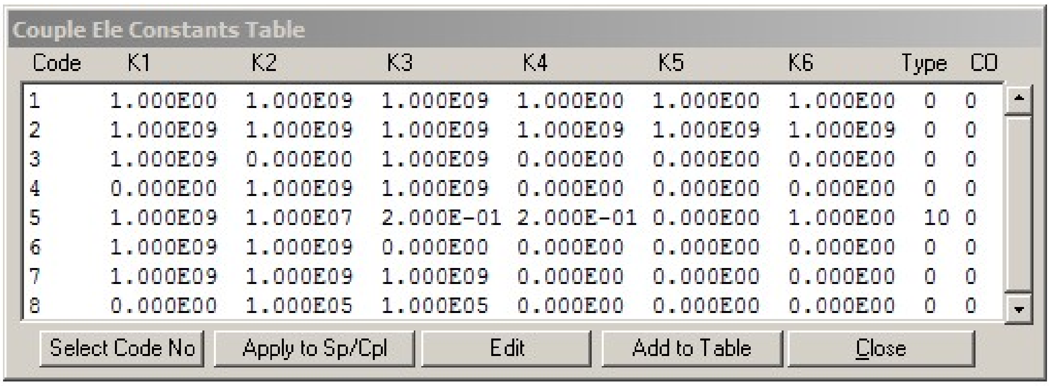
This Material Library Utility provides a more convenient method for modifying and viewing the data in Extended and Standard Material Property Libraries. This utility can also be started from the Windows Start menu.

-o-

5.4.3 Couple Property Tables

The Constants code attribute (Stiff in the Couple Element Defn box) of a Couple is used to identify the appropriate stiffness properties of a couple element by referencing it to an entry in the Couple Elements Constants Table. The data is defined using the Couple Elements Constants input box.

A property Table list can also be view from the Data menu. This list only shows the main properties. It is very useful for assigning properties to couples.


Couple Ele Constants Table

Code	K1	K2	K3	K4	K5	K6	Type	CO
1	1.000E00	1.000E09	1.000E09	1.000E00	1.000E00	1.000E00	0	0
2	1.000E09	1.000E09	1.000E09	1.000E09	1.000E09	1.000E09	0	0
3	1.000E09	0.000E00	1.000E09	0.000E00	0.000E00	0.000E00	0	0
4	0.000E00	1.000E09	1.000E09	0.000E00	0.000E00	0.000E00	0	0
5	1.000E09	1.000E07	2.000E-01	2.000E-01	0.000E00	1.000E00	10	0
6	1.000E09	1.000E09	0.000E00	0.000E00	0.000E00	0.000E00	0	0
7	1.000E09	1.000E09	1.000E09	0.000E00	0.000E00	0.000E00	0	0
8	0.000E00	1.000E05	1.000E05	0.000E00	0.000E00	0.000E00	0	0

The constants in the table are used differently depending upon the function of the Couple.

The couple may be used to define a node to node spring or a node to node gap.

Constants Used for Node to Node and Node to Ground Springs

In this case the constants define the spring stiffness. Six constants K1 to K6 may be entered representing the six freedom directions. The directions are in the local element axis.

K1 x translation

K2 y translation

K3 z translation

K4 x rotation

K5 y rotation

K6 z rotation

Zero values may be entered. A typical stiffness of 1E9 would be used to represent a rigid connection.

Constants Used for Node to Node and Node to Ground Gap Elements

Only the **3D NL/Pile Analysis** and **FS-DyNoFlex** modules are capable handling friction and specified gap clearance/interference. The 3-D Standard will only interpret gap elements as simple make or break contact elements.

In this case the constants are used to identify and define the parameters of a gap element. The following convention is used.

Type 10 Compression Gap

Type 11 Tension Gap

Type 12 Compression Gap (conservative friction)

K1 Gap stiffness - Normal direction

K2 Gap stiffness - Tangential direction

K3 Sliding Friction Coefficient - Coulomb friction

K4 Sliding Friction Coefficient Local Z

K5	Gap Size		
K6	Initial Gap Status	0	Open
		1	Closed

If Gap Size is specified as a negative value the element will produce forces in an unloaded structure equivalent to an interference fit of the gap magnitude . If the gap is defined with a positive value, the gap will require close that amount before it starts to contact.

The normal gap stiffness is the stiffness of the gap when it is closed. This can be adjusted to suit the model requirements. Typically a value of 1E9 would be a stiff gap for most structures created in S.I units. A good 'feel' for relative stiffness can be obtained by equating it to the axial stiffness of a beam i.e. AE/L.

The tangential gap stiffness is the sticking stiffness that is active for a sticking force less the limiting friction force. If specified as zero the default value is the normal gap stiffness

Orientation of gap elements

The sign convention for Gap elements is that positive local displacement will open a compression gap and negative displacement will close it. Positive local displacement in this context is the relative displacement of the I node relative to the J node in the direction from the I to the J node. Tensile gaps (hooks) actin a converse manner.

Node to Node Gaps: When these are referenced to an element or a coordinate system for their coordinate system then the numbering of the gap is not arbitrary, the numbering must relate the direction of the reference system.

Node to Ground Gaps: When these are referenced to an element or a coordinate system for their coordinate system then the positive direction should point away from the node such that in the cases of a compression gap the gap will opens for positive relative displacement.

It is more convenient to enter the gap data using the Gap Element Definition button in the Couple Element Constants input box.

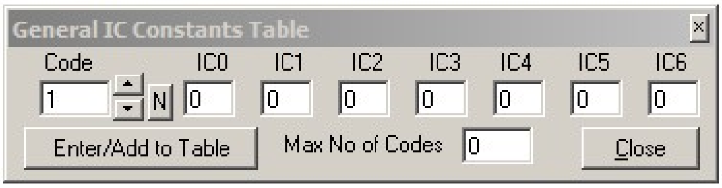
-o-

5.4.4 Constants Property

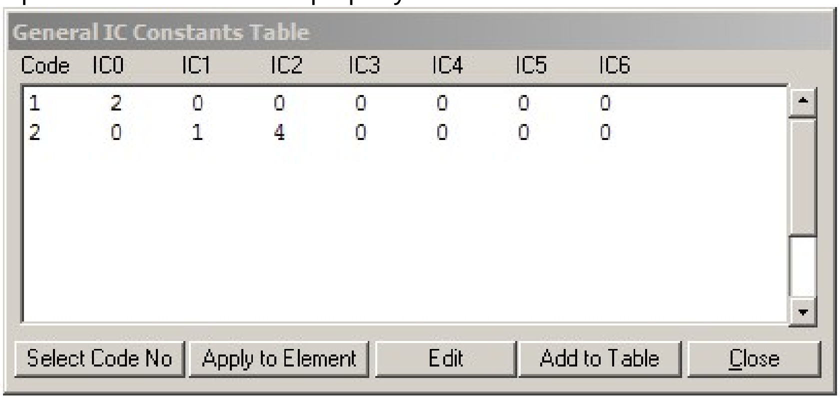
Constants are used to extend the element property definition for more specialised element often use in non-linear analysis.
The use of the constants can vary considerable and their use is element specific. Their use is described in the element descriptions in Section 5.
The beam element CO property (by element) or the couple CO property (by property code) are used to link to the constants property to the element.
There are two types:

- Integer Constants
- Real Constants

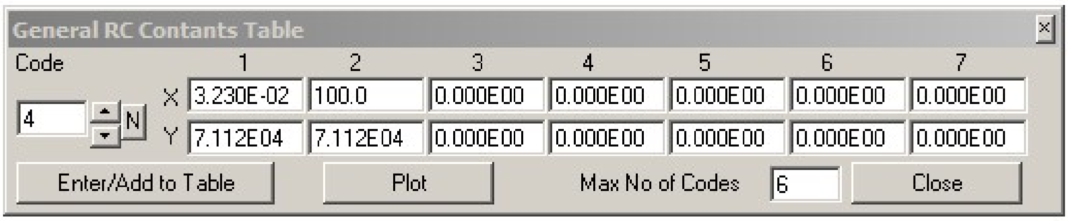
Integer Constants



The CO property in most cases would refer to an entry in the Integer constants table. Up to 7 entries can be defined for each integer constant entry.
Because the CO constant is often directly linked the the element's CO property the IC table entry can be copied to the elements CO property.



Real Constants



The RC property entries are used to define various property curves and other element properties. Up to 7 double entries can be defined for each real constant entry.

When RC table is used to define X-Y curves the curve is extended by the program. An extra point is added at the end (at X=Xn*1E6) which maintains the final defined slope between Xn-1 and Xn,

Defining Stress Strain Curves

Y represents Stress and X represents strain. The curve always starts at the origin in both axis.The RC-X1 and RC-Y1 are used to define the first point,which cannot be at the origin.The first point must produce a positive slope segment.

When curves are defined an extra point is added at the end the RC data (at Xe=Xn*1E6). This point maintains the final defined slope at Xn.

Note that the E value in the material properties should match the E value defined in the stress strain curve.

If a **Bi-linear** curve is being used for pipe elements it is advantageous to define the curve using only 2 points. The prevents the program from using more complicated iterative routines associated with piecewise curve definition in which strain hardening is a function of strain. Note that solids and shell elements use a plastic tangent modulus E_T (Geom Property Sy) to define a bi-linear curve.

Piecewise Curves - The first point must produce a positive slope segment. Other segments of the curve may be positive or negative. Segments of the curve cannot be vertical (infinite stiffness).

Ramberg-Osgood If the Stress Strain curve is represented by a Ramberg-Osgood relationship the following form of the relationship is used for parameter definition.
.

$$\varepsilon = \sigma/E + \alpha_r.\sigma_y/E.(\sigma/\sigma_y)^N$$

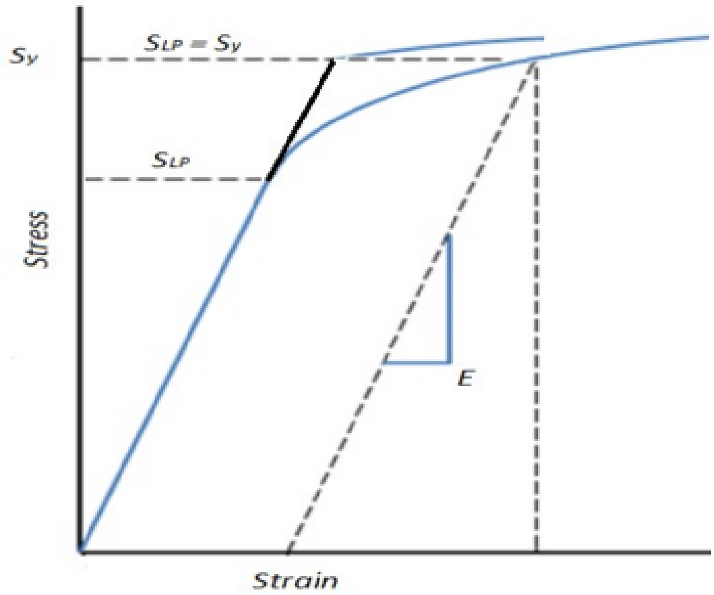
Where $\alpha_r = E/\sigma_y*\varepsilon_0 - 1$ N = 1+(Log($E/\sigma_{ULT}*\varepsilon_{ULT}$) - Log($\alpha_r$)/Log($\sigma_{ULT}/\sigma_y$) ε_0 = Material yield offset strain which is often 0.2% to 0.5% strain.

When Ramberg-Osgood type relationship is used for plastic solutions the curve is offset to provide a linear elastic relationship below a specific stress level. This is done to provide a distinct point for the solution's yield function evaluation i.e. defines the strain limit of proportionality above which yielding occurs

In the program the above Ramberg-Osgood equation is required to be converted to the following form in which the yield strength is a function of effective plastic strain ε_P .

$\sigma_Y = K(\varepsilon_P+\varepsilon_O)^M$ Where $K = 1/C^M$, $C = \alpha_r.\sigma_y/E/\sigma_y^N$ and M = 1 / M

The definition of a stress limit of proportionality, σ_{LP} enables the offset to be applied at any point on the curve. Strain offset $\varepsilon_O = \alpha_r.\sigma_y/E.(\sigma_{LP}/\sigma_y)^N$. A realistic plot of the actual curve and the strain levels of interest are required for to establish what value of σ_{LP} is appropriate and if indeed a value different from σ_y used for curve definition is a requirement.
Generally this is not a requirement and σ_{LP} is set to σ_y, which implies a strain offset of $\varepsilon_O = \alpha_r.\sigma_y/E$.



To identify that a Ramberg-Osgood is being defined the Y1 entry must be -1. The following data entries are used to define the Ramberg-Osgood parameters.

σ_y = X1 $\quad$ E = X2 $\quad$ α_r = X3 $\quad$ N = X4 $\quad$ σ_{LP} = X5
-1 = Y1

The following shows typical entries for a X65 pipe steel. This example has $\sigma_{LP} = \sigma_y$. i.e. a strain offset of $\alpha_r.\sigma_y/E$ is applied.

Code		1	2	3	4	5	6	7
1	X	4.480E08	2.050E11	1.31	25.61	4.480E08	0.000E00	0.000E00
	Y	-1.000E00	0.000E00	0.000E00	0.000E00	0.000E00	0.000E00	0.000E00

-o-

5.4 Loads

Creating Load Cases

Loading is applied to the model using referenced Load Cases. Load cases are groups of loads saved to a numbered load case file, **<Modelname>.L'n'** files, where n is the load case number.

The load case files are interpreted by the program. They may be created Interactively within FS2000 or be created by the user in a text editor using the Command Line Load definition commands. The latter method is for more advanced use of FS2000.

When creating load cases using command line instructions the load case still needs to be re-saved from within the Load Module of FS2000. This is necessary so that the load case is registered as existing. If a load case is not registered it will not appear in the load case selection lists. Unregistered load case files not appearing in the selection list can be retrieved by simply entering the load case number (if one exists).

Loading Types

The following types of loading may be applied

Concentrated Forces and Moments at Nodes - Global

Concentrated Mid-Span Forces and Moment on Elements - Global and Local

Uniformly Distributed Loads on Elements - Global and Local

Non-Uniformly Distributed Patch Loads on Elements - Global and Local

Thermal Loads - Loading due to differential expansion (Axial and Bending)

Pressure - Pressure loading on pipe elements

Acceleration Loads - Acceleration loads in the global directions e.g. gravity

Prescribed Displacements - Will yield forces or moments to produce the displacements of a Node

Element loading can be applied in either the global co-ordinate system or the local element co-ordinate system as shown in the element loading figures.

-o-

5.5 Mass Definition

The Load Definition TASK is used to create mass load cases that are used in the dynamic analysis, transportation analysis etc.

A mass case is defined and saved in an identical way to that of a static load case. A user defined dynamic gravitational conversion constant enables mass to be defined in terms of mass or force. This constant is entered in the analysis solution form and is called **Gravitational Constant/Mass Conversion**. The value of this would normally be 9.81 for SI units and 386 for US units

Only loading that is defined in the Y direction be converted to mass using the gravitational conversion constant. In most cases, this feature enables, conventional in-place load cases to be used directly for mass definition.

The following types of masses may be applied to a model

Self Generated Mass

This is the inherent mass due the area and density of an element. If a gravitational constant in the Y direction is defined i.e. a non zero value is defined, this mass effect will be included in the analysis. Note that the **Gravitational Constant/Mass Conversion** constant should be equal to the value of the load case gravitational constant otherwise the mass due to self weight will be factored.

Distributed Element Mass

Distributed mass can be applied to elements by applying them as Y direction (global only) distributed loads. The mass is evaluated by dividing the absolute value of the load by the dynamic gravitational conversion constant. In the case of patch loads the total load from the patch will be distributed evenly over the full length of the element. This is a requirement since only evenly distributed element mass can be used in dynamic analysis. Additional nodes could be included if a more accurate local mass distribution is a requirement. On large model this would be rarely a requirement. The mass is evaluated by dividing the load by the dynamic gravitational conversion constant

Element Concentrated Mass

Element point loads applied in the Y (global or local) direction will be converted to evenly distributed mass. The mass is then applied as though it were uniformly distributed over the length of the element. The mass is evaluated by dividing the absolute value of the load by the dynamic gravitational conversion constant.

Concentrated Nodal Mass & Inertia

Concentrated nodal mass can be applied to elements by applying them as Y direction concentrated loads. The mass is evaluated by dividing the absolute value of the load by the dynamic gravitational conversion constant.

Nodal masses can also used for defining mass directly (Note: the dynamic gravitational conversion constant is not applied Nodal masses). In SI units this mass would in kg and in US units Lbs/386 (W/g).

Nodal inertia (rotational mass) in the global directions can be applied as moments in the corresponding direction. The inertia is evaluated by dividing the absolute value of the moment by the dynamic gravitational conversion constant (T/g).

Distributed Mass defined by Geometric Property Code

Distributed mass can be applied to elements by applying them as Y direction distributed loads. The mass is evaluated by the program by dividing the absolute value of the load by the dynamic gravitational conversion constant

FS-Wave Generated Mass

FS-Wave has an option whereby hydrodynamic related data is use to create mass definition that would be used in a Time History analysis (DyNoFlex) or in a Frequency Solution. If active FS-Wave will create a binary file call .WMass. This file contains the mass effects due to:

- Contents

- Marine Growth
- Added Mass

If the file exists the element mass in the file will be combined with any other mass case used in the solution. The WMass file is only created by FS-Wave and will only be deleted if:

- The model is saved
- The FS-Wave WAV file is saved without the **Create Data for Fequency/Time History Analysis** being active

This file can be deleted any time by the user if the extra mass is not required.

-o-

5.5.1 Mass Modelling - Beam Elements

Combining the stiffness and mass properties of each elements of the model forms the stiffness and mass matrices used to evaluate the frequency modes of a model. The mass distribution in the model may to be defined by attributing masses to nodes or elements of the model. Creating load cases that define the mass does this.

The mass matrix can be one of two types

- Lumped Mass Model

In the lumped mass approach, the mass and mass moments of inertia of each element are lumped at the nodes of the elements. This results in a diagonal mass matrix, which is independent of frequency. Although this approach considerably simplifies the mathematics, it can represent a poor approximation to the actual mass distribution. To obtain an acceptable mass distribution and relatively accurate solutions, the user may be forced to include additional "artificial" nodes in the model. On large models where the total mass is distributed over a large number of elements or on models where the masses are concentrated, the lumped mass will give virtually identical results to the consistent mass model

- Consistent Mass Model

The consistent mass approach represents a compromise between the actual distributed mass and lumped mass approaches. This method is termed consistent since the mass matrix is derived using the same displacement functions used in the stiffness matrix derivation. The resultant mass matrix is independent of frequency but is a full matrix rather than a diagonal matrix as achieved with lumped mass option. Although the consistent mass matrix is an approximate mass representation, it is more accurate than the lumped mass matrix and retains the mathematical advantage of being frequency independent compared to that of the more complex distributed mass.

Beam End Release

The consistent mass approach assumes that the beam ends are rigid. If the beam mass is small the approximation will be insignificant. If however the beam mass is large then the beam releases should be modelled using couples to model the release.

-o-

5.6 Time Curves

Time curves are used to define the application of loading as a function of time or load step. They are used by the following FS2000 modules:

- Dynamic Non-linear Analysis - DynNoFlex (Maximum Number of curve is 100)
- Model Response Analysis (Transient Time History Response)
- Heat Transfer Analysis

The following describes their use in non-linear time history analysis. Their use is defined in the dynamic non-linear analysis (FS-DyNoFlex) form.

A time curve is simple a method by which loads cases can be factored by load factors that are a function of time i.e. they describe the dynamic application of load. Time curves are associated with loads cases using Load Case Combinations, therefore all loads defined in a load case will be factored by the curve value.

When a Time Curve is defined it is assigned a Curve ID Number. This ID number is subsequently associated with a Load Case by the combination Load Factor,i.e.a load factor that varies in time.

Time Curve commands are included within a standard load case called a History Curve Case. The command line definition for time curves is described in Section 9.10. This standard load case can be assigned any ID number and require no commands other than the time curve commands. The normal procedure would be to save a load case in the Load Definition TASK with the title "Time Curves" e.g. L50 (this registers the loads case in the model) and then use FS-Edit to add the time curve commands to the load case file L50. Note that FS-Edit can be accessed directly from the DyNoFlex input form.

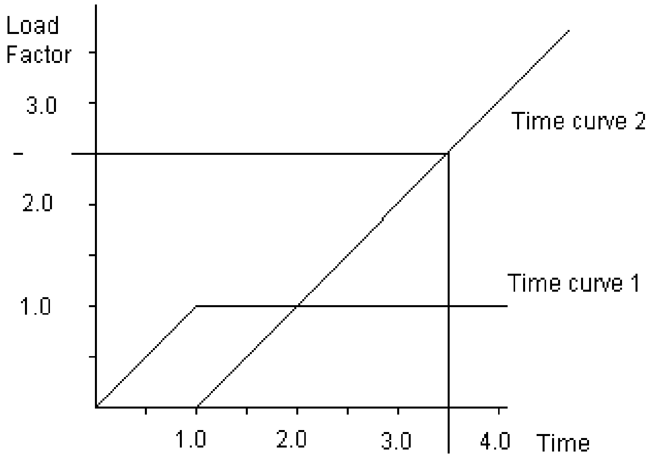
A good way to QA the curve definition is to create a simple model e.g. a cantilever (or any model) and use prescribed nodal displacements in a linear static solution to check that the desired output curve is obtained.

Example

The following illustrates the use of a Time Curves

The Load Cases combination is defines as:

Load Case Number	Load Factor (Effectively Time Curve No)
3	1
6	2



At t=3.5 the loading applied to the model will be: Load Case 3 x 1.0 + Load Case 6 x 2.5.

Linear interpolation is used to establish load factors at intermediate points i.e. between curve points'

The TCURVE command would be used to generate the above, the command line for this is:

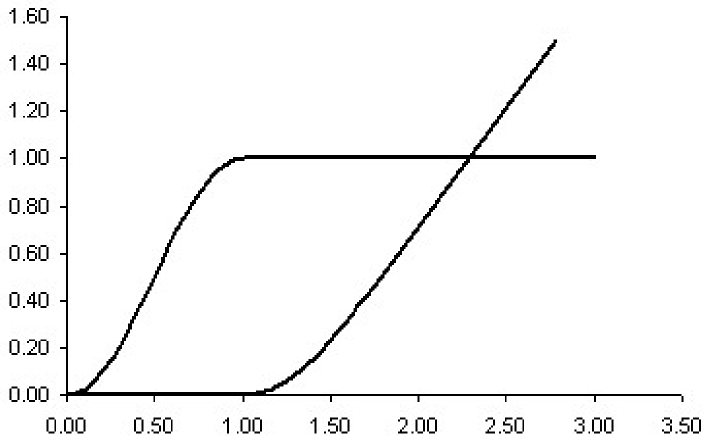
TCURVE, *No, NoPoints, Type*

Therefore the Time Curves definition commands for the above time curves shown above would be:

tcurve,1,3,1
0,0
1,1
10,1
tcurve,2,4,1
0,0
1,0
10,9

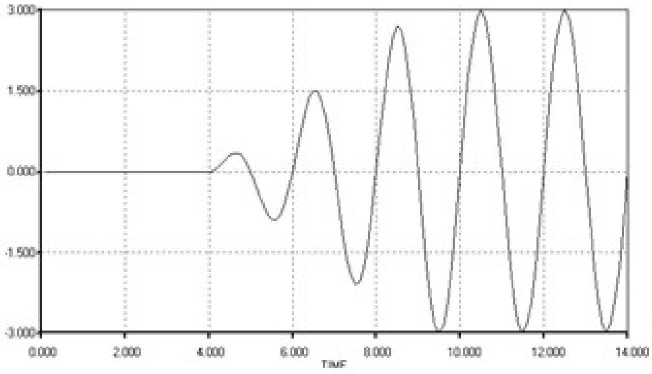
The following shows how the **SCURVE** or **SCURVE2** and **RCURVE** or **RCURVE2** command can be used to generate the above effect. These commands use a sinusoidal transition and have the advantage of reducing transient effects by applying the changes more gradually. **SCURVE2** and **RCURVE2** commands do not require the number of points to be defined (20 in the example below) and should be used in preference to the older commands.

scurve,1,1,1,0,20
rcurve,2,1,1,20



The following shows how the **RSCURVE** or **HCURVE2** can be used to generate a ramped sinusoidal curve which starts at t=4 and ramps up for 5 seconds

RSCURVE,1,3,3.142,0,5,4


3.000
1.500
0.000
-1.500
-3.000
0.000
2.000
4.000
6.000
8.000
10.000
12.000
14.000
TIME

-o-

6 Analysis Process

6.1 Analysis Procedure

Basic Analysis Procedure

FS2000 uses a number of program modules during the course of an analysis. A considerable number of menu commands are used to activate and control the use of these modules.

To provide logical access to these commands the analysis process has been divided into separate processes i.e. distinct stages of analysis. These stages are termed TASKS.

Tasks are initiated and terminated by using the TASK menu. The main title bar indicates the current task. For each task, a different set of menu commands become available.

A procedure for the basic analysis of a structure is given below. The procedure follows the basic menu list of the TASK menu

- Open a new model by defining its name and directory in the Primary Task
- Define the basic geometry of the model using in the Model Definition Task
- Define the loading (Load Cases) in the Load Definition Task
- Perform the matrix analysis in the Analysis Task to produce raw load case results
- Post-process the raw load case results to form Results Cases (can also be done during analysed)
- Inspect the results in the Results Task - Interactive plots and force listings
- Create Formatted Listing (Definition & Results) in the Reports Task - View or Print

Model Definition

In this TASK, the basic model geometry is defined. The GUI has some very efficient routines for creating model geometry and properties and is the approach that would always be used.

Note that model geometry may also be defined by either/or a combination of the following methods:

- Automatic generation using the specific model type generators.
- Command Line Definition Instructions.
- Importing a CAD DXF file (stick drawing of the structure for beam type elements).
- Spread Sheet Data - Node and element lists can be conveniently converted to command line instructions. Often used when converting other program structural geometry to FS2000 format.

Load Definition

In this TASK, the loading on the model is defined. Loading is categorized and saved as separate load cases. Load cases are identified by a reference number, an ID number and are also assigned a descriptive title.

Model load data may be defined or modified by using the text editor. If new load cases are created in a text editor they must then be opened and saved in the Load Definition TASK (registers the load case).

Basic Operation - Activation of the Analysis Process

FS2000 uses a number of program modules during the course of an analysis. The activation and control of these various modules may be achieved by Direct User Interaction using the menu commands or by the use Batch Mode command line instructions.

DIRECT USER INTERACTIVE OPERATION

In this mode of operation all program activities follow direct user action. For simple models with few load cases or load case combinations it is convenient to operate the program modules using this mode.

BATCH MODE COMMAND LINE OPERATION

For large models or models with numerous load cases or load case combinations it is often more convenient (almost essential) to operate the program modules using Batch Mode command line instructions. Command line instructions can be incorporated into batch list files to enable the user to set up multiple module processing that eliminates user intervention.

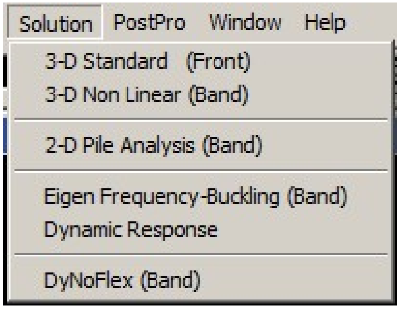
With batch mode operation the whole analysis process can be repeated with one single key stroke. An essential feature for the design of models with large numbers of load cases.

Batch processing if used in a multiple-processing environment can be run as a background process.

It is recommended that the user should become familiar with interactive operation prior to using batch operation.

-o-

6.2 Solution Options

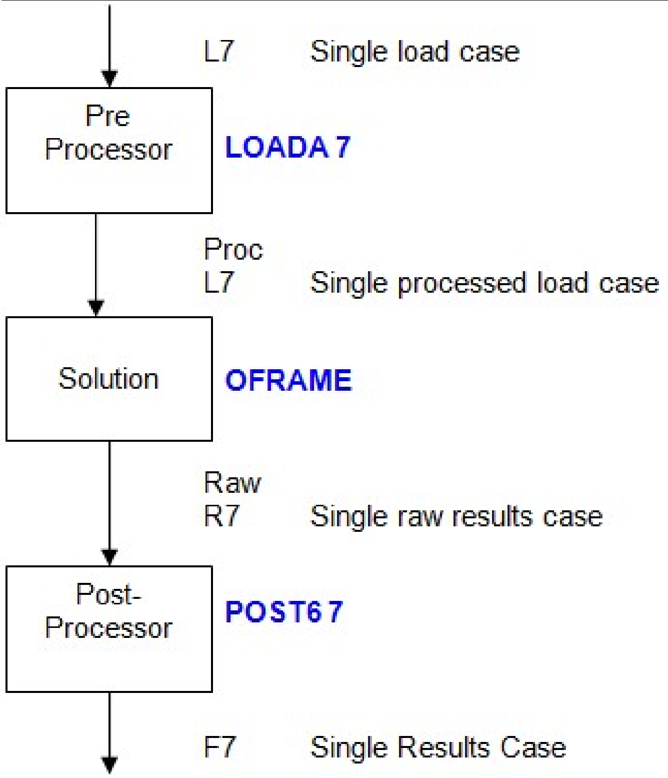


Current FS2000 analysis options are.

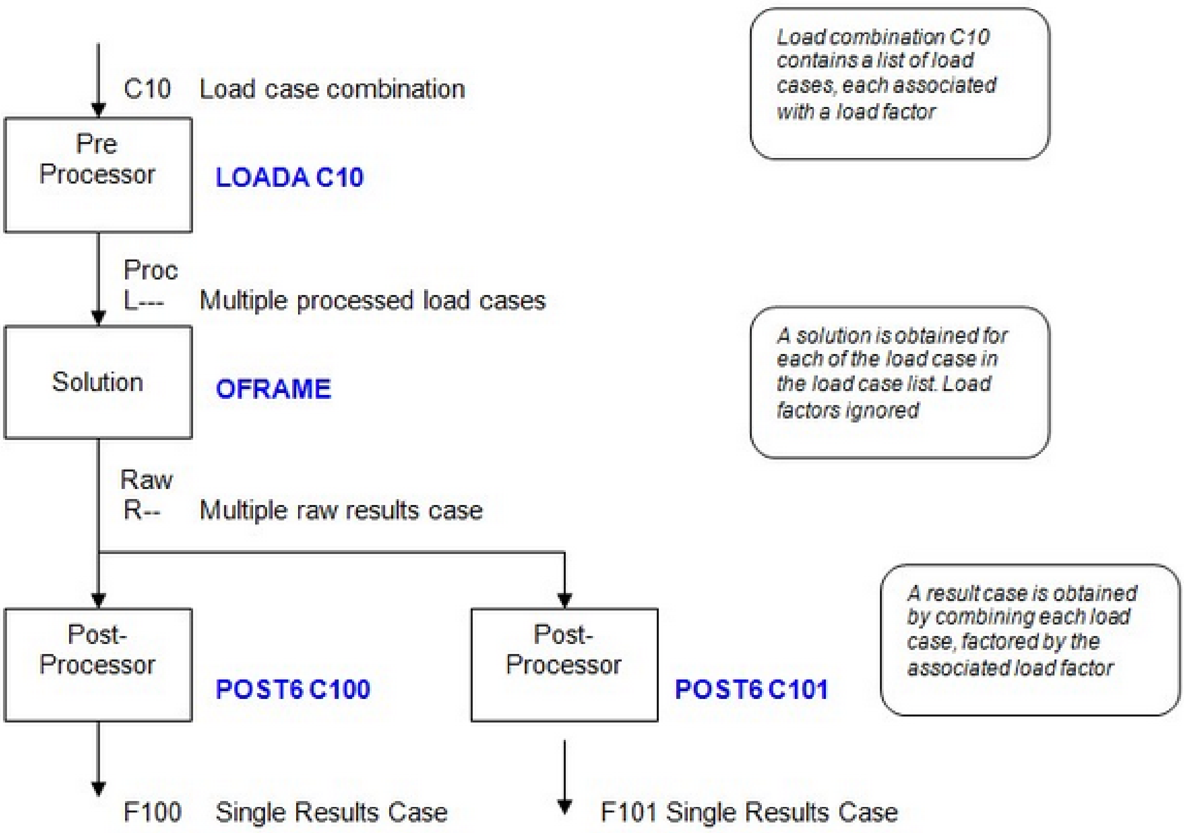
- 3-D Standard (Front) (Beams & Finite Elements) - P-Delta Effect, Contact elements, Tension/Compression Only
- 3-D Non Linear (Band) (Gaps/friction, Non-linear soil-pile interaction, frame plasticity)
- 2-D Pile Analysis (Specific 2-D Solver for model wil piles segments - Most pile solutions will employ the 3-D Non Linear solver)
- Frequency-Buckling (Band) (Modal analysis - (Vibration and buckling Eigen values)
- Dynamic Response (Modal response analysis)
- DyNoFlex (Band) - Incremental Linear/Nonlinear/Static/Dynamic Analysis.

The following describes how load cases and load case combination are used when the different analysis options are used. The blue text indicate the batch commands.

3D Standard Solution - SINGLE LOAD CASE

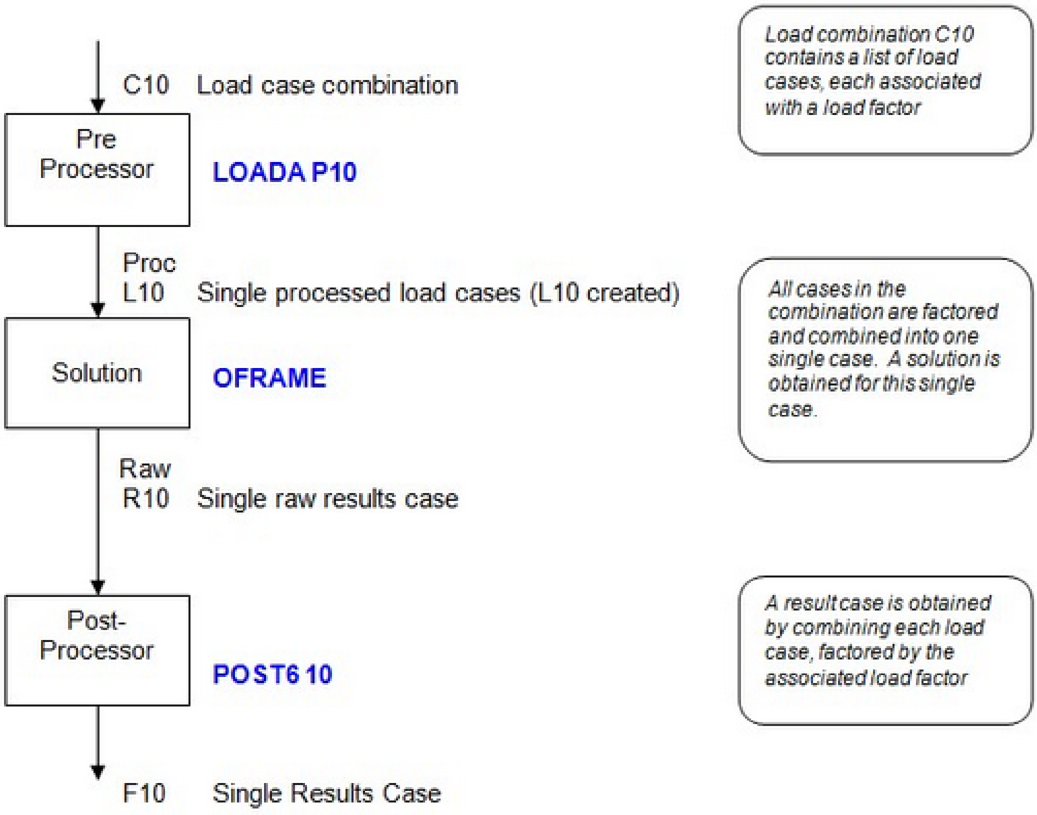


3D Standard Solution - LOAD CASE COMBINATION



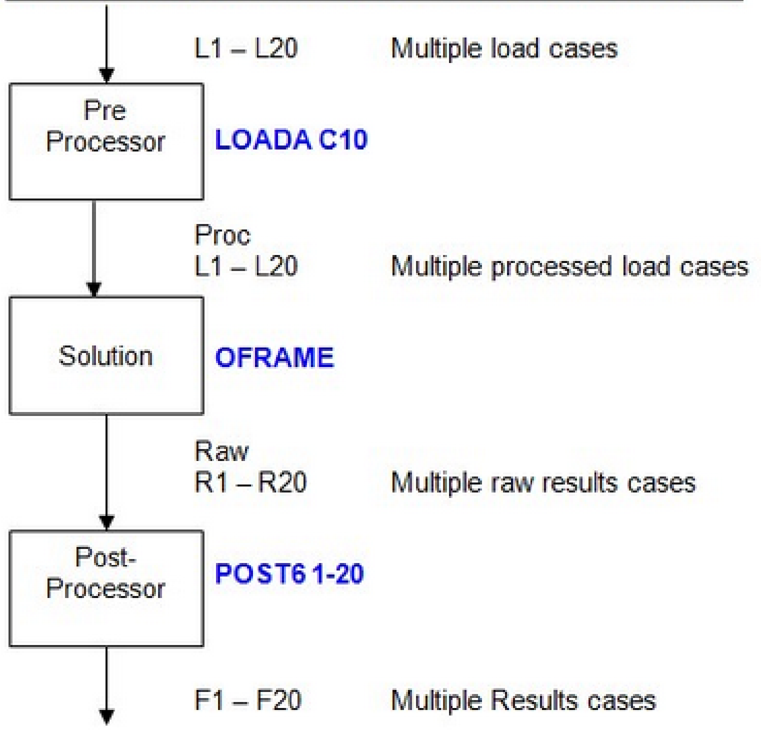
3D Standard Solution - LOAD CASE COMBINATION – PRE-COMBINED

This method has to be adopted when contact or P-Delta effects are required to be included in the analysis.

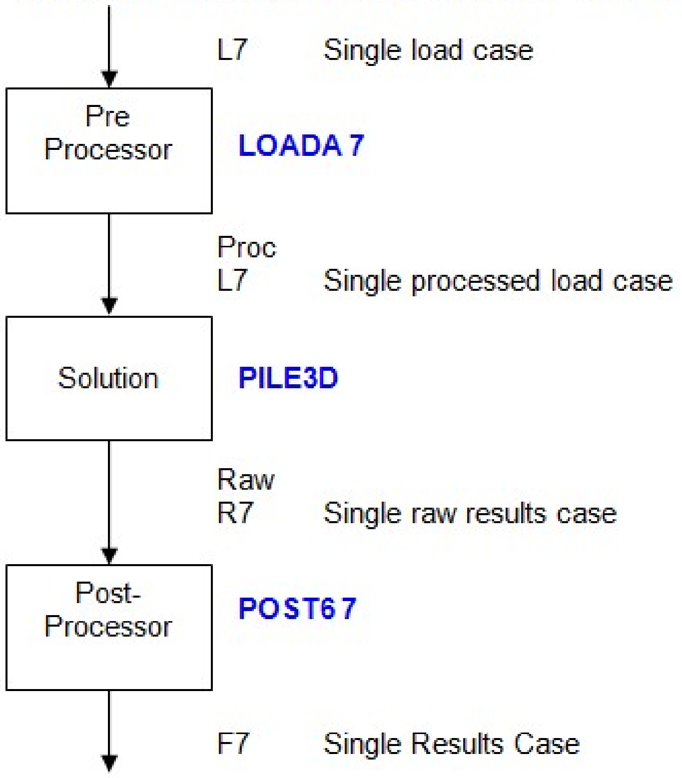


In this method a dummy Load Case (L10) is created using the same ID number as the Load Case Combination. For this reason there can be no load cases in the combination with the same ID number.

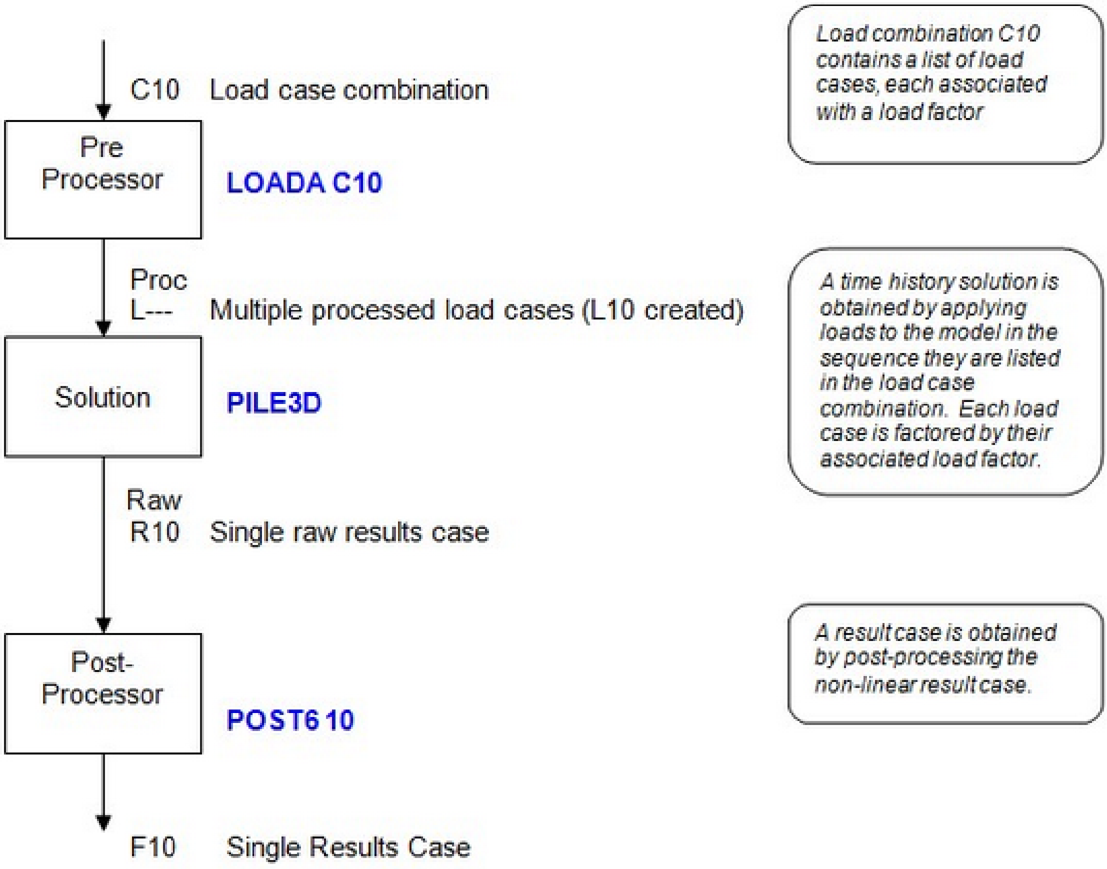
3D Standard Solution - MULTIPLE LOAD CASES



3D Non-Linear Analysis Solution - SINGLE LOAD CASE



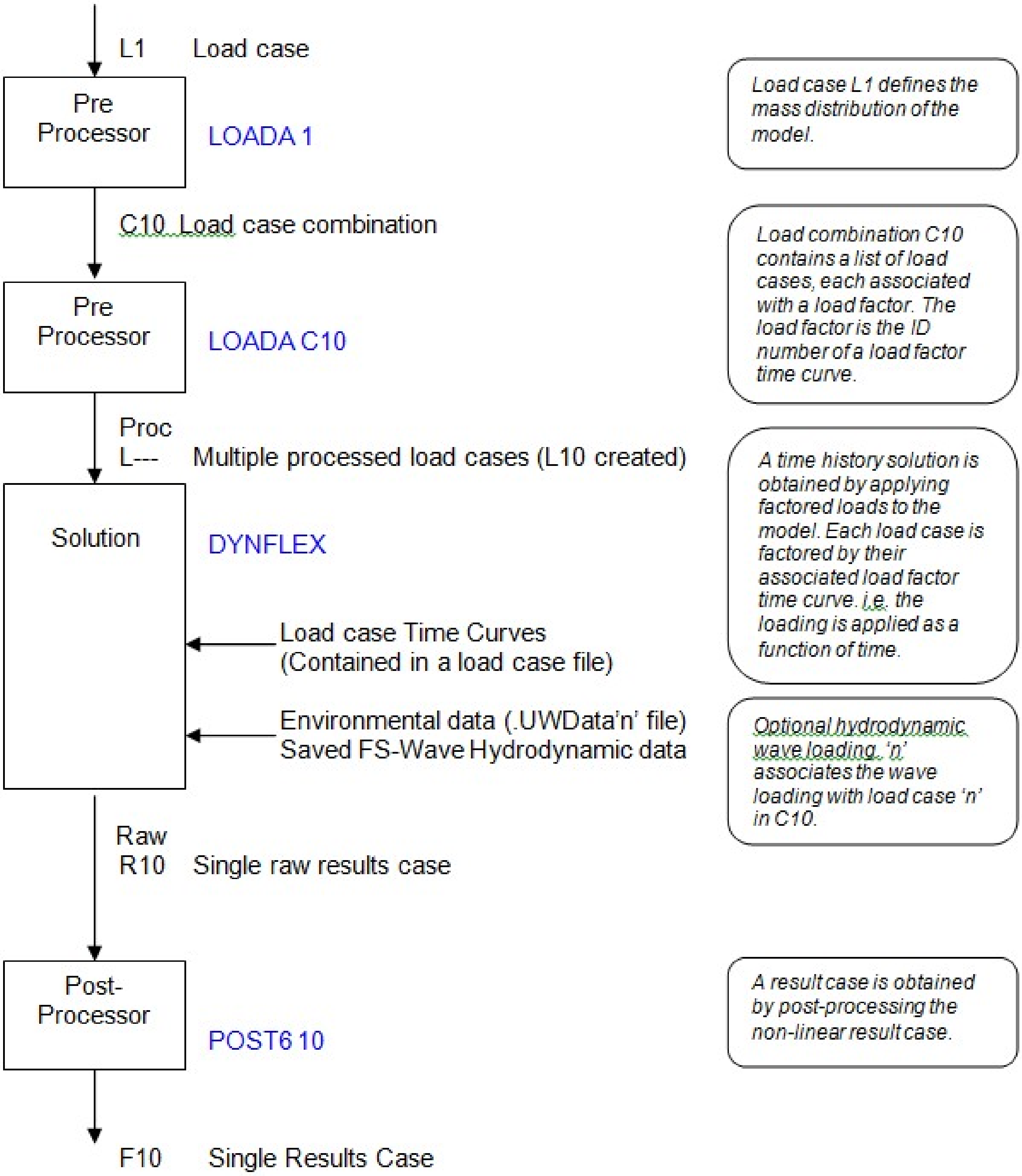
3D Non-Linear Analysis Solution - LOAD CASE COMBINATION



Post-processing can also combine raw results from other cases.

Load cases can be pre-combined as shown earlier for the Standard 3D solution.

DyNoFlex Solution - LOAD CASE COMBINATION



-o-

6.2.1 3-D Standard Analysis

3-D Standard Analysis

This analysis option is used for general linear analysis for both framed structures (beam and pipe elements) and finite elements. Beam and finite elements can be used within the same model.

The **3-D Standard Analysis module** is capable of accounting for the following type of non-linear behavior:

- **P-Delta Effects**
- **Tension/Compression Only Elements**
- **Contact (Make or break gaps elements)**

The program employs an iterative approach in which the load case is applied in one step and the solution repeated until a displacement convergence criteria is satisfied. After each solution the status of the elements are checked and if necessary updated for the next iteration.

Load cases or load case combinations are submitted for solution using the 3-D Standard Analysis form.

The non-linear effects require to be activated using the non-linear setting form activated from the 3-D Standard Analyis option forms.

The non-linear option settings are a global model settings. The are contained in the **<modelname>.UPT** file. When running in batch, they can be made load case specific by using the MFCopy command to re-create the setting file prior to solution.

- **P-Delta Effects**

In the program option for this effect is termed **Stress Stiffening (P-Delta)**.

Normal linear analysis does not take into account the interaction between member axial loads and lateral deflection. P-Delta analysis takes this effects into account by using an iterative approach and modifying the member stiffness matrix by using the axial force from the previous iteration. Tensile axial force in a beam element increase the lateral stiffness and compressive force decrease the lateral stiffness.

P-Delta effects can be used to predict elastic instability and tension stiffening effects in structures.

A typical approach to use P-Delta analysis is to predict global sway stability of structures using the 3-Standard solver would be;

> Analyse the structure with the basic loading plus a small lateral force in the direction of predicted instability. If the basic load does have loading in the direction of collapse the additional lateral load will not be required.
>
> Repeat the analysis but with all basic loading increased by a suitable factor.
>
> Repeat the last step until deflections become excessive.

If a structure is loaded above its collapse load it may be impossible to obtain a solution or the solution will be unrealistic. Always check the trend by monitoring the displacements as the load is increased.

Two beam element formulations are available,

- Beam Stability Functions (Ref:Structural Analysis, R.C. Coates et al)
- Beam Geometric Stiffness Matrix (Ref:Theory of Matrix Analysis, J.S. Przemieniecki).

The latter is the most commonly used formulation used by structural analysis programs but the former will produce more accurate bending moment results in tension controlled structures.

If the Z Direction Shear is specified as zero then Beam Stability Functions will be implemented otherwise Beam Geometric Stiffness will be implemented.

In the program option for this effect is termed **Stress Stiffening (P-Delta)**.

P-Δ or P-δ? P-Δ is the global structure displacement effects(sway) and P-δ is the local member (span) effects. In terms of program implementation there is no difference between the two and depending upon mesh density both effects will be included. At least one mid-span node is required for P-δ to be included. If double curvature exists in the span 3 nodes will increase the accuracy.

Important Note:

If P-Delta effects are dominant in an element then the mid-span forces and stresses evaluated in the post-processing module may be incorrect. Only the forces and stresses at the nodes will be correct. Plotting mid-span moments will show the magnitude of the error (discontinuities in curve). Setting the Number of Points on Span (Scaling) to 2, to plot only nodal values will remove the error. Consider using additional mid-span nodes to assess mid span sensitivity

- **Tension/Compression Only Elements**

All beam elements apart from bend elements are considered as Type 0 elements in this solution option and can be designated as as 'tension only' or 'compression only' elements.

- **Contact (Make or break gaps elements)**

Gap elements ie Couple Types 10, 11 & 12 are interpreted as simple make or break contact elements. Defined gaps and friction effects are ignored.

-o-

6.2.2 Modal (Frequency & Buckling) Analysis

The equations of motion for undamped free vibration and the equations of static equilibrium for linear buckling are similar in that they both form an eigenvalue problem and can both be solved using the same eigenvalue analysis routines.

For undamped free vibration:

$[K + Kg] - \omega^2 [M] = 0$

$\omega =$ Frequency (vibration modes)
K = Linear stiffness matrix
Kg = Geometric stiffness matrix (P-Delta effects) - Use is optional
M = Mass matrix (consistent or lumped)

For linear buckling:

$[K] - \lambda[Kg] = 0$

$\lambda =$ Load factor (buckling mode)
K = Linear stiffness matrix
Kg = Geometric stiffness matrix (P-Delta effects)

Eigen Value Shift

The Eigen value solution may be shifted using the following:

$\lambda = \lambda_0 + \lambda_S$

$\lambda =$ Desired Eigen value
$\lambda_0 =$ Eigen value SHIFT
$\lambda_S =$ Solution Eigen value

When this is done the [K] in the solution is replaced by $[K]_{EFF}$

$[K]_{EFF} = [K] - \lambda_0[M]$ or

$K]_{EFF} = [K] - \lambda_0[Kg]$

The use of this shift can be beneficial in the following two cases (generally but not always):

- Shifting Up $\lambda_0 > 0$ To avoid negative Eigen value being present in a symmetrical buckling solution e.g. pure shear panel.
- Shifting Down $\lambda_0 < 0$ To avoid having a singular stiffness matrix in frequency solution. This enables frequency solutions to be obtained on non restrained models e.g a floating structure.

In FS2000 not all elements types have been incorporated into the eigenvalue solution routines. The list below summarises those that are.

- All beam elements (note that all beam types apart form Type 16 will be treated like Type 0 but with P-Delta effects included)
- Type 16 Beam elements
- Couples (all couple elements will treated like Type 0 couples)
- Type 15 Spar elements

- Type 50 and 53 Shell elements

-o-

6.2.2.1 Modal Frequency Analysis

The eigenvalue analysis modules provide the capability to undertake frequency and modal shape analysis of 3-D structures. See Section 6.2.2 for element capability.

Three basic analysis methods are used by FS2000 - the Jacobi method, the Subspace Iteration method and Determinant Search method.

The module is started from the Eigen Frequency-Buckling command in Solution menu in the Analysis TASK. This command makes the analysis form described in Sect 8.15 visible.

The Jacobi method will evaluate all the frequency modes in a model i.e. one for each degree of freedom. The memory requirements of this method restrict its use to small models.

The Subspace Iteration method is described in the reference quoted below. It evaluates a selected number lowest frequency modes.

The Determinant Search method employs a determinant search technique and is described in the reference quoted below. It evaluates a selected number lowest frequency modes.

The Subspace Iteration and Secant methods used dynamic memory allocation but the size allocated to the matrix storage may may require to be increased for large models (see Sect 2.4)

Both methods use a banded solution method, so use the **Bandwidth** command in the Reseq Menu (Analysis TASK) to internally renumber the model for efficient solution.

Including P-Delta Effects in an Eigen Solution

This is optional in a frequency solution.

The tension stiffening effect (piano string) and compression softening effects can be included. To do this it is first necessary to run the **3-D Standard Analysis** with the **P-Delta** option active for the loading case that represents the loading in the structure when the frequency solution or buckling solution is required. This result case is then defined in the frequency analysis form.

Using an Eigen Value SHIFT

It is possible to obtain a modal solution for a model that does not have any form of restraint by using a negative Eigen value shift.

Modal Analysis - Output

The results from the eigenvalue analysis are the fundamental modes of vibration. The modes can be listed or plotted.

The output is identified by the mass case number for a frequency solution.

The fundamental modes are listed in the sub-case output data file **<Modelname>.FREQ.O'n'** where 'n' the number of the Mass Case.

The fundamental shapes are listed in the sub-case output data file **<Modelname>.FREQS.O'n'** where 'n' the number of the Mass Case.

It will only be visible in the standard analysis results selection list if a standard output case also exists.

The modes can be plotted using the Vibration Modes command of the Plot menu in the Output/Results TASK.

When this command is selected the user is required to enter the Case Number and the Vibration mode to be plotted.

The Vibration Case corresponds to the Load Case Number used to define the mass distribution. Vibration mode 1 is the lowest mode in the case. The Result Case scroll button can be used to quickly view the different modes.

The vibration plot may be animated by selecting the Animate command. Animation is stopped by re-selecting the animate command.

Command Line Operation

Providing that an appropriate <modelname>.UFO option file exists the module may be operated using command line (Batch) operation. The UFO file is created when the Module is run of the Batch button is pressed

The solution option settings in the **<modelname>.UFO** file are a global model settings. When running in batch, they can be made load case specific by using the MFCopy command to re-create the setting file prior to solution

Two command lines are used to run the frequency analysis. These are:

LOADA C1/C2/C3/

C1 Load (mass) Case to be analysed

Prefix with a P to pre-process a combination

C2 Soft Spring Option 0 - Inactive 1 - Active

C3 P-Delta Result Case Number

and one of the following (DYNFRAMB is always recommended)

DYNFRA2B 3-D Jacobi (Small Models)

DYNFRAMB 3-D Subspace Iteration or Determinant Search

Files Used

The only model definition files unique to the Frequency Analysis module are the Option file and the Modal Results files.

The Option file has the model filename with the extension <Modelname>.UFO

The Frequency Results files have the model filename with the extension <Modelname>.H'n' where n is the number of the Load (Mass) Case File. The files are binary files and can only be read by other FS2000 modules.

Ref: Bathe, K. J., Solution Methods for Large Generalised Eigenvalue Problems in Structural Engineering, University of California - Berkeley, Report No UCSESM 71-20, 1971.

-o-

6.2.2.2 Modal Buckling Analysis

The eigenvalue analysis modules provide the capability to undertake linear buckling analysis of 3-D structures. See Section 6.2.2 for element capability.

The eigenvalue analysis methods are used by FS2000 - is the Subspace Iteration method.

The module is started from the Eigen Frequency-Buckling command in Solution menu in the Analysis TASK. This command makes the analysis form described in Sect 8.15 visible.

The Subspace Iteration and Determinant Search methods can be used to evaluate a selected number lowest buckling modes. The ability to only evaluate the lowest modes is very computational efficient because only the lowest modes are of significance in real engineering systems.

The solution method use dynamic memory allocation but the size allocated to the matrix storage may may require to be increased for large models (see Sect 2.4)

Both methods use a banded solution method, so use the **Bandwidth** command in the Reseq Menu (Analysis TASK) to internally renumber the model for efficient solution.

Forming The Geometric Stiffness Matrix Kg

To enable the geometric stiffness matrix to be formed it is first necessary to run the **3-D Standard Analysis** with the **P-Delta** option active for the loading case that represent the loading in the structure when the buckling solution is required. The buckling solution case and this initial case are the same case number i.e. it is necessary the run the load case first and then the eigenvalue case.

Negative Eigenvalues

The Eigen solution methods in FS2000 are not capable of evaluating non-positive eigenvalues. These occur when the buckling load in a structure can is equal and can act in either direction. Examples of this are a X braced frame and a pure shear panel.

To prevent the solution from failing due to this effect, the Eigen solution can be positively shifted to avoid eigen values having identical but different signed values. The shift should be less than the lowest Eigen value. The Determinant Search method is often more effective in such situations.

Using Linear Buckling Mode Shapes in Non-Linear Buckling Analysis

When undertaking non-linear buckling analysis there will often be the requirement to define initial geometric imperfections. The choice or distribution may not be obvious for more complex structures. A practical approach is to use the linear buckling shape as the basis for the imperfections. This can be done by scaling the mode shape to the required magnitude and then add these displacements to the initial nodal geometry definition. The DNF button can be used to accomplish this task. When a mode shape is plotted the **DNF** button will add the scaled mode shape to the node geometry definition and create a file that can be interpreted in the Model Definition TASK to create a structure with initial imperfections. The **Displacements scaling factor** (**True scale deflections** - active) is used to scale the mode shape displacements. The displacements from the eigenvalue solution can be established using the **Deflections** command from the **Insp** menu (requires the same Results case to be open).

Modal (Buckling) Analysis - Output

The results from the eigenvalue analysis are the load factors to produce buckling ie a value of 2 would indicate that the load loading pattern would have to factored by 2 for buckling to occur. The buckling modes can be listed or plotted.

The output is identified by the load case number.

The fundamental modes are listed in the sub-case output data file **<Modelname>.FREQ.O'n'** where 'n' the number of the Force Case.

The fundamental shapes are listed in the sub-case output data file **<Modelname>.FREQS.O'n'** where 'n' the number of the Force Case.

It will only be visible in the standard analysis results selection list if a standard output case also exists.

The modes can be plotted using the Buckling Modes command of the Plot menu in the Output/Results TASK.

When this command is selected the user is required to enter the Case Number and the buckling mode to be plotted.

Buckling mode 1 is the lowest mode in the case. The Result Case scroll button can be used to quickly view the different modes.

The buckling plot may be animated by selecting the Animate command. Animation is stopped by re-selecting the animate command.

Command Line Operation

Providing that an appropriate **<modelname>.UFO** option file exists the module may be operated using command line (Batch) operation. The UFO file is created when the Module **Solve** button is clicked or the **Batch** button is pressed.

The solution option settings in the **<modelname>.UFO** file are a global model settings. When running in batch, they can be made load case specific by using the MFCopy command to re-create the setting file prior to solution.

Two command lines are used to run the buckling analysis. These are:

LOADA C1/C2/C3/

C1 Load Case to be analysed

Prefix with a P to pre-process a combination

C2 Soft Spring Option 0 - Inactive 1 - Active

C3 P-Delta Result Case Number

and the following

DYNFRAMB 3-D Subspace Iteration or Determinant Search

Files Used

The only model definition files unique to the Frequency Analysis module are the Option file and the Modal Results files.

The Option file has the model filename with the extension <Modelname>.UFO. This file would need to be copied and renamed in the the batch file if different options were required during a batch run.

The Frequency Results files have the model filename with the extension <Modelname>.H'n' where n is the number of the Load Case File. The files are binary files and can only be read by other FS2000 modules.

-o-

6.2.2.3 Obtaining Loads and Stresses from a Modal Solution

The magnitudes of the model displacements from an Eigen value solution are normalised and have no physical meaning.

However, there are situations where the relative loading distribution within a structure is of interest for a given mode shape.

There are two ways to obtain a Standard Results case from a modal displacement shape:

- Undertaking a Modal Response analysis, or
- Using the the modal displacements as a initial displacements if a DyNoFlex solution.

Modal Response Analysis

The FNCASE command is the easiest way to accomplish this.

FNCASE, *CaseNo, Mode*

This command will create a results case based on the modal displacements corresponding to the frequency of a specified mode.

The results case created is a standard results case must be post-processed in FS2000 like any other result case.

CaseNo Results Case Number

Mode Mode Number

DyNoFlex Initial Conditions

The DNF button can be used to accomplish this task.

When a mode shape is plotted the **DNF** button will do two things.

1. It will add the scaled mode shape to the node geometry definition and create a file that can be interpreted in the Model Definition TASK , and
2. It will create a file <model>.UNIDISP that contains the DyNoFlex INIDISP commands that describe the mode shape.

If the INIDISP command are added to a DyNoFlex History Curve Case file then this can be used to obtain a solution based on the displaced shape. Note that convergence of displacements, not force must be used, not force, because there are is no external loading.

-o-

6.2.4 Modal Response Analysis

The Dynamic Response Modules of FS2000 uses the normal mode method to provide dynamic response solution to the following types of vibration problems.

- Harmonic
- Transient Time History
- Response Spectra (Seismic)

The harmonic and transient response module is started form the Solution menu in the Analysis TASK.

The Response Spectra (Seismic) Analysis is started from the FS2000 menu in the Windows START menu.

They each have their own context sensitive Help files.

-o-

6.2.5 Static Non-linear Analysis

The **3-D Non-Linear** analysis option is used for general non-linear analysis. It supports the same element types as the Dynamic Non-linear solution option which is described in more details in the next section.

The **Static Non-linear solution module** is capable of accounting for the following type of non-linear behavior:

- **P-Delta Effects**
- **Plasticity**
- **Tension/Compression Only Elements**
- **Contact (friction and gaps)**
- **Non-linear soil/pile interaction (see FS-Pile Help)**

The module supports all elements type with the only restriction is that it does not support large displacement (updated geometry). Non-linear descriptions are given in Section 6.2.6.

The program employs an iterative approach in which the load case is applied in a number of steps. At each load step the solution is repeated until a convergence criteria is satisfied. After each step the status of the elements are checked and if necessary updated for the next load step. A description of the solution method is given in Section 6.2.6.

Load cases or load case combinations are submitted for solution using the 3-D Non-linear Analysis form.

The non-linear effects require to be activated using the 3-D Non-linear Option forms.

The non-linear option settings are a global model settings. The are contained in the **<modelname>.UPI** file. When running in batch, they can be made load case specific by using the MFCopy command to re-create the setting file prior to solution.

Load Steps & Incremental Solution

The Static Non-linear solution module employs an incremental solution method in which the loading is applied gradually in a series of loads steps.

A load case can sometimes be applied in a single step but often a solution can be more easily obtained by applying the loads in a series of load steps. In a model in which the internal load distribution is path dependent it is essential to apply the loading gradually, in a number of steps. An example of a path dependent structure is the spread of plasticity through a multiple redundant framed structure i.e. the load is distributed through the structure as the plastic hinges form which may be very different from the elastic distribution. Friction effects due to loading will also result in load path dependency. An example of this is where friction provides lateral restraint. This requires the contact force to be applied prior to the lateral force..

When a Load Cases combination is submitted for solution the load cases in the combination are applied successively in the order they have in the combination list. The load factor associated with the load case is applied during the solution phase. If load steps are specified they will be applied to each load case in the combinations. The resulting output will have the same number as the combination number.

If a load case combination is Pre-processed then the load will be combined as defined in the combination and then solved as one load case.

-o-

6.2.6 Dynamic Non-linear Analysis

FS-DyNoFlex is a general-purpose analysis option of FS2000 for the linear, nonlinear, static and dynamic analysis of structural systems.

The two predominant modes of structural non-linear behavior are:

- Geometric non-linearity (large displacement and contact)
- Material non-linearity (plasticity).

Geometric non-linearity exists when the deflections of a frame under loading become significant with respect to the original undeformed dimension of the frame such that the stiffness of the deformed structure is different from that of the undeformed structure or contact stiffness changes.

Material non-linearity exists when the elastic limit capacity (plastic limit) of individual elements is exceeded and local yielding results in redistribution of internal forces within the structure.

Non-linear analysis of structures is not a trivial task and an understanding of the method of analysis used by the program is essential if meaningful results are to be obtained. It is also essential to understand the behavior of the structure under given loading so that program options for the control of the analysis are appropriately set.

-o-

6.2.6.1 Background Theory

The finite element discretisation process yields a set of simultaneous equations:

[K]{u} = {W}

where:

[K] = stiffness matrix

{u} = displacement vector

{W} = vector of applied loads

If the stiffness matrix **[K]** is itself a function of the displacements then the above is a nonlinear equation.

The Newton-Raphson method is an iterative process of solving the nonlinear equations and can be written as:

[K]{u} = {W} – {F}

where:

{F} is the Newton Raphson equilibrium restoration vector (internal element forces)

The general algorithm proceeds as follows:

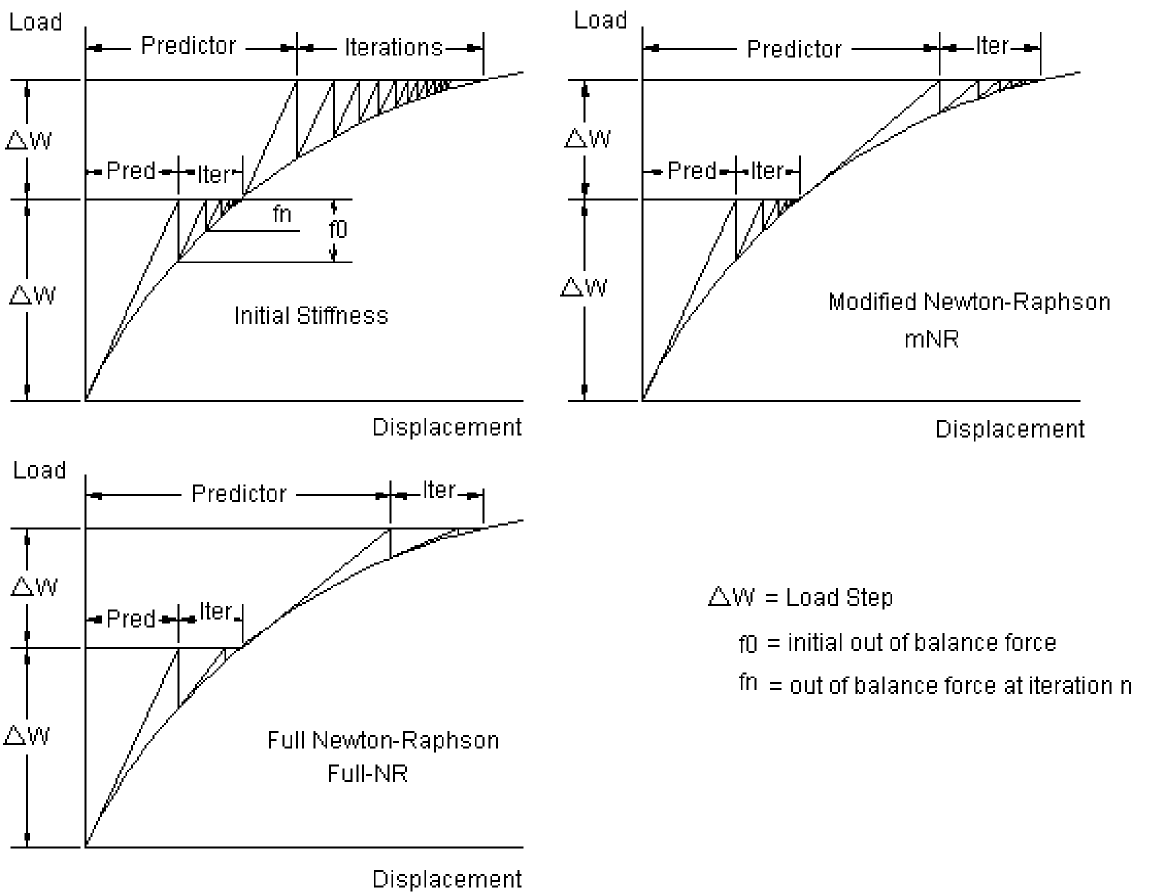
1. Assume $\{u_0\}$. $\{u_0\}$ is usually the converged solution from the previous time step. On the first time step, $\{u_0\} = \{0\}$.
2. Compute the updated tangent matrix **[K]** and the restoring load **{F}** from configuration $\{u_i\}$.
3. Calculate $\{\Delta u_i\}$ from **{u} = [K]/({W} – {F})**
4. Add $\{\Delta u_i\}$ to $\{u_i\}$ in order to obtain the next approximation $\{u_{i+1}\}$.
5. Repeat steps 2 to 4 until convergence is obtained.

Incremental Solution- Equilibrium Iterations

The approach adopted in nonlinear analysis is to apply the loading in a series of incremental steps. In static analysis the steps effectively equate to load factors. In dynamic analysis the steps equate to actual time increments. On completion of each step adjustments are made to account for the effects of the nonlinear behavior. The adjustments made are dependent on the type of elements and the defined solution control options.

The program uses the Newton-Raphson (**Full NR**) or the Modified Newton-Raphson (**mNR**) method for equilibrium iterations. The Newton-Raphson (**NR**) method updates the stiffness matrix at each equilibrium iteration. The Modified Newton-Raphson (**mNR**) method either never updates the stiffness matrix (Initial Stiffness method), updates it at every time step or updates it infrequently (at specified load step increments).

The number of time steps between reforming K and performing equilibrium iterations can be defined.



Convergence

The convergence criteria is based on the ratio $\Sigma\Delta D^{**}2 / \Sigma D^{**}2$ being less then Tol**2 where D represents forces or displacements or both.

Dynamic Equilibrium Equations

Transient dynamic analysis (time history analysis)

The basic equations solved are:

[M]{a] + [C]{v} + [K]{u} = {W(t)} – {F}

[M} is the Mass matrix

[C] is the Damping matrix

[K] is the Stiffness matrix

{a} is the Acceleration vector

{v} is the Velocity vector

{u} is the Displacement vector

{W} is the external load vector

{F} is the Newton Raphson equilibrium restoration vector (internal element loads)

The solution of the above dynamic equations employ direct implicit time integration methods. The program uses either the Newmark, WBZ Alpha or Wilson Theta methods. At any time t the equations are solved as an ‘effective static solution’ using Newton-Raphson equilibrium iterations.

The mass matrix **[M]** can be a lumped mass (diagonal) matrix or a consistent mass matrix.

The damping matrix **[C]** is in the form $C = \alpha M + \beta K + Cd$, where α and β are the Rayleigh damping coefficients. The Rayleigh damping coefficients relate to the more usual 'critical damping ratio' γ by the expression $\gamma = (\alpha + \beta\omega^{**}2) / 2\omega$. Cd is assembled from concentrated nodal dampers (Type 1 Couples) or Defined Element damping with Type 3 Couples and Type 15 Beams. Specific stiffness proportional damping may also be individually defined for Type 15 beams (Spars).

During a non-linear step by step solution for nonlinear elements the following matrix operations are undertaken:

- The Mass matrix is constant.
- The Stiffness matrix varies.
- The Damping matrix varies only if either the Beta_{NL} is specified or Defined Element damping is specified.

Newmark Method

This is the most commonly used algorithm used in transient analysis. The integration constants in the Newmark method take the form:

$\delta = 0.5 + \gamma$ and $\alpha = 0.25(0.5 + \delta)^{**}2$ where γ = amplitude decay factor

In the program the Integration Parameter equates to δ. If $\delta = 0.5$ ($\gamma = 0$) the Newmark method becomes the Average Acceleration method. Increasing the magnitude if the Integration Parameter above 0.5 will add a level of numerical damping which is often required to filter out unwanted high frequency response associated with discrete FE modeling. Setting $\gamma > 0$ does degrades the Newmark approximation therefore ensure that this does not effect the applicability of the solution. The value of δ is usually in the range 0.5 to 0.6 but can be increased when necessary to larger values. A warning will be given if this value exceeds 0.75. By default the minimum value for δ is 0.505 i.e $\gamma = 0.005$ regardless of the user defined value.

WBZ Alpha Method

This method introduces an additional parameter into the Newmark equations - α_m. This parameter is used for the interpolation of the acceleration between the time step. The value of is made equal to γ, the amplitude decay factor. If this value was set to zero the WBZ, method would be identical to the Newmark method. This method is reported to have increased numerical damping at hight frequencies with a lesser effect on lower frequencies.

Wilson Theta Method

In this method the Integration Parameter represents θ and the value for θ is usually in the range 1.39 to 2.01. As with the Newmark method increasing values of θ increase the level of numerical damping. It is reported that this method not so popular due to its tendency to overshoot.

Static Analysis

For static analysis or in cases where the mass and damping effects are insignificant the basic equations reduce to

$$[K]\{u\} = \{W(t)\} - \{F\}$$

Ref; E.L. Wilson,K.J.Bathe & I Farhoomand, "Non-Linear Dynamic Analysis of Complex Structures", Earthquake Engineering and Structural Dynamics, Vol 1, 241-252 (1973).

-o-

6.2.6.2 Time History Analysis

In some cases it is possible to apply the load in one step and the solution will converge to the correct solution. However, in most instances with non-linear analysis it is necessary to apply the loading gradually, in a series of steps.

Often there is also a requirement is to apply the loading in a sequential sequence to reflect an actual loading history.

This section summarises how conventional FS2000 load cases can be applied as sequential time history loading. The approach is equally applicable to both static and dynamic analysis.

In static analysis the steps are loads steps, in dynamic analysis the steps are time steps. They are both usually always referred to as time steps.

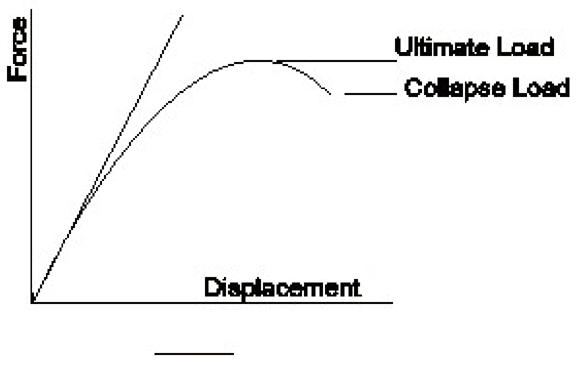
A summary of the basic method used is:

- Loading is defined and saved as conventional load cases.
- Time Curves are used to control how the applied loads defined in a load case(all) vary with each time step.
- Load Cases are associated with Time Curves using Load Case Combinations (the Load Factor references the time curve number) i.e. the time curve is a time varying load factor.
- As the Solution progresses the loading according is applied according to the referenced Time Curves.

The manner in which load are applied is specified by a **History Curve Case**. A History Curve Case is simply a load case combination in which the loads factors normal used in static analysis refer to Time Curves. One or more load cases can be contained in a combination and the loading is applied in sequence dictated by the time curves and not by the position the the combination list.

DyNoFlex solutions can also interpret a History Curve Case in the same manner as a **3-D Non-Linear** solutions i.e. use the load factors in the combination as load factor and apply the loading in the sequence dictated by the position the the combination list. This is done by preceding the History Curve Case ID with a C. See Section 8.

Load/Displacement Control



A typical load/deflection characteristic (buckling) of a non-linear structure is shown above. Using positive load increments (load control) it is only possible to trace the load path to the top of the curve (point of instability), ie the maximum load. At this point the solution fails to converge and a mechanism is formed.

If displacement control is applied it is often possible to trace the curve beyond the point of collapse. Therefore in many cases prescribed displacements may be used to trace beyond the point of total instability.

-o-

6.2.6.3 Non-Linear Options

Three basic non-linear options are available. These option are:

- Stress Stiffening/P-Delta
- Large Displacement/Nodal Displacement Updating
- Plasticity
- Contact/Friction

Each of these options may be enabled or disabled by the user as required.

6.2.6.3.1 Stress Stiffening/PDelta Analysis

In structural analysis P-Delta analysis is the general term used to describe analysis that takes into account the effects of lateral deflections and axial force (geometrical stiffening).

P-Delta effects are based on small displacement theory and their main use is for predicting axial stiffening effects and axial buckling instability.

Two formulations are available, Beam Stability Functions (Structural Analysis, R.C. Coates et al) and Beam Geometric Stiffness Matrix (Theory of Matrix Analysis, J.S. Przemieniecki)

In the program the option for this effect is termed ***Stress Stiffening(P-Delta)***.

6.2.6.3.2 Large Displacement

When the deflections are large it is possible to account for the change in stiffness by updating the global stiffness matrix based on the shape of the deformed structure. The program uses a corotational (convected coordinate system) approach to large displacement(Rankine & Brogan). In this approach the displacements that cause stresses are separated from those due to rigid body motion. The implicit assumption in this method is that whilst rotations can be large the rotations that cause stress must be small. Both displacements and rotations are updated.

The program option for this effect is termed ***Large Displacement***.

6.2.6.3.3 Beam & Pipe Element (Plasticity)

Beam Type Elements (see also Type 6 element description)

For beam elements (Frame Plasticity) the program uses the 'initial stress' method in which strain reversal and unloading is modeled ie yielded beams will retain their plastic strain even when the load is removed. In this method the stiffness matrix is kept constant and the plastic hinge effects are introduced by modification of the load vectors.

This method has the advantage that the global stiffness matrix is only assembled once. This disadvantage is that more iterations (but faster as only back substitution is employed) are required particularly when approached collapse.

If beam plasticity is the only type of non-linear behavior then the solution options should be set so that the stiffness matrix is not reformed. This will give the fastest possible solution. The Time step predictor and/or mNR Acceration and Line Search options are very effective for this type of non-linear behavior.

The non-linear material option for non-pipe beam elements always assumes ideal beam plasticity i.e. no strain hardening effects. The elastic load limit is defined in terms of a plasticity modulus and a material yield stress, which is associated with the element property code for that element. If the load level is below the yield limit the element behaves elastically if the load exceeds the limit the load in the element will be limited to that value.

Beam Yield Functions

To account for combined loading effects the program has four basic types of plastic beam element. The difference is the between the three is the criteria for establishing the plastic limits using different interaction surfaces. This is available for Type 6 beam element.

Pipe Plasticity

Plastic behavior of pipe beam elements can also be defined using a specified stress strain curves (Bi-linear, multi-linear or Ramberg-Osgood). This type of plasticity is described in the Type 6 beam description. Plastic strains can be obtained using the ETABLE utility

Moment Curvature

Beam non-linear material behaviour can also be accounted for by using moment-curvature curves. A utility to evaluate the moment curvature relationship for a specific pipe types (single or multi-layer) is also available. Moment curvature is described in the Type 6 beam description.

6.2.6.3.4 FE Solid Plasticity

These properties are input using the Non-Linear Geometric Properties form.

Shell Elements

They can only be run with the Non-Linear option active, even for linear elastic solutions.

For non-linear plastic solutions the stresses at the nodes are the Gauss point stresses i.e. no extrapolation to nodal locations.

Plasticity Model

The Von-Mises yield model is always used.

Properties

Yield Stress defines the yield limit

Plastic Sz defines the number of layers.

Plastic Sy defines Et or identifies a RC constants table for the definition of a multi-linear or Ramberg-Osgood curve. If Sy is a negative value it is interpreted as a RC table entry. The bi-linear hardness parameter is defined by H = Et / (1 – Et/E). For perfect plasticity Et should be set to zero.

Element Type	**Description**
T51	Flat Shell (QM6 formulation)
T52	Flat Shell (MITC4 formulation)
4 Node (Soln Opt 0)	

2-D & 3-D Solid Elements

The following elasto/plastic finite elements types are available for solutions with DyNoFlex. They can only be run with the Non-Linear option active, even for elastic solutions.

The degrees of freedom has to be set to **2 for 2-D** or **3 for 3-D**. No others elements can be used in the model apart from then following:

2-D Type 0 and Type 12 couples.

3-D Type 0 couples

For non-linear plastic solutions the stresses at the nodes are the Gauss point stresses i.e. no extrapolation to nodal locations. Plastic strains can be obtained using the ETABLE utility.

No reactions given. Consider node to ground couples to provide restraint if reactions are required.

No large displacements effects..

Plasticity Model

If the nonlinear Material is active then the following can be used to the define the material model.

The Geometric Property Code Type defined the plasticity model

1	Tresca
2	Von-Mises

3	Mohr-Coulomb
4	Drucker-Prager

Only the Von-Mises should be used for plane stress solutions.

Properties

Yield Stress defines the yield limit (For cohesive soils YS=SQRT(3)*Cohesion).

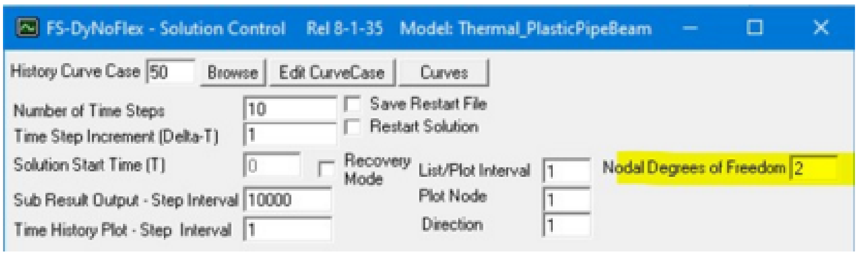
Plastic Sz defines the internal soil friction angle Phi.

Plastic Sy defines Et or identifies a RC constants table for the definition of a multi-linear or Ramberg-Osgood curve. If Sy is a negative value it is interpreted as a RC table entry. The bi-linear hardness parameter is defined by H = Et / (1 – Et/E). For perfect plasticity Et should be set to zero. The Plastic Torsion St defines the internal soil dilation angle Omega (Mohr-Coulomb plasticity). If Omega 0 = 0 then Omega = Phi.

Type	**Description**
T30	2-D Plane Strain/Plane Stress (Run with 2 nodal degrees of freedom)
	4 Node (Soln Opt 0), or
	8 Node and 6 Node parabolic
T40	2-D Axy-symmetric (Run with 2 Nodal Degrees of Freedom)
	4 Node (Soln Opt 0), or
	8 Node and 6 Node parabolic elements
T70	3-D Solid (Run with 3 Nodal Degrees of Freedom)
	8 Node Element (Soln Opt 0)
Soln Opt 0	QM6 (Extra shape functions)

WARNING - Ensure that the Degrees of Freedom option is set to 2 or 3. If not the solution will fail indicating a mechanism has been formed (Error Neg Pivot).

.



-o-

6.2.6.4 Time Steps

The time step is the fundamental control variable for dynamic and non-linear analysis. In static analysis the time step is effectively used to define load increments (load steps or load factor increments). In dynamic analysis it defines an actual time increment (usually seconds).

A solution duration is specified by defining:

- A constant time step (Δt), and
- The total number of steps to be repeated (Nt).

The solution duration Td = Δt.Nt

If a Time History curve is defined and specified for use then this curve will dictate how the loading is applied to the model during solution. If no curve is defined the time increment will be applied as a unit ramp.

Time Step Size

It should be noted that if convergence of the solution fails then either the load level is above the load limit of the structure or the load increments are to large and the solution is numerically unstable.

For dynamic analysis the time set requires to be sufficiently small to capture the response. The Newmark and Wilson Theta method are sufficiently stable such that the time step can be based on the response rather than the shortest natural period of the structure. Typically a time step of 1/10 to 1/20 of the period of the structures response should suffice. For non-linear systems may require a shorter time step.

The time step may be changed for re-start analysis.

Auto Time Step

If the Auto time step option is active the program will adjust the time step if the solution fails to converge.

Associated with this option is a Target Iterations parameter. The value of this parameter established the adjustment process.

Target Iterations = -1 (or any negative value)

> If this value is set to -1 the load step will be halved if the solution does not converge. The solution will stop if the accumulative reduction from the original time step is greater than 200 and convergence is not achieved. The solution will continue until the solution duration total is completed based on the original Δt i.e. Td = Δt.Nt.
>
> If this value is set any negative non zero value e.g. -1.1 then the load step will be halved on non convergence and following convergence, the absolute value will be used to factor the load step at each time step until the original time step is reached. This enables the time step to recover backs to its original value. A value of 1.1 would be considered an upper bound parameter.

Target Iterations = 0

> If the Target number of iterations is set to zero and the actual number of iterations exceed the maximum then the program will reduce the time step using the following criteria.
>
> If ITER > ITERMAX then DTNew = DTOld * Set Conv Tol / Actual Conv Tol
>
> Within the limit 0.5 => DTNew/DTOld <= 0.1

Target Iterations > 0

> The method should produce small increment when the response is most linear and large increments when the response is most linear. Difficulties can arise with this method when sharp changes in the response case exist and in cases where the response is flat e.g. plastic limit.
>
> If the Target number of iterations is set to value > 1 then the program will attempt to adjust the time step such that this number of iterations is maintained. It does this using the following criteria.
>
> DTNew = DTOld * (Target Iterations/Actual Iterations)**0.5
>
> Within the limits 2.0 > (Target Iterations/Actual Iterations)**0.5 < 0.5

-o-

6.2.6.5 Loading

Loading cases are created and applied in the same manner as would be done when using FS2000 for linear static analysis. The only difference is, is that Time Curves are used to define how the loading (Load Case) is applied. In dynamic analysis loads have to be defined as a function of time. It is possible to define specific nodal loads at each time step. This uses a ULData file and it's use is described in Moving Loads.

The output from the analysis will always be a single case with the same number as the load case or load case combination being solved. Intermediate results cases (sub-cases) at selected time intervals may also be obtained.

Initial Motions

Initial Motions may be defined using the **INIDISP**, **INIVEL** or **INIACC** commands. The command line syntax is of the form:

INIDISP*, Node, Displacement, Direction*

Node	Node to which condition is applied
Displacement	Applied displacement (m)
Direction	Global Freedom Direction (1 to 6)

These initial condition commands require to be included in the solution's History Curve Case (Load case).

Load Cases

When a single load case is analysed it is always assigned to Time Curve ID 1, this eliminates the requirement to use a combination when only a single load case is being used. If a History Curve Case specified then it should contain an ID 1 Curve. If no History Curve Case is defined or the History case does not contain time curve commands then a unit ramp curve will be applied. Note that if the History Curve Case is blank it will also have the same effect i.e. a unit ramp curve will be applied.

Pre-Combined Load Case Combinations

Pre-combined load case combination are pre-processed in the same manner as other analysis modules of FS2000. Pre-processing creates a new load case at the time of analysis based on the load case and load factors listed in a Load Case Combination. It enables different load case results to be combined into a single load case i.e. merges load cases at analysis time. During analysis it will be treated as would be a single load case,

Load Case Combinations and Time Curves

Load Case Combinations in FS-DyNoFlex are treated very differently to load case combinations in other FS2000 solution options. Load Case Combinations are use to associate Time Curves with Load Cases i.e. they are used to define how the load is applied.

When combinations are used for the purpose the Load Factor is interpreted as a Time Curve ID Number and the load case is than factored by the load curve.

Time Curve commands are contained within a load case called a History Curve Case. It is good practice to save a load case with the description 'Time Curves' or something similar, this will ensure that the case will be visible in FS2000's load case lists. The load case file can be later modified to include the required time curve definition commands using FS-Edit.

Time curves are described in more detail in Section 5. The time curve commands and their command line arguments are given in Section 9.

Load Combination Limits

There is no maximum limit on the number of load cases in time history combination but it is suggested that this be minimised by combining all linear loading effects into a singe case.

Large Displacement Restrictions – Beams/Pipes

An important restriction is that element structural loads e.g. UDLs, apart from those listed below do not follow the elements when the Large Displacement option is active

- Pipe pressure
- Thermal strain
- ESTR
- SEFO

This restriction may be ignored in cases where the Large Displacement is active but the element rotations are not too significant .

In case where large rotations are experienced the loading should be applied only as nodal loading. The **Lumped Load** option if active will convert all beam/pipe element loads eg self weight etc to nodal loads.

Prescribed Displacements

If prescribed displacement are used they must be defined in the first load case of the load case combination used to define the load history. The magnitude of the displacements can vary between load cases in the combination but they can only be applied to the same degrees of freedom identified in the first load case. It is not essential to include PDs in all load cases in a combination (this was a requirement in older versions of FS2000).

Finite (Solid) Elements

Only the following loading type are currently available in DyNoFlex for finite elements are:.

- Finite element gravitational loads
- Nodal loads

-o-

Moving Loads

Moving Loads - Specific Nodal Time History Loads for DyNoFlex Solutions

Moving Loads is a term used to describe user defined nodal loading that are defined for each and every load step. Such loading can employed in either static or dynamic solutions. This type of loading would be advantageous in cases where the loading pattern changes both in location and magnitude and would require the use of too many load cases and associated time curves. This type of load file may be created by a user or by the use of FS2000's Moving Load Generator.

The moving loads are contained in binary file called **<modelname>.~ULData'N'** , where **N** is load case to which these load are associated. These loads will be added to current loading applied being applied to the model but will not be factored by any load factor (time curve) that is associated with Load Case **N.** The association with load case **N** is merely a ID trigger, if a combination contains case **N** then the ULData file will be read.

The loading file **<modelname>.~ULData'N'** will be **deleted** if Load Case **N** is deleted or re-saved in the Load Definition TASK.

Moving loads are not added to a loading history i.e. they are added to the current nodal loads at each time step during the solution. They have to be defined for all solution time steps over the range of interest.

The time step defined in a moving load data file MUST BE the same as that defined for the solution.

The **ULData** file is read sequentially for each load step and there must be sufficient data in the file (**NSteps**), to match the number of solution time steps, if not the solution will fail. The exception to this is when the Format Type = 1, in such cases a single step termination line is used to signify that no more data is to be read and the solution will continue.

The **Start Time** parameter can be used to delay the time in the solution when the moving load definition starts being applied.

ULData Binary File Format

The loads are defined by node label. One or more nodal load definitions can be defined in a load step block.
A negative N value, any value, delimits the load step block.
This step termination line must have the 6 zero values defined.
The number of time(load) step blocks in the file must be equal or greater than those defined in the DyNoFlex option form i.e. if 1000 time step are used the file must contain 1000 -ve step terminators.
Loading has to be defined at each time step otherwise no loading will be applied, it is not held constant, it is incremented to the existing loads at that node.

Note that the commas below are only there for readability, each line is a random access record.
The format for each line the must be : **Long Integer, 6 x Single Precision**

NSteps, StartTime(s), TimeStep ,0, 0, 0, 0

N_1, Fx, Fy,Fz, Mx, My,Mz

N_n, Fx, Fy,Fz, Mx, My,Mz

Within the program the **FormatType** is designated as as 0 or 1.

If **NStep**s is +ve then **FormatType** = 0 else **FormatType** = 1.

If **FormatType = 0**, then time step terminators have to be included for every time step and loading can be applied to more than one node at each time step.

If **FormatType = 1**, then time step terminators are not included in the data and loading can be applied to only one node at each time step.

StartTime defines the solution time at which the time step loading should start being applied.

The **Time Step** is the time step applicable to to the ULData. This value has no influence on the solution time step but the solution time step must be the same.

N_1, Fx, Fy,Fz, Mx, My,Mz is the node and the associated loads.

A negative **N** value indicates the end of a time step for Format Type 0. For Format Type 1 a value of -1 indicates the end of the data.

This (**FormatType**=0) file example below shows a constant value of Fy being applied to a varying series of nodes for 510 time steps. In this example the StartTime=5s, TimeStep=0.1 and no loading is be applied at load step 4.

`510,5.0,0.1,0,0,0,0`	NSteps, StartTime, TimeStep,0,0,0,0
`1,0,-143.12,0,0,0,0`	Step 1
`-1,0,0,0,0,0,0,0`	End of Step 1
`31,0,-143.12,0,0,0,0`	Step 2
`32,0,-143.12,0,0,0,0`	
`-2,0,0,0,0,0,0,0`	End of Step 2
`31,0,-143.12,0,0,0,0`	Step 3
`32,0,-143.12,0,0,0,0`	
`33,0,-143.12,0,0,0,0`	
`-3,0,0,0,0,0,0,0`	End of Step 3
`-4,0,0,0,0,0,0,0`	Start & End of Step 4 - No Loading
`32,0,-143.12,0,0,0,0`	Step 5
`33,0,-143.12,0,0,0,0`	
`34,0,-143.12,0,0,0,0`	
`36,0,-143.12,0,0,0,0`	
`-5,0,0,0,0,0,0,0`	End of Step 5
`33,0,-143.12,0,0,0,0`	Step 6
`43,0,-143.12,0,0,0,0`	
`36,0,-143.12,0,0,0,0`	
`37,0,-143.12,0,0,0,0`	
`-6,0,0,0,0,0,0,0`	End of Step 6
.	
.	
.	
`-509,0,0,0,0,0,0,0`	End of Step 509
`31,0,-143.12,0,0,0,0`	Step 510
`32,0,-143.12,0,0,0,0`	
`-510,0,0,0,0,0,0`	Termination record

This (**FormatType**=0) file example below shows a varying value of Fy being applied at Node 5 for 6 time steps.

```
-1,0,0.1,0,0,0,0
5,0,-143.12,0,0,0,0
5,0,-100.00,0,0,0,0
5,0,-156.00,0,0,0,0
5,0,-167.00,0,0,0,0
5,0,-148.00,0,0,0,0
5,0,-106.00,0,0,0,0
-1,0,0,0,0,0,0,0        Termination record
```

-o-

Moving Load Generator

The Moving Load Generator pre-processor module creates a Moving Load time history definition case that can be included in a DyNoFlex time history solution. If can be used for both static and dynamic solutions.

The basic mode of operation of the load generator is:

- Using Group identification a path is defined along connecting elements line elements within a model.
- A load magnitude of finite length traveling at a specific velocity is defined - two loads per path can be defined.
- Nodal loads imposed by the moving loads are generated at each time step as they pass along each element.
- The resulting nodal loads are contained in a load step data file and can read by DyNoFlex for each solution time step.

The loading generated can include gravitational and or inertial (due to path curvature) load effects. Curvature is evaluated based on the definition node location.

For bend elements the load will traverse across the chord of the bend and the curvature will be taken as that defined for for the bend

The module is run in Batch mode using command line argument definition. The resulting output is a **Load Case N** and the file **<modelname>.~ULData'N' .** This is a binary file that is read by DyNoFlex during the solution. The use and format of the file is described in moving load data file.

When the moving load file is generated a standard load case will also be created. This load case contains no definition, only an echo of the command line parameters and some derived values.

When included in a time history case the variation is not dependent upon a time curve, the variation with time is within the moving load data file. However, in a Time History Combination the Moving Load case can and must be associated with any time curve being used in that history.

MOVELOAD C1/C2/C3/C4/C5/C6/C7/C8/C9/C10/C11/C12/C13/C14/C15/ - Command line switches in then Batch line editor.

Any number of moveload generated cases can be included in a DyNOFlex solution.

MOVELOAD
CaseNo/EntryNode/GroupSET/Group/STime/TimeStep/Velocity/L1/m1/L2/m2/NPoints/NSteps/ForceType/GCon/

1	*CaseNo*	Load Case Number
2	*EntryNode*	Start node for the element load path
3	*GroupSET*	Identifying Group SET for the load path
4	*Group*	Identifying element group for the load path (in Group SET)
5	*STime*	Time at which the train enters the path - If zero will start when the solution starts (even for restarts).
6	*TimeStep*	Time step - This must be the same as that of the subsequent DyNoFlex solution -- s
7	*Velocity*	Velocity of the moving train - m/s
8	*L1*	m1 length - m
9	*m1*	Mass m1 - kg/m

10 *L2* m2 length - m or Air gap between m1 train if m2=0 = 0 Advantageous for continuous trains to always specify as a -ve value (see below).

11 *m2* Mass m2 - kg/m

12 *NPoints* Number of load length divisions - A value of 1 would imply a concentrated load.

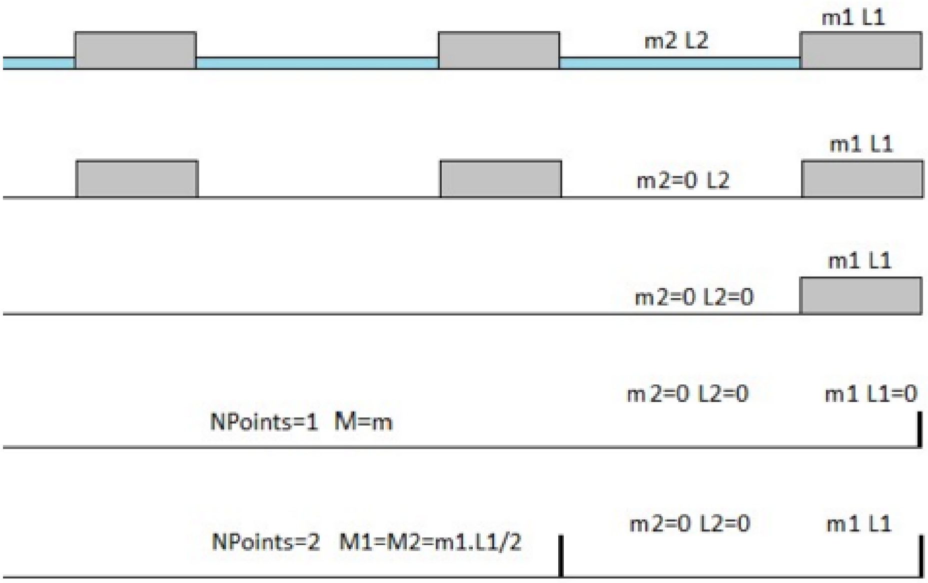
13 *NSteps* Number of of time steps - Optional Default=Number of steps for the train to enter and exist the element path

14 *ForceType* 0 - Both 1 - Gravity 2 - Inertia - Optional Default=1

15 *GCon* Gravitational constant - Optional Default=9.81 m/s

The above are self explanatory but the following may provide more clarity.

Basic train configurations. The top two are continuous trains.



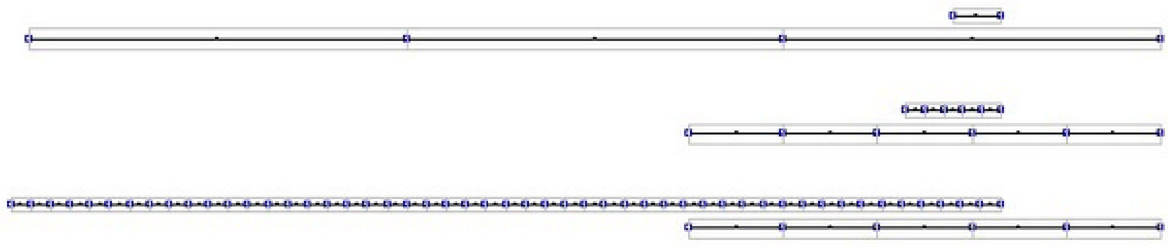
Start Time
The Start Time parameter can be used to delay the time in the solution when the moving load definition starts being applied. This should be borne in mind when using restart solutions.

TimeStep
The time step defined must be the same as that used in any subsequent DyNoFlex solution.
In static solution its size will be a function of velocity and load length considerations. In a dynamic solution response consideration may dictate.

Path Definition
The element define in the path definition group must be connected and to form a continuous line.
Individual element connectivity direction is arbitrary. The *EntryNode* defines path entry.

Lumped Load Considerations - ***NPoints***
The loads are applied over a defined length. This achieved by dividing the distributed load in to a series of consistent discrete point loads dictated by the variable *NPoints.* If this defined as 1, a single point load will represent the distributed load and this point point load will be distributed to the nodes of the element it passes as a function of its position within the element.
Increasing *NPoints* will improved the consistency of the distributed load and the number to be used will dependent up the ratio of the element length to the load length.
If the train length was say equal to an element length then there would be little benefit in increasing NPoint>5. However if the train was very much larger than an element then *NPoints* may have to be quite large.
A good guide for long trains is that the minimum number = (TrainLength)/(Smallest Element Length)*N

where N>5.
Under certain conditions the TrainLength i.e. (L1 + L2) need not include L2 ,see **Length L2** comments below.
If *NPoints* is to small a saw tooth response is likely (undertake a static soln to check).
There is no limit on *NPoints* and the value has no effect on DyNoFlex solution times. It only increases the processing time of MOVELOAD generator.



Inertia Load Evaluation-Curvature -Bends

The curvature of the path is evaluated at each node. The enables the mass inertia loading due to curvature using the relationship mV^2/R, to be evaluated. Curvatures cannot be evaluated for 90 degree sharp corners and when these are encountered a very large radius will result with an accompanying small inertia load. Only bends formed from segmented elements can account from curvature effects because nodal curvature is evaluated based on the position of the two adjacent nodes. It recommended that the bend tails with lengths equal in length to the bend elements be included when forming pipe bends.
If the curvature is insignificant it is computationally efficient to only evaluate gravitational loading i.e. *ForceType=1*

Mass m2 & Length L2 Considerations - Continuous Trains

If the L2 is defined as zero it implies a single m1 passage. At the first time step the m1 slug is at the entry to the path. Path initially unloaded.
If the m2 is small relative to m1 then it is computationally efficient to define the m2 as zero and thus eliminate the evaluation of m2 loading.
If the L2 is defined as a -ve value then the train is at the exit to the path i.e. starts with path loaded. This is very computationally advantageous if m2 is not zero. This is because in the program the m2 loading is applied as a constant and the m1 loading as a differential. This means that the **NPoints** value can be independent of the L2 length and thus be very computationally efficient for long L2 lengths.

Command Line Definition Data Echo

The command line definition data is echoed in the load case **L'N'** It includes the following derived data.

NStep	If NSteps=0 then NSteps=Path Time / DT
Path Length	The line length of the grouped elements.
Path Period	The time taken for single train to enter and exit the path.
Path Steps	The number of time steps required for single train to enter and exit the path.
Train Length	The length of the train pitch.
Train Period	The time taken for single train to pass a stationary point.
Train Steps	The number of time steps required for single train to pass a stationary point.
Slug1 Freq	Only if L2 > *0.*

-o-

6.2.6.6 Hydrodynamic Loading

DyNoFlex interfaces with FS-Wave for the definition of wave load related definition.

Unlike FS-Wave which creates static load case for specified wave condition the dynamic analysis method applies the wave and current loading as a dynamic time history load using Morrison's Equation.

The wave time history analysis can be run either statically or dynamically but should always be run as non-linear solution.

Element Loading

$W = 0.5Cd.\rho D|Ur|Ur + (1 + Ca).\rho.A.aw - Ca.\rho.A.a$ Load per unit length

Ur = relative velocity of water to structure

aw = acceleration of water

a = acceleration of the structure

Ca= added mass coefficient Note that Cm = 1 + Ca

In the program the last term in the above i.e **Ca.ρ.A.a** (RHS of the dynamic equation) is transferred to the LHS of the dynamic equation i.e. the effective mass becomes (mass + added wave/water mass). The added wave mass includes coating and contents defined using FS-Wave.

This LHS mass is applied as a nodal mass in all degrees of freedom, and is because of the dominance of lateral effects, based on the local y lateral direction added mass.

LHS mass effects above the LAT reference depth are not included in the structure mass case.

Nodal Loading

$W = 0.5Cd.\rho A|Ur|Ur + (1 + Ca).\rho.Vol.aw$

Unlike element loading the structure motion related added mass **Ca.ρ.Vol.a** is not automatically added to the dynamic mass case. Therefore the added mass **Ca.ρ.Vol** should be added to the structure mass case by the user (if considered to be dynamically significant).

When the Hydrodynamic model data is saved in FS-Wave a **Time History** option box is available that will, if active, create additional data required by DyNoFlex. This data relates to model hydrodynamic coefficients etc use for applied force evaluation and data used to define dynamic mass. Wave loading effects will only be active in DyNoFlex if this time history option is active.

RAO data can also be used to prescribe vessel type motions at specified node locations within the model.

Hydrodynamic and Mass Coefficients

All the data definition ie Cd Added mass etc.used by FS-Wave for linear quasi static analysis is available for DyNoFlex but with the following exceptions.

- Cd or Marine growth profiles are relative to the LAT reference depth. This depth must be defined (optional in linear quasi static analysis).
- The mid point of an element relative to the LAT reference depth is used as the submerged element criteria.
- Contents and added mass is zero above the LAT reference depth.
- The contents elevation limit is ignored i.e. if specified, contents are applied regardless of elevation up to LAT.
- The structural added mass components of defined nodal volumes i.e. **Ca.ρ.Vol** should be manually added the the mass load case (if considered to be dynamically significant).

Added mass, contents mass and coating masses defined in FS-Wave will be applied as lumped masses or consistent masses and be added to all mass cases. Note that LHS mass data is saved as a global model data (<modelname>**.WMass)** when the WAV file is saved in FS-Wave (time history active) and it will be added to all mass case cases e.g. modal analysis. This mass data is always deleted when the model is saved or when the WAV file is saved without the time history option being active.

Note that this additional mass file will also be used when undertaking Eigen frequency modal analysis.

The Wave Spreading Factor and Current Blockage Factor are not applied.

Buoyancy and Gravitational Loading

Buoyancy loading and gravitational loading due to contents and coatings will be applied as lumped loads and are not therefore required to be added to standard FS2000 load cases.

If the Include Buoyancy /Weight Effects option is not active in FS-Wave then these load effects will be neglected (Properties tab in FS-Wave) and only hydrodynamic loading will be applied. The Exclude Buoyancy/Weight in the Environmental tab in FS-Wave has no effect in a time history solution.

A good strategy when applying wave loading is the apply the wave load case with the initial model loading e.g. gravity and then use the **Wave Start Time** parameter in the UWData file to initiate the wave action at a specified time. Using this approach the buoyancy is always on and the wave starts at a specific time in the solution.

Environmental Data

Only linear (Airy) wave theory is currently available in DyNoFlex. The wave data is required to be defined in the following model file.

<modelname>.**UWData'N'**

If the file is exists it will activate wave loading and the wave loads will be associated with Load Case **N**. A dummy load case can be set up ie a load case with no load definition to be associated with the wave load. Note that this file will be deleted if Load Case N is deleted or re-saved.

The format for the text file is:

Water Depth
Wave Height
Wave Period
Wave Direction Angle
Current Angle
Number of Current profile points
Elevation, Current < One entry for each profile point
Wave spectrum type *
Spectrum parameters *
No of spectrum blocks *
Random Seed *
Wave Start Time, Irreg Wave Start Time (secs)***
WTh, Order, Damp, NPts
Number of RAO data files**
RAO Data file IDs**

This data file can be created in FS-Wave by making the **Dynamic History Case** option active. Only do this once as this is only a utility to create the file, using a second time may overwrite modified parameters.

The *data that defines an irregular wave must be edited by the user. The default value of zero for the Wave spectrum type specifies a regular wave (see next section).

The **data is used to define prescribed motion using vessel RAO data. The is described more fully in Section 5.

The *** **Wave Start time** is used to define the start time for the wave and current motions (buoyancy comes on as per the time curve). Not that this does not shift the wave phase.

For dynamic solutions wave motions are always ramped up over a time interval equal to 2 times the wave period. For current only, the time interval is 8 secs. Note that this ramping is superimposed on any time curve load variation. The wave crest always starts at the model x/z origin for periodic waves. Solitary waves are not ramped and the wave crest is always offset half of the effective wavelength from the origin.

The **Irreg Wave Start** Time is used to define the start time in an irregular sea. This can be used to start the solution at the beginning of a particular section in a random seas i.e. select a section where high sea motion exists.

The following is a typical wave data file

```
25            Water depth
8             Wave height
9             Wave period
0             Wave angle
0             Current angle
3             No of current points
0      1.0    Current point
-20    0.8    Current point
-113.3 0.5    Current point
0             Wave spectrum type
0.4, 3.2, 500 Spectrum parameters
30            No of spectrum blocks
1             Random Seed
0.0, 0.0      Wave Start Time, Irreg Wave Start Time(s)
1, 9, 0.3, 62 WTh, Order, Damp, NPts
1             No of RAO Files (Max is 4)
URAO1_0 Model file extension
URAO2_0 Model file extension
URAO3_0 Model file extension
URAO4_0 Model file extension
```

Wave Phase

The wave phase is defined by the time step and increment. At t = 0 the wave crest is at the global model origin

-o-

6.2.6.1.1 Regular & Irregular Wave Loadings

Regular or irregular wave trains can be applied in a time history analysis by defining the wave height and the wave period and running the analysis over a suitable time interval.
If the a wave energy spectrum is defined then a deterministic time record can be obtained from the specific spectrum using the approach described below.

The section of the data in the **<modelname>.UWData'N'** that defines the type of wave train is shown below. See previous section for the full UWData description.

0	Wave spectrum type
0.65, 3.2, 500	Spectrum parameters W1, W2, NW
30	No of spectrum blocks
1	Random Seed
0.0, 0.0	Wave Start time (secs), Irreg Wave Start Time(s)

Wave Spectrum type - This defines the sea spectrum with the following options:
0 Regular wave
1 ISSC Spectrum
2 Pierson-Moskowtz PM (ISSC)
3 JONSWAP

The **ISSC** (International Towing Tank Conference) spectrum used by the program is independent of wave period and is the same as the PM for the case when Tp = 10secs.

The **Pierson-Moskowitz (PM)** spectrum is given by:

$$S_{PM}(\omega) = \frac{5}{16} \cdot H_S^2 \omega_p^4 \cdot \omega^{-5} \exp\left(-\frac{5}{4} \left(\frac{\omega}{\omega_p} \right)^{-4} \right)$$

where $\omega_p = 2\pi/T_p$ is the angular spectral peak frequency.

The **JONSWAP** spectrum is given by:

$$S_J(\omega) = A_\gamma S_{PM}(\omega)\ \gamma^{\exp\left(-0.5 \left(\frac{\omega - \omega_p}{\sigma\ \omega_p} \right)^2 \right)}$$

The spectrum is formulated as a modification of the Pierson-Moskowitz spectrum for a developing sea state in a fetch limited situation (parameter recommendations from DNV RP C205)

where
SPM(ω) = Pierson-Moskowitz spectrum
γ = non-dimensional peak shape parameter
s = spectral width parameter
σ = sa for $\omega = \omega$p
σ = sb for $\omega > \omega$p
Aγ = 1- 0.287 ln(γ) is a normalizing factor.

The following value for the peak shape parameter γ is applied:

Average values for the JONSWAP experiment data are sa = 0.07, sb = 0.09.

$$\gamma = 5 \quad \text{for} \quad T_P / \sqrt{H_S} \leq 3.6$$

$$\gamma = \exp(5.75 - 1.15 \frac{T_p}{\sqrt{H_s}}) \quad \text{for} \quad 3.6 < T_P / \sqrt{H_S} < 5$$

$$\gamma = 1 \quad \text{for} \quad 5 \leq T_P / \sqrt{H_S}$$

where T_P is in seconds and H_S is in metres.

W1 is the lower frequency cut-off and **W2** the upper cut-off point for the spectrum for the specific spectrum curve. The values of these two parameters are function of wave period and are very significant. It is very important that the **W1** and **W2** are correctly assigned. The following values represent good staring points:W1 = 3.3 / T W2 = 25 / T. A utility called seaway.exe located in the FS2000\System folder which plots spectrums can be used assess their suitability.

The **NW** parameter is the number of points used to define the spectrum curve. There should be no reason to change this value from the default of 500 (1000 max), it should always be high. It does not effect the solution time because it is only called once.

The equal energy approach is used to discretise the spectrum. Near zero values are removed (if W1 is set too low). The total area of the spectrum is evaluated (m_0) and a nominal equal area block based on the number of blocks is obtained (m_0 /**N**). This is effectively means that each block contains the same energy.

N is the **No of Spectrum Blocks** use to descretise the spectrum. Increasing this values increases the precision but at the expense of computational time (250 max). Because of the nature of random loading a pragmatic approach is to make slight changes to N or the random seed and look at the effect on the seastate (see below). Keep this value as low as practicable -default is 30.

The **Random Seed** is used to generate the random phase. The program will always generate the same sequence of random phases for the same seed number.

Wave Motions

Irregular random waves, representing a real sea state can be modeled as a summation of sinusoidal wave components. The random wave model is the linear wave model given by:

$$\eta_1(t) = \sum_{k=1}^{N} A_k \cos(k_k x - \omega_k t + \varepsilon_k)$$

$$A_k^2 = 2S(\omega_k)\Delta\omega_k$$

where εk are random phases uniformly distributed between 0 and 2π.

Velocities and accelerations are similarly obtained. Wheeler+ kinematic wave stretching is applied to the super-imposed wave motions. (z + d)' is used to replace (z + d) for use in the linear wave hyperbolic functions. It is used to reduce numerical inconsistencies observed during motion evaluations.

$(z + d)' = (d + \eta_i)*(z + d) / (d + \eta)$
$(z + d) = d*(z + d) / (d + \eta)$

η_i and η are the instantaneous water surface elevations

Seastate

The program will always generate a 3 hour time seastate. The **Wave Start** time is to specify at what time point the program should start the analysis. The surface time history is printed a file called **<modelname>.~WTrain** This file can be plotted in FS-Graph (or Excel). This enables the highest surface elevations to be identified so the maximum responses can be obtained by using the **Irreg Wave Start Time** and solution time interval to specify the requires analysis interval.

-o-

6.2.6.1.2 RAO - Respone Amplitude Operators

Vessel type motions can be incorporated into a time history analysis using RAOs (Response amplitude operators). RAO data defines amplitudes and associated phased as a function of wave frequency. The motions are defined in the 6 degrees of freedom.

RAO can be applied in both regular and irregular wave motions.

The irregular linear wave RAO model given by:

$$\eta_1(t) = \sum_{k=1}^{N} R_o A_k \cos(k_k x - \omega_k t + \varepsilon_k + \phi_o)$$

$$A_k^2 = 2S(\omega_k)\Delta\omega_k$$

where εk are random phases uniformly distributed between 0 and 2π.

R and ϕ are the RAO amplitude and phase operators. A positive RAO phase angle lags the wave.

The RAO prescribed motions are applied to a mode using Type 20 couples.

A Type 20 could be regarded as a Vessel element. The position and local orientation of the couple defines the position and orientation of the vessel using the standard FS2000 coordinate system conventions.

A more detailed description of RAOs and the use of theType 20 couple element is given in Section 5.

-o-

6.2.7 Heat Transfer

The solution of steady state (linear or non-linear) and transient heat transfer (linear or non-linear) solutions can be undertaken using FS2000's thermal solver.

The basic heat transfer equations solved are:

([K] + [H]){T} +[C]{Ṫ} = {Rh} + {Rq} + {RQ}

[K] is the Conductivity matrix

[H} is the Convection(Radiation) matrix (diagonal)

[C] is the Thermal Capacity matrix

{Rh} is the Convection heat flux vector

{Rq} is the Heat flux vector

{RQ} is the Heat generation vector

{Rhr} is the Radiation heat flux vector

{Ṫ} = dT/dt = 0 for steady State Heat Transfer

Ref: "Concepts and Applications of Finite Element Analysis", R.D. Cook, D.S.Malkus and M.E.Plesha, 3rd Ed, John Wiley & Sons.

The solution employs an implicit incremental approach similar to that used in structural solutions (see 6.2.6.1). The heat matrix is updated at every time step. If equilibrium iterations are specified this solution corresponds to a full Newton-Raphson (N-R) scheme. N-R equilibrium iterations can be undertaken at every time step or at a specified increment.

Structural 2-D, Axisymetric and 3-D solid elements can be used in thermal analysis. A structural model can be used in thermal analysis without any modification and can be use to run both thermal and structural solutions.

Model Properties

The following model properties are used to define the non-load case dependent thermal properties:

- K-Thermal Conductivity   Material Property: **RigidMod(G) -** Note that this property is not used by structural solid elements (no conflict with structural solutions).
- C-Thermal Capacity   Material Property: **Density** (not used for Steady State Heat transfer i.e. a non incremental solution)
- E Value This must be a non zero value

If K is defined a negative integer then K will be assumed to be temperature dependent and the value of K must be defined using an RC curve where K is the table entry. K is defined as a function of temperature using RC curves.

Load Case Data

The following thermal properties can be defined in a load case. Loading can also be define by reference to Load Case Combinations.
Using Load Case Combination enables loading affects to be varied using time curves.

Element Loads

Thermal load case conditions are defined by reference to element faces and are considered to be constant over the face of the element. The exception is heat generation which is based on element volume.

The following loading are define as structural FE Loads.

- Surface Convection.
- Specified Temperature - *Must be contained in the first load case of a combination if loading is specified using combinations.*
- Specific Heat flux.
- Adiabatic - Fully insulated boundary (default for all non specified element boundaries).
- Specified Element Heat Generation.

Nodal Loads

- Concentrated Heat Generation (Source/Sink) - Applied as a nodal Fx load.
- Nodal Temperature Definition - This is used to define non zero initial temperatures for transient heat transfer solutions. It is not used for Steady State Solutions (ignored).

Nodal Temperature Definition must be contained in the first load case of a combination if loading is specified using combinations. Note that initial temperature are not factored by the time curves.

Non-Incremental Solutions

A non-incremental solution is when the load case is applied in one step. Equilibrium iterations will be undertaken so non-linear effects will be included in this steady state solution (C is ignored). A steady state solution can also be undertaken using an incremental approach if the thermal capacity C is specified as zero.

Incremental Solutions

The time dependency of thermal loading in transient solutions is define using the DyNoFlex Solution Control form. The solution is not started from this, only the option parameters are saved.

A steady state solution is obtained if the thermal capacity C is specified as zero.

The load case loading can be factored using time curves to time vary the applied loading. If loading is specified using a load case combination then the load factor in the combination is used to reference a time curve which then factors that load case accordingly.

During the save process some secondary data associated with time curves is created. If the model is required to run in batch mode the command line to generate this data is **NLPARAM *'OptID'*** This would only require to be done if the model is opened from Archive.

Incremental Solution Option File

A saved Option file must have the same number as the the load case being solved. If a combination is used the Option file must have the same number as the combination number.

Only the following DyNoFlex option parameters are used:

Number of Time Steps	Defines the solution time scale using dt. Solution Time = Nstep* dt.
Time Step Increment (dt)	Vary to check accuracy Try dt < CL^2 / (4K) where L is the element length for transients solutions.
Time History Curve Case	Only Time Curve 1 contained in History Case can be used to vary the applied load when only one load case is solved. If History Case is blank the loading will be factored using the time step. If combinations are used up to 10 load cases can be combined each with a specified time curve.
Plot Node	This define a node that can be used to monitor the the solution. Time history plots can also be created is using FS-Graph's **Time**

	Solution Monitor plot option. Plot direction should always be 1.
List/Plot Interval	Interval for Plot Node output.
Number of time steps between equilibrium iterations	In heat transfer solutions rarely require equilibrium iterations. Set this to a high value to avoid and decrease solution time.
Maximum number of equilibrium iterations	To suit solution requirements
Convergence Tolerance	Increase if solution does not converge

Solution Output

A thermal solution creates a standard output case in which the nodal temperature are output as Z direction displacements.

To display temperature contour plots first make the Deflection Contours option active and then plot Z displacements from the **FE-Plots** menu.

An additional results file is also created, **<model>.UNT(n)**. This file contains the nodal temperature in the load case NT command format. This file can be used to undertake thermal expansion stress analysis or can be used to define initial temperatures for a transient solution base on a previous solution

-o-

7 Graphical User Interface (GUI) - Description

7.1 Graphical User Interface

This section describes the main operating features of the FS2000 Windows graphical user interface - GUI. The GUI enables virtually all of the analysis process to be undertaken in an interactive manner. All fundamental definition data can be input or generated in the GUI and all significant modeling parameters can be labeled or plotted.The interface also provides multiple windows, called Viewports that may be used to simultaneously view different areas of the model and/or different result cases.

Operation of the MOUSE

The **Left Mouse** button is used for:

- Single Selection- in this mode an item is selected by a single click of the left mouse button.
- Window Selection - In this mode the first corner of the window is located by clicking the left mouse button, the window is then sized and completed by pressing the left button again. If the shift key is held down prior to pressing the left button for the second time the window may be re-positioned using its current size.
- Entering node or element labels in input boxes. Double clicking the left mouse button in an input box where a node or element label is required will enter the previously queried node or element.

The **Right Mouse** button is used for:

- Cancelling an activity or operation indicated in the Activity Status Box (pressing the **[Esc]** key has the same effect).
- Menu - the right hand mouse menu gives quick access to some features and shows the 'hot keys' which can also be used to o access those features.

The **Mouse Wheel** is used for:

- Zooming the active viewport. Rotating the wheel moves the model in and out.
- Dynamic Viewing - If the mouse is pressed and the mouse is moved the model will rotate. If the **[Ctrl}** key is also pressed the model will translate. The toggle button can be used to switch the rotation and translation actions.

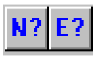
The GUI is divided into the following basic regions:

Menu Bar	Menu commands for program operation
Button Bar LHS	Below menu bar used for frequently used commands
Button Bar RHS	Below menu bar used for frequently used commands
Select Control	Floating options used to define how model entities are to be selected
View Control	Floating button pad used to control the model view orientation
General Toolbar	Quick access to various view settings
Pipework Toolbar	Quick access to piping analysis features
Results Toolbar	Quick access to result display features
ViewPorts	View ports showing the model
List Box	Bottom of the window, displays listed information

-o-

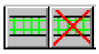
7.2 Button Bar LHS

The button bar contains access to the most frequently used controls. This described the left hand side. The buttons are described from left to right.

Query Buttons 

N? Node Query Used to list Node data to the List Box (use mouse to pick node). If two nodes are listed the distance between them will also be given. Nodes will be listed in their local coordinate system. Distances between nodes will be given in the active coordinate system. Hot key **N** also activates node Query.

E? Element Query Used to list Element and Couple data to the List Box (use mouse to pick element). If the Element Data (Full List Box) is visible it will be populated with more detailed element data. Hot key **E** also activates element Query.

Orthogonal Window Clip Planes  Clip planes are used to restrict visible portions of the model to those portions that are within defined planes. The window is positioned in the same manner as that for a zoom window. Successive clip planes may be defined.

The plane limits are defined by those nodes, which are within the window boundary. These planes are parallel to the principle axis (global coordinate system) and are defined using a mouse window. Since the planes are always defined parallel to the principle axis it is often more accurate to set the Viewport parallel to one of the principle axis of the model. Isometric clip can also be set uing the ISO Window Clip buttom.

The left Clip Button selects the clip planes. The right Clip Button cancels all clip planes.

Local Orientation Button  Used to display local element orientation

When shown graphically local orientation is represented by the equivalent of the webs of an I beam. The yellow edges indicate the fore end and the direction of the local Y axis. For finite elements a small orientaion triad is drawn - pink is the local z axis.

Labelling Buttons 

N Displays node labels

E Displays element labels

P Displays element geometric property code labels

M Displays element material property code labels

Le Displays entity labels - Label elemnt with specific analysis/design parametes - TASK dependent.

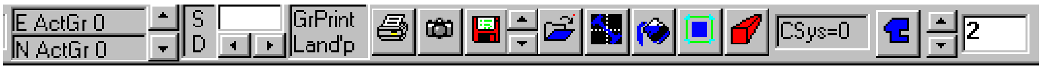
Activity Status 

Activity Status This text box indicates what action is required when user mouse activities are required.

-o-

7.3 Button Bar RHS

The button bar contains access to the most frequently used controls. This described the left hand side for the bar. The buttons are described from left to right.

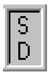


Active Group 

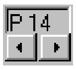
E ActGrp Changes and shows active element group

N ActGrp Changes and shows active node group

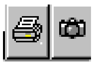
Use arrow buttons to scroll through groups

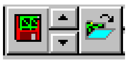
Combine Displays 

The **SD/AD** button is used to switch between Single Display and Add Display mode. This is used when selective plotting by group or property attribute is being undertaken using the **Selective Entity Plot**. If SD is active only one entity will be visible. If AD is active then the selected entity will be added to those already visible. The Display menu also provides the capability to add and additionally remove from current display.

Selective Entity Plot 

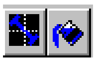
The Arrow buttons are used to increment selective plots when plotting by group (G), element geometric property(P) or element material property(M) attribute. To initiate the plot the RH mouse menu , the Hot key or the Element Display form must be used first, since the arrow buttons have no effect when the box is blank.

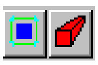
Printer and Camera Button  The printer button sends the current Viewport to the printer. The camera button saves the current Viewport to a BMP picture file. When saving BMP file is advantageous to set the screen to a white background using the Menu:View:Screen commnd.

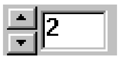
Save and Open Views 

The left and right buttons are used to save and open views from disc files.

The arrow buttons are used to scroll through the views stored on disc.

Virtual Beam View and **Element Solid Fill**  The VBV button change the appearance of beam elements from the default wire-line view to a ghost view which shows the outline of the element. The Solid Fill button is used to colour fill the ghost view to produce a solid view. For more information.

Shrink Elements View and  The shrink elements view changes the appearance of finite element meshes by shrinking the element boundary from the nodes. The Perspective View adds perpective to the view. The focal position is set when setting clip planes or explicitly from View:View Setting menu.

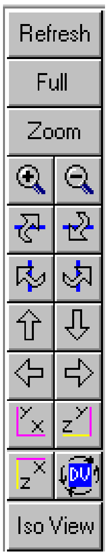
Load case/Results Scroll 

These arrow buttons at the RH end of the button bar are used to scroll through Load Cases or Results cases. They only become active when in the appropriate TASK. They can also be used to scroll through vibration mode shapes

-o-

7.4 View Control

This View Control button pad is used to control view orientations of the active View Port. Only the active Viewport is affected. The following describes the functions of the buttons.



Refresh — Re-draws the current view

Full — Centralises the current view entities of the model

Zoom window — Used to zoom into specific areas of the model using a mouse

- — Moves the view port away from the model
+ — Moves the view port towards the model

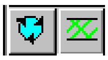
The 4 curly arrows are use to rotate vertically and horizontally around the model. The right hand mouse menu can also be used to do this using define angular increments. The button increment can be set from the Dynamic View Control.

The 4 straight arrows are used to move vertically and horizontally about the model.

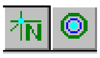
The XY, YZ and ZX buttons are used locate the view axis down the respective principal plane

DV — Makes the Dynamic View Control visible

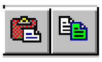
Iso View — Resets the view angle to its defaults and centralises the current view. The default can be set from the Dynamic View Control.



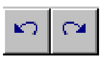
The left button toggles the rotation/translation options when using the mouse wheel (pressed) for dynamic viewing.
The right button, the **Isometric Window Clip Planes button** is similar to Orthogonal Clip Planes button but uses node locations within depth direction of the viewport to define visible entities not the principle axis.



The left button (**Nodes-Only by Elem Assoc**) makes nodes that are attached to elements that are visible always visible regardless of the node visibility option currently set.
The right button (**Set Centre of Rotation**) require the uses to select a node which will be used as the center of rotation when the model view is rotated. It will also move the model so that this node point is in the centre of the viewport.



The lower left and right buttons are used to capture and restore views from memory. Use this to temporarily save a view for future recall. Note that the view in memory is overwritten when a view is saved or recovered from file (see Button bar)



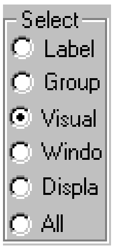
These button are the undo and redo buttons. These buttons are only visible when in the Model Definition TASK

.

-o-

7.5 SelectBy Control

This control is used to define how entities are to be selected in the various routines that require entity selection.



Label Select by label range

Group Select by Group attribute

Visual Select by individual mouse pick

Windo Select by mouse Window . For best results use when viewing parallel to one of the principal planes. For critical selections first use the window to select entities to be displayed. Re-draw the display and if the display is correct, select by that display. This approach gives the most reliable visible feedback that the selection is correct.

Displa Select by current display entities. This is not restricted to those entities that are visible on the screen but those that are in the display settings i.e. zooming does not change the Display. Use the Full View button to ensure that the full display is visible. Use this option with CAUTION

All Select all entities in the model. Use this option with CAUTION

The user is required to confirm all selection activities apart from when the Visual option is used.

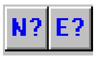
Useful Tip

When using the latter two options it is good practice to always reset the option to the one of the other options since they require additional user input. Hence, if the wrong method is current it can be quickly re-set.

-o-

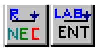
7.6 General Toolbar

This is a moveable toolbar which gives single click access to some of the most commonly used menu commands.

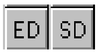


N? Node Query Used to list Node data to the List Box (use mouse to pick node). If two nodes are listed the distance between them will also be given. Nodes will be listed in their local coordinate system. Distances between nodes will be given in the active coordinate system. Hot key **N** also activates node Query.

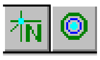
E? Element Query Used to list Element and Couple data to the List Box (use mouse to pick element). If the Element Data (Full List Box) is visible it will be populated with more detailed element data. Hot key **E** also activates element Query.



The left button makes the Model Display switches visible.
The right button makes the Entity Labeling switches visible.



The ED button makes the Element Display form visible.
The SD button makes the Couple Display form visible

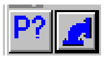


The left button (**Nodes-Only by Elem Assoc**) makes nodes that are attached to elements that are visible always visible regardless of the node visibility option currently set.
The right button (**Set Centre of Rotation**) require the uses to select a node which will be used as the center of rotation when the model view is rotated. It will also move the model so that this node point is in the centre of the viewport.

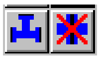
-o-

7.7 Pipework Toolbar

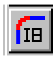
This is a moveable toolbar which gives single click access to some of the most commonly used menu commands generally associated with pipework analysis. Apart from the P? button the buttons are only visible in the Model Definition TASK



P? Pipe Query requires the user to select and element. When an elements is selected and if they exist, the pipework coefficients will be displayed in the List Box.
The right button gives quick access to the Pipe Bends command.



The left button gives quick access to the Pipe Tee/Connection command.
The right button removes the pipework coefficients from the element.

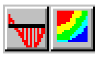


This button gives quick access to the Insert Bend command.

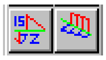
-o-

7.8 Results Toolbar

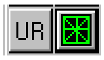
This is a moveable toolbar which gives single click access to some of the most commonly used menu commands used in the Results TASK



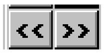
The left button gives quick access to the Line Plot Setting form used for beam element plots.
The right button give quick access to the Contour Setting form used for FE plots



The left button switches the magnitude labels on or off.
The right button is a toggle for plotting major, minor or both axis beam action plots.



The left button gives quick access to the Unity Ratio (UR) form used for beam element plots.
The right button removes the result plot from the current view

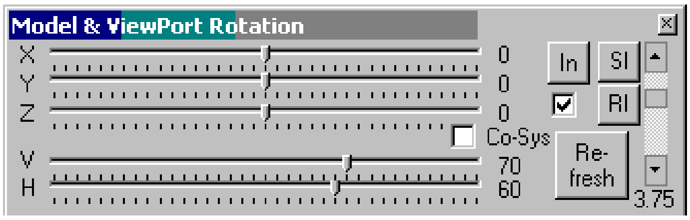


These two button are used to change the scaling factors used in displacement and stress/load action plots. They also change the upper range limit for UR plots.

-o-

7.9 Dynamic View Control

This control is used to control the view angle of the Viewport and the Visual orientation of the model. The control is made visible using the Dynamic Model View button on the ViewControl.



When the sliders are moved an outline envelope of the model in global coordinates moves to reflect the current orientation of the model set by the slider position.

The **X, Y & Z Slider** controls are used to change the orientation of the model.

The **V & H Slider** controls are used to change the orientation of the Viewport is a similar way to the ViewControl Pad Buttons.

The **In** initialisation button resets **X, Y & Z Slider** the to the Zero position.

The **Option** box below the **In** button disables the model orientation transform i.e. the effect of the X, Y & Z sliders.

The **SI** button sets the default orientation for the Iso View button on the View Control Pad.

The **RI** button initialises the default orientation for the Iso View button on the View Control Pad.

The **Co-Sys** option box selects the type of orientation triad to be displayed when the model outline is moved. If the is option is checked the Active coordinate system triad will be displayed otherwise it will be the global system.

The **Ref** button is used to re-draw the view in a similar way to the Refresh button on the View Control Pad.

The right end slider sets the rotation increments used by the Viewport rotation buttons on the View Control Pad and the Mouse Dynamic view.

-o-

8 GUI Menus & Menu Commands

8.1 Menus and Menu Commands

TASK Orientated Menu Commands

A considerable number of menu commands are available for the different stages of analysis. To provide logical access to these commands the analysis process has been divided into separate TASKS i.e. distinct stages of analysis. Some menu commands are global others are TASK related.

Global Menu Commands

Right Hand Mouse Menu

Menu:File

Menu:View

Menu:Data

Menu:Display

Menu:Group

Menu:TASK

Menu:Comb

Menu:Window

TASK Related Commands

Model Definition

Load Definition

Design Parameters

Analysis

Output/Results

-o-

8.2 Right Hand Mouse Menu

If the right hand mouse button is pressed the following menu will appear.

'Hot keys' The commands in this menu can also be activated without the menu being visible by pressing the short cut key shown in the menu e.g. N to label and list a single node.

Label Node	N	This will label the node and list its attributes in the List Box
Label Element	E	This will label the element and lists its attributes in the List Box. Use the Element Data (View Full List Box) in the Data menu to see more element related data.
Label Couple	S	This will label the spring/couple and lists its attributes in the List Box
Show Geom	G	This will restrict the displayed elements to those element with the particular geometric property code. Re-entering with zero will re-set to all elements. -1 will show elements not assigned to any group. -1 can only be set using the Element Display command in the Displ Menu.
Show Mat	M	This will restrict the displayed elements to those element with the particular material property code. Re-entering with zero will re-set to all elements.
Show GRP	G	This will restrict the display to all entities to those entities with that particular group attribute. Re-entering with zero will re-set to all entities
Active Group	A	Sets both the node and the element active group
Active Coord	C	Sets the active co-ordinate system
ViewRot Horiz	H	This will rotate the viewport horizontally by a specific amount in degrees.
ViewRot Vert	V	This will rotate the viewport vertically by a specific amount in degrees.

-o-

8.3 Menu:File

The menu options and commands functions of the file menu are TASK dependent.

New	Primary TASK	Initiates a new model by defining a unique model name in an existing directory
	Load Defn TASK	Initialises all current loads in memory
Open	Primary TASK	Define the default model by selecting an existing model
	Primary TASK	Retrieves a model from an Archive file
	Load Defn TASK	Opens and existing Load Case
	Results TASK	Open a Results Case
Merge Loads	Load Defn TASK	Merges load cases into current memory
Save		Saves current data related to active TASK
Save As	Primary TASK	Copies default model.
Save Loads & Comment		Save Loads and comments 'added' loads
	Model Defn TASK	Copies default model
Save(Archive)	Primary TASK	Merges all definition data to a Model Archive File
Purge Results		Deletes all results related files
Delete	Primary TASK	Deletes the default model
	Load Defn TASK	Deletes a selected Load Case
Copy Sub Mod	Model Defn TASK	Copies sub-models (by group association) to Model Clipboard
Copy Entities	Model Defn TASK	Copies node and elements to Model Clipboard
Paste	Model Defn TASK	Appends sub-models to current model
Interpret File	Model Defn TASK	Interprets a command file
	Load Defn TASK	Interprets a command file
InterpComd	Model Defn TASK	Interprets a single command
	Load Defn TASK	Interprets a single command
Printer Settings		Sets printer setting for report printing
Print(Grahics)		Sets printer settings for Graphics printing
Run Editor		Starts FS-Edit a general purpose text editor
Run Graph....		Starts FS- Graph
Run Appl.....		Starts a user defined application eg Windows calculator

EXIT	Closes FS2000

-o-

New command (Menu:File)

Clears the all current definition data (not saved data) so data may be defined from scratch.

In the **Primary TASK** the New model creates the model workspace. The workspace is simply the Windows folder where the various model files that are used by FS2000 are located. To create the workspace the user is required to select an existing Windows folder and define a new model name. The model name must be unique for the selected model directory.

In the **Loads Definition TASK** the command will delete all current loads in the GUI (imediate memory). If these loads are not save to a Load Case then they will be lost. Note that this command will not delete any existing load cases i.e. load cases that have been previously saved.

-o-

Open command (Menu:File)

Primary TASK

This is used to select the current default model from existing Standard models or the retrieve a model from an Archive file. A Standard model is a model for which a workspace currently exists. An Archive files is a single file that contains all definition data relating to the model.

Selecting Standard Model or Archive Models (MOD Files) from the List of File Types selects the appropriate model option.

Standard models are opened by selecting files that have an XYZ file extension.

Archive models are opened by selecting files that have a MOD file extension.

Load Definition TASK

This is used to open an existing load case i.e. retrieve load data from an existing Load Case into current memory.

Unless the load case was previously saved in the FS2000 it will not appear in the current load case list since it will not be in the FS2000 registry. This would be the case if the load case file were created in a text editor. In such cases the load case can be opened by entering only the load case number.

Results TASK

This is used to open an existing Results Case. The result case will be loaded into the current Viewport. If more than one Viewport is used, then more than one result case may be loaded. The title bar of the Viewport will show the results case description currently loaded.

-o-

Merge Loads(Menu:File)

This command will merge an existing load case with loads currently in memory. The command may be activated more than once to merge multiple cases.

A load factor is required to be entered when merging. The default value is 1. When load cases are merged using Load Combinations the load factors in the combination are used

The following merge options are available:

By Selection

Load cases are selected individually from the load case list.

By Combination

Load cases are merged using an existing Load Case Combination

By Combination - Save and Comment

With this option the load cases are merged and then saved. When the load case is saved comments lines containing the load case number, title and load factors of the component load cases are inserted in the recipient load case. To use this option the load case must be saved first so as to define a recipient load case number and title.

Warning

This method is a static load case merging process. This means that if at a future date a component load case is updated then the merging process will have to be repeated to ensure that the merged load case is updated. Use Pre-Processing to provide dynamic load case merging.

-o-

Save command (Menu:File)

The save command is a TASK dependent command and is used to save the state of the current definition data appropriate to the active TASK.

Note: The Save (Archive) in the Primary TASK is used to save ALL definition data to a single MOD file.

Model Definition TASK

Saves all geometric type model data relating to model currently in memory. On selection the user may also define reference data e.g. description etc. relating to the model. This data will be echoed on selected output listings.

Model Backup - When a model is saved the previous model definition file (.MDL) is copied to (.MLA) as a backup. If an erroneous save was done the model may be recovered to its previous state by interpreting the .MLA backup file.

Load Definition TASK

Saves all load data currently in memory to a specific Load Case. On selection a list box will appear showing all existing Load Cases. To save load data to a load case simply enter the load case number and its description and click the OK button.

Load Case Backup

When a load case is being saved, and if a load case already exists with the same name, then the original will be copied to a backup file called <modelname>.BAK.L(n). Thus if an erroneous save was done the load case may be recovered to its previous state by interpreting the load case backup file.

Design Parameter TASK

Saves model definition data related to the option structural design code checkers. Note that current data will always be opened when the TASK is made active.

Member design data is written to a <modelname>.ELN file.

Tubular Joint data is written to a <modelname>.ECI

-o-

Save As command (Menu:File)

Primary TASK & Model Definition TASK

Copies the default model. On selection the user is required to select an existing directory and define a new model name.

The model name must be unique for the selected model directory.

-o-

Save Loads and Add Comment

This command enables comment lines to be added to the load definition file. Comment lines are appended to the load file before the 'added loads' are appended. 'Added loads' are those load added to the model since the last save. Each time a comment line is added the loads are saved.

Note that existing loads and comments always retain their current position in the load case file.

For comment lines to be added the load case must be saved using the standard Save Loads command or be an existing load case that has been opened. On selection a text box for the comment line will appear. When the OK button is pressed the load case will be saved.

Comment lines may be added, deleted or modified to any load case using a text editor. Note that the REFORMAT command must not be present in a commented load case file. Similarly if the REFORMAT command is added a commented load file all the comments will be removed.

Only 'added loads' can be sorted in commented load file.

When the commented load cases are saved using the usual Save Loads command an amendment comment will be added. These will not be included in the Formatted Load Definition File unless the Amend characters are removed. They can be used for the placement of comments i.e. add them when saving and then edit them later in an editor.

-o-

Save(Archive) command (Mneu:File)

Primary TASK command

Saving to Archive merges **All** model definition data into a single file. Archive files have the file extension .MOD.

If an archive file exists in the model directory it will not be deleted when the model is deleted.

It should also used when the use of a model is completed and disc space is required. In such cases the model is first archived and then deleted using the Delete command of the File menu.

To archive a model the user is required to select an existing folder, usually a backup (network) drive for the archive file. Archive files can be saved under a different name to that of the model but when an Archive file is opened the original model name will be adopted.

Opening Archived Models

Archive models are opened using the File menu Open command. By electing Archive Models (MOD Files) from the List of File Types the MOD file can be selected.

User Defined Data

If a user creates files for special purposes e.g. files containing model definition commands then these file will always be archived if the file has the same name as the model and the extension starts with the letter u(U).

Files with two extensions will not be archived. ie

<Filename>.UM_Test   will be archived

<Filename>.UM.Test   will NOT be archived

-o-

Purge Results command (Menu:File)

This command is used to delete all existing result cases. It would be generally used after saving the model in the model definition TASK. This would ensure that all previous result case data relating to earlier conditions of the model are eliminated.

By default results are always purged when the model is saved. An option to cancel the results purge is available when saving the model in the Model Definition TASK.

-o-

Delete command (Menu:File)

Primary TASK

Delete the current default model. Use with caution.
The delete command does not delete the archive file (<modelname>.MOD file).
Use this command to retrieve disc space by removing models that have been archived.

Load Definition TASK

Deletes an existing Load Case selected from the Load Case List.
Current loads in memory are not affected.

-o-

Copy Sub Model Command (Menu:File)

Model Definition TASK

This command is used to copy selected elements and attached nodes to the Model Clipboard. Its main use is to extract sub-Models from the current model.

When the command is selected all elements associated with the Active Group will be copied to the Clipboard. The node and elements numbering will be re-sequenced and all label gaps removed. Property code data associated with the elements will only be copied if the option to copy property codes is accepted.

If the property code data is not copied to the Model Clipboard ensure that the option not to read the property code data is selected when pasting from the Clipboard otherwise the element's property codes may be incorrectly assigned.

When model data is assigned to the Clipboard any existing model data in on the Clipboard will be overwritten.

Model data in the Clipboard can be pasted to any current model from within the Model Definition Task.

The Copy Entities undertake a similar procedure.

The **Model Clipboard** is a text file that contains model definition commands. The file is called buffer.mdl and can be found in the user's FS2000 directory.

-o-

Copy Entities Command (Menu:File)

Model Definition TASK

This command is used to copy selected node and elements to the Model Clipboard. This differs from Copy Sub Model command in that the node and element labels are not re-numbered and **all** property codes in the model are copied if the option to copy property codes is accepted.

If the property code data is not copied to the Model Clipboard ensure that the option not to read the property code data is selected when pasting from the Clipboard otherwise the element's property codes will be incorrectly assigned.

When the command is selected all nodes and elements associated with the active groups will be copied to the Clipboard.

Warning Note: Nodes attached to elements will only be copied if they are in the active group.

When model data is assigned to the Clipboard any existing model data in on the Clipboard will be overwritten.

Model data in the Clipboard can be pasted to any current model from within the Model Definition Task.

The **Model Clipboard** is a text file that contains model definition commands. The file is called buffer.mdl and can be found in the FS2000 directory.

-o-

Paste Command (Menu:File)

Model Definition TASK

The **PASTE** command is used to append/merge model data from the Model Clipboard or any other file containing valid model command line instructions to the current model. If a model is in the Clipboard an option to retrieve this will be given, alternatively a command file may be selected.

Node and elements labels in the appended model will be changed so as to follow and be sequential with those of the current model.

Property code data associated with the appended models will be similarly appended. If the option not to read Property data is selected element property codes will not be re-assigned.

The appended model will be assigned to the current active groups. An option to define the active groups will be given. Groups assignment is useful for identification e.g. node translation etc.

-o-

Interpret File command (Menu:File)

Model Definition TASK

Reads and interprets command line instruction files relating to model definition. This process will overwrite existing model data i.e. data with the same label or code numbers.

To append data to the model use the Paste command.

It is recommended the text file containing the instructions should adopt the file extension .UM? where ? is an ID character(s). This will ensure that file is archive with the model.

Model definition files (.MDL) may also be interpreted. If an MDL file is to be interpreted an option to **re-initialise** the basic model will be given. **Re-initialisation** deletes all data in current memory and replaces it with that defined in the MDL file.

Re-initialisation should only be done if the interpreted MDL is solely required to define the model.

Load Definition TASK

Reads and interprets command line instruction files relating to load definition.

It is recommended the text file containing the interactions should adopt the file extension .UL? where ? is an ID character. This will ensure that file is archive with the model.

Load definition files (.L?) from other models may be interpreted.

Command Line Instructions

A full description of all Command Line Instructions is given in this Help file. These commands are not indexed within the Help. This enables the whole of the commands to be printed using the help print button since it is in the printed form that they are most useful.

-o-

InterpCommand command (Menu:File)

The interpreter is made visible from the Menu:File/Interpret Command command.

Model Definition TASK - Interprets a command line instruction relating to model definition.

Load Definition TASK - Interprets command line instruction relating to load definition.

The most common use of this is when copying and pasting commands from an exiting command file (mdl or um file). Multiple lines can be copied into the box

When the Return key is pressed any valid commands in the box will be interpreted

Command Line Instructions

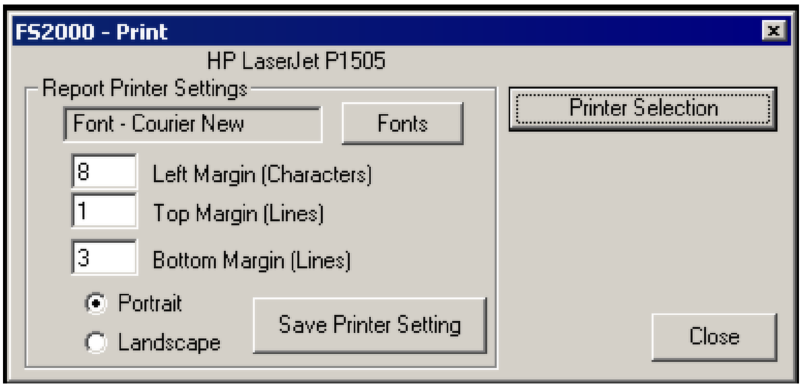
A full description of all Command Line Instructions is given in this Help file. These commands are not indexed within the Help. This enables the whole of the commands to be printed using the help print button since it is in the printed form that they are most useful.

-o-

Printer Settings command (Menu:File)

Global TASK - Used for to set the printer options

On selection this command will make the Report Printer Settings dialogue box is made visible.



It is used to define the output format for printed listings. A printed listing is a formatted text output file produce by the various output modules.

The **Font** command button allows the selection of the default font. Since all output is in tabbed columns only non-proportional (fixed width) fonts can be selected. If changing printers always check that the current font is available.

If available the Line Printer or Consolas fonts will give good results. These are compact fonts all allow a large **Left Margin** (20) to be used

Warning - If proportional fonts are used the tabbed columns will not be aligned.

To ensure the whole line is printed when using large fonts the **Left Margin** may have to be reduced.

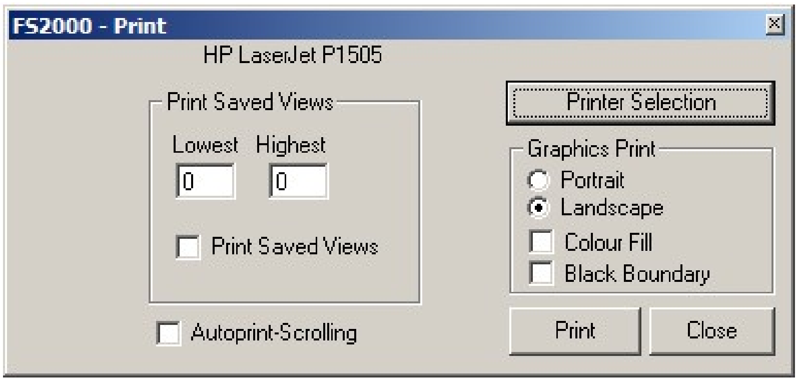
The **Printer Selection** command button allows the user to select the output printer (default printer) and corresponding set up.

The **Save Printer Settings** button only saves the setting related to the Report Printing Settings.

-o-

Print Graphics command (Menu:File)

On selection this command will make the Graphics Printer Settings dialogue box is made visible.



The **Portrait/Landscape** option buttons are used only for graphics printing. This option can also be set by a button the main FS2000 tool bar.

The **Colour Fill (Beam)** is used to obtain colour plots. If this is not checked the plot will be in black and white.

The **Print Saved Views** options enable the successive printing of Saved Views. If the **Print Saved Views** option is checked all view within the range will be printed when the **Print** button is clicked.

The **Autoprint-Scrolling** if active will print each loaded view when the Save View Scroll button is used.

-o-

Run Editer... command (Menu:File)

Global TASK

Runs FS Edit.

This is a general purpose text editor. It primary function is to provide text editing capabilities for FS2000 related model files.

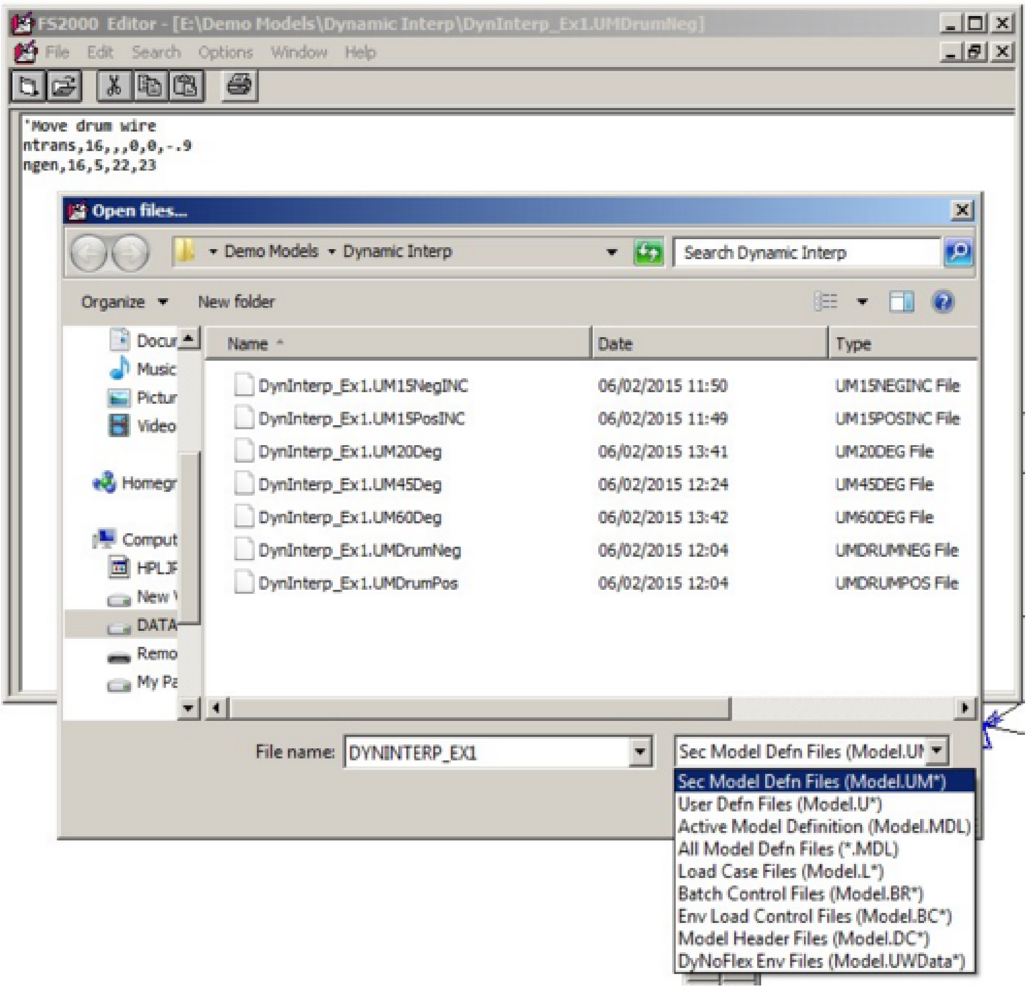
The Open dialogue box list the most commonly edited files.

Note that if user files are required to be archived with the model they must have U prefix.

User crreated files should be limited to:

<modelname>.UM* are secondary model definition files and can be included in in formatted definition data lists.

<modelname>.U* are used for non specific files that will be archived.



-o-

Run Appl.... command (Menu:File)

Global TASK

Used to start any used defined application e.g. the Windows calculator. (c:\windows\calc.exe)

-o-

8.4 Menu:View

This menu is used to select what model attributes are to be displayed and set the various options and parameters related to the model display.

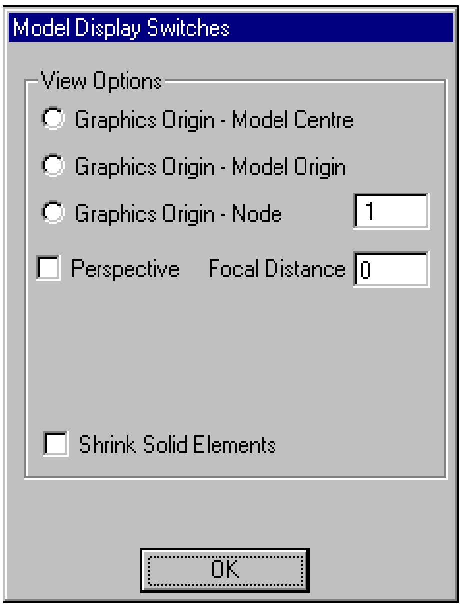
The menu options are:

Initialise	Global TASK	Reset all view settings
View Settings	Global TASK	Sets view parameters, Perspective etc
Model Display Switches	Global TASK	Defines model attributes to be displayed
Loading Display Switches	Load Defn TASK	Defines types of loading to be displayed
Entity Label Switches	Global TASK	Labels elements with selected entity- TASK dependent.
Open View-Update All Settings	Global TASK	If activated the current display settings (labels etc) will be updated to those of the view being loaded.
Dimension	Global TASK	Add dimensions to a model
Caption Prompt	Global TASK	Caption Toggle switch for graphic if active will activate a prompt for a title (Caption) each time a graphics plot is printed.
Screen Label Fonts	Global TASK	Select font type and size for model labels
Printer Label Fonts	Global Task	Select font type and size for model labels (graphics print)
Screen(Black/White)	Global Task	Sets a white background. Useful when creating bitmaps.
View/Select Controls	Global TASK	Hides/shows the controls on the LHS of the screen
Toolbars	Global TASK	Sub menu to make the the following toolbar visible/invisible General Toolbar, Pipework Toolbar and Results Toolbar

-o-

View Settings

This form is used to define View settings. The graphics origin is Viewport dependent the other parameters are applied globally.



The graphics origin is the view origin, the point at which the model rotates when the view angle is changed and is the reference point for perspective views. By default it is set to the model origin. It can be re-define by:

Model Centre - Geometric centre of the visible nodes of the model

Model Origin - Global origin of the model

Node Reference - Based on the global coordinates of a selected node.

When clip planes are used the graphics origin is set to the geometric centre of the clipped model. It is only reset when a clip plane is re-applied or one of the above is options is used.

If **Perspective** is active the **Focal Distance** is used to set the distance from the graphic origin to the view position. Decreasing this value will increase perspective.

Shrink Solid Elements - This option will shrink the boundaries of solid elements thus making elements connectivity errors to be more obvious.

Graphical Representation of Elements

For beam or pipe elements three types of graphic representations are available. **Virtual Beam View** and **Fill Solid Elements** options set the view type. Buttons on the Button bar are used to set the options.

· Wire Line View - Default View Elements are represented by lines using symbols for features such as releases etc.

· Ghost View - Same as Wire Line but also includes the true outline shape of the beam.

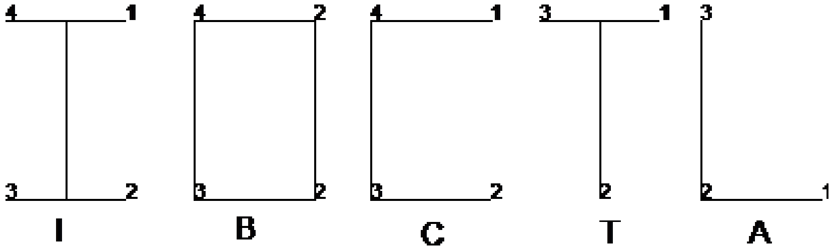
· Solid View-Solid virtual view (Ghost view with solid fill and hidden line removal)

For finite elements either Wire Line or Solid Views can be used.

The outline shape of beams is defined from the GT and stress point properties of the element Geometric Property Code. These are shown below.

Additional graphic Offset properties can be used to draw the beam offset in the local y or z directions. The main purpose of this is to include offset such as stringer beam so that a true virtual view may be produced when the solid view is active. Note: If structural Offsets are defined they will be included in the virtual view but only when the solid view is active.

GT Designation and Stress/Section View Properties

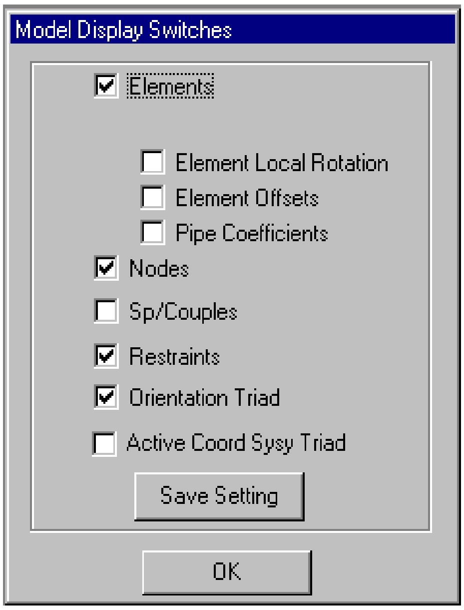


Properties with an OD defined will be drawn as tubes.

-o-

Model Display

This form is used to define the model attributes to be displayed. The options in the box are Viewport dependent.



Moment release symbols are small circles near to the node end of the element.

The **Local Rotation** (Orientation) of beam elements is represented by a symbolic plate which represents the major axis of the beam (similar to the web of an I-beam). Virtual Beam Views can also be used to show local rotation.

Element Offsets are shown drawn to scale. When active beam and shell plots will include and defined offsets. When then Virtual Beam View is active the effect of beam offsets are always included in the plate Beam force actions will always be drawn on the offset element.

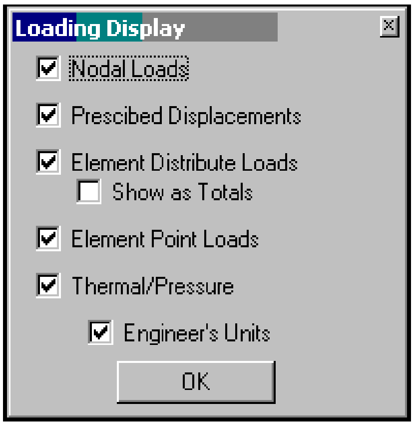
If **Pipe Coefficients** are assigned to elements the elements will be drawn in a different colour.

The **Save Setting** button will maintain the same settings whenever FS2000 GUI is opened with any model.

-o-

Loading Display

This form is used to define the type of loading to be displayed in the current view. The Entity Label buuton can be used to label the magnitude of the loads.



-o-

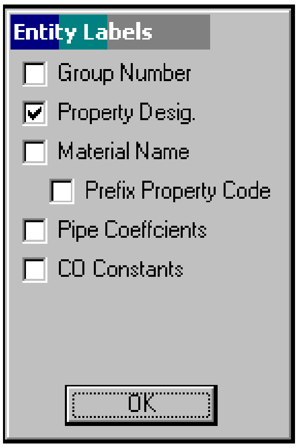
Entity Labeling

This form is used to define the element entity labels to be applied to the current view. The options in the box are TASK dependent.

The ability to plot the various element parameters enables most element property data to be listed and printed in a graphical context thereby providing an invaluable QA capability.

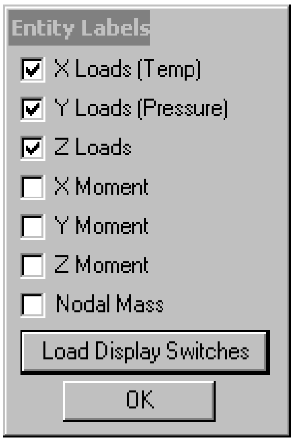
The following shows TASK dependent the entity label options

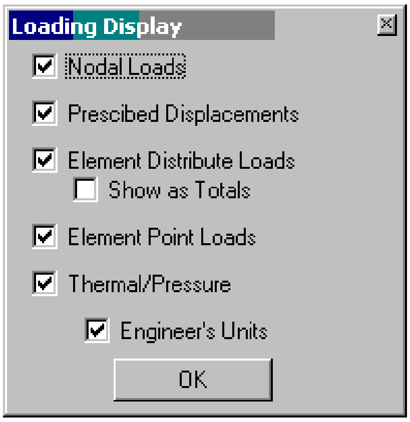
Primary, Model Definition & Results TASK



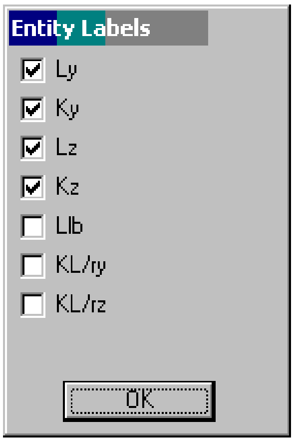
Load Definition TASK

The **Loading Display Switches** button enables selection of the type of load to be displayed





Design Definition TASK



-o-

Dimensions (Menu:View)

This command loads the Add Dimensions form. When this form is visible, dimensions may be added between selected nodes of the model. Only when the form is visible will dimensions be drawn. Closing the form will erase all dimensions (unless stored).

Dimensions are not saved with views - use **S** and **R** buttons.

When the command is selected the following form will become visible.

The **Dimension** option buttons are used to select the dimension co-ordinate to be added.

The dimensions are added by using the Node Query button to select the two nodes.

The **Active** check box can be used to disable the Add Dimension function i.e. allow the Node Query button to be used without adding dimensions.

The side of the model on which the dimensions are added depends upon the sequence in which the nodes are selected.

The dimensions are added in 3-D space. By default the location of the dimension line is 20% of the line length but can be positioned exactly by defining a co-ordinate position in the appropriate dimension box. The co-ordinate is relative to the first node picked.

The **Undo Last** button will remove the last dimension added (last in list). Re-Draw is required to see the effect. **Remove All** - removes all dimensions.

The **Align** to last check box is used to align the next dimension to be added with the previous dimension added.

The **Link** option enables dimensions to be added by clicking from node to mode. The default is to pick both nodes for the dimension to be added.

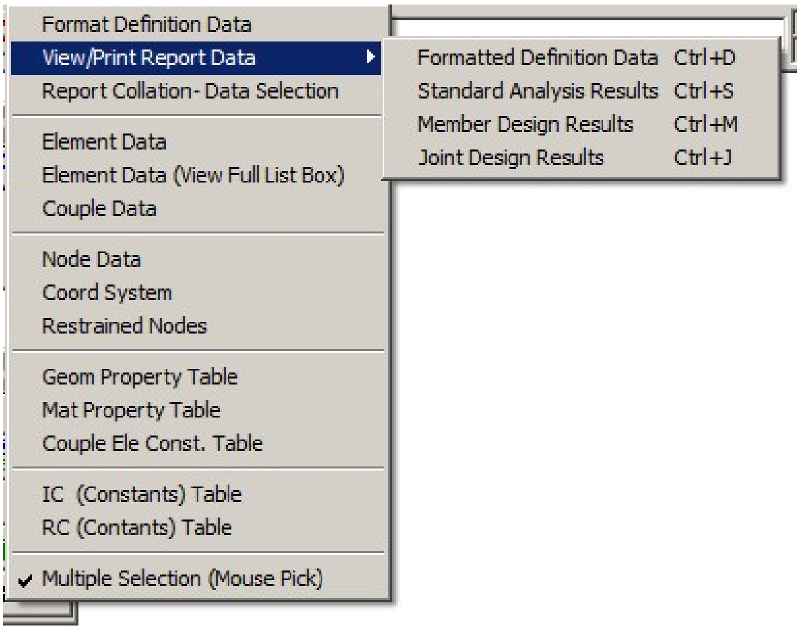
The **S** and **R** buttons are used to store and retrieve the current dimensions.

-o-

8.5 Menu:Data

This menu has commands which are used to view model definition data and formatted definition and results data. It is a Global TASK menu.

In the Model Definition TASK the lists can be used to edit and modify model attributes. The property Table lists are especially useful for assigning properties to elements.

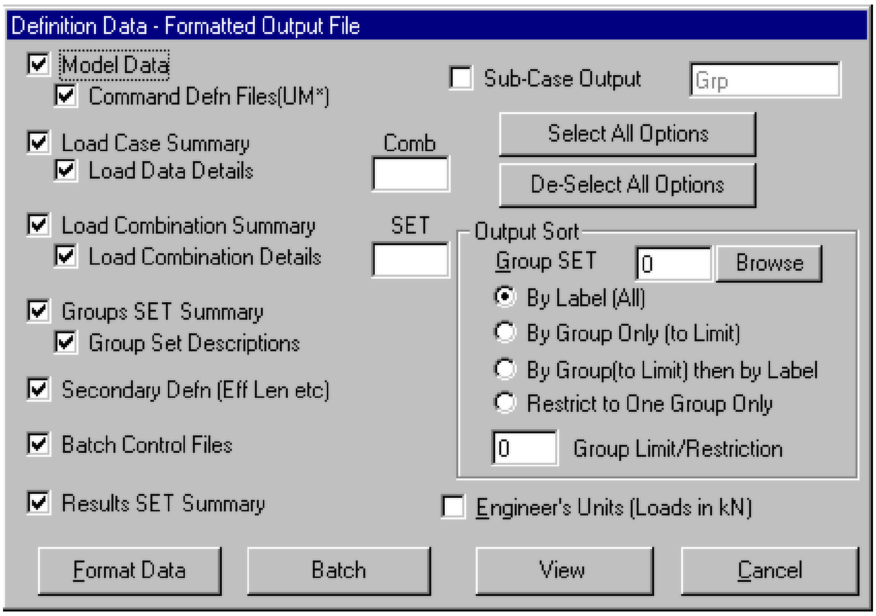


Formatted Definition Data	This command makes visible the Definition Data Formatted Output form.
View/Print Report Data	Selection from a sub-menu enables formatted definition & result data file to be viewed/printed.
Report Collation-Data Sel	Makes visible the Report form which is used to collate formatted data files.
Element Data (View Full)	The opens a list box shows more detailed data from an Element Query
Geom Prop Table	Makes the property table list visible
Coord System	Allows the active co-ordinate system to be changed
Multi Selection (Mou-pick.	If this is disabled the first node (by label) will be selected in an ambiguous

-o-

Definition Data Format

The Format Definition Data command of the Data menu makes the following form visible. It is used to create formatted data files which contains selected definition data. The files created (<Model>.MTM) can be viewed or printed at any time using the View/Print Data command of the Data menu.



The check boxes on the left-hand side of the form are used to select which type of definition data is to be included in the file. Generally for a final report all data will be included.

The **Sub-Case Output** option enables multiple files to be created. The term Sub-Case is used as this term is used in the Results modules which also include this capability. This may be used if it is desirable to create separate data files for the different output category options and various sort options available. If the option is checked then the file created will have the file name <Model>.Grp.MTM where Grp is the name entered in the Sub-Case description box. DC file will not be include with MTM Sub-Cases unless the Sub-Case name is 'Rep'.

The **Comb** box adjacent to the Load Data Details can be used to restrict the load cases to be included in the formatted output. If a Load Case Combination number is entered in this box then only Load Cases in that combination will be included.

The **SET** box adjacent to the Load Combination Details can be used to restrict the load case combinations to be included in the formatted output. If a Results SET number is entered in this box then only Load Combinations in that SET will be included.

If the **Engineers Units** check box is activated the units for the Load data will be formatted in kN for forces. If not checked the output will be in pure S.I. exponent format.

Output Sort

The **Group SET** box is used to define the group SET to be loaded. If a SET is loaded then all node and element labels will be accompanied by their respective group attribute. If this field is left blank or contains the number of a non existent group then only the basic node and element numbers will be used for reference in the lists.

The **By Label (All)** option will output all entities(nodes and elements) in ascending label order.

The **By Group Only (to Limit)** option will output entities in ascending Group order. Entities not assigned to groups or entities assigned to Groups greater than defined by the **Group Limit/Restriction** box will not be output. This is a restricted process option.

The **By Group(to Limit) then Label option** will output entities in ascending Group order. Entities not assigned to groups or entities assigned to Groups greater than defined by the **Group Limit/Restriction** will be output in label order following the sorted groups. All data is processed with this option.

The **Restrict to One Group** option is used to restrict entities to only those entities with the same group number as defined by the **Group Limit/Restriction** box (zero value indicates that all data will be shown). This is a restricted process option.

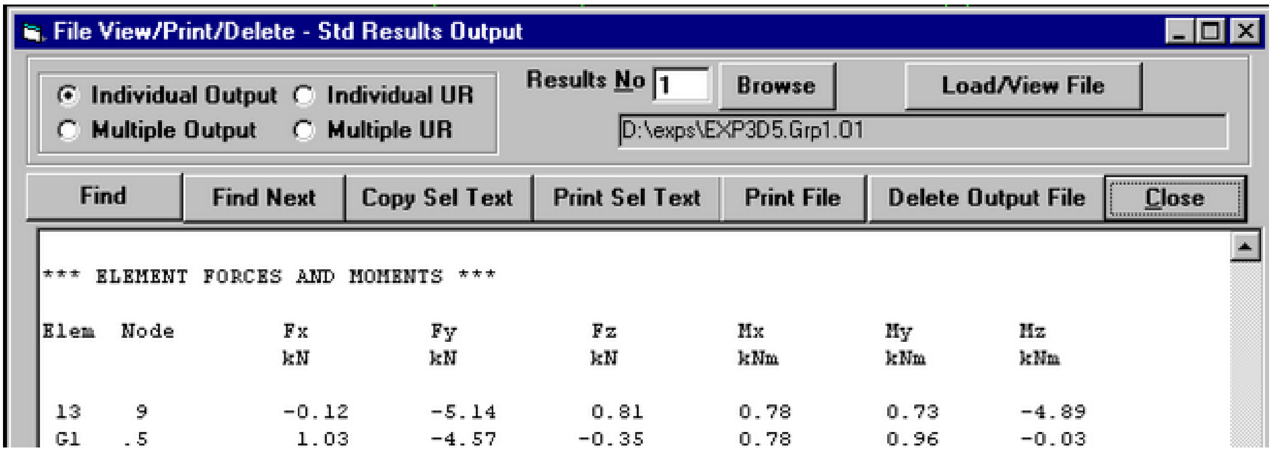
-o-

Output View/Print

The View/Print Report Data sub-menu commands of the Data menu make the following form visible. It is used to view, print or delete existing formatted files. The type of file that can be accessed is dependent upon the sub-menu command selected.

For the batch printing of multiple formatted report files see Report Collation

The top section of the form shown below will change depending upon the type of file being viewed



The option buttons are used to select the type of output to be viewed.

The **Results No** box is used to select the results case. The browse button may be used to select the case from a list of existing cases that match the results case type.

The **View/View File** button is used to load the selected results file into the viewer.

The **Search** button is used to search the file for the occurrence of defined characters. The search is not case sensitive. The search will always start at the top of the file.

The **Cont Search** button is used to continue the current search to the next occurrence or the bottom of the file if no match is found.

The **Copy Sel Text** button is used to copy selected text to the Windows Clipboard. Use this to paste the text to other Windows applications.

The **Delete Output File** button will delete the formatted file displayed in the **Results No** box. This will not delete the Processed Results Case.

The **Print** button will print the current Results file displayed in the **Results No** box. See Report Collation for the batch printing of report files.

-o-

Collate Reports - Output Selection (Menu:Data)

When the Report Collation- Data Selection command of the Data menu is selected the Collate Report-Output Selection form becomes visible.

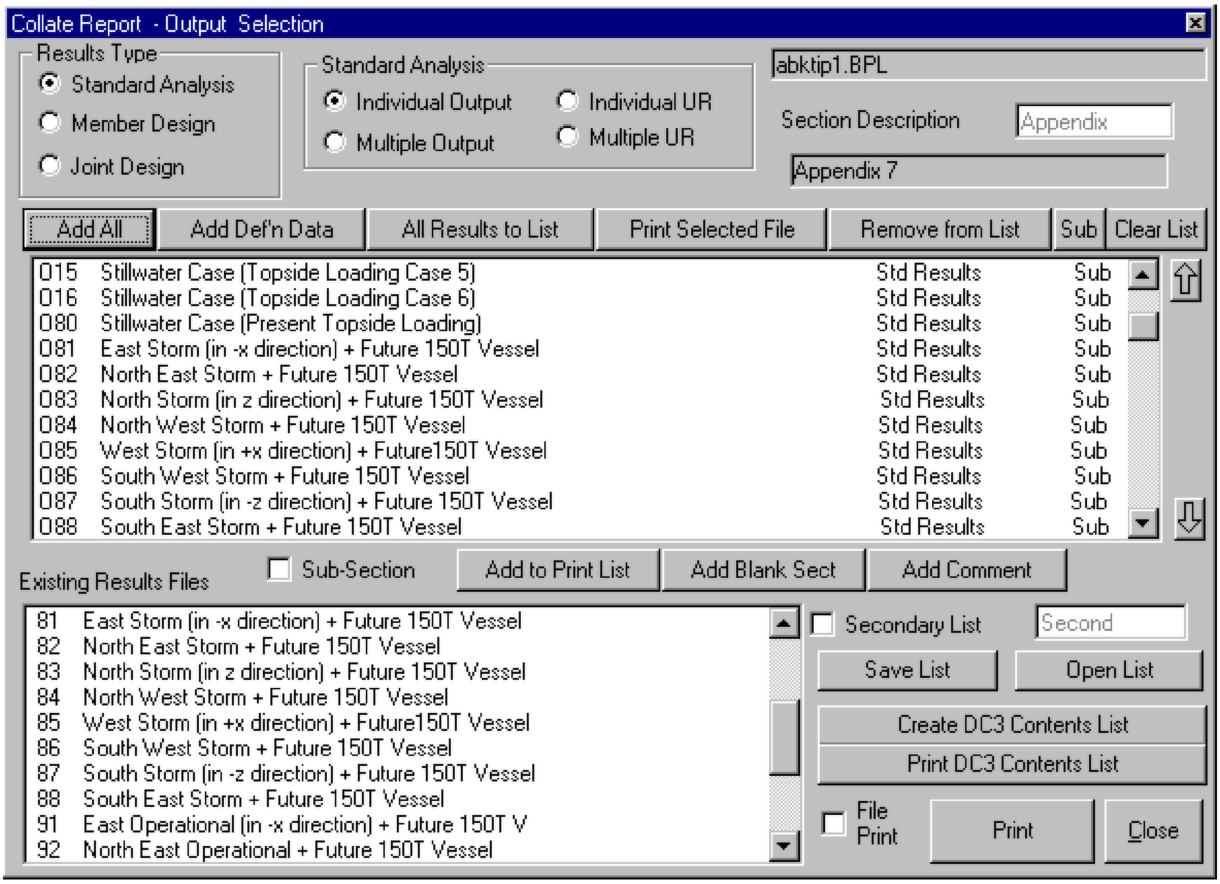
The form enables Analysis Reports to be collated and printed. It does this by creating Batch Print Lists. A Batch Print List is simply a list of formatted output files which can be grouped in sections. The list is created by selecting files from existing formatted files, these file may be definition files or results files.

When a Batch Print List is submitted to the printer all pages will be sequentially numbered in accordance with their section and sub-section number within that list. This enables complete analysis reports to be created and saved, which can be printed at any time.

The default Batch Print List (<Model>.BPL) file will always be opened when the form is made visible. The Secondary List check box may be used to Save and Open secondary Batch Print Lists. If the Secondary List option is checked then the file created will have the file name <Model>.Other.BPL where 'Other' is the name entered in the Secondary description box

This **Print List** shows the current contents of the Batch Print List.

The **Existing Results Files** show the existing formatted results files relating to the current Results Type.



The **Results Type** option buttons are used to select the category of Results to appear in the **Existing Results Files** list. If Sub-Case results exist, a secondary selection list will appear following selection of the results case. The option buttons to the right hand side are used to select the type of formatted file. These change according to the category type

The **Section Description** box is used to define the section page number type e.g. Section or Appendix could be used. As each file is printed its page would number would be Section **x.n** where **n** is the page number for the section and **x** is the section number which correspond to the position in the Print List. The section description can only be entered when the print list is empty. The page numbers always start at 1 in a new section unless it is designated as a Sub-Section.

The **Add All** will scan all formatted definition data and result data and add them to the list. If definition Sub-cases are used they will require to be selected manually from a list. Use the multiple select feature of the list to select more than one.

The **Add Def'n Data** will scan for alll formatted definition data files and add them to the list. If Sub-cases are used they will require to be selected manually from a list. Use the multiple select feature of the list to select more than one.

The **All Results to List** will scan all formatted results data and add them to the list. Sub-Case results are included in the list.

The **Print Selected File** will print individual files from the print list. Note that when printing Sub sections the page number of the sub section will requires to be entered.

The **Sub** button is used to change entries to and from Sub sections

The **Remove from List** button will remove the selected entry from the Print List. Double clicking an entry will also do this.

The **Clear List** button will remove all entries from the Print List.

The **Sub-Section** box is used link the section name and section numbers of the file to those of the preceding file. The Sub at the end of the line indicates a linked section. When a section is linked it has the same section number as the previous section. In the above example there are 5 sections, the 6th entry is a sub-section of section 5.

New entries to the Print List will be located at the position of the current line in the Print List box.

The **Add Def'n Data** button is used to append the formatted definition data file to the list.

The **Add to Print List** button will append the selected files in the Existing Results Files list to the Print List. Double clicking an entry will also do this.

The **Add Blank Sect** button is used to insert dummy sections into the print list. The main use of this function is to insert extra sections for external inserts e.g. graphics plots. It effectively inserts one blank page for each entry. If the sub-section box is unchecked then the inserted section will be a new section. If the sub-section box is checked the inserted section will be a sub section.

The **Add Comment** button is used to add comments to the list. It has no effect on the printed output

The **Save List** button will save the current Print List. If the Secondary List option is checked then the file created will have the file name <Model>.Other.BPL where Other is the current name entered in the Secondary description box.

The **Open List** is used to load secondary Batch Print List files.

The **Create DC3 Contents List** will create a DC3 text file that contains a list of the contents of the current print list. If required this content list may be edited in FS-Edit.

The **Print DC3 Contents List** will print the content list i.e. the DC3 file. This should be done prior to printing the contents of the Print List

The **Print** button submits the current Print List to the printer for batch printing. It will also action File Print if active.

Output files can be paginated if **Print to File** option is active. Paginated files are output files that include page breaks, page numbers and page headers, exactly what the printer produces. This can only be done when printing from the Report Collator. The paginated files will be located in a sub directory of the model directory called "<modelname> TEXT". The files will be merged to produce a single merged file for each section e.g. Appendix 1.TXT would contain all the output files within that section. The DC3 file should be used for an index for the merged files.

Identifying Results Files

If the user requires to process results files, e.g. import and parse the results into a Spread Sheet then the print list may be used to identify the files. All formatted output files are ASCII text files. The characters at the start of an entry are the file extension of the result file that takes the name of the model.

-o-

8.6 Menu:Display

This menu is used to selectively restrict the display of node and elements. All menu commands are global.

To **show** elements or nodes that are not assigned to a group enter -1 for the plot group. Note that this cannot be entered using the LH Mouse short cut menu.

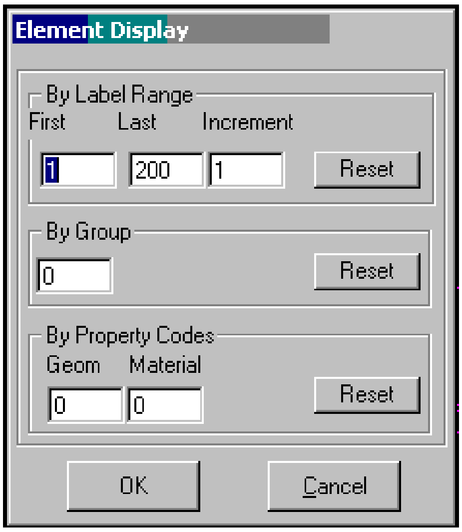
The commands that append and remove from the current display only apply to the current Viewport they are not Viewport dependent. If the Viewport is changed the display will re-set to the display set by the Viewport dependent settings ie window clip, label range or attribute (group or property). Only Viewport dependent settings can be saved.

Initialise	This will re-set the display so that all node and elements are visible.
Nodes-Only by Elem Assoc	Toggle Switch to Plot Element Nodes. If active enables nodes attached to elements to be plotted. This is useful when plotting selective element groups whose nodes are not in the same group.Quick access button available
Element Display	Defined elements (label range, group) to be displayed
Node Display	Defines nodes (label range, group) to be displayed
Couple Display	Defined spring/couples to be displayed
Commands listed below are not Viewport dependent	
Appends Nodes to Display	Adds to node to the current display group
Remove nodes from Display	Removes node from current display group
Appends Elems to Display	Adds to elements to the current display group
Remove Elems from Display	Removes elements from current display group
Appends Elems+Nodes to Display	Adds to elements & attached nodes to the current display group
Remove Elems+Nodes from Display	Removes elements & attached nodes from current display group
Window Clip - Iso View	Iso Clip planes are used to restrict the visible portions of the model to those portions that are within the mouse window. Unlike Othogonal Viewport clip planes Iso clips are effective in all view angles. Iso clip planes are not Viewport dependent and cannot be successively applied. .

-o-

Element Display (Menu:Display)

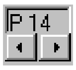
This form is used to define what elements (by label range, group etc) are to be displayed in the active viewport. The setting are viewport dependent.



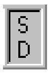
To **show** elements that are not assigned to any group enter -1 for the plot group. Note that this cannot be entered using the LH Mouse short cut menu.

Alternative Screen Method

The Group and Property Code displays can be more conveniently activated using the LHS mouse menu or the Screen Hot Keys. The controls on the button bar also enable scrolling and combining of views as described below.

Selective Entity Plot 

The Arrow buttons are used to increment selective plots when plotting by group (G), element geometric property(P) or element material property(M) attribute. To initiate the plot the RH mouse menu , the Hot key or this form must be used first because the arrow buttons have no effect when the box is blank.

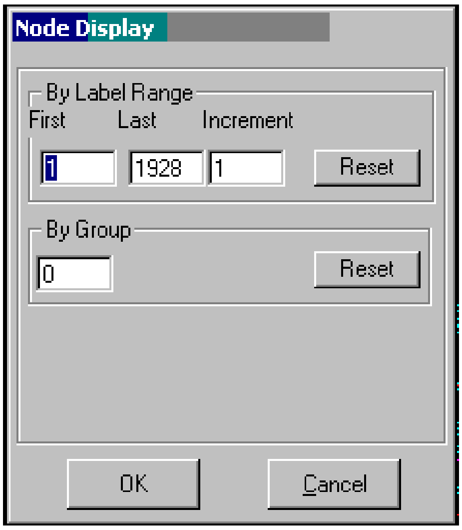
Combine Displays 

The **SD/AD** button is used to switch between Single Display and Add Display mode. This is used when selective plotting by group or property attribute is being undertaken using the **Selective Entity Plot**. If SD is active only one entity will be visible. If AD is active then the selected entity will be added to those already visible. The Display menu also provides the capability to add and additionally remove from current display.

-o-

Node Display

This form is used to define nodess (label range, group etc) to be displayed in the active viewport. The setting are viewport dependent.



To **show** elements that are not assigned to any group enter -1 for the plot group.   Note that this cannot be entered using the LH Mouse short cut menu.

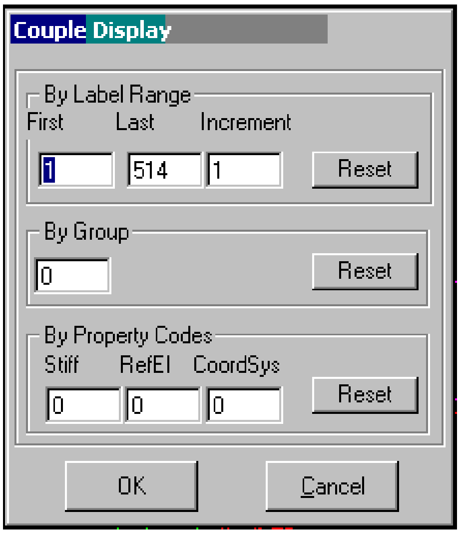
Alternative Screen Method

The Group displays can be more conveniently activated using the LHS mouse menu or the Screen Hot Key (G). The controls on the button bar also enable scrolling and combining of views.

-o-

Couple Display (Menu:Display)

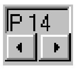
This form is used to define couples (label range, group etc) to be displayed in the active viewport. The setting are viewport dependent.



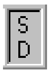
To **show** elements that are not assigned to a group enter -1 for the plot group. Note that this cannot be entered using the LH Mouse short cut menu.

Alternative Screen Method

The Group and Property Code(Stiff) displays can be more conveniently activates using the LHS mouse menu or the Screen Hot Keys. The controls on the top button bar also enable scrolling and combining of views as described below.

Selective Entity Plot 

The Arrow buttons are used to increment selective plots when plotting by group (G) or couple stiffness property(P) attribute. To initiate the plot the RH mouse menu , the Hot key or the above Couple Display form must be used first because the arrow buttons have no effect when the box is blank.

Combine Displays 

The **SD/AD** button is used to switch between Single Display and Add Display mode. This is used when selective plotting by group or property attribute is being undertaken using the **Selective Entity Plot**. If SD is active only one entity will be visible. If AD is active then the selected entity will be added to those already visible. The Display menu also provides the capability to add and additionally remove from current display.

-o-

8.7 Menu:Group

Groups are a powerful secondary number system for node and element identification. This menu is used to assign and save group attributes.

This menu is a Global TASK menu

To **show** elements or nodes that are not assigned any group enter -1 for the plot group using the Element Display . Note that this cannot be entered using the LH Mouse short cut menu.

Define Active Group	Defines the active node and active element groups
Initialise All	Removes all group assignments within the model
Initialise All - Maintain View	As previous but does not change visibility attribute previously set
Add Elements	Adds selected elements to the active element group
Remove Elements	Removes selected elements from the their current group
Add Elements & Att Nodes	Adds selected elems and their attached nodes to the active group
Remove Elements & Att Nodes	Removes selected elems and attached node from their current group
Add Elem by Attribute	Adds element to the active group by attribute identification
Add Nodes	Adds selected nodes to the active node group
Remove Nodes	Removes selected nodes from the their current group
Assign Nodes by Elem Assoc	Adds nodes to the active group that are connected to selected elements
Assign Elem Geom Code-All	Assigns the element geometric code to the group attribute
Assign Elem Geom Code-Selby	
Open Group SET	Retrieves group attributes from a previously saved SET and initilises current groups
Merge Group SET	Retrieves group attributes without initialisation
Save Group SET	Save the current group attributes to a group SET
Descriptions/Colors	Add descriptions and colors for each group within a Group SET

-o-

Define Active Group command (Menu:Group)

This command used to set the active node and/or active element group. On selection the Active Groups dialogue box will appear.

If a group is made active then all nodes and elements created will be assigned to their respective active group.

The current active groups are shown in the label buttons in the top toolbar. The Active Groups dialogue box will also appear if these buttons are clicked.

-o-

Initialise All (- Maintain View) command (Menu:Groups)

This command will remove all node group attributes and element group attributes from all nodes and elements in the model.

This is achieved by assigning all node and elements to group zero.

If any group display options are active they will be re-set and all elements will become visible.

The Maintain View option will not reset the visibility attribute of the element.

Group zero is ignored in all operations.

-o-

Add Elements command (Menu:Group)

Assigns selected elements to the active element group.

The selected elements are those elements defined by the Selection (SelectBy) Control.

Note: To transfer elements between groups use group selection to select those elements.

-o-

Remove Elements command (Menu:Group)

Removes selected elements from their current group

This is achieved by assigning the element(s) to group zero. Group zero is ignored in all operations.

The selected element are those elements defined by the Selection (SelectBy) Control.

Note: To remove only elements in a particular group then use group selection to select those elements.

-o-

Add Elements & Att Nodes command (Menu:Groups)

Assigns selected elements and their attached nodes to their respective active groups.

The selected elements are those elements defined by the Selection (SelectBy) Control.

-o-

Remove Elements & Att Nodes command (Menu:Group)

Removes selected elements and their attached nodes from their current group

This is achieved by assigning the element(s) to group zero.   Group zero is ignored in all operations.

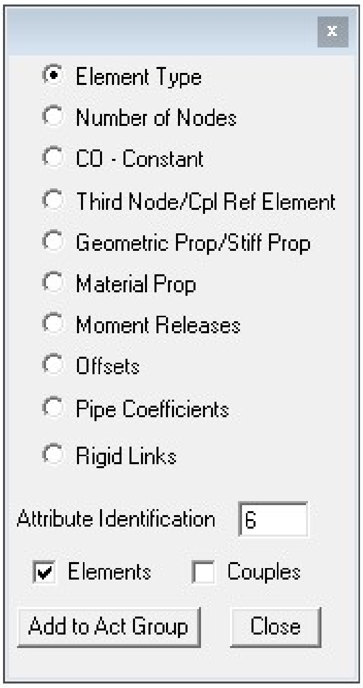
The selected element are those elements defined by the Selection (SelectBy) Control.

Note: To remove only elements in a particular group then use group selection to select those elements.

-o-

Add Element by Attribute (Menu:Group)

This form enables elements to be added to the active group by identifying elements with specific attributes.



The Attribute Identification is not used with all of the selection options.

The settings shown below will add all Type 6 elements to the active group.

-o-

Add Nodes command (Menu:Group)

Assigns selected nodes to the active node group.

The selected nodes are those nodes defined by the Selection (SelectBy) Control.

Note: To transfer nodes between groups use group selection to select those nodes.

-o-

Remove Nodes command (Menu:Group)

Removes selected nodes from their current group

This is achieved by assigning the node(s) to group zero.   Group zero is ignored in all operations.

The selected nodes are those nodes defined by the Selection (SelectBy) Control.

Note: To remove only nodes in a particular group then use group selection to select those nodes.

-o-

Assign Elem Geom Code command (Menu:Group)

Assigns the element Geometric Code Number to the Group attribute number.

Often users find it is desirable to group elements by reference to their property codes.

There are two basic options for this command;

Assign Elem Geom Code - All

This will assign the Geometric Property Code to the element group attribute of all elements. Note: This command will overwrite all current group attributes.

The group descriptions will be assigned to the property code names.

Assign Elem Geom Code - SelBy

This will assign the Geometric Property codes to only those elements selected by the current Select By method.

-o-

Save Group SET command (Menu:Group)

Saves the current node and element group attributes to a Group SET. Nodes and elements attributes are saved in the same SET.

Group SETs are identified by a SET number and an optional SET description

The valid SET ID range is 0 to 99. The description can be up to 50 characters.

When selected the Group SET box will appear. This list show all group SETs previously saved.

When saving SETs, existing SETs of the same ID will be overwritten without a prompt being given.

-o-

Open Group SET command (Menu:Group)

This is used to retrieve a previously saved group SET.

Group SETs are identified by a SET number and an optional SET description

When selected, the Group SET list box will appear. This list shows all group SETs previously saved.

When the SET is loaded all groups are initialised.

-o-

Merge Group SET command (Menu:Group)

This is used to retrieve a previously saved group SET and merge with the existing groups.

When a group SET is loaded it will overwrite any current group attribute for a node or element providing that the SET being loaded has a non zero value for that node or element. i.e. the existing group attributes will be left unchanged if the loaded SET does not have a group attribute for given nodes or elements.

This merging arrangement makes it possible to manipulate groups by using group SET storage to selectively overwrite.

Group SETs are identified by a SET number and an optional SET description

When selected, the Group SET list box will appear. This list shows all group SETs previously saved.

-o-

Descriptions & Colors command (Menu:Group)

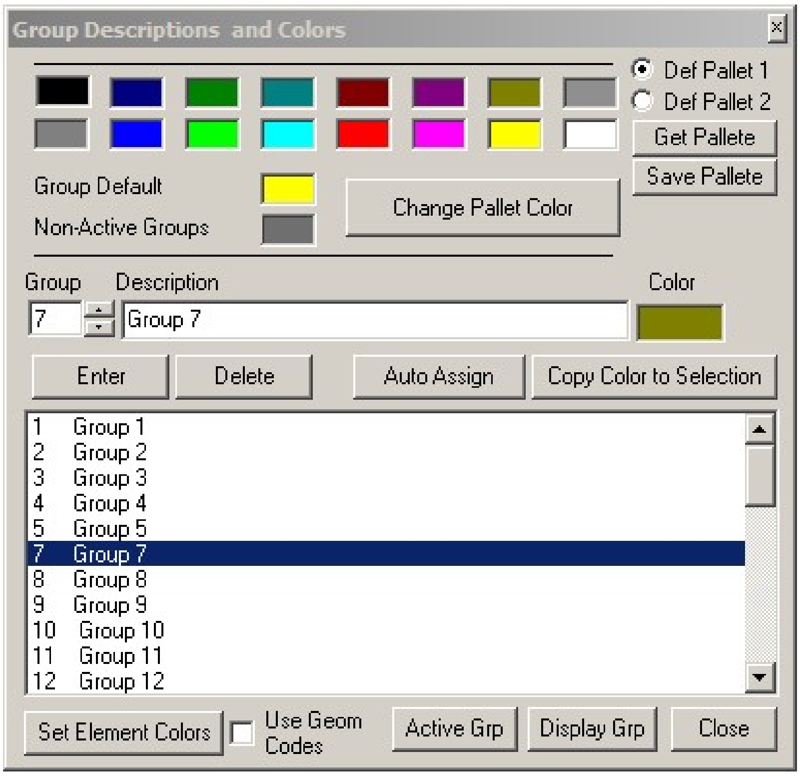
This command activates the Group Descriptions and Colors definition form. The colors are used for the solid fill view of beam and solid elements.

The descriptions are for the description of groups within a Group SET. They are saved with the Group SET. They will appear at the top of all output that uses the group attribute.

By default the element's colour is set by the Geometric Property Code colour. The element's colors can only be changed by use of this form and they are saved with the model.

Two model independent default pallets may be created and saved. This can be useful for setting up palettes for color and black & white (gray scales) printers.

The form shown below shows the case where the groups are based on the Geometric Property codes.



Group Descriptions

Group Descriptions are saved with the Group SET. When a SET is saved the the default group description will be assigned i.e. Group X

The **Enter** button is used to enter the description.

The **Delete** button will delete all selected descriptions and will be set to the default i.e. Group x when the SET is saved.

Group Colors

Element colors are defined on an element basis but they can only be assigned using the element group attribute. They are saved and opened by Group SET.

A palette of 16 colors is available for Group assignment. The palette colors can be changed at any time. If the palette is changed existing group and element colors will change. Palettes are saved with the Group SET.

The selected palette color will be assigned to a group when the **Enter** button is clicked. Clicking a palette colour will select it. A X symbol indicates the current selection. Re-clicking will de-select

The **Default Color** is the fill color for all elements.

The **Auto Assign** will assign the the active colour pallet to all groups (successive).

The **Copy Color To Selection** is used to assign the selected palette to selected groups .

The **Set Element Colors** button is used to assign colors to the element. If the **Use Geom Codes** is not active then the element's colours will be assigned the their group colour attribute. If the **Use Geom Codes** is active the the element's colour will be set back to that of the colour of the Property Codes (i.e. the default). When a new Group Set is opened the element colors will remain unchanged until this button is clicked.

The **Active Group** button is used to make the currently selected group the active group.

The **Display Button** is used to restrict the display to those elements in the currently selected group.

-o-

8.8 Menu:Comb

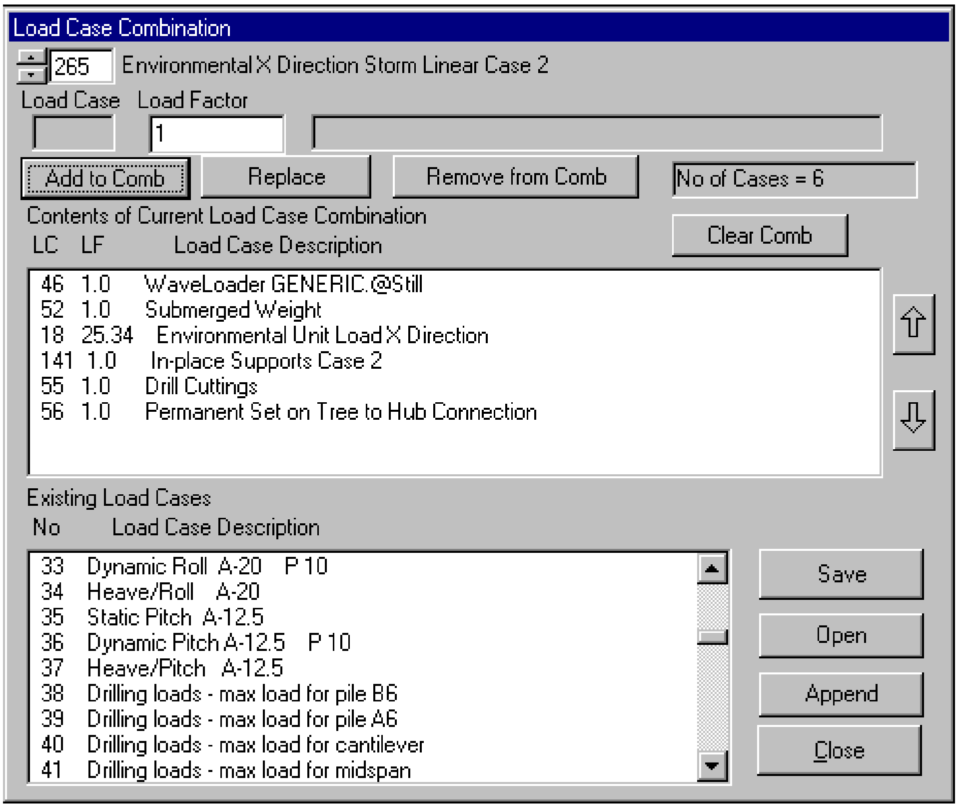
This menu is used for the creation and editing of Load Case Combinations and Results SETs. The menu is available in most TASKs.

Create Load Case Comb	Creates Load case Combinations
Edit/Delete Case comb	Edits or deletes existing Load Case Combinations
Create Results SET	Creates results SETs from existing results files
Edit/Deletes Results SET	Edits or deletes existing Results SETs

-o-

Creating/Editing Load Case Combinations

The form below is used to create/edit load case combinations



The **Add to Comb** button will append the entries shown in the **Load Case** box and the **Load Factor** to the contents list of the current load case combination.

The **Existing Load Cases** list must be used for the selection of load cases. When a load case is selected it will appear in the **Load Case** box. The Load Case Factor may be changed prior to pressing the **Add to Comb** button.

If a load case is double-clicked it will be appended directly to the list using the currently entered Load factor.

Selection of multiple load cases can be done using the Alt or Shift keys.

If a load case is selected from the combination list it can be removed by using the **Remove from Comb** button.

The **Replace** button is used to replace the selected entry in the combination list by the data shown in the **Load Case** and **Load Factor** boxes.

The **Clear Comb** button will remove all data from the combination list.

The **Arrow** buttons are used to move combination entries to different positions in the list.

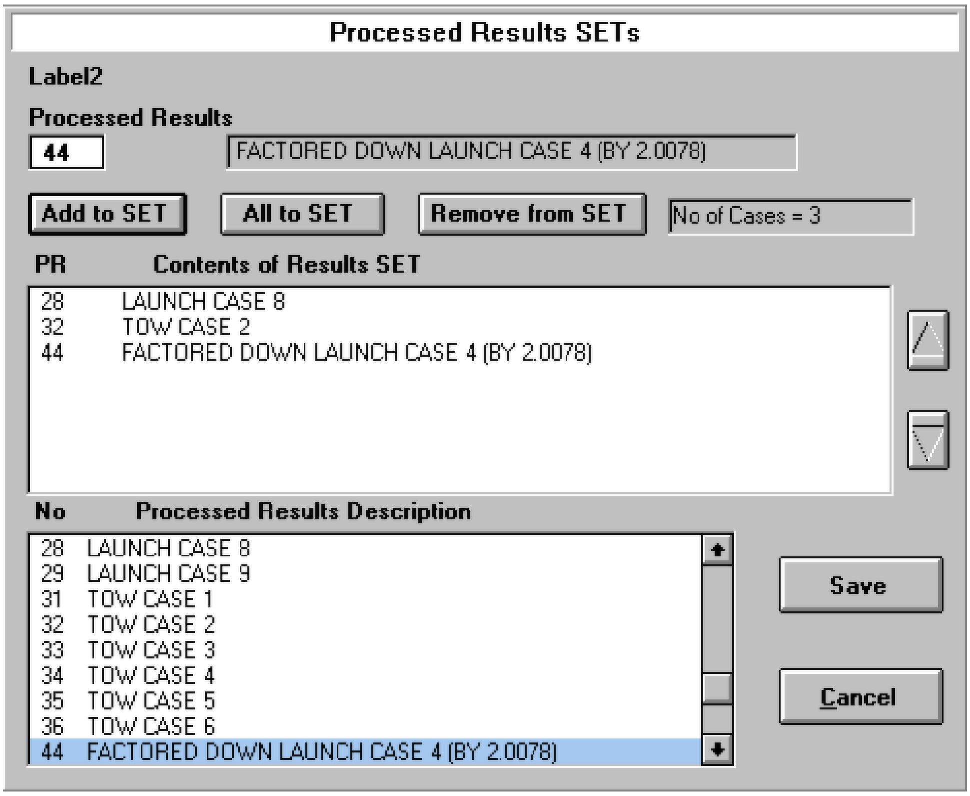
The **Save** button is used to save the current combination list. Load case combinations are identified by their reference number.

The **Append** button is used to merge existing combinations

-o-

Creating/Editing Results SETS (Menu:Comb)

The form below is used to create /edit Results Sets



The **Add to SET** button will append the entry shown in the **Processed Results** box to the results set list.

If any results cases exist, the **Processed Results** list may be used for the selection of the results cases. When a case is selected it will appear in the **Processed Results** box.

If a case is double-clicked it will be appended directly to the list.

If a results case does not exist the case number can be typed into the **Processed Results** box and than be added to the list. This is useful when setting up batch processing.

The **All to SET** button is used to add all existing results case to the set list

If a results case is selected from the set list it can be removed by using the **Remove from SET** button.

The **Arrow** buttons are used to move set entries to different positions in the list.

The **Save** button is used to save the current set list. Results SETs are identified by their reference number.

-o-

8.9 Menu:Window

FS2000 has the capability to open multiple windows of the current model. These windows are termed Viewports.

Then Window menu comprises of the following command:

Add View	Adds an additional Viewport to the current model
Remove View	Removes a Viewport (last added) on the current model
Single View	Removes all previously added Viewport to the model
Scale to Printer	Proportions the active Viewport to match the default printer
Cascade	Cascades all current Viewports
Tile Vertical	Tiles all current Viewports
Tile Horizontal	Tile all current Viewports

-o-

Viewport (Menu:Window)

A Viewport is a graphics window where the model is plotted. When the program is started only one Viewport is visible. Additional Viewports may be added as and when required.

Each Viewport has own settings. These setting includes those set using the Menu:Displ menu, the Zoom Windows facility and the Clip Plane Settings. This capability enables different areas of the model to be viewed simultaneous. The windows may be tiles or cascaded.

When selecting entities with a mouse it is possible to pick from any available Viewport which shows the entity to be picked.

Each Viewport can contain a different result case when viewing result cases.

-o-

8.10 Task Orientated Menu Commands

A considerable number of menu commands are available for the different stages of analysis. To provide logical access to these commands the analysis process has been divided into separate processes i.e. distinct stages of analysis.

These processes are termed TASKS. Tasks are initiated and terminated by command selection in the TASK menu. The main title bar indicates the current task.

For each task a different set of menu commands become available. Some menus/commands are available for all tasks. These are termed global menus or commands. The menus to the left of the TASK menu in the top menu bar are Global Menus. Those that appear to the right are TASK Related Menus (except Window & Help).

The TASK menu and the basic functions of the tasks are:

Primary	Used for general model management i.e. opening, archiving deleting etc.
Model Definition	Provides all the commands used to define a model.
Load Definition	Provides all the commands to define loading and load cases on a model.
Design Parameters	Enables design parameters associated with code checkers to be defined e.g. effective lengths
Analysis	Use to submit a model for solution
Output/Results	Used to, plot and interrogate analysis results on the screen. Prepare, view and print output listings - both definition and results listings
Model Checking	Model checking routines
Frame Wizard	Generation of common structural configurations

-o-

8.11 Primary

The primary TASK is used to for basic model management e.g. copying models, archiving models etc.

When certain TASKS are active memory is allocated to those tasks. When in the primary TASK only the basic model data is held in memory and command functions are limited to a only a few functions e.g. grouping and combinations.

When quitting the definition TASKS data should be saved. It is essential that any changes to the model data be saved otherwise the data will be lost as memory is reallocated as the program returns to the Primary TASK.

Definition TASKS are

- Model Definition
- Load definition
- Design Parameters

-o-

8.12 Model Definition

The following menus are available in this task.

Node	Node definition
Elemts	Beams and pipework definition
FE-Solids	Finite elements definition
Couple	Spring/Couple definition
Rest	Model Restraints definition
Prop	Definition of Property Tables

-o-

Model Definition:Menu:Node

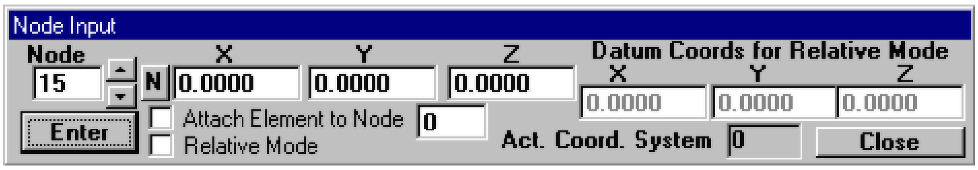
This menu provides the model definition commands associated with node definition

Input	Makes the primary Node Input dialogue box visible for node definition
Generate Between	Generates nodes (linear or arc) between two existing nodes
On Element	Adds one or more new nodes on an existing element by splitting and adding new elements
Move/Create on plane	Moves nodes or creates nodes at the current mouse location
Copy	Copies existing nodes
Delete	Deletes selected nodes
Re-Numbers	Removes gaps in nodal label sequence and re-numbers node labels
Translate	Moves existing node (translation or rotation)
Reflect	Reflects nodes across x, y or z principal planes
Align	Re-defines selected node coordinates of existing nodes
Node to Node	Moves one node to the location of another node
Move to Surface	Moves existing nodes on to surface intersections
Roll Up/Out	Translates node meshes between cylindrical and flat surfaces
Coord System	Makes the Co-ordinate System dialogue box visible
Convert to Act Sys	Converts selected nodes to the Global Active Co-ordinate system. Does not convert to Spherical.

-o-

Node Input (Menu:Node)

The following Node Input box appear when the Node Input command is selected.



To define a node simply enter the node number, its co-ordinates and click the enter button.

If an existing node number is entered in the Node box, its co-ordinates will be displayed.

If the **Node** box is double clicked with the **Left Mouse Button**, then the last node listed with the Node Query button will be entered. This can be useful for editing.

If the **N** button is pressed the next unused node label will appear without changing the co-ordinates. This can be useful for copying.

The **Act Coord System** box shows the currently active Co-ordinate System. The Y coord is in degees for a cylindrical or spherical type system. The Z coord is in degrees for a spherical type system.

If the **Attach Element to Node** box is checked and an attached Node is defined then an element will be attached from that node to the newly entered node. By default the last entered node will appear in the Attach to Node box. The properties of the elements will be the default values (change these in the Element Definition input form).

If the **Relative Mode** box is checked Relative Mode Input will be active and the dialogue box will extend to show the Datum Co-ordinates for relative mode input. These co-ordinates may be re-entered as required.

If a node is listed and then one of the datum co-ordinate boxes clicked the datum co-ordinates will take on those of the listed node.

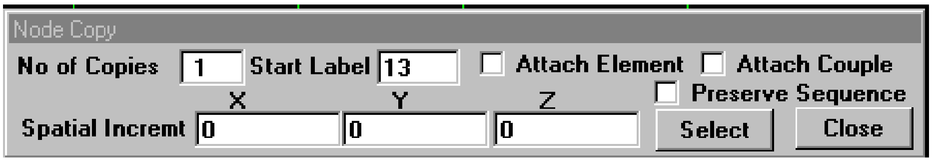
Using the **Attach to Node** option with relative mode input enables equally-spaced line elements to be generated by simply repeatedly clicking the **Enter** button.

-o-

Node Copy

When this command is selected the following box will appear. This input form is used to copy existing Nodes. The box is also used when copying elements with the Crate New Nodes option active.

Node Groups - When copying nodes the new node will be assigned the to the active node group. If no node group is active the node will be assigned to same group as the node being copied.



The **Spatial Increment** defines the co-ordinate increment between the nodes to be copied and the new nodes The spatial increments must be in the same coordinate system as the node being copied.

The **No of Copies** box defines the number of copies per selected node. The spatial increment of each copy is the product of the copy number and the defined increment.

The **Start Label** defines the first label of the new node. The default will be the next unused node label. This will always be updated when the box receives focus.

The **Preserve Sequence** option is used in cases where more than one node is selected e.g. using a window to select a group of nodes. In this instance if the option is selected the node label sequence of the nodes being copied will be preserved in the new nodes. If not, the new nodes will be sequential (no gaps in sequence) starting at the start label.

If the **Attach Element (Or Attach Couple)** box is checked then an element will be attached between copied node and the new node. The properties of the elements will be the default values (change these in the Element Definition input form). The Preserve Sequence will be activated if the number of copies exceed 1.

Extruding 2-D Frames

When copying beam elements the **Attach Element** option will result in beams being connected the nodes of the elements being copied and the nodes of the new element(s) e.g. 2-D frames may be extruded to 3-D frames.

When the **Select** button is pressed the nodes to be copied will require to be selected by the current selection method.

Unless the Cancel button is clicked the box will remain visible. Parameters may be changed at any time during the pick process.

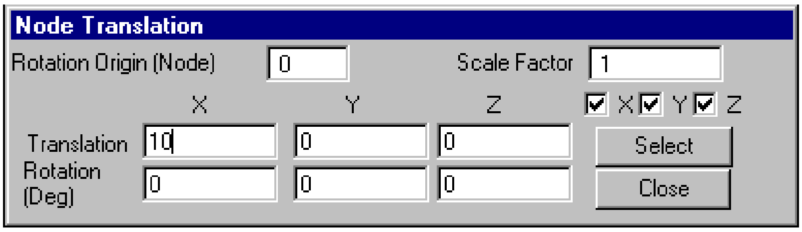
-o-

Node Translation

This command is used to move existing nodes to a different location. The nodes may be translated or rotationally transformed. When the command is selected the following box will appear.

Coordinate Systems

The translational and rotational increments must be defined in the local coordinate system of the node(s) being moved. When individual nodes are being moved the new coordinates of the node will be listed in the using the Active coordinate system.



The **Translation** boxes are used to define the distances to move the nodes in the local co-ordinate system of the node being moved.

If two nodes are listed with the **Node Query** button prior to selecting the command then the distance between these nodes will be entered in the **Translation** boxes when the boxes are **Double clicked**. The node pick sequence dictates the translation direction. **Double clicking** will enter the last used increments if translation was the last operation.

The **Rotation** boxes are used to define the rotational transforms. These are in the global co-ordinate system. If the **Rotation Origin** is set to zero the transform will be about the origin. If a node number is entered in the **Rotation Origin** then the transform will be about that node. **Double click** this box will enter the last node listed with the Node Query button.

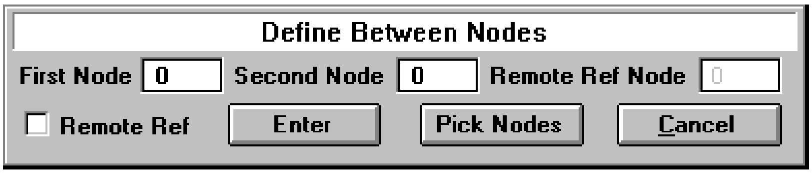
The **Scale Factor** can be used to factor node co-ordinates. The X, Y and Z check boxes are used to selectively apply the scaling factor.

When the **OK** button is clicked the nodes to be transformed should be selected by the current selection method.

-o-

Node Generation - Between Nodes

This generation routine will generate one or more nodes between two existing nodes.

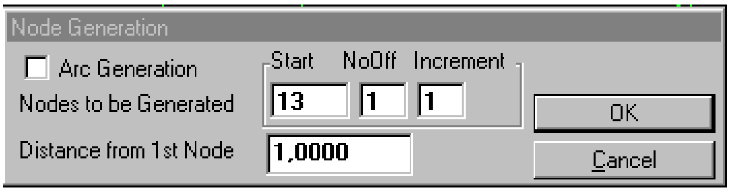


The existing nodes may be identified by label using the **First Node** box and the **Second Node** box and clicking the **Enter** button. Alternatively, they may be visibly picked if the **Pick Nodes** button is clicked.

See Node Generation-Remote Reference for description of the Remote Ref function.

The **Remote Reference Node** is only active if the **Remote Ref** box is checked. If a node is listed prior to selecting the Between Node command this node will become the default Remote Reference Node. If the origin triad is visible this may also be picked. This enables reference to the origin to be defined in models where nodes are not defined at the origin.

Following selection the following box will appear (providing Remote Ref is not checked).

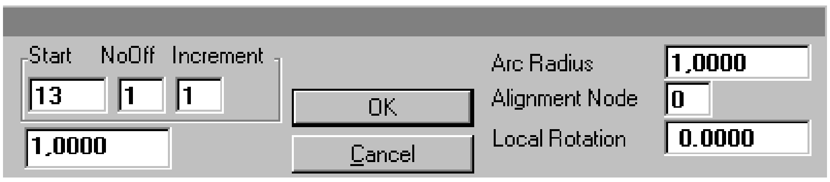


This box is used to define the number of nodes to be generated. The default **Start** node is the next unused node label. The **Increment** defines the node increment of the new nodes.

If only one node is to be generated the distance from the First Node requires to be entered. The default is mid way. If a negative length is specified the new node will be placed in line with the two nodes but on the opposite side of the First Node. If the distance is larger than the distance between the two nodes the new node will be positioned at that point and be aligned with the two nodes. This is a very useful construction technique for projecting nodes.

If more than one node is specified the nodes will be generated evenly between the two nodes.

If the **Arc Generation** box is checked the box will extend to allow entry of the following.



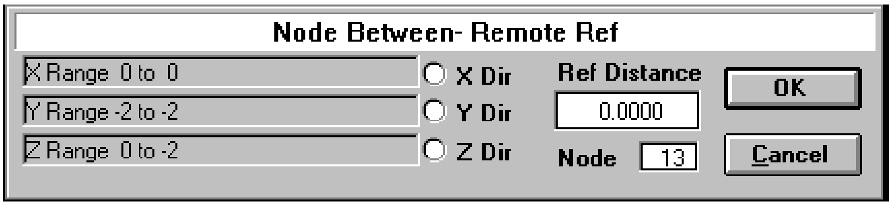
The default **Arc Radius** is the radius for a 90 bend. The **Alignment Node** or alternatively the **Local Rotation** angle is used to define the rotational orientation of the bend. The sign convention for orientation is identical to that for Local Rotation of beam elements.

-o-

Node Generation -Remote Reference

This definition method is used to create a node between two existing nodes but by dimensional reference to a third node on the model i.e. a remote node. It is also used by the Node On Element generation routine.

Following selection of two between node the box below will appear.



The position of the node is defined by specifying only one reference distance which is in the plane of the selected co-ordinate direction.

Shown next to the directions are the dimensional ranges between the two existing nodes (or element nodes) and the reference node.   Always chose a direction that has a non zero range.

If the **Ref Distance** is specified as a value outside the specified range then the new node will be located at a point outside the two nodes but aligned with the two nodes.   Negative values will place the node on the opposite side of the First Node.

When generating a node on an element ensure that the **Ref Distance** is within the ranges otherwise the resulting elements will be overlapped.

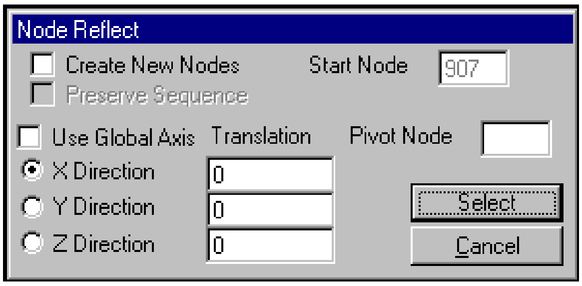
The **Node** box is used to define the new node. By default it will be the next unused node label.

-o-

Node Reflection (Menu:Node)

This command is used to reflect nodes across the principal planes i.e. make a mirror image. Existing or copies of existing nodes can be reflected. If elements are attached to the nodes they will also be reflected.

When the command is selected the following box appears.



To reflect existing nodes simply select the reflection direction. When the **OK** button is clicked the nodes to be reflected will be selected by the current selection method.

The nodes will be reflected across their own local definition coordinate system. Nodes in a cylindrical system will be rotated 180 when the Y direction is checked (don't check the X direction in a cylindrical system). Spherical systems will be ignored.

If the **Use Global Axis** is active the reflection will be done across the global axis regardless of the nodes coordinate system.

The **Translation** boxes are used to move the reflected nodes by the defined translation increment. Note that these values will be re-set each time a pivot node in entered.

The **Pivot Node** if defined will mirror the reflected nodes about the defined pivot node. This should only be used if the subject node is defined in the global coordinate system. Double click to enter the last node listed with the Node Query button. This will enter the appropriate translation offsets.

If the **Create New Node Set** box is checked then copies of the selected nodes will be reflected, the selected nodes will not be changed. The node label sequence will be maintained in the node copies if the **Preserve Sequence** option is active, otherwise the nodes will be sequentially numbered.

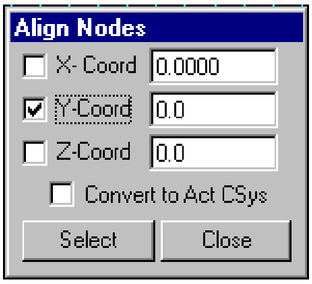
The **Start Label** box is used to define the start label of the new nodes when the **Create New Nodes** box is checked. The default for this is the next unused node label.

-o-

Node Alignment (Menu:Node)

This command is used to re-define the position of existing nodes. It differs from using the Node definition box in that the only selected coordinates re-defined. Its main use is to align nods onto surfaces

When the command is selected the following box appears.



Node coordinates dimensions will only be updated if the dimension is checked. The coordinates will be interpreted in the coordinate system of the selected node.

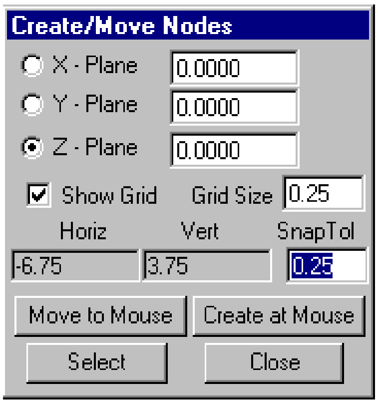
If the **Convert to Active CSys** is checked the coordinates of the selected node will be converted to the active coordinate system when at its aligned position.

-o-

Move/Create on plane (Menu:Node)

This command is used to move existing nodes or create new nodes at the current location of the mouse. The plane in which the node is moved or created must be parallel to one of the principal planes of the global cartesian coordinate system. Any node which is moved or created will be assigned to this coordinate system.

When the command is selected the following box appears.



The **X, Y or Z** option buttons are used to select which plane of the global cartesian coordinate system is to be active. The corresponding data box is used to define the coordinate location of the plane. ie if Z was active and assigned a value of 10 all moved or created nodes will have a Z coordinate of 10.

The current coordinates of the mouse location are shown in data boxes. The **Snap Tol** can be used to provide additional control of coordinate definition. Note: The mouse position will not move according to the snap tolerance setting the snap tolerance only effects the node coordinate definition.

A grid based on the Snap Tol can be displayed if the **Show Grid** box is checked. If the grid density becomes too great it will be disabled.

If the **Create at Mouse** button is clicked a new node will be created at where the mouse is subsequently clicked. The process is terminated by pressing the ESC key or the RH mouse button.

If the **Move to Mouse** button is clicked then any node which is selected (clicked on) will be moved to the next location where the mouse is clicked. The process is terminated by pressing the ESC key or the RH mouse button.

-o-

Node to Node (Menu:Node)

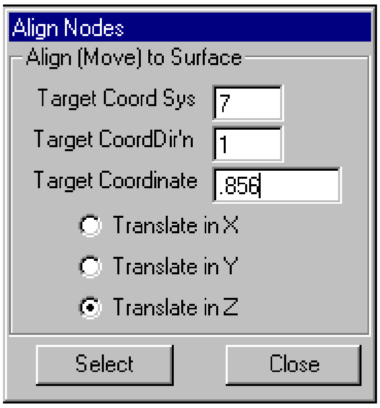
The command is used to move one node to the location of another node. The node coordinate system of the node being moved will be changed to that of the stationary node.

The moved node is the node which is picked first.

-o-

Move to Surface (Menu:Node)

This command is use to move existing nodes on to a surface defined by a coordinate system. Its main use is to move nodes to the intersection points of surfaces.



The **Target Coord Sys** is used to identify the coordinate system of the surface.

The **Target Coord Dir'n** is coordinate direction of the Target coordinate. This should be 1, 2, or 3 for x, y or z respectively. This would be set to 1 for a cylindrical surface i.e. the target would then be in the radial direction.

The **Target Coordinate** defines the coordinate magnitude on the surface on which to attach the node. This has to be a non-zero value. This would often be the radius of a cylinder. If moving onto flat surface e.g. a cartesian system, use a very small non-zero value if zero is the target requirement.

The **Translation** options are used to select the direction in which the node will move. This direction is in the coordinate system of the node being moved. If the case of intersecting cylinders the setting could be Y for moving tangentially or Z for moving axially.

N? Node Query Used to list Node data to the List Box (use mouse to pick node). If two nodes are listed the distance between them will also be given. Nodes will be listed in their local coordinate system. Distances between nodes will be given in the active coordinate system.

Hot key **N** also activates node Query and Hot key **C** sets the active coordinate system.

-o-

Roll Up/Out (Menu:Node)

This command is used to develop cylindrical surface node meshes into flat 2-D surface meshes or to roll flat surface meshes into cylindrical meshes.

In the description below the surface is simply a collection of nodes in the same coordinate system and could be in its simplest form be just one node.

Roll Out

When a cylindrical surface is developed into a 2-D surface (Roll Out) it will be developed onto the x-z plane of the global cartesian system.

- The x direction coordinate equates to the r-theta direction ($r.\varphi$).
- The z direction coordinate equates to the axial coordinate(z)
- The y direction coordinate equates to the radial coordinate(r).

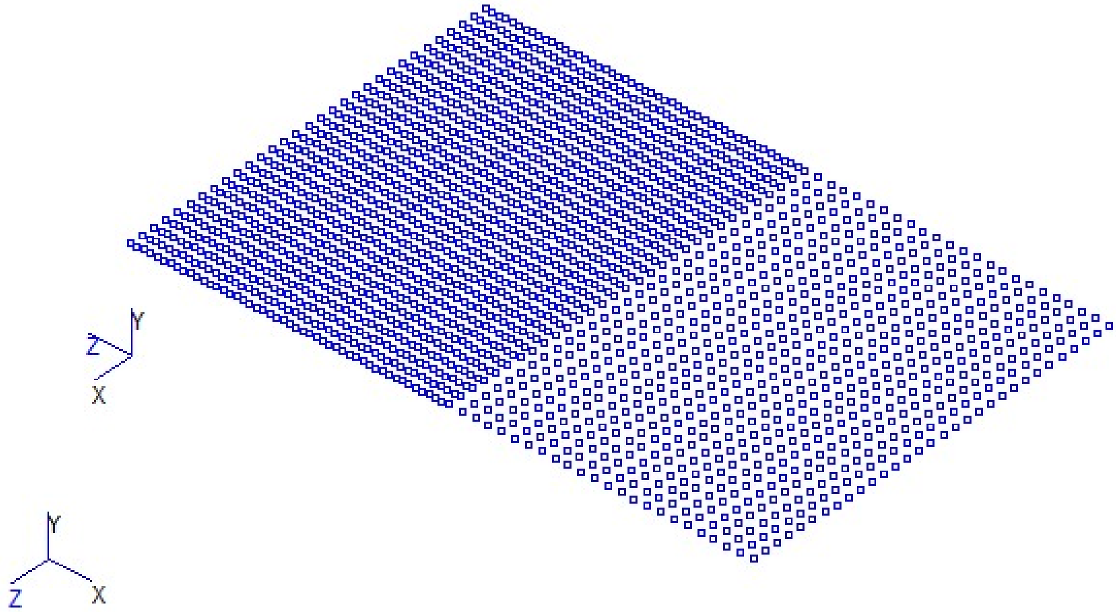
The co-ordinate system of the developed flat nodes will be the cartesian global coordinate system.

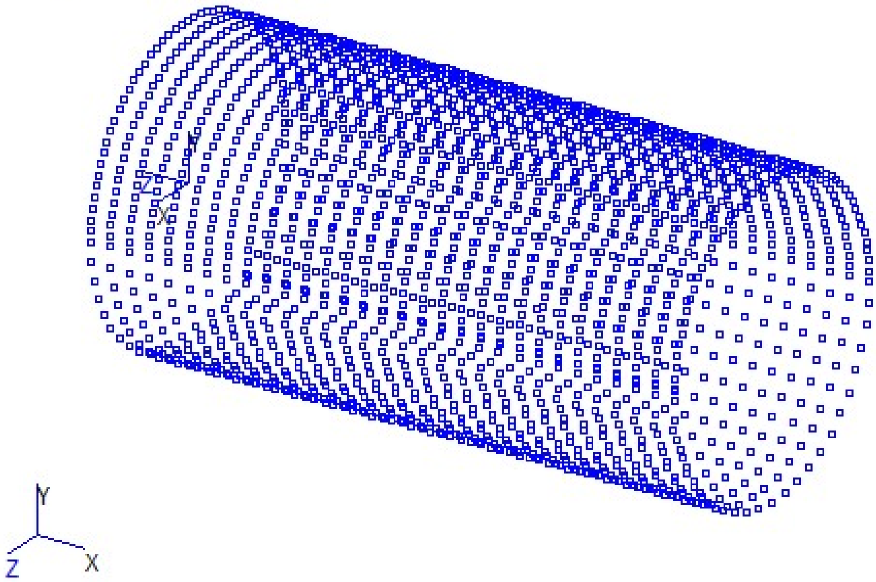
Roll Up

When a flat surface is required to be rolled into a cylindrical surface (Roll Up) the **Active Coordinate** system MUST be set to the required local cylindrical coordinate system of the cylinder and the nodes to be rolled up must be in the global coordinate system. The following transform rules will be applied following the roll up.

- The x direction coordinate will become the theta cylindrical coordinate (x/r).
- The y direction coordinate will equate to the radial coordinate (r).
- The z direction coordinate will equate the axial coordinate (z).

The example below shows how a simple flat mesh (elements not displayed) can be rolled up to form a cylindrical mesh.





-o-

Node Deletion (Menu:Node)

This command option is used to delete unwanted nodes. The nodes to be deleted are selected using the current selection method.

Nodes that are connected to elements cannot be deleted. Although they will become invisible after deletion they will be automatically recovered when the model is saved.

The nodes are not actually deleted during this process only their internal status flag is modified. The node status flag has three states,

- Undefined
- Defined
- Connected to an element.

Undefined nodes labels are apparent by the gaps they produce in the Node Data Lists.

When a node is defined it is always saved but it is only visible in the Model Definition Task.

If a node is deleted the actual co-ordinates will still exist until the program is quit. To recover the node simply re-enter the label in the Node Input box.

-o-

Adding Nodes to an Existing Element (Menu:Node)

This definition method is used to add nodes to an existing beam/pipe element by inserting both new nodes and new elements. This is probably the most used method to create nodes, or should be when creating model using the GUI.

The new elements created take on the attributes of the original element. The original element becomes part of the new string with its original label.

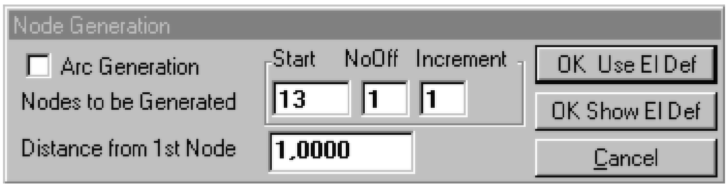
On selection of this command the element and reference node are required t be visibly picked in that order.

If the **Reference Node** is not one of the nodes of the element to be split Remote Reference will be assumed.

See Node Generation-Remote Reference for the remote reference function.

Following selection the element and reference node the following box will appear (providing Remote Ref is not active).

The Distance from 1st node will initially be the element mid point, thereafter the last used value fro subsequent picks. Double click the input box to use the last picked element mid point.

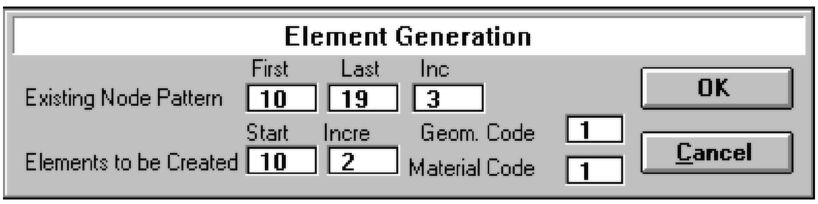


This box is described in Node Generation-Between Nodes.

If the **OK Use El Def** button is pressed the nodes will be generated and the new element(s) will be assigned to the next unused element label.

If the **OK Show El Def** button is pressed the nodes will be generated and the following will appear for confirmation of element labels.

The user will be asked if Group SETs require updating. If the answer is yes than all group Sets will be scanned for the element which is being segmented. If the element exists within a SET then new elements will be assigned to the same group within the SET. This prompt will only occur once in a Model Definition TASK session.



The defaults are the next unused element labels. The default property codes are those of the elements to being segmented.

Note that when segmenting elements the Design effective lengths are applied to the new segments of the span i.e. the new elements

-o-

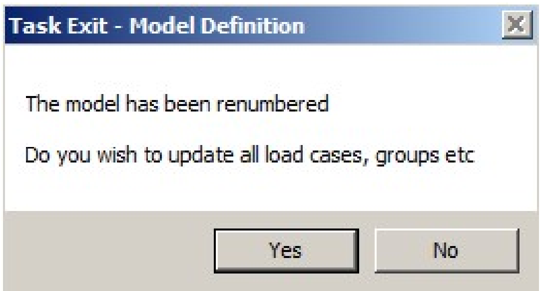
Re-Number Nodes (Menu:Node)

This command is used to 'pack' nodes. Its use is optional.

The command will remove all gaps from the node label sequence and re-number all nodes within the specified node label range. All contiguous label ranges will maintain their label sequence.

If the Model Definition TASK is exited without saving the model, all renumbering operations will be canceled.

When the model is saved an option will be given to update all secondary definition e.g. loads cases, effective lengths, groups, piles, waveloader etc



If this update is not undertaken all such secondary definition may require to be re-defined depending upon the current re-numbering range.

It is recommended that the model by archived before any renumbering operations are undertaken.

The renumbered nodes are listed in the scatch file <model>.~RENN. The RHS column is the new number and the LHS the previous.The first line lists the old max and the new max.

-o-

Co-ordinate Systems (Menu:Node)

This dialogue box enables Co-ordinate Systems to be created or existing systems to be made active. Co-ordinate Systems can only be created when in the model definition TASK. Their use is in node definition and FE stress output.

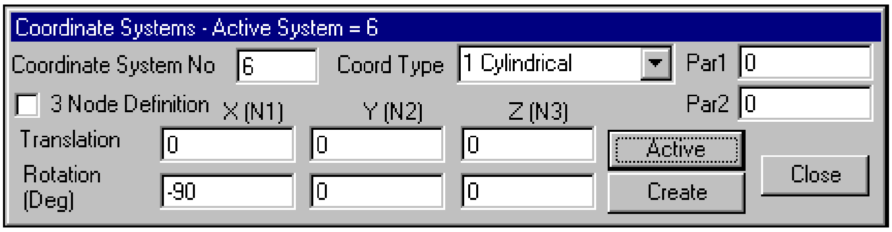
They can be defined by two methods

- Origin definition and three rotations (Y-Z-X transformation)

  The origin is defined by defining the X, Y & Z co-ordinates. The angular orientation is defined using rotations (degrees) in a Y-Z-X transform.

- 3 Nodes

  The origin is defined by the co-ordinates of N1. The x-axis goes towards N2 from N1. The x-y plane is defined by N3. The Z axis is formed using a RH orthogonal system.



The **Coord System Number** box is used to enter the System number. Number 0 to 5 are pre-defined. Number 6 to 20 are for user defined systems.

The predefined systems are;

0	Global Cartesian (x,y,z)
1	Global Cylindrical (r, theta,z)
2	Global Spherical (r,theta,phi)
3-5	Not used (unavailable)

The **Coord Type** is used to specify the type of coordinate system. The options are:

- **Cartesian**
- **Cylindrical**
- **Spherical**
- **Conical**

The **Par1** & **Par2** parameters are used in the definition of a Conical system. Par1 is the Radius at Z=0 and Par2 is the cone angle (from Z axis). A positive cone increases the radius in the z direction. These two parameters make the R and Z coordinates interdependent i.e. defines a conical surface where x(r) or z and y(theta) define the position. z takes precedence over x unless z is zero.

If the **3 Node Definition** is active the X, Y and Z boxes are used to define the node label that defines the x-y plane of the system. In this case the Rotation boxes are not used.

The **Create** button will be activated if the System Number is greater than 5. When this is clicked the co-ordinate system will be created using the defined origin parameters. The **Create** button will also change the parameters of an existing system.

Caution - It is not advisable to change parameters of an existing co-ordinate system if nodes are already assigned to that system. However, if this is necessary the affected node can be converted to the new system by using the Convert to Active command in the Nodes menu.

The **Active** button is used to make an existing Co-ordinate System the active system. This button will only be active if the co-ordinate system exists.

When the above form is not visible the currently active system will be indicated in the Activity Status box when the Node Query button is clicked.

-o-

Model Definition:Menu:Elemts

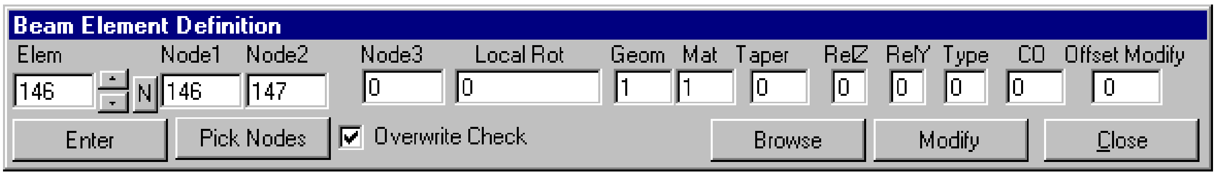
This menu provides the model definition commands associated with element definition

Input/Modify	Makes the primary Element Input dialogue box visible for element definition
Line Generate on Nodes	Defines elements by label pattern on an existing node label pattern
Line generate between Nodes	Generates nodes and elements (linear or arc) between two existing nodes
Copy	Copies selected elements and optionally, attached nodes
Delete	Deletes selected elements
Move	Re-connect a selected element to different nodes
Merge	Eliminates coincident elements and/or coincident element nodes.
Re-Number	Remove gaps in label sequence and re-numbers all elements including solid elements
Connect Intersecting	Connect two intersecting elements with a node at the intersection point.
Reverse Connectivity	Switched the node numbering direction.
Insert Bend	Inserts a bend between to connected elements
Insert End Spring/Couple	Inserts a spring couple
Moment Releases	Used to define end moment releases on selected elements
Local Rotation	Used to define local element rotation on selected elements
Offsets	Used to define end offsets on selected elements
Offsets List	Permits element offset definition to be listed (mouse pick)
Cables/Catenaries	Generates catenary properties and element mesh based on shape
Pipework	Defines pipework stress and flexibility data

-o-

Element Input/Modification (Menu:Elemts)

The following input box will become visible when the Element Input/Modify command is selected. This form is used to define Element related data.



When the box is first visible the default element label is the next unused element label.

To define an element, enter the element number and the appropriate definition parameters and click the **Enter** button. It is more often convenient to define the connectivity of the element i.e. Node1 and Node2 of the element by picking nodes from the screen. To do this, simply click the **Pick Node** button and then select Node1 and Node2 for the element from the visible nodes. When picking nodes any data in the Node1 or Node2 box will be ignored. When picking nodes the **Enter** button is not to be used.

The **Overwrite Check** is a safeguard to prevent the re-definition of existing elements.

If an existing element number is entered in the **Elem** box, its parameters will be displayed in the other boxes.

If the **N** button is pressed, the next unused element label will appear without changing the other parameters. This is useful for copying parameters from existing elements.

If the **Elem** box is double clicked using the **Left Mouse Button**, the last element listed with the Elem Query button will be entered. This is useful for editing existing elements.

The **Node3** and **Local Rot** boxes are used to define the local orientation of the element

The **Geom** box is used to assign the geometric properties to an entry in the Geometric Property Table. If the **Geom** box entry is a -1 then the element will be defined as a Rigid Link

The **Mat** box is used to assign the material properties to an entry in the Material Property Table.

The **Taper** box is used to define Tapered Beams by defining a property code for the aft end of the element.

The **Offset Modify** box is an exception to the other parameters in that it can be only be used when modifying elements. When used it effectively copies the offsets from the element label defined in the box to the element selected for modification. Offsets are deleted by entering a value of zero. Note that offsets are defined using the Offset Definition form.

The **RelZ** and **RelY** are used to define moment release codes.

It is often more convenient to define the **Local Rot** (Local Element Rotation) and the **RelZ** & **RelY** (Moment Releases) parameters after the elements have been created. Definition methods specific to these tasks have been developed and are available as command options of the Menu:Elemts.

The **Type** box is used to specify the element type.

The **CO** property is an additional property used by certain element types. The use of this property is described in the element description for elements that use this property.

The **Browse** button is used to browse the current property tables of the model. The tables can be modified when browsing.

Most of the parameters of the input box when entered become default values e.g. used when attaching elements to nodes as the nodes are being created.

The **Modify** button is used to copy these defaults to existing elements. The modification process is selective, the Element Modification form will appear so the attribute(s) to be modified can be identified. If the parameter box is not checked it will not be modified.

Tension Only or Compression Only elements are defined in terms of their Geom property code.

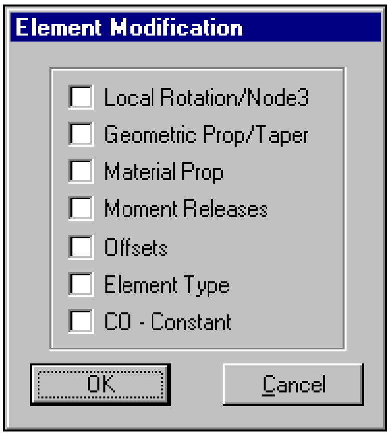
-o-

Element Modification (Menu:Elemts)

The properties of existing elements may be modified using the Modify button on the Element Definition form. The values that will be applied to the elements are the current values of the shown in element deifintion form.

Use this to feature delete or copy Element Offsets. First enter element number to be copied (0 to delete) in the Offset Modify box in the Element Definition form.

Element Type will also copy Bend Radius for bend elements.

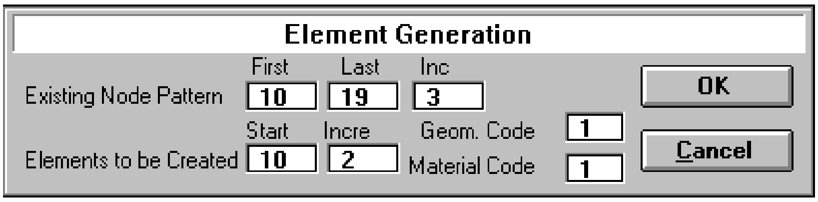


When the **OK** button is clicked the elements to be modified are selected by the current selection method. This requires to selected before the box is visible.

-o-

Elements - Line Generate on Nodes (Menu:Elemts)

The generation routine is used to generate a set of elements on a set of existing nodes. Both sets require to be defined by label pattern. The following input box will become visible when the Generate on Nodes command is selected.



-o-

Elements - Line Generate Between Node (Menu:Elemts)

The generation routine will generate a set of in line elements between two existing nodes. The nodes between the elements will also be created.

When the command is selected the first task is to define the node(s). The input options for this stage are identical to that for Node Generation - Between Nodes existing nodes. Note that the number of nodes to be generated are then number of elements required -1.

The next task is to define the elements label for the elements to be created. The input for this is identical to the Elements - Line Generate on Nodes but the node pattern is pre-set.

-o-

Merge (Menu:Elemts)

This command is used to merge the nodes of elements.

It does it by merging coincident nodes to a single node and then connecting all elements attached to the eliminated node(s) to that node. Coincident nodes are nodes within a defined spatial tolerance of the subject node.

If the **Merge(Node + Elements)** command is selected (most common use), coincident elements will also be removed. If the **Merge(Nodes)** command is selected only the nodes will be merged.

It is a very useful command to use after copying and translating elements to different location on a model since it can easily remove duplicate elements and nodes which can often occur when undertaking such tasks.

Elements to be processed are selected by the current SectBy method.

When the command is selected a coincident node tolerance is required to be defined. If the spatial position of adjacent nodes is within this tolerance the nodes are regarded as coincident. The default value is 0.01.

For a selected element the merging process works as follows.

> All nodes are scanned to identify if coincident nodes exist. If one does exist, the element node will be re-assigned to the coincident node with the lowest label. If the eliminated node is not attached to any other part of the model it will be deleted.
>
> All elements are scanned to identify coincident elements. If one does exist the element with the highest label will be deleted from the model.

Merging Solid Elements with Beam Elements

Only if the label numbers of the solid elements are less than those of the selected beam elements will the nodes of beam and solid elements be merged using this command. In cases where they this is not the case and the elements are required to be merged use the Merge command from the FE-Solid menu to merge the elements. Using the merge command successively from both the FE-Solids menu and the Beam Elem menu will ensure that all elements are merged.

-o-

Elements - Copy (Menu:Elemts)

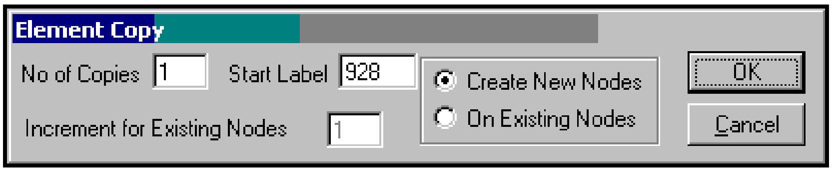
This routine is used to copy existing elements. New nodes can be created as the elements are copied or the elements may be copied onto an existing node pattern (rarely done).

If new nodes are created it is more than likely that the Merge command will be required to be used once the elements are created. This is because the new elements will not use common nodes between the existing elements or between new element blocks when multiple copies are done.

Element & Node Groups - When copying elements and nodes the new entities will be assigned the to the respective active group. If no group is active they will be assigned to same group as the element and node being copied.

Coordinate Systems - When creating new nodes the spatial increments must be in the same coordinate system as the node being copied. This implies that all node of the element being copied must be in the same coordinate system.

On selection of the command the element(s) to be copied require to be selected. The current selection method is used for selection. After selection the following input box will become visible.



The **No of Copies** box is used to define the required number of copies of the selected element(s).

The **Start Label** is used to define the start label of the new elements. By default the next unused element label will be shown.

The **Node Option** boxes are used to dictate whether an existing node pattern is to be used or new nodes are to be created. If new nodes are to be created the Node Copy input form will appear. This enables the start node label and spatial increments to be defined.

If an existing node pattern is to be used the **Increment** box is used to define the label increment between the copied element pattern and the recipient node pattern. This recipient node pattern must be identical in pattern sequence to that of the nodes connected to the elements that are being copied otherwise unpredictable results will occur.

If an existing node pattern is to be used the **Increment** box is used to define the label increment between the copied element pattern and the recipient node pattern. This recipient node pattern must be identical in pattern sequence to that of the nodes connected to the elements that are being copied otherwise unpredictable results will occur.

Extruding 2-D Frames

When copying beam elements the **Attach Element** option of the Node Copy form will result in beams being connected between the nodes of the elements being copied and the nodes of the new element(s) e.g. 2-D frames may be extruded to 3-D frames.

-o-

Re-Number Elements (Menu:Elemts)

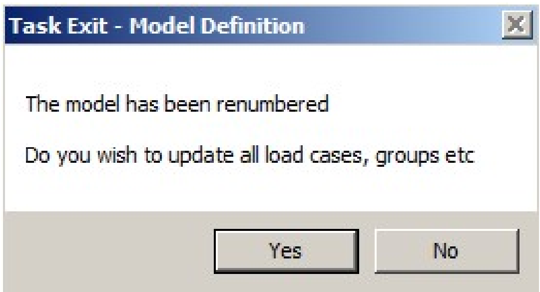
This command is used to 'pack' or compress elements. Its use is optional.

This command will remove all gaps from the element label sequence and re-numbers all elements within the specified element range. All contiguous label ranges will maintain their label sequence.

If the Model Definition TASK is exited without saving the model, all renumbering operations will be canceled.

When the model is saved an option will be given to update all secondary definition e.g. loads cases, effective lengths, groups,piles, waveloader etc.

The CO attribute for elements (EType 20 and CType 8) that use the CO parameter for node or element ID will not have that attribute renumbered.



If this update is not undertaken all such secondary definition may require to be re-defined depending upon the current re-numbering range.

It is recommended that the model by archived before any renumbering operations are undertaken.

The renumbered elements are listed in the scatch file <model>.~RENE. The RHS column is the new number and the LHS the previous. The first line lists the old max and the new max.

-o-

Connect Intersecting (Menu:Elemts)

This will connect two intersecting elements with a common node at the intersection point. A slight tolerance is applied to allow for some degree of misalignment. If this is exceeded, a message that the elements are not connected will be given.

-o-

Insert End Spring/Couple

This command is used to insert a couple element at the end of a beam element.

If a beam and node end is selected by mouse pick a couple element will be inserted at that end of the beam. A new coincident node will be created and used for the beam connectivity. The inserted couple will connect the coincident nodes.

The orientation of the couple will be by element reference. If the default **Reference Element** is non zero this will be used. Otherwise the subject beam element will be used.

The default **Property Code** (Stiff Code) will be used.

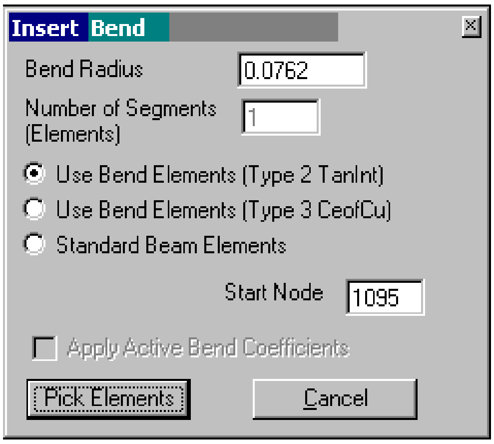
Note that the default **Reference Element** and **Property (Stiff) Code** are those currently defined in the Couple Element Definition form. If this form is visible the defaults can be changed prior to picking an element.

When inserting couple at the end of beams it is always more convenient to use the subject element as the reference element. This ensures that the couple's degrees of freedom align with the beam's degrees of freedom.

-o-

Insert bend (Menu:Elemts)

This routine will insert a bend between two connecting elements. A bend element (Type 2 or Type 3) or a segmented bend formed from a series of straight elements (Type 0) may be used to form the bend.



The **Bend Radius** defined the mean radius of the bend.

The **Use Bend Elements** selects the type of bend to be inserted. A Type 2 has the bend third node at the tangent intersection point. A Type 3 has it at the centre of curvature.

The **Number of Segments** defines how many elements will be used to form the bend. The option applies only to Type 3 bends and standard beam elements.

The **Apply Active Bend Coefficients** will apply the currently active bend coefficients. This will only be active if values have evaluated using the Bend Coefficient Input form.

When the **Pick Elements** button is clicked, two elements will be required to be selected. If the elements are connected a bend will be inserted between the two elements at the common node.

When a bend element is inserted the common node will be moved to the centre of curvature of the bend. This node is used for the third node definition of the bend and should NOT be moved.

When straight elements are used to form the bend the local rotation of the elements forming the bend will be defined by the third node method using the end node of first element selected. This ensures that the local y-axis lies in the in-plane direction of the bend. This is a requirement if Flexibility and Stress Intensification Factors are employed. The common node may be deleted following the element generation process.

-o-

Element Delete (Menu:Elemts)

This command option (Model Definition:Elem Menu) is used to delete unwanted elements. The elements to be deleted are selected using the current selection method.

-o-

Element Moment Releases (Menu:Elemts)

The Moment Releases command is used to define Moment Releases on existing elements. When this command is selected the element to be released requires to be picked, after which the node end to be released is picked.

After selection the axis of the element will be drawn and the following input box will become visible.

The blue square indicates which end of the element was selected.

To define the release simple check the appropriate input box. The release code will be updated accordingly.

When entry is complete click the **OK** button.

Torsional Releases

It torsional releases are required they can be implemented by specifying a zero or near zero value for Ix value or the G modulus value

Partial Fixity - Defined End Stiffness

Couple elements can be used to model more complex end conditions. Using couples enables both linear and non-linear effects can be included in any of the element degree of freedom.

The Insert End Spring/Couple command can be used to insert such couples.

-o-

Element - Local Rotation (Menu:Elemts)

The Local Rotation command is used to individually select an element and its third node for local Rotation Definition.

When the command is selected the mouse is used to pick and element an then pick its third node.

After picking the local rotation angle for the element will be drawn.

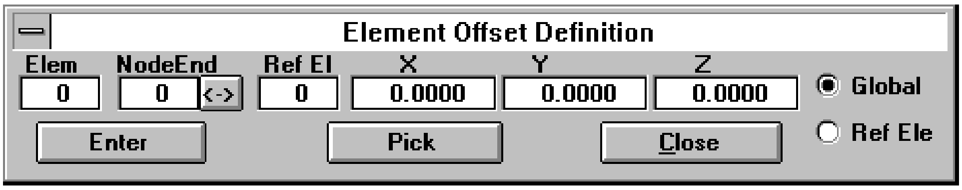
To copy rotation angles or third nodes enter them as defaults and use the Modify button of the Element Input box.

-o-

Rigid Element Offsets (Menu:Elemts)

The Offset command is used to define rigid beam Offsets

When the Offsets command is selected the following input box will become visible.



Offsets are defined on an element end basis. An element and its node end for the offset are identified and the offset is specified by defining the x, y and z distances from the node end to elastic element end.

Offset co-ordinates are defined using either the global axis or the local axis of a selected element, the Ref El. There are some restrictions when using the subject element as the reference element.

Keyboard Entry

The **Elem**, **NodeEnd** and **Ref El** boxes are used with the **Enter** button to the define the Offset by label reference. The last element listed with the Element Query button will be entered if the boxes are double clicked.

The **<- >** button can be used to switch the offset definition to the opposite node end.

Graphical Entry

If the **Pick** button is clicked the element and its node end require to be picked in that order. The **Global** and **Ref Ele** options are used only when the element is to be selected by the pick method. If the **Ref Ele** box is checked then a reference element for the co-ordinates system is also required to be picked.

The Offset will be drawn on completion of definition.

If the local rotation of an element with an offset is defined using a third node, the local rotation angle will be evaluated on the orientation of the offset position of the element.

Editing Offsets

Existing offsets are edited by entering the element label into the input box. To convert local element reference to global reference enter zero for the **Ref Ele.**

Copying Offsets

To copy Offsets use the Offset Modify box and the Modify button of the Element Input box.

The Offset modify should contain the element label of the element to be copied. Note that both ends will be copied.

If the element to be copied uses itself as a reference element then the recipient elements will each reference themselves.

Deleting Offsets

Offsets can only be deleted by using the Modify button of the Element Input box. To delete Offsets enter zero in the Offset Modify box and select the elements for which the offsets are to be removed.

-o-

Offset List (Menu:Elemts)

This command is used to list element offset data. When the command is selected the elements to be listed are picked.

Definition data relating to the element will be listed in the Listing Box at the bottom of the screen.

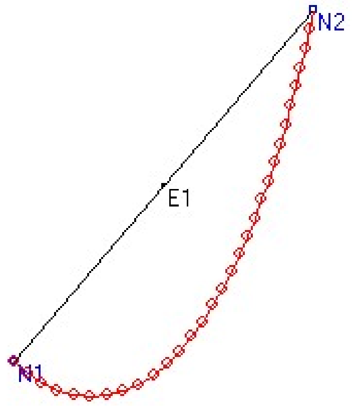
-o-

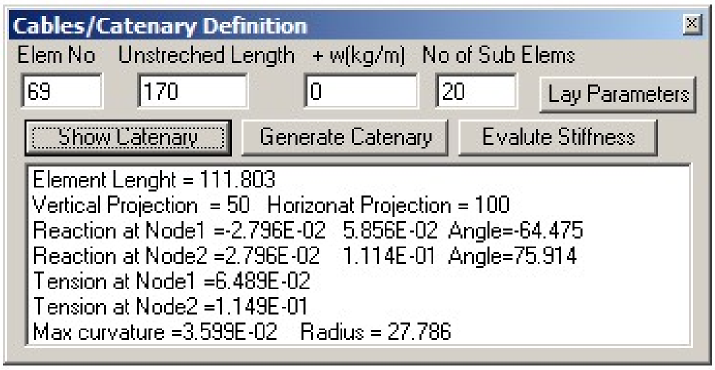
Cables/Catenaries

This routine is used to generate the properties and shape of free a catenary. It's main use is to generate a mesh of beam elements that take the form of a hanging catenary. The catenary definition is achieved by specifying the two anchor points and the initial unstrained length. An option to evaluate the anchor point locations and initial length based on a bottom tension and height is also available. This feature is useful for anchor chain or pipe lay applications.

This mesh can used is subsequent analysis solution to asses the deflections and loading in a model that includes catenary effects i.e. tension stiffening. This solution requires a large displacement or a P-Delta/ large displacement solution. They can be solve using the Standard 3-D solver with P-Delta active.

This routine can also be used evaluate loading and properties of a catenary in an interactive mode. By adjusting the initial un-stretched length any number of catenary shapes can be generated.





To use the utility at least one Type 15 beam element must first be defined. The element length represents the chord distance between the two anchor points. The

Elem No is element label of the subject Type 15 beam element.

Unstretched Lenght is the initial unstrained length of the catenary. This should always be larger then the element length (catenary chord length).

+ w(kg/m) is the additional weight of the cable. This value will be added to the self generated weight.

No of Sub defined the number of sub-divisions (elements) used to generate the catenary. Only the accuracy of the evaluated curvature is dependent upon the no of sub divisions.

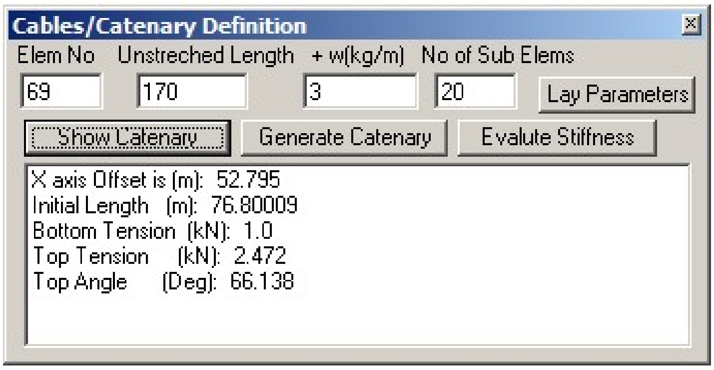
The **Show Catenary** button is used generate the properties and show the shape of the catenary

The **Generate Catenary** button will generate a mesh of Type 0 Beam elements.

The **Evaluate Stiffness** button will display the stiffness coefficients for the catenary. Usefull if the catenary action

The **Lay Parameter** button is used to evaluate the parameters for a catenary with a defined bottom tension and and zero angle e.g. anchor chain/pipelay touchdown configuration. The catenary height is taken to be that of the Y coord difference between the nodes that define the subject element. A user prompt will request the bottom tension. Note that only the weight and Elem No are required for this facility.

An option to update the setting will also be given. If this is selected the aft node will be moved to the offset location and the **Unstretched Length** will be updated, this enables the lay catenary to be easily shown/generated.



-o-

Pipework Definition (Menu:Elemts)

This command gives access to the Pipework Definition Sub-menu This menu enables pipe flexibility and stress intensification factored to be included into the analysis.   This factors are defined in accordance with ASME B31.3 and are applicable to most national piping design codes.

See Pipework Orientation for the convention for in-plane and out of plane coefficients.

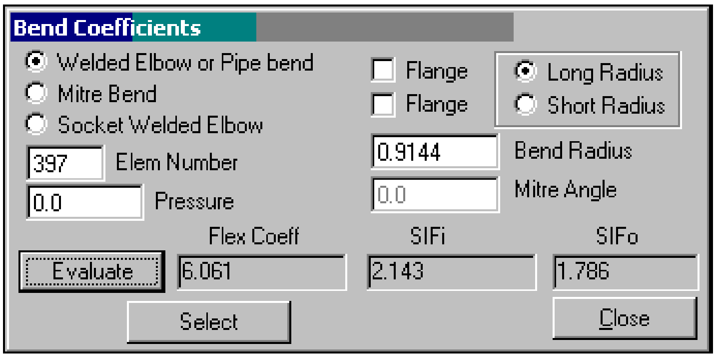
This menu has the following commands.

Bends	Defines data relating to pipe bends
Tee/Connections	Defines data relating to tee and other standard connections
ListPipe	List pipework coefficients in List Box
Delete Coefficients	Removes the pipework coefficients from the element(s)

-o-

Pipework Bends

This form is used to define factors associated with bend elements.



The **Elem Number** is used to enter the element number of the bend element.

If the **Elem Number** box is double clicked the last element listed with the Elem Query button will be entered.

The **Evaluate** button is used to evaluate the factors based on the data and option selected on the form . The factors are evauated in accordance with ASME B31.3 -2006. These will be shown in the boxes. This button does not apply the factors to the element. It uses the element as a template for factor evaluation. Note that both element Geometric and Material properties must be defined for the factors to be evaluated.

The **Pressure** box is used to include the effects of pressure stiffening. The units for this are N/m2 if SI units are being used.

The **Select** button is used to copy the factors to elements using the current SelectBY method. Active the Pipe Coefficients Display Switch to highlight elements with coefficients.

The **Long** and **Short Radius** options will automatically enter the Band Radius based on the OD of the pipe. The radius may be re-entered. For large diameter pipes (API 14" (355mm) and above) this will produce the correct radius since the nominal pipe size is the OD. For smaller API pipes and for other pipe specs ensure that the Radius is correct (API uses a nominal pipe diameter for small pipes)

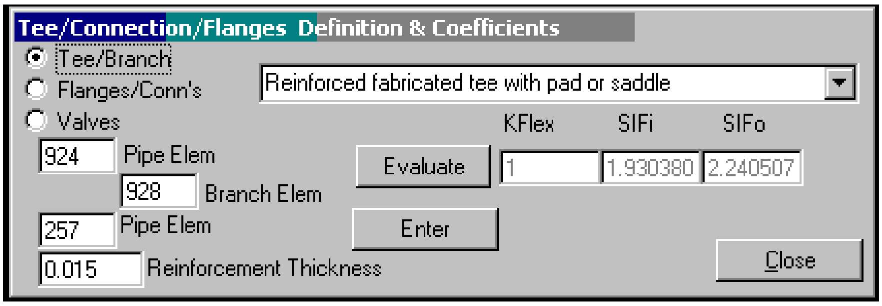
See Pipework Orientation for the convention for in-plane and out of plane

-o-

Pipework Tee/Connections

This form is used to define coefficients to branch connection and in-line connections.

The form below shows the form with **Tee/Branch** option active.



The pull down list is used to select the type of connection

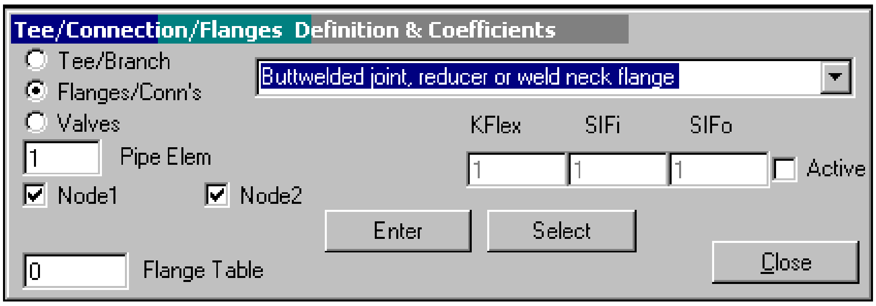
The **Pipe Elem** and **Branch** boxes are use to define the elements that form the branch.

If the **Pipe Elem** and **Branch** box are double clicked the last element listed with the Elem Query button will be entered.

The **Evaluate** button is used to evaluate and display the coefficients using the current settings. The factors are evauated in accordance with ASME B31.3 -2006.

The **Enter** button is use to assign the coefficients to the defined elements.

The form below shows the form with **Flange/Connection** option active. This form is also used for **User Defined** factors definition.



The pull down list is used to select the type of connection. Note that there is a **User Defined** option.

The **Pipe Elem** is use to define the element with the connection. If the **Pipe Elem** box is double clicked the last element listed with the Elem Query button will be entered. This would not be required to be done if the **Select** option is used.

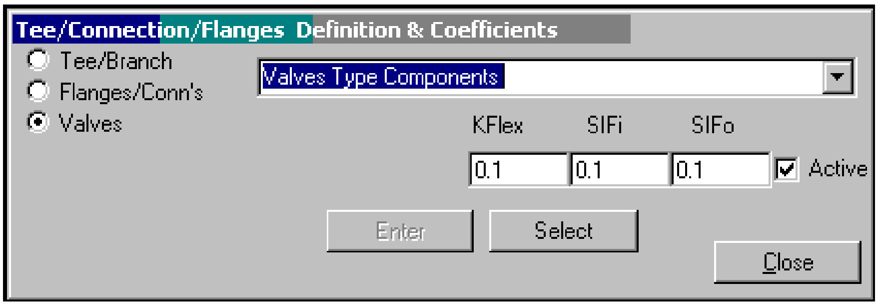
The **Node1** and **Node2** check boxes are use to define to which node end of the element the factors are to be applied.

The **Active** option enables the factors to be editied used only when **User Defined** fittings are selected.

The **Enter** button is use to assign the coefficients to the defined elements.

The **Select** button can be used as an alternative method of assignment. When clicked the user is required the select the nodes representing the flanges.

The **Flange Table** is used to reference entries in the Flange Design module (optional use). Note that the Flange Design module (FlangeCHK.exe) is started from the FS2000 Windows menu.



The form above shows the form with **Valve** option active.

When the valve option is initially activated the KFlex factor will be set to 0.1 i.e. the valve is made 10 times the stiffness of the equivalent pipe and it has no mass(weight). The user MUST define the weight effects of valves by definition in a load case. Note that if extended properties are not in use the weight of the valve will be that of the equivalent pipe.

When the **Select** button is clicked the user selects the elements that are to be assigned as valves.

See Pipework Orientation for the convention for in-plane and out of plane

-o-

Pipework Orientation:In-plane/Out-Plane

When defining pipe stress factors it is necessary to define in-plane and out of plane directions terms of the element local orientation.

The convention adopted by the program is that In-Plane is in the plane of the major axis and Out of plane is in the plane of the minor axis.

This means that it will often be necessary to rotate the elements to ensure that this is the case.

-o-

ListPipe Display Pipework Coefficients

When elements are selected the pipework coefficients will be displayed in the in List Box

-o-

Delete Coefficients

Removes the pipework coefficients from the element(s). Elements are selected using the current SelctBy method

-o-

Model Definition:Menu:FE-Solids

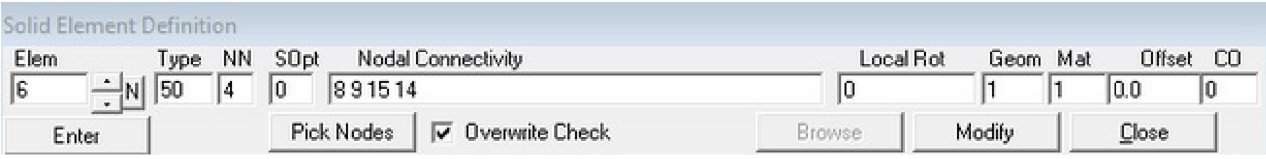
This menu provides the model definition commands associated with element definition

Input/Modify	Makes the primary Element Input dialogue box visible for element definition
Sub Mesh	Sub divides existing elements into a mapped mesh
Sub Mesh-Transitions	Sub divides quad elements into specific mesh patterns
Copy	Copies selected elements and optionally, attached nodes
Delete	Deletes selected elements
Move	Re-connect a selected element to different nodes
Merge (Nodes)	Eliminates coincident nodes and warns of coincident elements
Re-Number	Remove gaps in label sequence and re-numbers all elements
Extrude 2-D to 3-D	Form 3-D elements from 2-D elements
Reverse Normals	Renumber the elements so that the local z-axis is reversed

-o-

FE-Solids Input/Modify (Menu:FE-Solids)

The following input box will become visible when the FE-Solid Input/Modify command is selected. This form is used to define Solid Element related data.



When the box is first visible the default element label is the next unused element label.

Important: When selecting nodes for the element, always ensure the node order sequence is correct. Information for each element type can be found in the element descriptions.

Keyboard Definition

To define an element, enter the element number and the appropriate definition parameters and click the **Enter** button.

Graphical Definition

It is more often convenient to define the connectivity of the element i.e. nodes of the element by picking nodes from the screen. To do this click the **Pick Node** button and then select the nodes for the element from the visible nodes. When picking nodes any data in the Node Connectivity box will be ignored. When picking nodes the **Enter** button is not to be used.

The **Type** box is used to define the element type to be input - see element description for details. The adjacent box defines the number of nodes of the element. Use the **Browse** button to select the element type'

The **SOpt** box is used to specify the solution option for the element type - see element description for details. 0 is the default and is unlikely to be required to be changed.

The **Overwrite Check** is a safeguard to prevent the re-definition of existing elements.

If an existing element number is entered in the **Elem** box, its parameters will be displayed in the other boxes.

If the **N** button is pressed, the next unused element label will appear without changing the other parameters. This is useful for copying parameters from existing elements.

If the **Elem** box is double clicked using the **Left Mouse Button**, the last element listed with the Elem Query button will be entered. This is useful for editing existing elements.

The **Local Rot** box is used to define the local orientation of the element.

The **Geom** box is used to assign the geometric properties to an entry in the Geometric Property Table. The element thickness is specified in the Area(t) box of the Geometric Property input form.

The **Mat** box is used to assign the material properties to an entry in the Material Property Table.

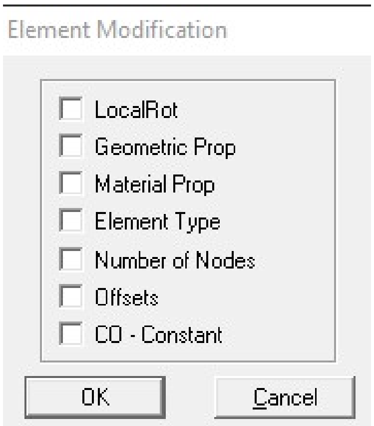
The **Offset** box is used the define rigid offsets for shell elemnts. The offset is in the local z coordinate and is applied to all elements nodes. It is only applicable to to shell elements.Offsets can be copied or deleted(entry=0) using the element **Modify** button.

The **CO** box is used to define the Winkler support stiffness for shell elements.The element CO property indirectly defines the value of k by referencing an entry in the RC Table in which RC-X1 defines the magnitude of k. If k is defined as a -ve value than the foundation will only apply compressive support (non-linear solution only).

The **Browse** button is used to browse the current property tables of the model. The tables can be modified when browsing.

Most of the parameters of the input box when entered become default values e.g. used when attaching elements to nodes as the nodes are being created.

The **Modify** button is used to copy the defaults to existing elements. The modification process is selective using the active **SelectBy** method.The following will appear so the attribute(s) to be modified can be identified. If the parameter box is not checked it will not be modified.



When the OK button is clicked the elements to be modified are selected by the active selection method.

-o-

Sub Meshing (Menu:FE-Solids)

The Sub Mesh dialogue box enables a mesh of elements to created within the boundary profile of an existing element (region element). The region element can be either 4 or 8 node 2-D quadrilateral element for area meshes, an 8 node solid element for volume meshes or a 2 node line element. If an 8 node quad element is used the mesh will be mapped to a curved parabolic profile based on the mid side nodes.

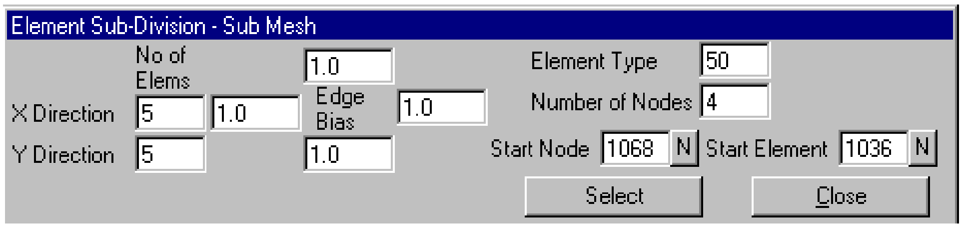
The region element is be deleted when the mesh is created.

Coordinate Systems The generated mesh will be mapped to the active co-ordinate system therefore always ensure that the nodes of the region element are in defined in the same coordinate system as the active system.

Element Properties The element will be assigned the default properties of the element input form.

A basic procedure for model generation is:

- Create coarse mesh of region element to define the basic regions of the model (Archive this model).
- Use the Sub mesh tool to mesh each of the regions (elements) taking care to match the boundary node of adjacent regions.
- Use the Merge command to join adjacent regions to each other.



The **Element Type** box is used to define the type of element to be meshed. This is independent of the region element type.

The **Number of Nodes** box is used to define the number of node per element.

The mesh tool is capable of generating 3 Noded, 4 Noded and 8 Nodes area meshes in 3-D space and 8 noded volume meshes.

The **Start Node** and **Start Element** define the start labels for the mesh.

The **No of Elems** boxes are used to define the number of elements on the boundary of the mesh. The x and y directions refer to the local axis of the region element.

The **Edge Bias** boxes are used to produce a mesh with elements of varying size. If a value of 3 is used then the elements at that side will be 3 times larger than the side using a unity bias. If 8 noded region elements are used, additional bias can be applied by offsetting the mid side node.

When the **Select** button is clicked the user is required to pick an individual element for meshing. On selection the resulting mesh will be drawn. User confirmation is required before the mesh is converted to actual nodes and elements.

Meshing Basic Pipe Sections

The Frame Wizard has a utility that will generate basic pipe sections using quad shell elements. The utility can generate the following basic pipe related shapes:

- Pipe Bend (Elbow)
- Straight pipe
- Conical pipe

The Frame Wizard is started from a command in the **TASK(Model Definition)** menu.

-o-

Merge (Nodes) (Menu:FE-Solids)

This command is used to merge elements edges and faces. It does it by merging coincident nodes to a single node and then connecting all elements attached to the eliminated node(s) to that node. Coincident nodes are nodes within a defined spatial tolerance of the subject node.

This command will also warn if coincident elements exist.

It is a very useful command to use after copying and translating elements to different location on a model since it can easily remove duplicate elements and nodes which can often occur when undertaking such tasks.

Elements to be processed are selected by the current SectBy method.

When the command is selected a coincident node tolerance is required to be defined. If the spatial position of adjacent nodes is within this tolerance the nodes are regarded as coincident. The default value is 0.01.

For a selected element the merging process works as follows.All nodes are scanned to identify if coincident nodes exist. If one does exist, the element node will be re-assigned to the coincident node with the lowest label. If the eliminated node is not attached to any other part of the model it will be deleted.

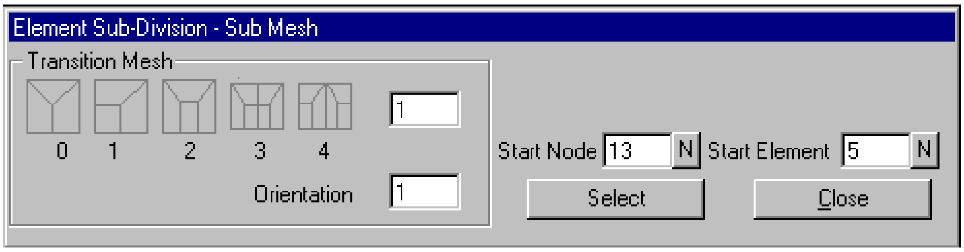
Merging Solid Elements with Beam Elements

Only if the label numbers of the beam elements are less than those of the selected solid elements will the nodes of beam and solid elements be merged. In cases where they this is not the case and the elements are required to be merged use the Merge command from the beam elements menu to merge the elements. Using the merge command successively from both the from FE-Solids menu and the Beam Elem menu will ensure that all elements are merged.

-o-

Sub Mesh - Transitions (Menu:FE-Solids)

This routine is used to sub divide existing quad elements into specific mesh patterns. It main use is to create transition meshes between areas of different mesh density.



If the **Pattern** is clicked the pattern reference number will be entered into the input box.

The **Orientation** number is used to define the mesh orientation relative to the start node of the element. The datum point on the mesh is the lower LHS.   A value of 1 will align this to the first node of element being meshed. A value of 2 will align it to the second node and so on.

-o-

Extrude 2-D to 3-D (Menu:FE-Solids)

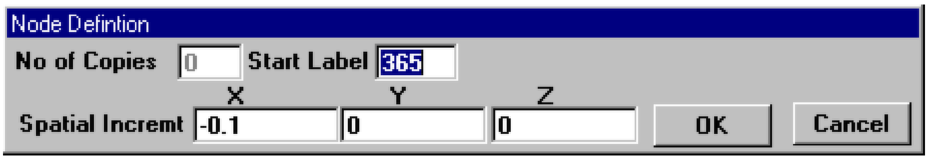
This routine is used to form 8 Nodes brick elements from 4 Node flat elements by extruding the 2-D plane shape into a 3-D solid shape. The third dimension is defined by specifying x, y, & z spatial increments. The increments are interpreted in the coordinate system of the nodes of the element therefore always ensure that all nodes are in the same coordinate system

When specifying the direction of extrusion ensure that the element numbering sequence is correct.

The element to be extruded can be of any element type with 4 nodes. The resulting element will be a Type 70 - 8 noded brick element.

Only a single layer can be extruded. If additional layers are required then the Copy routine should be subsequently used.

When the command is selected the used is required to select the elements to be extruded using the current SelectBy method. Following selection the node increment form will appear.



The **Start Label** is used to specify the start label for the new nodes.

The X, Y & Z boxes specify the spatial increments for the new nodes. The increments are in the active co-ordinate system. e.g. if a Type 1 (Cylindrical) system is active then Y would be defined in degrees.

-o-

Reverse Normals (Menu:FE-Solids)

This will renumber the elements so that the local z-axis is reversed. It does by simply reversing the node numbering direction. Its most common use is to ensure that the top face of shell elements are in a consistent direction. This is often a requirement when shell elements are reflected across an axis.

It can also be used to correct 2-D solid elements that have been incorrectly number in a clockwise direction.

When applied to 3-D solid element it will reverse the IJKJ and MNOP faces. MNOP faces always have to be in the local positive Z direction relative to the IJKL face.

-o-

Copy SelectBy (Menu:FE-Solids)

This routine is used to copy existing elements. The elements may be copied on to existing node patterns or new nodes can be created as the elements are copied.

If new nodes are created it is more than likely that the Merge command will be required to be used once the elements are created. This is because the new elements will not use common nodes between the existing elements or between new element blocks when multiple copies are done.

Element and Node Groups

When copying elements the new elements and nodes will be assigned to the respective active group. If no groups are active they will be assigned to the group attribute of the element and node being copied.

On selection of the command the element(s) to be copied require to be selected. The current selection method is used for selection. After selection the following input box will become visible.

The **No of Copies** box is used to define the required number of copies of the selected element(s).

The **Start Label** is used to define the start label of the new elements. By default the next unused element label will be shown.

The **Node Option** boxes are used to dictate whether an existing node pattern is to be used or new nodes are to be created.

If an existing node pattern is to be used the **Increment** box is used to define the label increment between the copied element pattern and the recipient node pattern. The recipient node pattern must be identical in pattern sequence to that of the nodes connected to the elements that are being copied otherwise unpredictable results will occur.

If **New Nodes** are to be created the Node Copy input form will appear. This enables the start node label and spatial increments to be defined. The spatial increments must be defined in the same coordinate system as nodes of the element. All element nodes of the element being copied must be in the same coordinate system.

-o-

Model Definition:Menu:Couple

This menu provides the model definition commands associated with spring/couple definition

Input/Modify	Makes the primary Couple Input dialogue box visible for couple definition
Generate on Nodes	Defines couples by label pattern on an existing node label pattern
Copy	Copies selected couples and optionally, attached nodes
Delete	Deletes selected couples
Re-number	Used to 'pack' or compress all couple elements
Move	Move existing couple by selecting new nodes
Connect Coin Nodes	Connects selected coincident nodes using the default couple properties and a specified capture tolerance
Reverse Connectivity	Switches the node connectivity on couple (Mouse pick)
Select Reference Element	Define reference elements for couple using Mouse picks

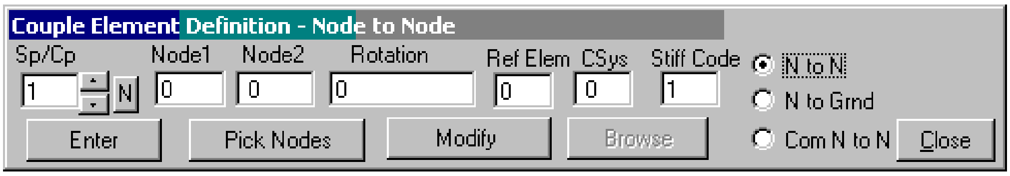
-o-

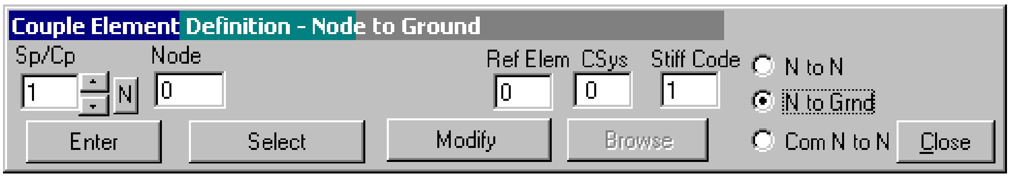
Couple Input/Modification (Menu:Couple)

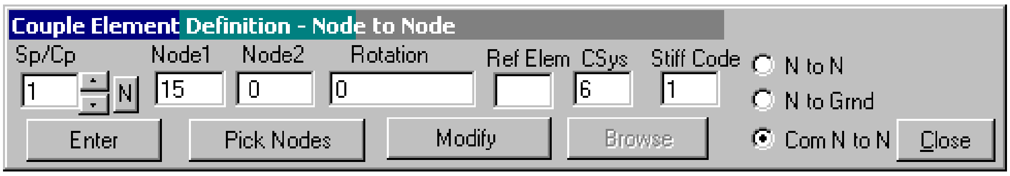
The command enables couple elements to be defined. Section 5 presents a description of couple elements.

The type of couple element is governed by the CType property in the couple constants property table. The most common type are linear spring couples (CType=0) elements and Gap elements.

The following input box will become visible when the Couple Input/Modify command is selected.







The **N to N** or **N to Grnd** are used to set the type of Couple element to be define. If the **N to N** option is active a Node to Node couple is assumed and two nodes are required to be defined (or picked). If the **N to Grnd** is active a Node to Ground Couple is assumed and only one node is required to be defined (or picked). If the **Com n to N** is active a Node to Node couple is assumed, the entry in Node1 defined the start node of the couple and the only one node is required to be defined (or picked).

When in the Node to Ground mode the form is as shown above. In this case the single node for each couple is selected using the current SelectBy method. When in the Node to Node mode the method is the next paragraph is used. In this case the **Select** button is replaced by a **Pick Nodes** button. Note that Node to Ground couples are identified by defining Node1 and Node as the same node and can be defined by the latter mode by picking the same node twice.

To define a spring/couple, enter the element number and the appropriate definition parameters and click the **Enter** button. It is more often convenient to define the connectivity of the element i.e. Node1 and Node2 of the element by picking nodes from the screen. To do this, simply click the **Pick Node** button and then select Node1 and Node2 for the element from the visible nodes. When picking nodes any data in the Node1 or Node2 box will be ignored. Do not use the Enter button when picking nodes.

When the box is first visible the default couple label is the next unused couple label.

If the **Spring** box is double clicked using the **Right Mouse Button**, the next unused element label will appear without changing the other parameters. This is useful for copying parameters from existing elements.

If the **Spring** box is double clicked using the **Left Mouse Button**, the last spring/couple listed with the Elem Query button will be entered. This is useful for editing existing spring/couples.

If an existing element number is entered in the **Spring** box, its parameters will be displayed in the other boxes.

The **Local Rot** box is used to define the local orientation of the couple when its orientation is defined by its connecting nodes.

The **Ref Elem** box is used to define the local orientation of the couple by referencing it to the orientation of an existing beam or pipe element.

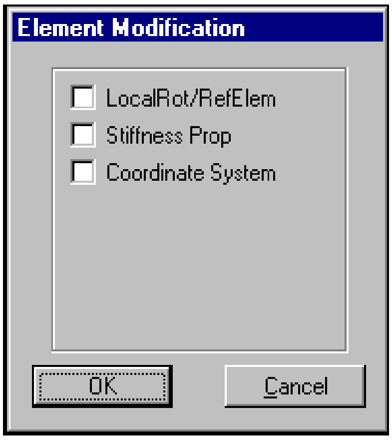
The **CSys** box is used to define the local orientation of the couple by referencing it to an existing coordinate system.

If the nodes of the couple are coincident and the value **Ref Elem** and **CSys** entries are zero the couple will be aligned with the global axis.

The **Stiff Code** box defines the stiffness properties of the couple by referencing it to the Couple Contants Tables. Gap elements are defined by these constants.

The **Browse** button is used to browse the current couple constants table of the model. The tables can be modified when browsing.

Most of the parameters of the input box when entered become default values e.g. used when attaching elements to nodes as the nodes are being created. The **Modify** button is used to copy these defaults to existing elements. The modification process is selective, the following will appear so the attribute(s) to be modified can identified. If the parameter box is not checked it will not be modified.



When the **OK** button is clicked the spring/couples to be modified are selected by the current selection method.

-o-

Couples - Line Generate on Nodes (Menu:Couple)

This is identical to the Line Generation of beam elements on nodes. It is seldom used for couple definition.

-o-

Couple - Copy (Menu:Couple)

This is identical to the Elements - Copy for standard beam elements.

-o-

Couple - Delete (Menu:Couple)

This command option (Model Definition: Couple menu) is used to delete unwanted couples. The elements to be deleted are selected using the current selection method.

Use the View Menu:Model Display Switches command to make the spring/couples visible.

-o-

Model Definition:Menu:Rest

This menu provides the model definition commands associated with model restraint definition.

Define	Makes the primary Restraint Input dialogue box visible for restraint definition
Delete	Removes the restraints from selected nodes

Restraints and Prescribed Displacement Reactions

There are sometimes slight difference between the output obtained for restraint and prescribed displacement reactions depending upon whether the solution was undertaken using the linear solver or a non-linear solver. In both solvers such a load would no effect of the displaced solution and the loads and stresses would be identical.

If Nodal Forces are applied in a restrained direction (same DOF)

Linear: *It will be included in the restraint list even though it results in no element loading. The Nodal Forces will be included in the restraint summation.*
Non-Linear : *It will not be included in the restraint list. The Nodal Forces will not be included in the restraint summation and will therefore be different from the input load summation.*

If Prescribed Displacements added to a Restrained Node (different DOF)

Linear *PDs will be included in the Restraint Reaction List and vica-versa.*
Non-Linear *PDs will only be shown in the PD List*

Non-Linear Solution with Solid Elements

If the model contains shell elements (Type 50 or 53) then the defined element load contribution e.g. gravity to a restraint on an element from that element will not be included in the restraint force (small effect for practical meshes). This only effects direct loads not moments.

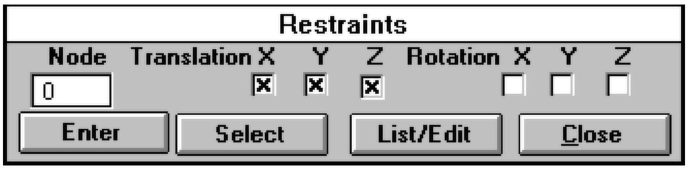
If the model contains solid elements, the reactions will not be evaluated or listed when using a non-linear solver.

If reaction magnitudes are a solution requirement then the model should be use Ground Couples to provide the model restraint. These will be listed and can be plotted in the same manner as restraints.

-o-

Restraint Input (Menu:Rest)

When the restraint Define command is selected the following input box becomes visible.



The check boxes are used to identify the restrained freedoms.

The **Node** box and the **Enter** button are used to identify the node to be restrained by label reference.

If the **Select** button is clicked the nodes to be restrained will be selected by the current selection method.

The restraints will be plotted as they are applied.

This **Edit/List** button enables existing nodal restraints to be listed, edited or copied. When the button is pressed the mouse is used to select the restraint (node) is to be listed.

The restraint freedoms can now be modified or copied as required using label reference or the current selection method.

-o-

Restraint Delete (Menu:Rest)

This command option is used to remove unwanted restraints (all freedoms) from selected nodes. The nodes are selected using the current selection method.

-o-

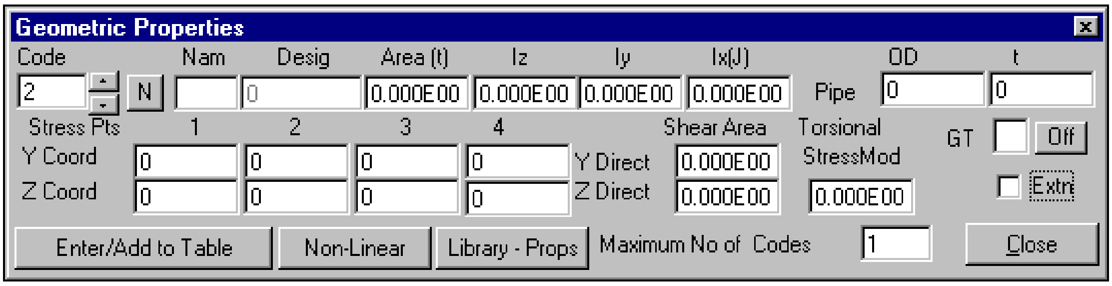
Model Definition:Menu:Prop

Geometric	Makes the primary Geometric Properties Input dialogue box visible for property definition
Non-Linear Properties	
Property Libraries	
Property Generation Utility	
Material	Makes the primary Material Properties Input dialogue box visible for property definition
Couple Ele Const	Makes the primary Couple Properties Input dialogue box visible for constant definition
IC(Constants)	Makes the IC(Constants) Input dialogue box visible for constant definition
RC(Constants)	Makes the RC(Constants) Input dialogue box visible for constant definition
Delete Unused Properties	Removes all unused properties from all property tables.

-o-

Geometric Property Tables (Menu:Prop)

The command is used the make the Geometric Properties input box visible. The input box is used to add/edit entries in the Geometric Property Table.



The **Code** box identified the table entry. The default is the next unused code.

The number of entries in the table is shown in the **Maximum No of Codes** box. When a model is saved this value will be updated (and the table extended) to show the maximum code referenced by the elements of the model.

If an existing code number is entered in the **Code** box, its parameters will be displayed in the other boxes.

If the **N** button is pressed the next unused Code label will appear without changing the data properties. This can be useful for copying existing code data.

The **Nam** box uses a 3 character ID to library reference (libraries use a 3 character name). Properties not selected from libraries will have USD (User Defined) or PIP (Pipe) entered. The **Desig** box is used to identify the library entry. Neither of these can be user defined.

The remaining boxes are used to enter the properties.

The thickness of ***shell and plane stress elements*** are defined using the **Area(t)** box.

If entries are made to the **OD** and the **t** boxes then all the entries will be updated to show the properties relating to the corresponding pipe size.

The **GT** box is used to define the graphic element type. This property and the stress point data defines how the element will appear when the Beam Virtual View is active. The **Off** button enables a graphic offset to be defined. For more information on graphic views.

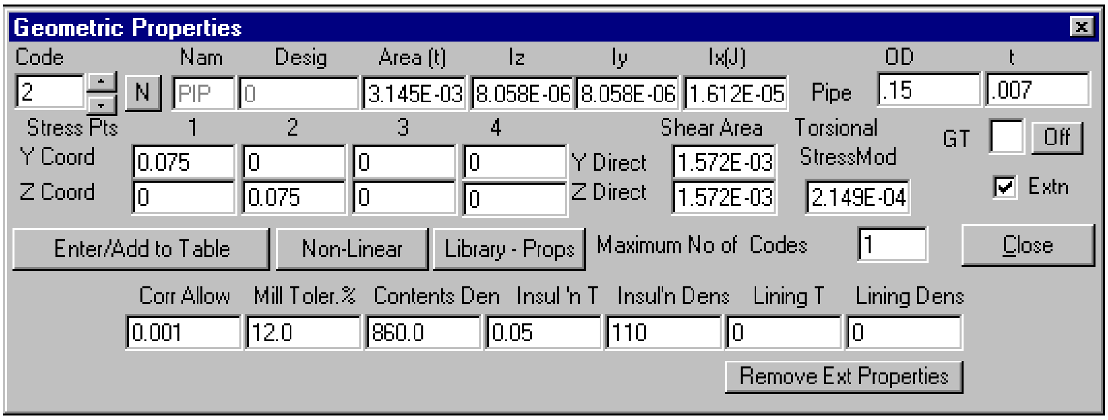
The **Enter/Add** to Table button is used to enter the current setting into the table.

The **Non Linear** button is used to access additional Non-Linear Properties relating to the some non-linear properties.

The **Library Prop** button is used to access the Geometric Property Libraries

The **Ext** option is used to extend the form and show extended geometric property data that is more specific to pipework analysis. If the extended form is visible and the model is then saved within the current Model Definition Task additional data relating to the following properties will be written to the model data files and included in the analysis. The **Corrosion Allowance** and the **Mill Tolerance** data are only used by the pipework design module FS-Pipe. The other extended data is related to mass loading and will be used in any type of analysis that involve load effects. The **Insul'n** is effectively an external coating and its effect on hydrostatic and hydrodynamic loading (OD and displacement) is accounted for in FS-Wave/Wind.

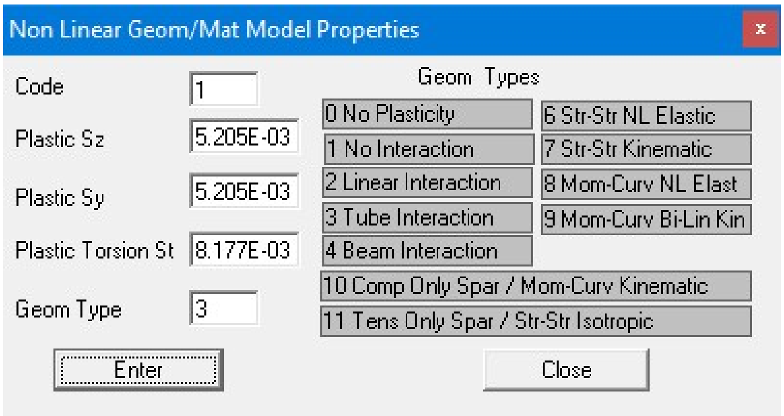
The **Remove Ext Properties** button can be used to remove all existing extended property data.


Geometric Properties
Code
Nam
Desig
Area (t)
Iz
Iy
Ix(J)
OD
t
2
N
PIP
0
3.145E-03
8.058E-06
8.058E-06
1.612E-05
Pipe
.15
.007
Stress Pts
1
2
3
4
Shear Area
Torsional
GT
Off
Y Coord
0.075
0
0
0
Y Direct
1.572E-03
StressMod
Z Coord
0
0.075
0
0
Z Direct
1.572E-03
2.149E-04
Extn
Enter/Add to Table
Non-Linear
Library - Props
Maximum No of Codes
1
Close
Corr Allow
Mill Toler.%
Contents Den
Insul 'n T
Insul'n Dens
Lining T
Lining Dens
0.001
12.0
860.0
0.05
110
0
0
Remove Ext Properties

-o-

Sect 8 Non Linear Geometric Properties

The input box is used to define non-linear behavioral properties used in non-linear analysis that are associated to the geometric property codes. No data in this box is not used in linear analysis.



The **Code** box identified the table entry.

The plastic properties for library selections will be entered automatically.

The **Geom Type** box is used to identify:

- The non-linear characteristics of a Type 6 beam element.
- Compression/Tension Only action in all Beam Elements Types (only applicable to 3-D Standard Solution).
- Compression/Tension Only action in Type 15 beam (Spar) elements.
- The plastic properties for Type 15 beam (Spar) elements.
- The plastic properties for finite elements (see below).

DyNoFlex Plasticity Model for Solid & Shell Elements

The **Geom Type** defined the elasto-plastic model. A zero value implies the Von-Mises model.

Only the Von-Mises option should be used for plane stress solutions

1 Tresca (YS & Et) : For Soils YS=√3.C where C is the soil cohesion

2 Von-Mises (YS & Et) : For Soils YS=√3.C where C the is soil cohesion

3 Mohr-Coulomb (YS & ϕ) : YS=C where C is the soil cohesion

4 Drucker-Prager(YS & ϕ) : YS=C where C is the soil cohesion

Properties

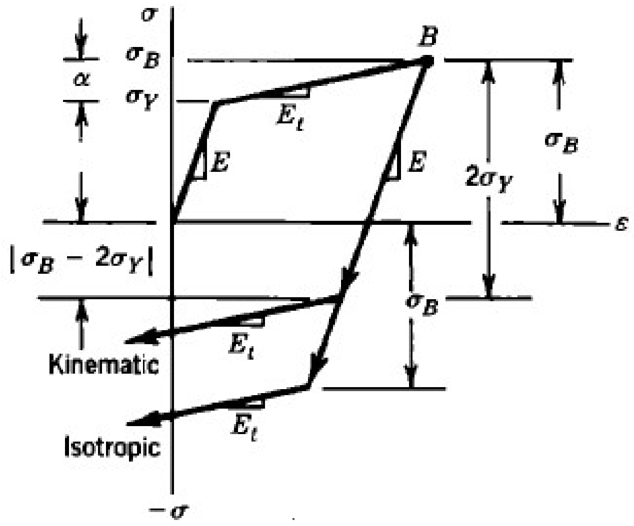
The material property code Yield Stress defines the YS limit parameter.

Plastic Sz defines the friction angle ϕ or the Number of Layers in a Type 51/52 shell element.

Plastic Sy defines Et which is used by the program to evaluate the linear hardness parameter H (H = Et / (1 - Et/E)) for a bi-linear stress-strain curve.

If **Sy** is entered a negative value then for 2-D and 3-D Solids (not shells) it is interpreted as an entry in the RC table for definition of a piecewise curve or a Ramberg-Osgood stress-strain type relationship .

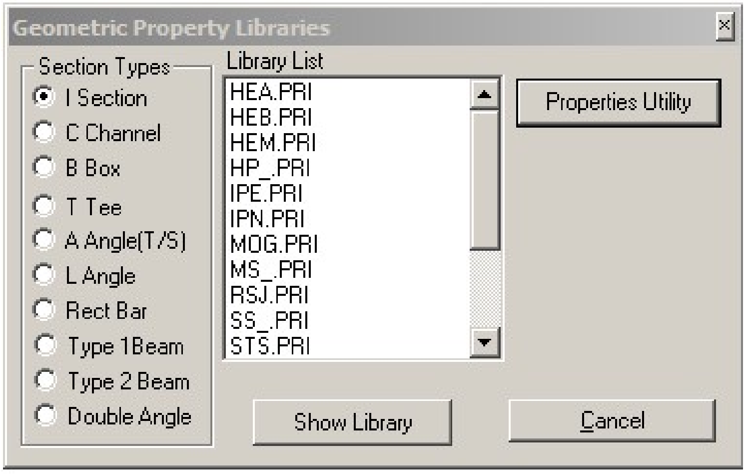
Plastic Torsion St defines the Dilation angle



-o-

Geometric Property Libraries

This dialogue box is used to enable the selection of property data from existing Geometric Property Libraries.



The library type is selected from the **Section Types** option box. Existing libraries for the chosen section type are shown in the **Library List** box.

To select a library, simply highlight it with the mouse then click the **Show Library** button. Alternatively, double click the library entry.

When a library is selected the library list box will appear. The section properties to be loaded are selected from the list of entries in the library. Selection only loads the properties into the Geometric Properties box. To add the entry to the model property table the Enter/Add to Table button in the Geometric Properties box requires to be pressed.

If model dependent libraries exist they will be shown in the Library List under the model name.

The **Properties Utility** button, if clicked, will open the Geometric Property Generation Utility. This utility generates section properties for various section configurations.

The USD Buffer library is a scratch library that contains only one entry, the entry last added to it from within the generation utility. The USD Buffer is used to transfer properties from the Properties Utility to the model without using permanent library files. This is not a recommended method since the data that defined the properties is not traceable.

-o-

Geometric Property Generation Utility

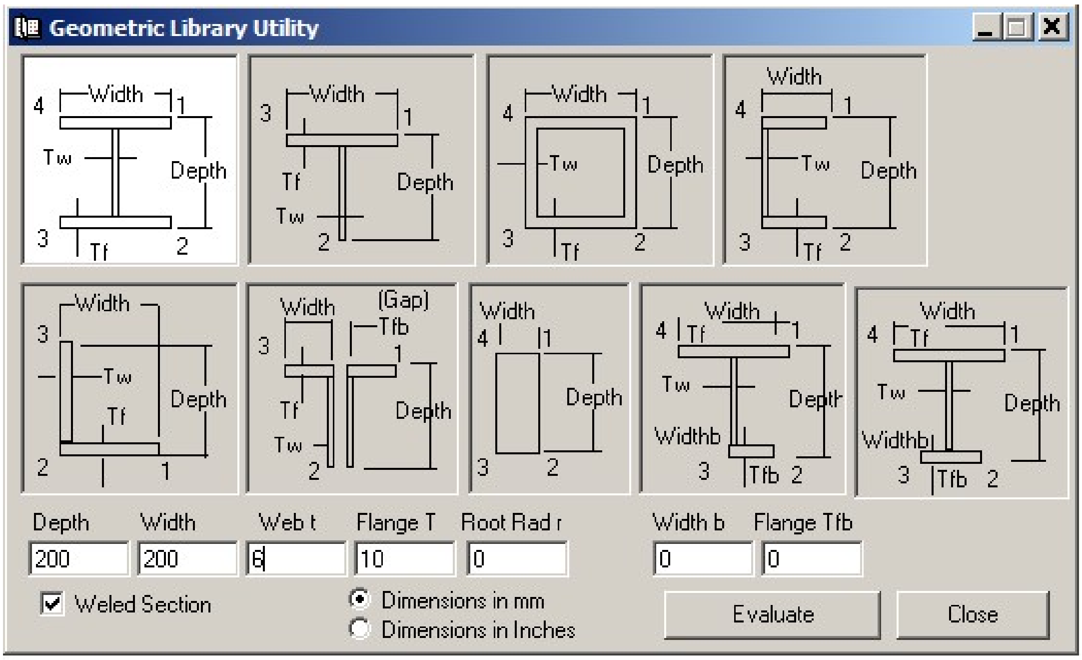
This utility is used to generate sections properties and add them to user defined Geometric Section Libraries. If libraries do not exist they will be created.

The utility is started from the Geometric Properties Libraries form or from the Windows Start menu.

When the utility is started the following input box will become visible.

Note: This utility is an external windows application. If focus is lost the window may become a background window and be unable to be seen.

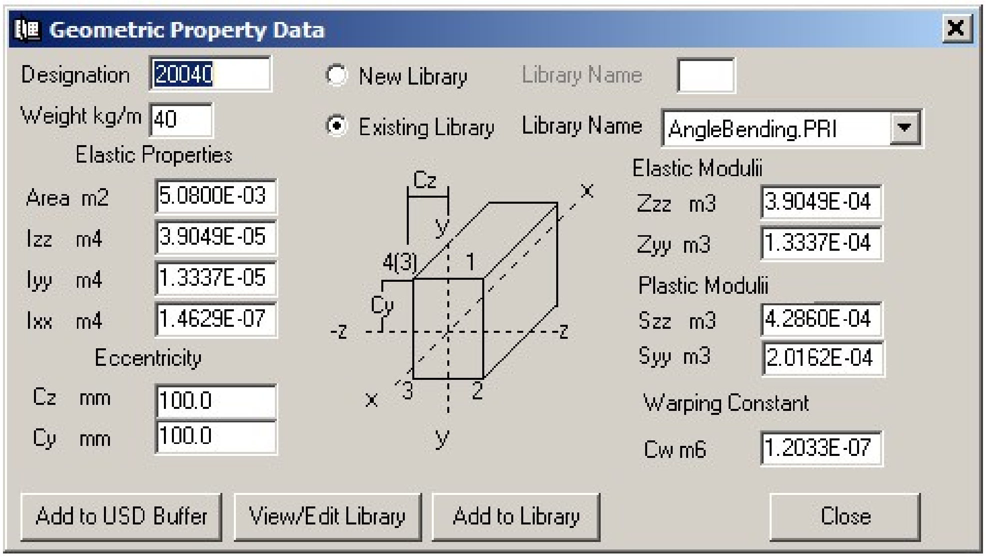
See Geometric Property Tables for a commentary on property evaluation.



To select a section type simple click the image frame. The Stress Point numbering convention is shown in the image.

Enter the appropriate data and click the **Evaluate** button to obtain the properties.

On evaluation the following dialogue box will be visible.



The **Designation** box is used to add a unique numeric library designation (up to 9 digits).

If the **New Library** option is checked the library name must be entered in the **Library Name** box. The library name must be a 2 or 3 character name.

The library name and the designation are used in the program as a description for the section e.g UB 40617874 could be seen for a section selected from the standard UB library.

If the **Existing Library** option is checked the library can be selected from the **Library Name** list box. Read only libraries will not be shown. The model dependent library is shown under the model name. This will always be present whether it exists or not.

The **Add to Library** button is used to add the data to the selected library.

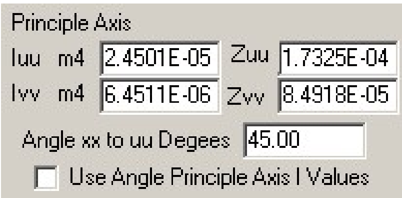
The **View/Edit Library** button enables the library to be viewed. When being viewed section can be deleted from the library.

The **Add to USD** Buffer adds the data to the USD buffer. The USD buffer would be used when properties are to be transferred to the model without saving the source data.

The **Previous** button returns to the previous data entry dialogue box.

Angle Principles Axis

When angle properties are generated the is an option to use the principle axis I values as opposes the to usual geometric axis I values.

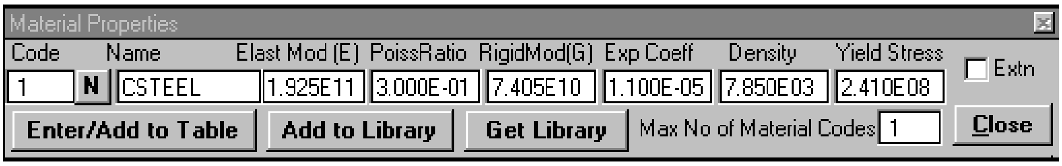


Note that tis does not affect the design code checking of angles which always use the principle axis.

-o-

Material Property Tables (Menu:Prop)

The is command is used to make the Material Properties input box visible. The input box is used to add/edit entries in the Material Property Table.



The **Code** box identified the table entry. The default is the next unused code.

The number of entries in the table is shown in the **Maximum No of Codes** box. When a model is saved this value will be updated (and the table extended) to show the maximum code referenced by the elements of the model.

the other boxes.

If the **N** button is pressed the next unused Code label will appear without changing the data properties. This can be useful for copying existing code data.

The **Name** box is used to enter an ID name (up to 8 characters).

If a non zero value is entered for Poisson's Ratio (γ) the Modulus of Rigidity G will be evaluated by the expression:

$$G = E / 2(1 + \gamma)$$

Shear stiffness property in beam elements are based on the value of G. Shear stiffness properties in solids elements are based on the value of γ i.e.G is evaluated as $E / 2(1 + \gamma)$ during solution.

Rigidity(G) is also used to define thermal conductivity for heat transfer solutions for solid elements (G is only used in beams solutions)

The **Enter/Add** to Table button is used to enter the current settings displayed in the boxes into the table.

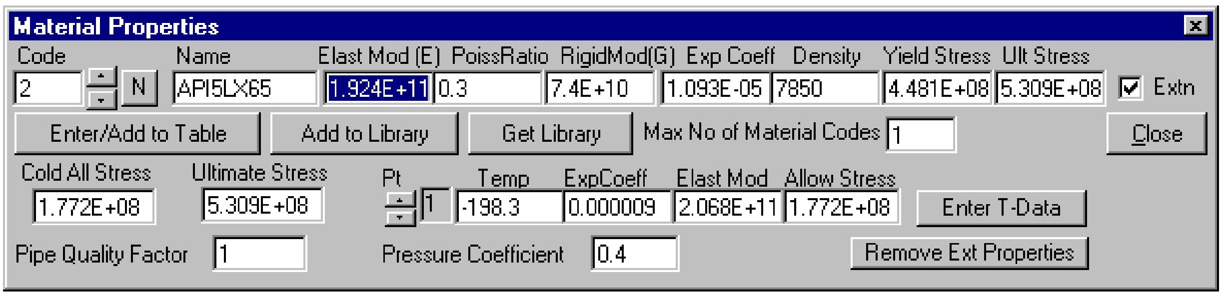
If an existing code number is entered in the **Code** box, its parameters will be displayed in the boxes.

The **Add to Library** is used to add the current settings to the Material Library . Use the **Get Library** button to check the current library.

The **Get Library** button loads the Material Library form. Entries from the current library can be selected from this form.

The **Ext** option box is used to extend the form to show additional properties relating to temperature dependent material properties used in pipework analysis. If **Ext** box is activated and the model is then saved within the current Model Definition Task additional data relating to the extended properties will be written to the model data files and included in the analysis. The **Remove Ext Properties** button can be used to remove all existing extended property data.

If the box is extended the additional data will also be added or recovered from the appropriate Material Library when the library buttons are activated.



The thermal dependent properties (up to 15 points) are entered into a data buffer using the **Enter T-Data** button. The data contents of the buffer may be viewed using the Pt box spin buttons.

Properties are only entered to the table when the **Enter/Add to Table** button is pressed.

A procedure for the entry of extended properties is

Enter the Code Number

Enter the properties for the **CAS**, **QF** and **Press Coeff etc**

Enter the properties corresponding to a temperature point and press the Enter T Data button (this adds data to the form buffer). Do this for each temperature point. Use the Spin buttons to check the data.

Press the Enter/Add to Table to add the all data to the model Material Property table.

The **Add to Lib** button will add the standard and extended (if form is extended) properties directly to the library.

The following summarises how the extended properties are used by the program:

Colld All Stress - is only used by FS-Pipe for piping design checks.

Ultimate Stress - is only used by FS-Pipe for piping design checks

Pipe Quality Factor - is only used by FS-Pipe for piping design checks.

Pressure Coefficient - is only used by FS-Pipe for piping design checks.

ExpCoef - used to evaluate thermal strain based on the load case element temperature - This should be defined as mean value .e.g. B31.3 Table C-3

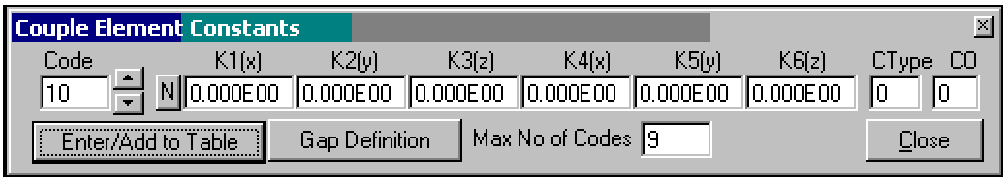
ElastMod - not used. Model stiffness is always based on cold E modulus i.e. **Elast Mod (E).**

AllowStress - is only used by FS-Pipe for piping design checks.

-o-

Couple Constants Tables (Menu:Prop)

When the Couple Ele Const command is selected the Couple Elements Constants input box will become visible. This box is used to add/edit entries in the Couple Constants Table.



The **Code** box identifies the table entry. The default is the next unused code.

If the **N** button is pressed the next unused Code label will appear without changing the data properties. This can be useful for copying existing code data.

The number of entries in the table is shown in the **Maximum No of Codes** box. When a model is saved this value will be updated (and the table extended or) to show the maximum code referenced by the couple elements of the model.

For standard linear couples (CType=0) the constants K1 to K6 are used to define the stiffness of the couple in terms of its local co-ordinate system. i.e.

K1	x translation
K2	y translation
K3	z translation
K4	x rotation
K5	y rotation
K6	z rotation

Zero values may be entered. A typical stiffness of 1E12 would be used to represent a rigid connection.

The **CType** box is used to specify the couple element type.

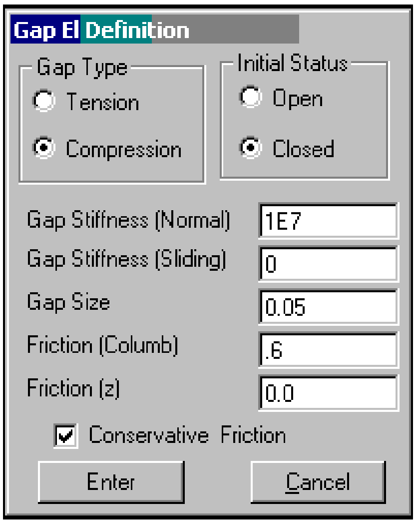
The **CO** property is an additional property used by certain couple types. The use of this property is described in the element description for elements, that use this property.

The **Enter/Add** to Table button is used to enter the current settings into the table.

Gap Elements

Gap elements are a very common type of couple element and have their own input form.

The **Gap Definition** button is used to define the properties of a Gap Element. When this button is clicked the following box will become visible.



Gap Type is used to define whether the gap is a compression gap or a tension (hook) gap.

Initial Status is used to define the gap status for the first iteration.

Gap Stiffness defines the connecting stiffness when the gap is closed. This can be adjusted to suit model requirements. A typical gap stiffness of 1E8 would be a reasonable stiff gap for most structures created in SI units. A good 'feel for relative stiffness can be obtained by equating it to the axial stiffness of a nearby element i.e. AE/L . Avoid using very stiff gaps they can is some cases prevent solution convergence.

Gap Size defines the initial gap size. The gap must move by this amount before it will close. If a negative value is specified the gap elements will induce loading into the structure by simulating an interference fit. A zero gap infers an element 'just' in contact. If the gap size is set to value greater than 1E4 than the gap size will interpreted as the distance between the element nodes

Friction (Coulomb) defines the relationship between normal force and the lateral frictional resistance.

Friction (z) enable friction to be defined independently in the local lateral directions. It this value is zero the lateral frictional force will be the resultant force i.e. Coulomb friction.

Conservative Friction Conservative friction assumes no energy is lost and the gap will unload down the same path (CType 12). If more realistic unloading is a requirement this should not be active ((CType 10).

-o-

8.13 Load Definition

The following menus are available in this task

NL	Used to define nodal loads
ND	Used to prescribe nodal displacements
El.Pt	Used to define element point loads (mid span concentrated loads)
El.Dl	Used to define distributed element loads
FE-Solids	Used to define pressure and edge loads on solid elements
Tp/Pr	Used to define pressure (pipe elements) and loading due to thermal expansion
PropLDS	Used to define temperature and pressure (pipes) by Geom property reference
Grv	Used to define gravitational constants
Non-LinEff	Used to define non-linear loading effects used in non-linear solutions
Sum	Evaluates the total loading and centres of force on visible nodes and beam elements.

Load Generators

The **TASK** menu is extended to include menu commands that start the most commonly used load generators.

- FS-Wave Wave Loading
- FS-Wind Wind Loading
- CMotion Vessel Motions
- FS-Seismic Static Equivalent Seismic Loading

The load generators each have their own Help system.

-o-

Load Definition:Menu:NL

This menu provides the load definition commands associated with nodal load definition.

Concentrated (point loads) and moment couples may be applied to any node. They can only be defined in the principal directions of the global co-ordinate system.

Nodal masses may also be defined. These masses represent concentrated mass at the node. If accelerations are specified e.g. gravity then forces will be applied in the appropriate direction. Such a facility is useful for motion related loading since the load direction is then only a function of defined acceleration.

More than one defined load may be applied to a node. If a node has more that one load will be necessary to reference the load using the Display List when copying, deleting etc.

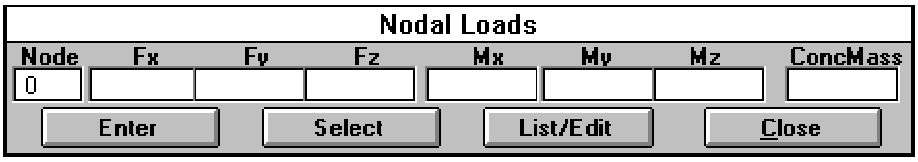
The menu commands used to define nodal loads are:

Input	Makes the primary Nodal Load Input dialogue box visible for load definition
Delete	Remove nodal loads from selected nodes
Copy by Node	Copies loads by reference to the node to which they are attached using a defined label increment
Share	Equally distributes a defined load to selected nodes.
Display List	Displays the Nodal Load list box. List box can be used for edit or deletion selection.

-o-

Input Nodal Load

The following nodal load input form becomes visible when the Input command is selected.



Fx, Fy, and Fz define the concentrated loads in the global co-ordinate system. Note downwards is -ve y direction.

Mx, My, Mz define the couples in the global co-ordinate system.

ConcMass defined a concentrated mass at a node (applied force is mass x g where g is the gravitational constant in either the x, y or z direction). When undertaking dynamic analysis the force method of defining nodal mass is recommended. If load cases are to be pre-merged avoid using nodal mass because the mass may be unwantedly combined.

The **Enter** button is used in conjunction with the **Node** box to apply the loads by label reference.

The **Select** button enables the recipient node or nodes to be selected using the current selection method. The activity status box will indicate when this is active. When selected the displayed (defined) loads will be applied to the selected nodes.

This **List/Edit** button is used to list, edit or copy nodal loads by selecting loads by visual pick. When a node with a load is selected its label will be shown (greyed) in the **Node** box. This label cannot be changed until the **Enter** button is clicked. This prevents the load from being applied twice the same node. If the Node box is required to be used to apply the loads to a different node then simply hit the Enter button to re-enter the load and then use the Node box in the normal manner to define the load to other nodes. The **Select** button of the form can be used to copy the loads to other nodes in the normal manner.

If more than one nodal load is applied to a node the Display List is used to select the load for edit.

Thermal Loads - Heat Generation

When undertaking heat transfer analysis the form can be used to define concentrated heat generation (source/sink) by define the heat magnitude as a Fx load.

-o-

Delete Nodal Loads

This command option is used to delete nodal loads. The nodes with the loads to be deleted are selected using the current selection method.

If more than one load is applied to a node and the node is selected by mouse pick, the specific load will have to be selected from the load list. The list will show highlighted entries. The load entry to be deleted should be selected and the list delete button clicked.

If more than one load is applied to an node and the node is selected using a multi-selection method e.g. mouse window, then the window will have to be applied more than one to delete the multiple loads on the node.

-o-

Copy by Node (Nodal Loads)

This command is used to copy multiply loads using a label increment. It would be used in situations where identical loads are to be applied to repeated node patterns within a structure e.g. roof loads applied to identical roof truss frames within a structure.

It can be used with any selection method since the routine simply copies nodal loads from the selected node(s) (SelNode) to another node(s) (SelNode + Label Increment). The routine checks for nodes with loads within the selection and copies them if they exist. Generally the routine will be most effective using the select by Label Range or select by Window.

When the command is selected the label increment between the nodes pattern is entered. The nodes with the loads are then selected using the current selection method.

-o-

Share Loads

This command is used to evenly distribute (share) defined loads to selected nodes or elements.

In the case of shared nodal loads the defined load is divided equally between the selected nodes.

In the case of shared element loads the defined load is applied as a distributed load to all selected elements.

The loads are defined by entering them into the Share Loads input form. The loads that are entered are the total loads that will be applied to the model. They can only be defined in the global axes.

The input form has two control buttons. The **Select** button is used to initiate the selection process which is by the current SelectBy method. When the selection process is complete the **End Selection** button should be pressed. The loads will be distributed to those entities selected.

-o-

Display List (Nodal Loads)

This command is used to view the load data in a list format. List entries may be selected for editing etc. This is the only method to select loads when more than one load is applied to a single node.

The list has the following command buttons

Delete Deletes the selected entry.

Edit Displays the Input form with the selected entry as the current form entry.

Sort Sorts the load list by label.

Update Updates the load list. To be used following data entry or editing.

-o-

Load Definition:Menu:ND

This menu provides the load definition commands associated with the prescribed displacement of nodes. When a displacement is prescribed the analysis will evaluate the force required to produce that displacement. Nodal displacements can only be defined in the principal directions of the global co-ordinate system.

Prescribed displacements cannot be defined on a node more than once.

The menu commands used to define nodal loads are:

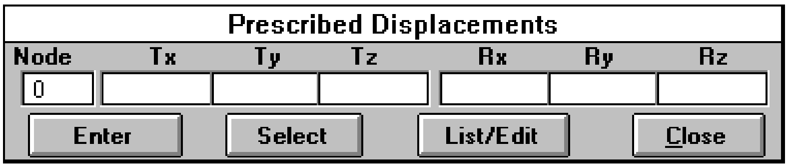
Input	Makes the primary Prescribed Displacement Input dialogue box visible for displacement definition
Delete	Remove displacements from selected nodes
Copy by Node	Copies displ. by reference to the node to which they are attached using a defined label increment
Display List	Displays the Prescribed Displacement list box. List box can be used for edit or deletion selection.

For further information on Prescribed Displacements

-o-

Input Nodal Displacements

The following nodal displacement input form becomes visible when the Input command is selected.



Tx, Ty, and Tz define the node translations in the global co-ordinate system.

Rx, Ry, Rz define the node rotations (radians) in the global co-ordinate system.

The **Enter** button is used in conjunction with the **Node** box to apply the displacements by label reference.

The **Select** button enables the recipient node or nodes to be selected using the current selection method. The activity status box will indicate when this is active. When selected the displayed (defined) displacements will be applied to the selected nodes.

This **List/Edit** button is used to list, edit or copy nodal displacements by selecting nodes by visual pick. When a node is selected its label will be shown (greyed) in the **Node** box. This label cannot be changed until the **Enter** button is clicked. This prevents the displacement from being applied twice the same node. If the Node box is required to be used to apply displacements to a different node then simply hit the Enter button to re-enter the displacements and then use the Node box in the normal manner to define the displacement to other nodes. The **Select** button of the form can be used to copy the displacements to other nodes in the normal manner.

For further information on Prescribed Displacements

-o-

Delete Nodal Displacements

This command option is used to delete nodal displacements. The nodes with the displacements to be deleted are selected using the current selection method.

-o-

Copy by Node - Nodal Displacements

This command is used to copy multiply displacements using a label increment. It would be used in situations where identical displacements are to be applied to repeated node patterns within a structure.

It can be used with any selection method since the routine simply copies nodal displacements from the selected node(s) (SelNode) to another node(s) (SelNode + Label Increment). The routine checks for nodes with displacements within the selection and copies them if they exist. Generally the routine will be most effective using the select by Label Range or select by Window.

When the command is selected the label increment between the nodes pattern is entered. The nodes with the displacements are then selected using the current selection method.

-o-

Display List - Nodal Displacements

This command is used to view the load data in a list format. List entries may be selected for editing etc.

The list has the following command buttons

Delete	Deletes the selected entry.
Edit	Displays the Input form with the selected entry as the current form entry.
Sort	Sorts the load list by label.
Update	Updates the load list. To be used following data entry or editing.

-o-

Load Definition:Menu:El.Pt

This menu provides the load definition commands associated with Element Point (mid span concentrated) Load definition. Element point loads may be defined in either the global co-ordinate system or the local element co-ordinate system.

Input Makes the primary Element Point Load Input dialogue box visible for load definition

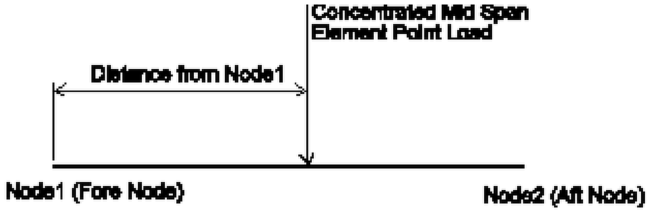
Delete Remove Point Loads from selected elements

Copy by Elem Copies loads by reference to the element to which they are attached using a defined label increment

Display List Displays the Element Point Load list box. List box can be used for edit or deletion selection.

-o-

Input Element Point Load



The following Element Point Load form becomes visible when the Input command is selected.

Fx, Fy, and Fz define the concentrated loads in the selected co-ordinate system. Note global downwards is -ve y direction.

Mx, My, Mz define the couples in the selected co-ordinate system.

Elem Length shows the length of the element whose label is in the Elem box

If an element is listed using the element query button and then the **Elem** box is double clicked, the listed element will appear in the box and its length will be also be shown. Note that the Node1 end of a listed element is circled white.

The **Dist From Node1** box defines the length from the first node of the element to the position of the loads. If by error it is referenced to the wrong end (Node2), the **<->** button can be used to reverse this distance to the correct end.

If a value of zero is entered in the **Dist From Node1** box then **Remote Distance Reference** mode will become active when the **Enter** button is pressed.. In this mode the postion of the load is can be defined relative to any node on the model. This reference node **must** be picked using the node Query button prior to pressing the **Enter** button. The distance in this mode is entered in a similar manner to that for Node Generation.

The **Global** and **Local** option buttons define whether the loads are to applied in the local element co-ordinate system of the global co-ordinate system.

The **Enter** button is used in conjunction with the **Elem** box to apply the loads by label reference.

The **Select** button enables the recipient element(s) to be selected using the current selection method. The activity status box will indicate when this is active. When selected the displayed (defined) loads will be applied to the selected elements.

This **List/Edit** button is used to list, edit or copy loads by selecting loads by visual pick. When an element is selected its label will be shown (greyed) in the **Elem** box. This label cannot be changed until the **Enter** button is clicked. This prevents the load from being applied twice the same element. If the Elem box is required to be used to apply the loads to a different element then simply hit the Enter button to re-enter the load and then use the Elem box in the normal manner to define the load to other elements. The **Select** button of the form can be used to copy the loads to other elements in the normal manner.

If more than one point load is applied to an element the Display List is used to select the load for edit.

-o-

Delete Element Point Loads

This command option is used to delete element point loads. The elements with the loads to be deleted are selected using the current selection method.

If more than one load is applied to an element and the element is selected by mouse pick, the specific load will have to be selected from the load list. The list will show highlighted entries. The load entry to be deleted should be selected and the list delete button clicked.

If more than one load is applied to an element and the element is selected using a multi-selection method e.g. mouse window then the window will have to be applied more than one to delete the multiple loads on the element.

-o-

Copy by Element - Element Point Loads

This command is used to copy multiply loads using a label increment. It would be used in situations where identical loads are to be applied to repeated element patterns within a structure e.g. roof loads applied to identical roof truss frames within a structure.

It can be used with any selection method since the routine simply copies element loads from the selected element(s) (SelEle) to another element(s) (SelEle + Label Increment). The routine checks for elements with loads within the selection and copies them if they exist. Generally the routine will be most effective using the select by Label Range or select by Window.

When the command is selected the label increment between the element pattern is entered. The elements with the loads are then selected using the current selection method.

-o-

Display List - Element Point Loads

This command is used to view the load data in a list format. List entries may be selected for editing etc. This is the only method to select loads when more than one load is applied to a single element.

The list has the following command buttons

Delete Deletes the selected entry.

Edit Displays the Input form with the selected entry as the current form entry.

Sort Sorts the load list by label.

Update Updates the load list. To be used following data entry or editing.

-o-

Load Definition:Menu:EL.Dl

This menu provides the load definition commands associated with Distributed Elements Load definition.

Two basic input modes are available, UDL and Non-uniform.

UDL is for uniform loads over the entire span. If the direction is global, these loads are save as UDL command loads.

Non-uniform is for patch/trapezoidal loads.

The two modes of input are provided to simplify the definition of uniform span loads. When loading is defined in one mode it will also exist in the other mode.

The loading may be defined relative to the local element or global co-ordinate system.

Note that the loading is per unit length of the element. For snow loading which is normally global and referenced to the projected length, simply factor the load by the ratio of projected/true to reduce it slightly.

Input-UDLs Makes the primary Element UDLs Input dialogue box visible for uniform load definition

Input-NonUniform Makes the primary Element Distributed Loads Input dialogue box visible for load definition

Delete Remove Distributed Loads from selected elements

Copy by Elem Copies loads by reference to the element to which they are attached using a defined label increment

Share Equally distributes defined loads to selected elements as UDLs.

Load Distribution Applies distributed loads to selected elements based on linear distribution defined by global dimensions and directions

Display List Displays the Element Distributed Load list box. List box can be used for edit or deletion selection.

.

-o-

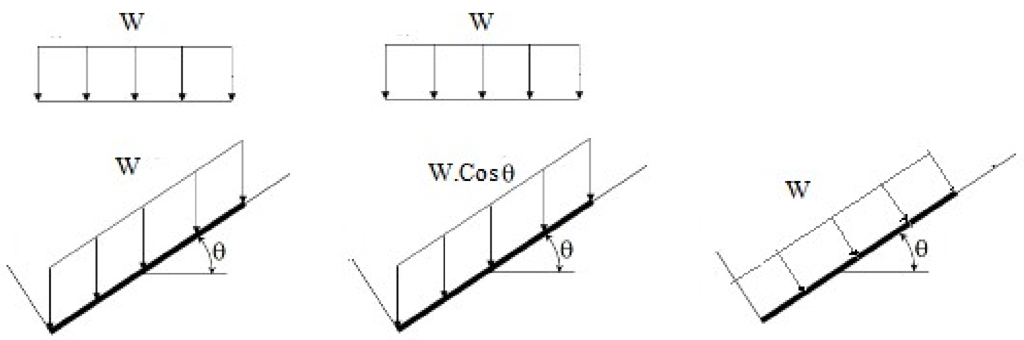
Input Element UDLs

The following Element UDLs Load form becomes visible when the Input-UDLs command is selected.

wx, wy, and wz define the uniform distributed loads in the selected co-ordinate system. Note Global downwards is -ve y direction.

The **Global, Projected Global** and **Local** option buttons define whether the loads are to applied in the local element co-ordinate system of the global co-ordinate system. If the direction is Global or Projected Global, the loading will be saved using the **UDL** command. Otherwise it will saved using the **ED** command in the load case command file.

Global Projected Global Local



The **Enter** button is used in conjunction with the **Elem** box to apply the loads by label reference.

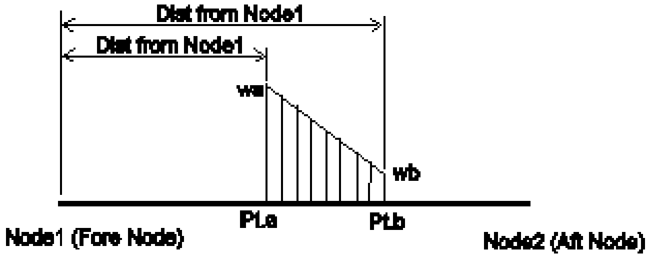
The **Select** button enables the recipient element(s) to be selected using the current selection method. The activity status box will indicate when this is active. When selected the displayed (defined) loads will be applied to the selected elements.

This **List/Edit** button is used to list, edit or copy loads by selecting loads by visual pick. When an element is selected its label will be shown (greyed) in the **Elem** box. This label cannot be changed until the **Enter** button is clicked. This prevents the load from being applied twice the same element. If the Elem box is required to be used to apply the loads to a different element then simply hit the Enter button to re-enter the loads and then use the Elem box in the normal manner to define the load to other elements. The **Select** button of the form can be used to copy the loads to other elements in the normal manner.

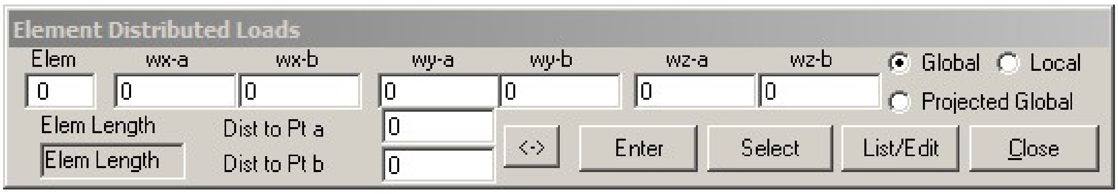
If more than one load is applied to an element the Display List is used to select the load for edit.

-o-

Input Element Non-Uniform Dist Loads



The following Element Distributed Loads form becomes visible when the Input-NonUniform command is selected.



wx, wy, and wz define the distributed loads in the selected co-ordinate system. Note global downwards is -ve y direction.

The -a and -b suffix refer to the start end (Pt a) and the finish end (Pt b) of the distributed loads.

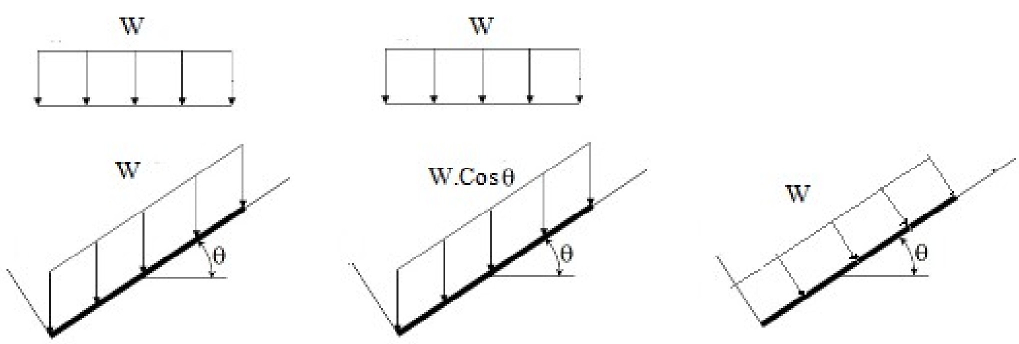
Elem Length shows the length of the element whose label is in the Elem box

If an element is listed using the element query button and then the **Elem** box is double clicked, the listed element will appear in the box and its length will be also be shown.

The **Dist to Pt a** box defines the length from the first node of the element to the start point of the load. The **Dist to Pt b** box defines the length from the first node of the element to the end point of the load. If by error they are referenced to the wrong end (Node2), the **<->** button can be used to reverse the distances to the correct end.

The **Global, Projected Global** and **Local** option buttons define whether the loads are to applied in the local element co-ordinate system of the global co-ordinate system

Global Projected Global Local



The **Enter** button is used in conjunction with the **Elem** box to apply the loads by label reference.

The **Select** button enables the recipient element(s) to be selected using the current selection method. The activity status box will indicate when this is active. When selected the displayed (defined) loads will be applied to the selected elements.

This **List/Edit** button is used to list, edit or copy loads by selecting loads by visual pick. When an element is selected its label will be shown (greyed) in the **Elem** box. This label cannot be changed until the **Enter** button is clicked. This prevents the load from being applied twice the same element. If the Elem box is required to be used to apply the loads to a different element then simply hit the Enter button to re-enter the loads and then use the Elem box in the normal manner to define the load to other elements. The **Select** button of the form can be used to copy the loads to other elements in the normal manner.

If more than one load is applied to an element, use the Display List to select the load for edit.

-o-

Delete Element Distributed Loads

This command option is used to delete element distributed loads. The elements with the loads to be deleted are selected using the current selection method.

If more than one load is applied to an element and the element is selected by mouse pick, the specific load will have to be selected from the load list. The list will show highlighted entries. The load entry to be deleted should be selected and the list delete button clicked.

If more than one load is applied to an element and the element is selected using a multi-selection method e.g. mouse window then the window will have to be applied more than one to delete the multiple loads on the element.

-o-

Copy by Element - Element Distributed Loads

This command is used to copy multiply loads using a label increment. It would be used in situations where identical loads are to be applied to repeated element patterns within a structure e.g. roof loads applied to identical roof truss frames within a structure.

It can be used with any selection method since the routine simply copies element loads from the selected element(s) (SelEle) to another element(s) (SelEle + Label Increment). The routine checks for elements with loads within the selection and copies them if they exist. Generally the routine will be most effective using the select by Label Range or select by Window.

When the command is selected the label increment between the element pattern is entered. The elements with the loads are then selected using the current selection method.

The **X Dir, Y Dir & Z Dir** option buttons are used to select the co-ordinate direction used during interpolation.

The **Face** box is used to specify the face of the elements

The **Normal/Tangential IP/Tangential OP** options are for the definition of edge loads.

The **Select** button enables the recipient element(s) to be selected using the current selection method. Load will not be applied to element outside the **Coord1** and **Coord2** co-ordinate range.

The **Node Group** button is used to apply face pressures to solid elements (Hexahedral & Tetrahedral only) by face node identification. If the all the face nodes of an element are in the active group when button is clicked then the defined pressure loads will be applied to the faces of all such elements.

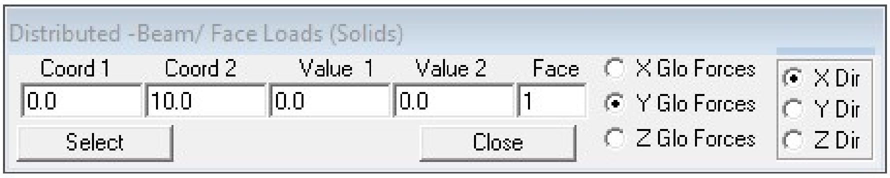
-o-

Load Distribution

This command will apply distributed loads to selcted beam elements as function of position of the element using linear distribution defined by global dimensions and directions

A typical use of this command would be to apply linearly varying load distributions to a series of elements e.g a triangular distribution.

A trapezoidal load distribution is defined using two co-ordinate points (**Coord1** & **Coord2**) each with associated corresponding load values (**Value1** & **Value2**). When an element is selected the load applied to the element is the interpolated value corresponding to the position of the element node.   The node co-ordinates used are the local co-ordinates.



The **X Dir, Y Dir & Z Dir** option buttons are used to select the co-ordinate direction used during interpolation.

The **Face** input box is not used for beams.

The **X Glo Forces, Y Glo Forces & Z Glo Forces** entries define the magnitude and direction of the distributed loading.

The **Select** button enables the recipient element(s) to be selected using the current selection method. Load will not be applied to element outside the **Coord1** and **Coord2** co-ordinate range.

-o-

Display List - Element Point Loads

This command is used to view the load data in a list format. List entries may be selected for editing etc. This is the only method to select loads when more than one load is applied to a single element.

The list has the following command buttons

Delete	Deletes the selected entry.
Edit NonUnifrom	Displays the Input form with the selected entry as the current form entry.
Edit UDL	Displays the Input form with the selected entry as the current form entry.
Sort	Sorts the load list by label.
Update	Updates the load list. To be used following data entry or editing.

-o-

Load Definition:Menu:FE-Solids

This menu provides the load definition commands associated with specification of distributed face (pressures) and edge loads on solid elements.

Input/Edit	Makes the Fe-Solids Load Input dialogue box visible for load definition
Delete	Remove Loads from selected elements
Pressure Dist	Applies face based on linear distribution define by global dimensions and directions
Copy by Elem	Copies loads by reference to the element to which they are attached using a defined label increment
Display List	Displays the Element Point Load list box. List box can be used for edit or deletion selection.

-o-

Input/Edit FE Loads

This input form is used to define structural face loads on solid elemenst.

It is also used to define thermal loading by the definition of heat transfer properties described in the next section.

Face and Edge Loads are applied by reference to the Face Number of the element. The Face numbers of elements are labeled when the Element Query button is used (only in Load definition TASK).

Edge loads are applied in force/unit length (N/m). Edge load can be normal to the edge, in-plane tangential to the edge or out of plane tangential to the edge.

Normal loads are positive when going into the element. Tangential in-plane loads are positive when in the same direction as the element node sequence. Tangential out of plane loads are positive when in going into the element. The sign convention for the tangential out of plane loads is based on the sign of the load on the first node of the edge.

For shell faces and 3-D elements the loads are applied as face pressures (N/m2). By sign convention positive pressures go into the element.

For shells, Faces 1 and 2 are the Top and bottom surfaces respectively and the edges are formed from faces 3 to 6 which follow the node sequence.

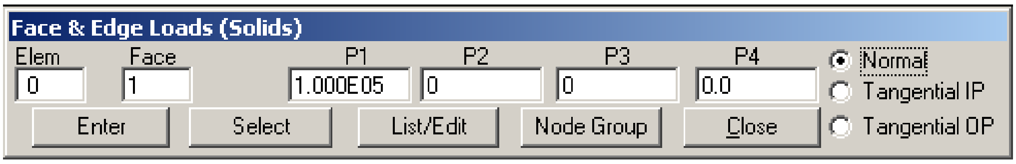
2-D elements have faces which follow the node sequence.

Use the **Local Rotation/Coord** System button to show the local co-ordinate system of the element.

Face loads can be defined differently at the 4 corner node. If only one node is defined it will be applied to all nodes. If mid side nodes are present the applied load will be the mean of the corner nodes.

Edge load can be defined differently for each edge nodes (up to 3). If only one node is defined it will be applies to all edge nodes. If mid side nodes are present and loaded a parabolic distribution will be applied.

The following load definition form is used for the specification of distributed load on solid elements.



The **Face** box is used to specify the face of the elements

The P1 to P4 boxes are used to specify to load at the face nodes. The P1 refers to the first node on the face or edge in terms of the element connectivity list. P2 to the second an so on. Zero values will be assigned the value specified in P1.

The **Normal/Tangential IP/Tangential OP** options are for the definition of edge loads.

The **Enter** button is used in conjunction with the **Elem** box to apply the loads by label reference.

The **Select** button enables the recipient element(s) to be selected using the current selection method. The activity status box will indicate when this is active. When selected the displayed (defined) loads will be applied to the selected elements.

The **List/Edit** button is used to list, edit or copy loads by selecting loads by visual pick. When an element is selected its label will be shown (greyed) in the **Elem** box. This label cannot be changed until the **Enter** button is clicked. This prevents the load from being applied twice the same element. If the Elem box is required to be used to apply the loads to a different element then simply hit the Enter button to re-enter the loads and then use the Elem box in the normal manner to define the load to other elements. The **Select** button of the form can be used to copy the loads to other elements in the normal manner. If more than one load is applied to an element the Display List is used to select the load for edit.

The **Node Group** button is used to apply face pressures to solid elements by face node identification. If the all the face nodes of an element are in the active group when button is clicked then the defined pressure loads will be applied to the faces of all such elements.

-o-

Input/Edit FE Loads Heat Transfer

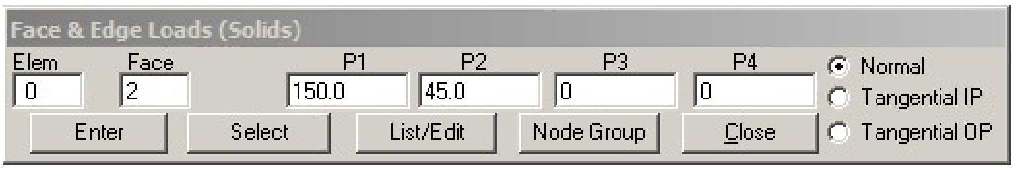
This input form is used to define thermal element boundary properties and heat generation (source/sink) used in heat transfer solutions.

Concentrated Heat Generation (concentrated source/sink) can be applied as a nodal force by definition in the X direction. **Heat Generation** would be more usually defined as a uniform element body force as described below

Boundary properties are specified with reference to an element face and are taken to be constant across the face of the element.

Note that face number of an element is displayed when an element is queried (only in Load Defintion TASK).

Any combination of consistent unit may be used. Shown below are those used if the model in meter length units and SI unit are being used.



Surface Convection **Normal** is active

P1 = Tf Bulk fluid temperature - deg C(K)

P2 = h Surface(film) heat transfer coefficient - W/m^2 deg C(K) (Heat flux q = h*(Tf - T))

If h is defined a negative integer then h will be assumed to be temperature dependent (Tf - T) and the value of h must be defined using an RC curve where h is the RC table entry.

Specified Temperature **Normal** is active

P1 = T Face temperature - deg C(K)

P2 = 0 This must be set to zero other wise interpreted as h

Specific Heat flux **Tangential IP** is active

P1 = HF Heat flux. - W/m^2

Specified Element Heat Generation **Tangential OP** is active

P1 = HG Element heat generation - W/m^3

Surface Radiation **Tangential OP** is active

P1 = Tf Bulk fluid temperature - deg K

P2 = $\Psi\sigma$ Surface(film) heat transfer coefficient - W/m^2 deg K (Heat flux q = hr*(Tr - T) where hr =$\Psi\sigma$(Tr^2 + T^2)(Tr + T) and Tr is known.

This assumes that the radiation source area>>than the incident surface area and that Ψ includes view factors and emissivities for the subject surfaces.

If radiation is included all temperature must be define as absolute temperatures i.e. in SI unit Tabs = T + 270

Adiabatic - Fully insulated boundary

All element boundaries are take to be fully insulated unless one of the above is specified.

The **Enter** button is used in conjunction with the **Elem** box to apply the loads by label reference.

The **Select** button enables the recipient element(s) to be selected using the current selection method. The activity status box will indicate when this is active. When selected the displayed (defined) loads will be applied to the selected elements.

The **List/Edit** button is used to list, edit or copy loads by selecting loads by visual pick. When an element is selected its label will be shown (greyed) in the **Elem** box. This label cannot be changed until the **Enter** button is clicked. This prevents the load from being applied twice the same element. If the Elem box is required to be used to apply the loads to a different element then simply hit the Enter button to re-enter the loads and then use the Elem box in the normal manner to define the load to other elements. The **Select** button of the form can be used to copy the loads to other elements in the normal manner. If more than one load is applied to an element the Display List is used to select the load for edit.

The **Node Group** button is used to apply face pressures to solid elements by face node identification. If the all the face nodes of an element are in the active group when button is clicked then the defined pressure loads will be applied to the faces of all such elements.

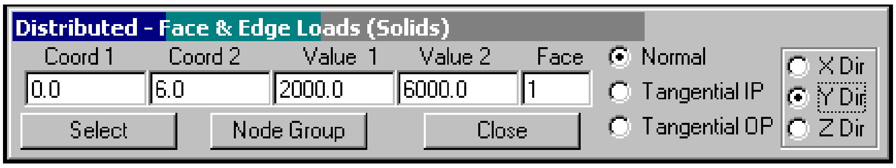
-o-

Pressure Distribution

This command will apply face and edge loads to solid elements as function of position of the element. A similar form is used for beam loadings.

A typical use of this command would be to apply linearly varying pressure distributions (typically representing a hydrostatic load) to the faces of shell elements.

A trapezoidal load distribution is defined using two co-ordinate points (**Coord1** & **Coord2**) each with associated corresponding load values (**Value1** & **Value2**). When an element is selected the load applied to the element is the interpolated value corresponding to the position of the element node. The node co-ordinates used are the local co-ordinates.



The **X Dir, Y Dir & Z Dir** option buttons are used to select the co-ordinate direction used during interpolation.

The **Face** box is used to specify the face of the elements

The **Normal/Tangential IP/Tangential OP** options are for the definition of edge loads.

The **Select** button enables the recipient element(s) to be selected using the current selection method. Load will not be applied to element outside the **Coord1** and **Coord2** co-ordinate range.

The **Node Group** button is used to apply face pressures to solid elements (Hexahedral & Tetrahedral only) by face node identification. If the all the face nodes of an element are in the active group when button is clicked then the defined pressure loads will be applied to the faces of all such elements.

-o-

Delete

This command option is used to delete distributed element loads. The elements with the loads to be deleted are selected using the current selection method.

If more than one load is applied to an element and the element is selected by mouse pick, the specific load will have to be selected from the load list. The list will show highlighted entries. The load entry to be deleted should be selected and the list delete button clicked.

If more than one load is applied to an element and the element is selected using a multi-selection method e.g. mouse window then the window will have to be applied more than one to delete the multiple loads on the element

-o-

Copy by Elem

This command is used to copy multiply loads using a label increment. It would be used in situations where identical loads are to be applied to repeated element patterns within a structure e.g. roof loads applied to identical roof truss frames within a structure.

It can be used with any selection method since the routine simply copies element loads from the selected element(s) (SelEle) to other element(s) (SelEle + Label Increment). The routine checks for elements with loads within the selection and copies them if they exist. Generally the routine will be most effective using the select by Label Range or select by Window.

When the command is selected the label increment between the element pattern is entered. The elements with the loads are then selected using the current selection method.

-o-

Display List - FE Loads

This command is used to view the load data in a list format. List entries may be selected for editing etc. This is the only method to select loads when more than one load is applied to a single element.

The list has the following command buttons

Delete	Deletes the selected entry.
Edit	Displays the Input form with the selected entry as the current form entry.
Sort	Sorts the load list by label.
Update	Updates the load list. To be used following data entry or editing.

-o-

Load Definition:Menu:Tp/Pr

This menu provides the load definition commands associated with loading due to thermal expansion and pressure effects (pipe elements only).

The pressure is definition is defined in terms of internal pressure Pi and external pressure Po.

The units for pressure should be consistent with the stress units.

Thermal strain is based on $\varepsilon t = \alpha.\Delta T$ where $\Delta T = (T - Tamb)$

Pressure strain is based on $\varepsilon p = (1 - 2.\mu)(Pi.Ai - Po.Ao) / (EAs)$

Input — Makes the primary Pressure & Temperature Input dialogue box visible for load definition

Delete — Remove pressure and thermal loads from selected elements

Copy by Elem — Copies loads by reference to the element to which they are attached using a defined label increment

Display List — Displays the Element Pressure & Temp list box. List box can be used for edit or deletion selection.

Nodal Temp (Fe) — A sub menu for the definition of nodal temperature of solid elements

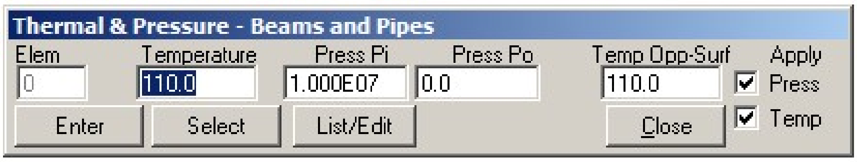
Line Temp Profile (ETPR Cmd) — The ETPT command is used to generate a temperature profile on line elements defined by an element group

Ambient Temp — Defines the Ambient Temperature.

-o-

Input Element - Thermal & Pressure

The following Element Thermal & Pressure Load form becomes visible when the Input command is selected. It is used to define beam & pipe element temperatures and pressure in pipe elements.



Temperature defines the temperature of the element. The differential temperature used to evaluate element axial strain is the differential between this value and the **Ambient Temp** defined temperature. i.e.

$\varepsilon t = \alpha.\Delta T$ where ΔT = (T - Tamb)

Temp Opp-Surf would normally be set to the same value as the **Temperature**. If this is defined as a different value, the difference defines the differential temperature between the top surface and bottom surface of the element. See below regarding moment loading due to differential thermal strain.

Press Pi defines the internal pressure for pipe elements. The units should be consistent with the stress units i.e. N/m2 for SI units.

Press Po defines the external pressure for pipe elements. The units should be consistent with the stress units i.e. N/m2 for SI units. If this is define as zero Pi becomes the differential pressure i.e. the gauge pressure.

The **Apply** options can be used to restrict load type application.

The **Enter** button is used in conjunction with the **Elem** box to apply the loads by label reference.

The **Select** button enables the recipient element(s) to be selected using the current selection method. The activity status box will indicate when this is active. When selected the displayed (defined) loads will be applied to the selected elements.

This **List/Edit** button is used to list, edit or copy loads by selecting loads by visual pick. When a element with load definition is selected the element Thermal & Pressure input form will become visible with that element and its associated definition shown. When an element is selected its label will be shown (greyed) in the **Elem** box. This label cannot be changed until the **Enter** button is clicked. This prevents the load from being applied twice the same element. If the Elem box is required to be used to apply the loads to a different element then simply hit the Enter button to re-enter the load and then use the Elem box in the normal manner to define the load to other elements. The **Select** button of the form can be used to copy the loads to other elements in the normal manner.

Differential Thermal Loading

Ambient Temperature

If the Ambient temperature if defined the differential pressure defined above will be reduced by the **Ambient Temp** temperature. When an **Ambient Temp** temperature is defined it implies that the differential temperature is an absolute value, not a differential value.

Bending In Beams due to Differential Thermal Strain

If a differential temperature is defined for the top and bottom surface of an element a moment will be induced due to the differential strain across the section. The magnitude of which depends on the depth of the section and the end moment stiffness of the element. The top surface is the taken to be the positive major axis and the bottom surface the negative major axis. The depth of the section is taken to be the sum of the Y co-ordinates that define the stress points at Points 1 and 2.

Axial in Beam Elements

The axial force due to thermal strain is based on the mean of the top and bottom temperature relative to the ambient temperature ie $\Delta T(eff)$ = T - Tamb.

-o-

Delete Element - Thermal & Pressure

This command option is used to remove thermal and pressure definition from an element(s). The elements with the loads to be deleted are selected using the current selection method.

-o-

Copy by Element - Thermal & Pressure

This command is used to copy multiply loads using a label increment. It would be used in situations where identical loads are to be applied to repeated element patterns within a structure e.g. roof loads applied to identical roof truss frames within a structure.

It can be used with any selection method since the routine simply copies element loads from the selected element(s) (SelEle) to another element(s) (SelEle + Label Increment). The routine checks for elements with loads within the selection and copies them if they exist. Generally the routine will be most effective using the select by Label Range or select by Window.

When the command is selected the label increment between the element pattern is entered. The elements with the loads are then selected using the current selection method.

-o-

Display List - Element Thermal & Pressure Loads

This command is used to view the load data in a list format. List entries may be selected for editing etc.

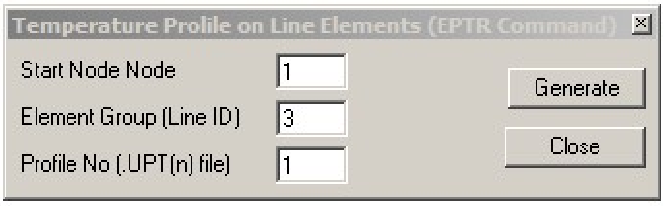
The list has the following command buttons

Delete	Deletes the selected entry.
Edit	Displays the Input form with the selected entry as the current form entry.
Update	Updates the load list. To be used following data entry or editing.

-o-

Line Temp Profile (ETPR Command)

The following form becomes visible when the above menu command is selected. This enables easy entry and execution of the ETPR command parameters.



The ETPT command is used to generate a temperature profile on line elements defined by an element group. This is a wizard command which should be used interactively in the Load Definition TASK. The routine generates TEPR load commands and should only be used once. The profile is applied by linear interpolation from a profile curve defined by a number of discrete points.

Node Start node for the profile (attached element must be in group)

EGroup Identifying element group for the profile - must be in current SET

ProfileNo Profile ID number. The profile is defined a user defined text file **.UTP(N)** where N is the profile ID number. The format for the file is:

Number of points

Distance1, Temperature1

….. , ……

DistanceN,TemperatureN

If the element group in not continuous along the line element, the profile definition will terminate. If a branch is encountered it will follow the grouping. If groupings are common at a branch it follow the element with the lowest label.

The profile can be plotted using the **TP** button located in the Pipework Toolbar.

-o-

Load Definition:Menu:Tp/Pr:Nodal Temp(FE)

This menu provides commands for the definition of nodal temperatures of solid elements. Nodal temperatures defined here are only used for thermal expansion analysis and heat transfer (non zero initial temperatures for transient heat solutions).

Thermal strain is based on $\varepsilon t = \alpha.\Delta T$ where $\Delta T =$ (T - Tamb) and Tamb is the ambient temperature

If a nodal temperature is not defined at a node it will be assigned a zero value or the value of the Ambient Temperature if one is defined i.e. the node will be at assumed to be at the ambient temperature unless specifically defined.

Nodal temperatures have no effect on beam or pipe elements. They must be defined on an element basis using the Tp/Pr menu.

Input	Makes the primary Nodal Temperature Input dialogue box visible for load definition
Delete	Removes temperature definition from selected nodes
Copy by Node	Copies temperatures by reference to the node to which they are assigned using a defined label increment
Display List	Displays the Node Temperature list box. List box can be used for edit or deletion selection.

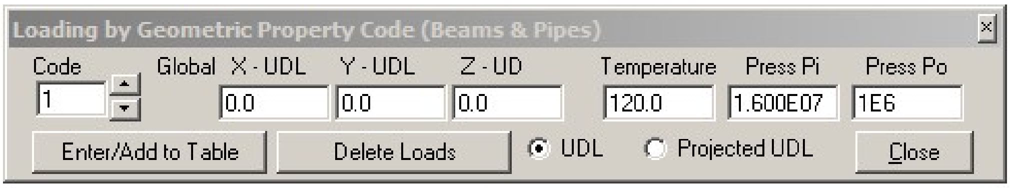
-o-

Load Defintion:Menu:PropLDS

This menu provides the input capability to define UDLs, differential temperature and differential pressure related loading on elements by reference to the Geometric property code of the element. If element temperature and pressure is defined by direct element reference then this will take precedence over property code definition. Property referenced UDLs are accumulative to other defined loading.

Input/Edit   Makes the Loading by Geometric Property Code dialogue box visible for load definition.

Display List   Displays the Geometric Property Loading list box. The List box can also be used for the deletion of loading.

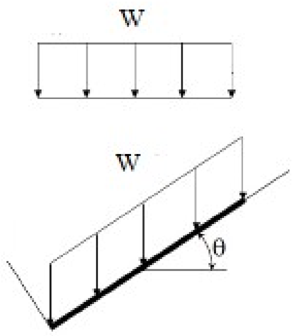


Code is the Geometric Property Code

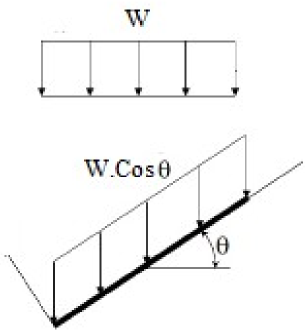
X-UDL, **Y-UDL**, and **Z-UDL** define the uniform distributed loads in the global co-ordinate system.

The **UDL or Projected UDLl** option buttons define how the global loads are applied.

UDL



Projected UDL



Temperature defines the differential temperature of the element.   The differential temperature used to evaluate axial strain is the differential between this value and the **Ambient Temp** defined temperature.

Press Pi defines the internal pressure for pipe elements. The units should be consistent with the stress units i.e. N/m2 for SI units.

Press Po defines the external pressure for pipe elements. The units should be consistent with the stress units i.e. N/m2 for SI units. If this is define as zero Pi becomes the differential pressure i.e. the gauge pressure.

-o-

Load Definition:Menu:Grv

This is a command that enables the gravitational constants (accelerations) to be defined using the gravitational constants dialogue box.

Gravitational constants can only be defined in the principal axis of the global co-ordinate system. Use component definition for other planes. Note downwards is -ve y direction.

The gravitational constants are used by the following expression to evaluate loads

Fg = Den x g x csa

where Den = Material density defined in the Material Property Table

g = Gravitational constant (acceleration)

csa = area defined in the Geometric Property Table

Centrifugal acceleration can be applied to axi-symmetric elements. For these elements the X direction constant is interpreted as the angular velocity in rads/s.

-o-

Load Definition:Menu:Load Definition:Menu:Non_LinEff

The menu enables specific non-linear loading effects to be defined. These definitions will be ignored when undertaking linear analysis.

A sub menu enables the following load commands to be defined.

ESTR The **ESTR** command is used to define beam element axial strain. This command can be used with the large displacement option ie the strain rotates with the element. This command is very useful when used with spar (Type 15) elements for simulating winch wires where large strains can be use represent winch wire movement.

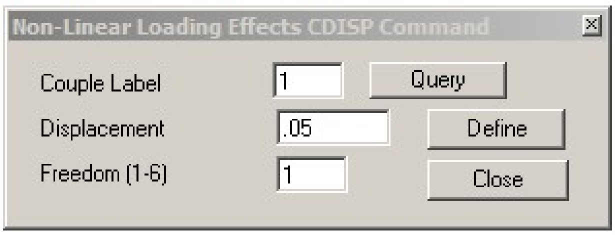
SEFO The **SEFO** command is used to define element axial loads with Type 15 elements.The axial stiffness of the element should be zero or an extremely small value to assist convergence. This can be done setting the E or Area to near zero. Setting the E value to zero has the advantage that the evaluated stresses will be low.

CDISP The **CDISP** command is used to define relative movement between the coupled degrees of freedom of a Type 7 couple element. It effectively applies prescribed displacements between connected nodes in a similar manner to which nodal prescribed displacement do between a node and the boundary. Note that Type 7 couple are large displacement couple when associated with a large displacement element.

CFACT The **CFACT** command is used to factor the stiffness (all components) of a Type 7 couple element. Its main use is to add or remove couple elements (birth & death) during a time history solution, i.e. for making or breaking connections during dynamic simulations.The stiffness of the couple is directly factored by the value of the associated time curve at the current time step. The command is added to any of the load cases (only one) in a time history combination.

More information on the above commands is given in Section 9.7 under Additional Load Commands.

The input form shown below is for the CDISP command and is typical for the other commands.

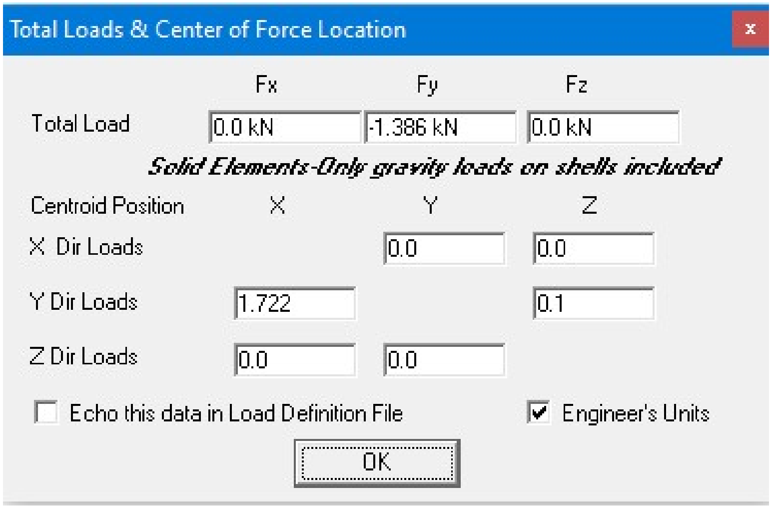


The Query button can be used to check for any existing definition on elements identified with the label entry (only first one entry shown for multiple CDISP definition).

-o-

Load Definition:Menu:Load Definition:Menu:Sum

This command evaluates the total loading and centres of force on all visible nodes and visible elements.



Only gravitational loading on shell elements (Types 50-69) are included in the above valuation.

Other type of loading on shells and loading on other finite (solid) elements are excluded.
If a load case has such, the only way to obtain the loading contribution from these elements is from the Reaction Summation which is evaluated when a Result Case is formatted.

-o-

8.14 Design Parameters

This Task enables additional model definition data used by the optional design code checkers to be defined. Unless code checker are to be used there is no requirement to define this data.

Data files associated with design definition are loaded into memory when the Task becomes active.

The Task is used to define member and tubular joint design parameters. Modified elements are shown in red. Modifies cans are shown in pink with only the side of the member with the modified joint shown such.

The Task has two modes, one for member definition and one for joints definition. The mode is made active using the active command in the respective menu. The active mode is indicated by a check mark. Modified elements will only be indicated if the appropriate mode is active. Data can only be saved in the active mode. Use the File Menu Save command to save design data.

Important - Any data changes will be lost unless the data is saved prior to quitting the Task.

The following menus are available in this task.

MemDesign	This menu is used to define member design parameters e.g. buckling lengths
CanDesign	This menu is used to define tubular joint design parameters e.g. can thickness

-o-

Menu:MemDesign

This menu provides the commands used for the definition of member design parameters i.e. effective lengths and length K factors used by the member code checkers for buckling checks.

The operation of the menu commands are similar to data entry and editing of load case data.

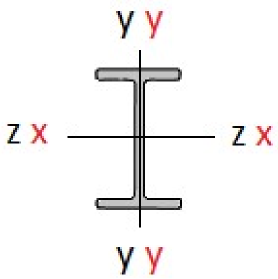
Define	Makes the primary Member Design Parameter dialogue box visible for data input
Remove Defn	Removes design parameters definition from selected elements
Display List	Lists elements with modified parameters. Default elements are NOT listed
Active	Makes the member design parameter input mode active

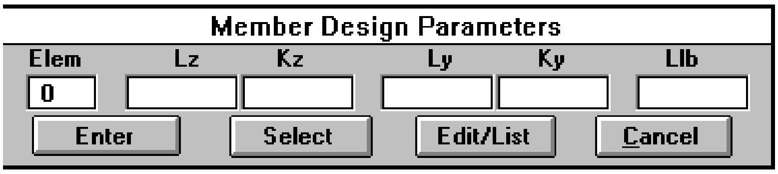
-o-

Define - Member Design Parameters

The following Member Design Parameters Input form becomes visible when the Define command is selected from the MemDesign menu.

The convention used for definition is the normal beam convention where x is the axial direction. This is different from the normal Design Book convention (the same as used in the FS2000's member design code checkers) where z would be axial.





Lz	Major axis compressive buckling length, about the z-z axis.	(Design Book x-x axis)
Kz	Major axis compressive buckling K factor	
Ly	Minor axis compressive buckling length, about the y-y axis.	(Design Book y-y-axis)
Ky	Minor axis compressive buckling K factor	
Llb	Unsupported flange length (lateral buckling in beams)	

Zero values are ignored and infer that the default values will be used in cases where no previous definition exists. Exiting values will not be changed in a zero value is entered in a box. The default values for the lengths are the actual element lengths. The default values for the K factors are 1.

If the **Select** button is clicked the elements will be selected by the current selection method.

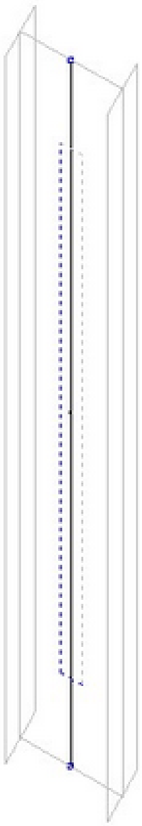
This **Edit/List** button enables existing element data to be listed, edited or copied. When the button is pressed the mouse is used to select the element is to be listed. The data can now be modified or copied as required using label reference or the current selection method.

Visual Selection

Lengths may be assigned visually by picking two nodes. If two nodes are picked (Node Query) the the length between the nodes is listed in the List Box. If a length box in the above form is then double clicked the length will be entered into the box.

Effective Length Sign Convention Example

Note that the convention below is the Design Book convention. This is an extract from a Member design check.



Buckling about the minor axis (y-y) is dominant Lx-Ly=7.0

```
Section Classification                    Design strength = 345.00 N/mm2
Radius of Gyration     rx  = 8.49 cm      ry  = 2.37 cm
Actual Member Length   Lx  = 7.00 m        Ly = 7.00 m
Eff Length Factor K    Kx  = 1.00          Ky = 1.00
Slenderness            SLx = 82.46        SLy = 295.26
Eff. Slenderness   (KL/r)x = 82.46    (KL/r)y = 295.26
                       Cc  = 108.301
Euler Stress           Fe  = 155.3 N/mm2  Fe  = 12.1 N/mm2
Compressive Strength   pcx = 129.2 N/mm2  pcy = 12.1 N/mm2
```

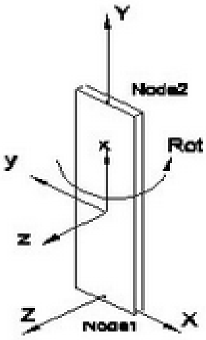
Buckling about the major axis (x-x) is dominant
Minor axis supported at 1/2 points Ly=1.75

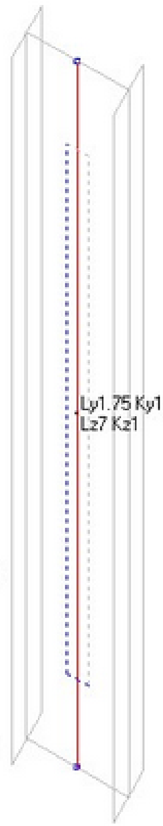
```
Section Classification                    Design strength = 345.00 N/mm2
Radius of Gyration     rx  = 8.49 cm      ry  = 2.37 cm
Actual Member Length   Lx  = 7.00 m        Ly = 1.75 m
Eff Length Factor K    Kx  = 1.00          Ky = 1.00
Slenderness            SLx = 82.46        SLy = 73.81
Eff. Slenderness   (KL/r)x = 82.46    (KL/r)y = 73.81
                       Cc  = 108.301
Euler Stress           Fe  = 155.3 N/mm2  Fe  = 193.7 N/mm2
Compressive Strength   pcx = 129.2 N/mm2  pcy = 140.7 N/mm2
```

Member Design Parameters

Elem	Lz	Kz	Ly	Ky	Llb
1	7	1	1.75	1	7

Enter | Select | Edit/List | Cancel





-o-

Menu:CanDesign

This menu provides the commands used for the definition of tubular joint design parameters i.e. wall thickness and diameters of stubs and cans used by the tubular joint code checker.

The operation of the menu commands are similar to data entry and editing of load case data.

Define	Makes the primary Tubular Joint Can data dialogue box visible for data input
Remove Defn	Removes design parameters definition from selected elements
Display List	Displays the Tubular Joint Can Data list box. List box can be used for edit or deletion selection.
Active	Makes the tubular joint design parameter input mode active

-o-

Design Parameters:Menu:CanDesign

The following Tubular Joint Can Data Input form becomes visible when the Define command is selected from the CanDesign menu.

Wall N1 Member wall thickness at Node1 of the element

OD N1 Member outside diameter at Node1 of the element

Wall N2 Member wall thickness at Node2 of the element

OD N2 Member outside diameter at Node2 of the element

Zero values are ignored and infer that the property code values will be used in be used in cases where no previous definition exists. Exiting values will not be changed if a zero value is entered in a box.

If the **Select** button is clicked the elements will be selected by the current selection method.

This **Edit/List** button enables existing element data to be listed, edited or copied. When the button is pressed the mouse is used to select the element is to be listed. The data can now be modified or copied as required using label reference or the current selection method.

Visual Selection

If an element is listed with the Element Query button the First Node of the will be circled. This identifies to which boxes (N1 or N2) the data should be entered.

-o-

8.15 Analysis

The following menus are available in this task

Reseq	Used to see the effects of re-sequencing the internal numbering used in the matrix solution
Solution	Used to initiate load case solution
PostPro	Used to initiate post-processing

-o-

Analysis:Menu:Reseq

This menu provides the commands to enable the effect of re-sequencing the internal numbering system used during the analysis solution to be assessed. Options to save the renumbered sequence for subsequent solution are also available.

Re-sequencing does not effect the labeling defined by the user when creating the model.

It is not essential to renumber the model each time an analysis is run since the solver will read any previously saved re-sequence file. Only when a model is saved is the existing re-sequence file deleted.

These re-sequencing options are also available in the analysis submission forms.

Why Re-sequence?

With relatively small models re-sequencing is not required but as models increase in size the benefits of re-sequencing increase. The benefits are that the solution is more efficient both in speed and memory usage (larger models possible). Many users re-sequence models if the number of nodes exceed about 30. With very large models it may essential to re-sequence so as to fit the model into memory or in extreme cases (ill-conditioning) to obtain an accurate solution. Its good habit to always re-sequence.

FS2000 employs two basic methods of matrix solution:

- Frontal(Wavefront)
- Banded Solvers.

In Frontal solvers the sequence of the elements is critical. The WaveFront Optimiser is used with these solvers.

In Bandwidth solvers the sequence of the nodes is critical. The Bandwidth Optimiser is use with these solvers.

The Analysis form indicates which method is used for the selected solution method. The form shows the input options for either the WaveFront Optimiser or the Bandwidth Optimiser.

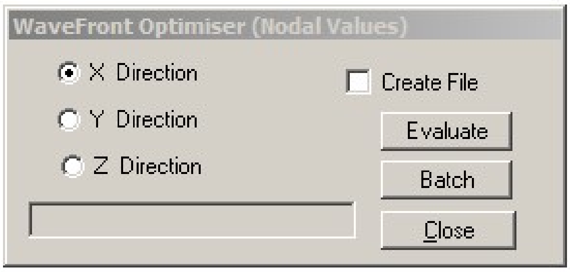
The following menu commands are used to activate the appropriate re-sequencing utility.

WaveFront Optim	Wave Front Re-sequencing for the Frontal solution methods
Bandwidth Optim	Bandwidth Re-sequencing for the Banded solution methods

-o-

WaveFront Optimiser

The following form will become visible when this command is selected.



Generally the lowest front will be obtained by specifying the direction running along the longest length of the model i.e. on a tower structure the direction would be vertical.

If in doubt, specify each of the global directions i.e. x, y and z in turn to establish which gives the lowest front.

The front will only be evaluated when the **Evaluate** button is clicked. The front width will then be shown in the Text box.

Create the renumber file for the lowest front direction by clicking the Create File check box and re-evaluating.

On large models the evaluation of the front may take a few minutes.

When the re-sequence file is saved the model need not be re-sequenced again unless the model is re-saved. In such cases the direction for re-sequencing will be known and it will be more convenient to initiate the re-sequencing prior the solution using the options in the Analysis form. This command would always be at the top of a Batch files ie before the solution.

Using the Bandwidth Re-order Sequence

In some cases such as in a U shaped structure it may be more desirable to renumber the elements based on a the nodal bandwidth re-numbering scheme. This can be don by using the Bandwidth Optimiser with the Wavefront Option checked.

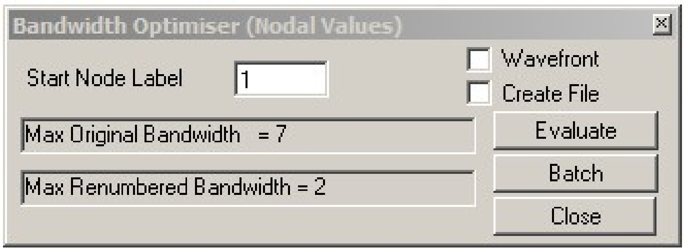
Maximimum Wavefront

If the solver warns that the model wavefront is too large that the solver wavefront capacity can be extended by creating or changing the wavefront settings file.

-o-

Bandwidth Optimiser

The following form will become visible when this command is selected.



Generally it is most efficient to define a corner node of the model. If there is no obvious choices simply try different nodes on a trial and error basis and choose the one with the lowest bandwidth.

Note - Multiple start nodes may be defined when running the Batch module 'BAND' . This can be useful for guiding the numbering sequence. See the Batch Process Module Help for details

Create the renumber file for the lowest bandwidth by clicking the **Create File** check box and re-evaluating.

The **Wavefront** option will re-order the elements for the frontal solver based on the node ordering for a banded solver. Generally this will be less efficient than the Wavefront Optimser but in some case such as in a U shaped structure it may be more efficient.

Warning - If the model has Rigid Links do not create the Renumber File. If the model has rigid links the Batch Module 'BAND' must be used to renumber the model. See the Batch Process Module Help for details.

On large models the evaluation of the bandwidth may take a few minutes.

When the re-sequence file is saved the model need not be re-sequenced again unless the model is re-saved. In such cases the start node for re-sequencing will be known and it will be more convenient to initiate the re-sequencing prior the solution using the options in the Analysis form. This command would always be at the top of a Batch files ie before the solution.

Maximum Bandwidth

If the solver warns that the model bandwidth is too large that the solver bandwodth capacity can be extended by creating or changing the bandwidth settings file.

-o-

Analysis:Menu:Solution

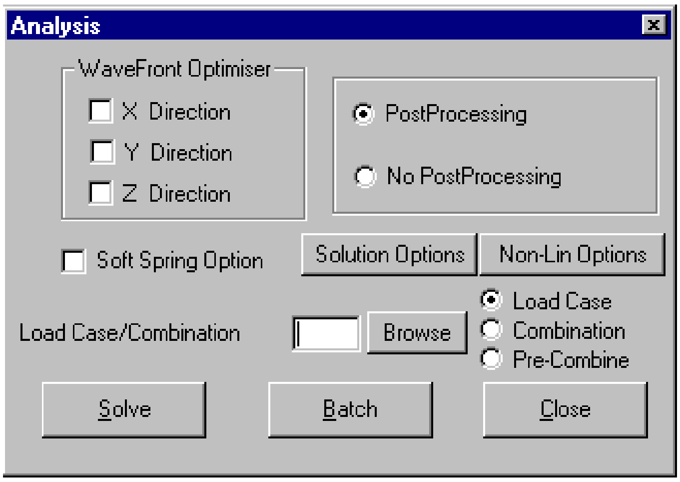
This menu provides the commands used to submit a load case or a combination of load cases for the solution stage of analysis using an appropriate solution option. The solution options available are:

2-D Plane Frame (Banded)	2-D frame analysis.
3-D Light (Banded)	3-D frame analysis. Not usually used if standard is available. For small models only
3-D Standard (Frontal)	**3-D Standard** analysis solver**.**
Non-Linear Analysis (Band)	**3-D NL** and Pile-Soil Interaction Analysis
Eigen Frequency-Buckling (Banded)	Modal Frequency Analysis. This option has its own Help File
Dynamic Response	Modal Response Analysis. This option has its own Help File
FS-DyNoFlex (Banded)	Non-linear static/dynamic analysis.

-o-

3-D Standard Analysis

When the 3-D Standard option is selected the following Analysis form becomes visible.



The 3-D Standard solution employs a Frontal matrix solution in which the element ordering dictates the efficiency of the solution. The **WaveFront Optimiser** options are used to optimise the solution.

If the **Post Processing** option is checked the load case or load case combination will be post-processed directly following analysis. This is only a requirement when a single load case being analysed or the combination submitted for analysis is also to be used to create a processed results case.

The **Load Case/Combination** box is used to enter the Load Case or Load Case Combination (multiple load cases) to be analysed. If the number is preceded by a **C** or **c** the analysis module will analyse multiple load cases in succession. If the number is preceded with a **P** or **p** then the load combination will be merged.

WARNING - With successive load case runs **DO NOT** include load cases that have prescribed displacements unless all load cases in the same multiple run are subject to the same prescribed displacements in the same freedoms. For more information on prescribed displacements

The **Browse** button is used to show a list of the existing Load Cases. If either the **Combination** or the **Pre-Combine** options are checked the browse button will show existing Load Case Combinations. The **Combination** option will prefix a **C** to the selected case. The **Pre-Combine** will prefix a **P** to the selected case.

The analysis modules can apply a soft spring (1E-10) to all element degrees of freedom if the **Soft Spring Option** is checked. This eliminates the possibility of the user creating a 'pure' mechanism by either incorrectly specifying too many end releases at a node or neglecting to define all stiffness parameters. In practical terms the structure will still form a mechanism since any loading in the freedom of the soft spring will create huge deflections. If these deflections are not apparent then the solution is likely to be a valid one.

The Solutions Options & Non-Lin Options buttons are use to set non standard analysis options.

Pre-Processing Load Case Combinations

Pre-processing is used to merge the individual Load Cases of a Load Case Combination into a new single Load Case. The new merged Load Case is assigned the same number as the Load Case Combination. Load case factors are taken into account in the merge process.

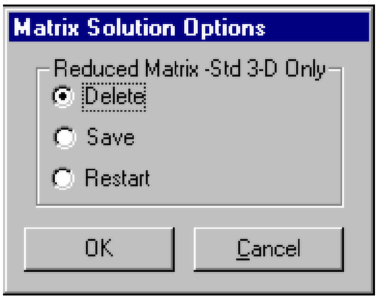
The merged load case will not contain any load definition commands but its description will identify that it is a pre-merged combination and show the combination number.

-o-

3-D Standard - Solution Options

The following forms are used to set the solution options for the 3-D Standard solution option.

Matrix Options



This provides a restart capability by saving the reduced model stiffness matrix.

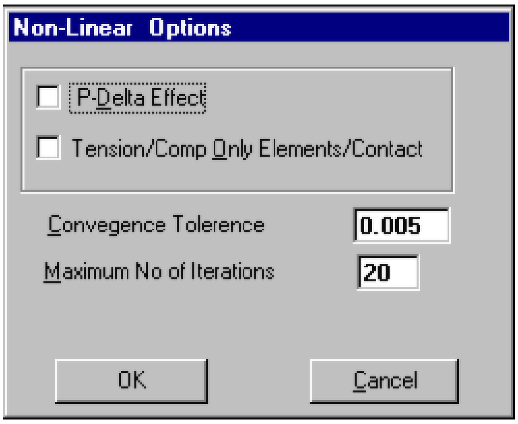
The restart feature allows the user to analyse load cases using the global stiffness matrix assembled during a previous solution. In a large model this can be save considerable analysis time since the most of the time used in analysis is spent assembling the matrix.

The analysis has to be run once with **Save** active so as to create the restart file.

Subsequent load case solutions are then run with **Restart** active.

In batch opperation the above matrix operations are sset in the LOADA command.

Non-linear Options



The **P-Delta** check box is used to make the P-Delta effect active.

> P-Δ or P-δ? P-Δ is the global structure displacement effects(sway) and P-δ is the local member (span) effects. In terms of program implementation there is no difference between the two and depending upon mesh density both effects will be included. At least one mid-span node is required for P-δ to be included. If double curvature exist in the span 3 nodes will increase the accuracy.

The **Tension/Compression/Contact** check box is used to make this solution option active. This option activates tension/compression only elements and gap elements. Note that in the 3-D Standard Solver the Gap elements only behave as 'make or break' contact elements.

The non-linear solution is an iterative one based on the comparison of displacements between iterations. The solution is complete when either the convergence tolerance is satisfied or the maximum number of iterations are completed.

Stored Settings

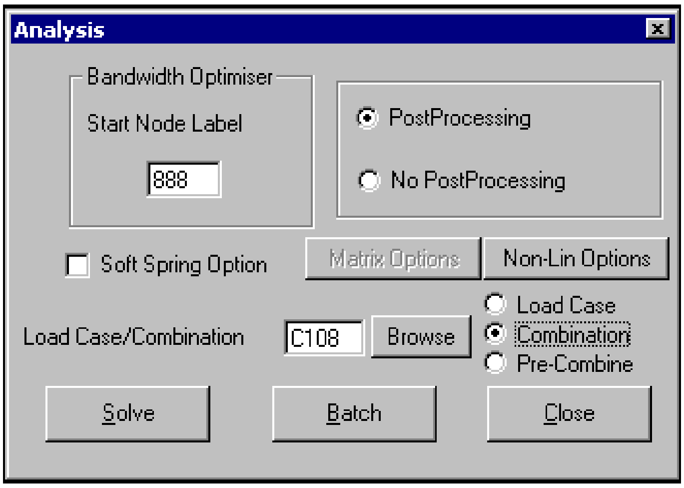
The non-linear option settings are a global model settings. The are contained in the **<modelname>.UPT** file. When running in batch, they can be made load case specific by using the MFCopy command to re-create the setting file prior to solution.

See Analysis Options for more information on this type of analysis.

-o-

Non-Linear Analysis

When the Non-linear Analysis option is selected the following Analysis form becomes visible.



The Non-linear solution employs a Banded matrix solution in which the nodal ordering dictates the efficiency of the solution. The **Bandwidth Optimiser** options are used to optimise the solution. If a zero value is entered for the Node label no renumbering will take place

If the **Post Processing** option is checked the load case or load case combination will be post-processed directly following analysis.

The **Load Case/Combination** box is used to enter the Load Case or Load Case Combination (multiple load cases) to be analysed. If the number is preceded by a **C** or **c** the analysis module will analyse multiple load cases in succession. If the number is preceded with a **P** or **p** then the load combination will be merged. The maximum case ID is 999.

The **Browse** button is used to show a list of the existing Load Cases. If either the **Combination** or the **Pre-Combine** options are checked the browse button will show existing Load Case Combinations. The **Combination** option will prefix a **C** to the selected case. The **Pre-Combine** will prefix a **P** to the selected case.

The analysis modules can apply a soft spring (1E-10) to all element degrees of freedom if the **Soft Spring Option** is checked. This eliminates the possibility of the user creating a 'pure' mechanism by either incorrectly specifying too many end releases at a node or neglecting to define all stiffness parameters. In practical terms the structure will still form a mechanism since any loading in the freedom of the soft spring will create huge deflections. If these deflections are not apparent then the solution is likely to be a valid one.

The Non-Lin Options button is use to set the analysis options for the solution.

Pre-Processing Load Case Combinations

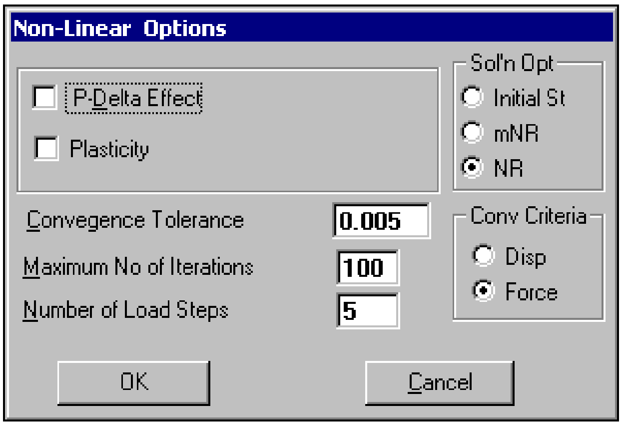
Pre-processing is used to merge the individual Load Cases of a Load Case Combination into a new single Load Case. The new merged Load Case is assigned the same number as the Load Case Combination. Load case factors are taken into account in the merge process.

The merged load case will not contain any load definition commands but its description will identify that it is a pre-merged combination and show the combination number.

-o-

Non-Linear - Solution Options

The following form is used to set the solution options for the 3-D NL solution option.



The following give some guidance on the use of the above solution options but more detailed information is given in Section 6.

Convergence Criteria

It is always recommended to use **Force** for the **Conv Criteria** . If convergence difficulties are experienced and the solution fails then using **Disp** should be tried as this generally converges more easily.

Solution Option

The **Initial St** option can be used for Frame Plasticity (Perfect Plasticity) or Pile Analysis. If other non-linear effects are also included then mNR or NR are recommended.

If the model contains gap elements the **Sol'n** option should be set to **NR**, using other setting with contact elements may result in convergence difficulties.

Convergence Tolerance

This should be set to suit the required accuracy. The higher the value the more easily convergence is achieved. If a higher value is being used to obtain a solution always do element force checks to confirm the validity of the solution in the area of interest. The convergence criteria is based on the ratio ΣΔD**2 / Σ D**2 being less then Tol**2 where D represents forces or displacements or both.

Maximum Number of Iterations

Generally setting a high value will not aid convergence with a model that does not converge. Plastic solutions near the collapse limit

Number of Loads Steps

This parameter has the most significant effect with respect to non-convergence. If a model does not converge increase the number of loads steps.

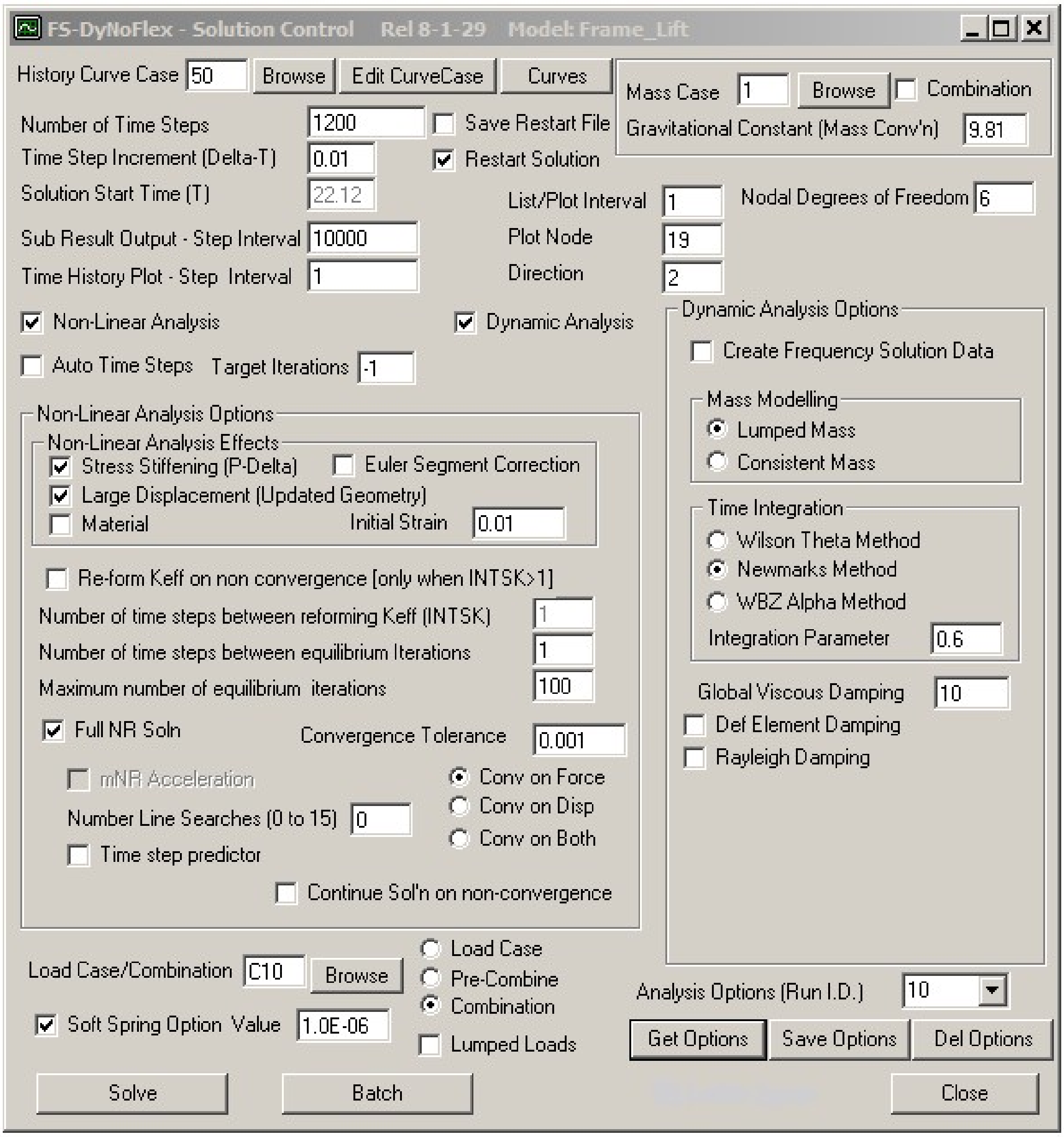
Stored Settings

The non-linear option settings are a global model settings. The are contained in the **<modelname>.UPI** file. When running in batch, they can be made load case specific by using the MFCopy command to re-create the setting file prior to solution.

-o-

Dynamic Non-Linear Analysis

When the Dynamic Non-linear Solution option is selected the following Analysis form becomes visible. DyNoFlex is a non-linear incremental solver and can be used for both static and dynamic solutions.



The above form has the three basic input sections. The form settings are required to be save using an Option ID number, they can then be recalled for subsequent analysis runs or will be loaded when running the solution in Batch. The basic input section are:

- Common Analysis Setting
- Non-linear Solution Settings
- Dynamic Solution Setting

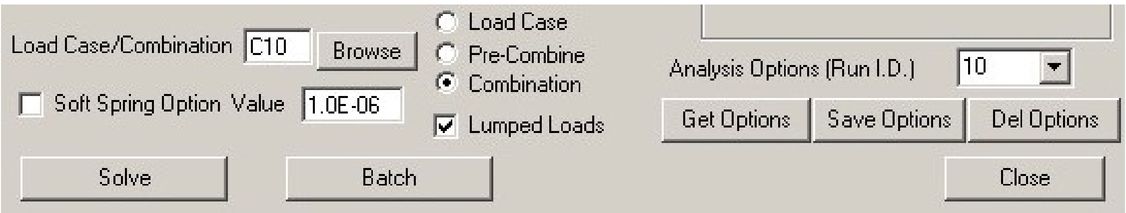
These are described in the above sub-sections.

When running a linear solution (**Non-Linear Analysis** option not active) it is recommended that only linear element be used in the model. The exception is a Type 6 beam which will behave as a linear Type 0 element. It would be unusual to use DyNoFlex for a linear solution unless of course it was also dynamic solution.

-o-

Common Analysis Setting

Lower Section



The **Load Case/Combination** box is used to enter the Load Case or Load Case Combination (multiple load cases) to be analysed. If the number is preceded by a **C** or **c** the analysis module will analyse multiple load cases in succession. If the number is preceded with a **P** or **p** then the load combination will be merged. See Section 6 for more information on loadings. The maximum case ID is 999.

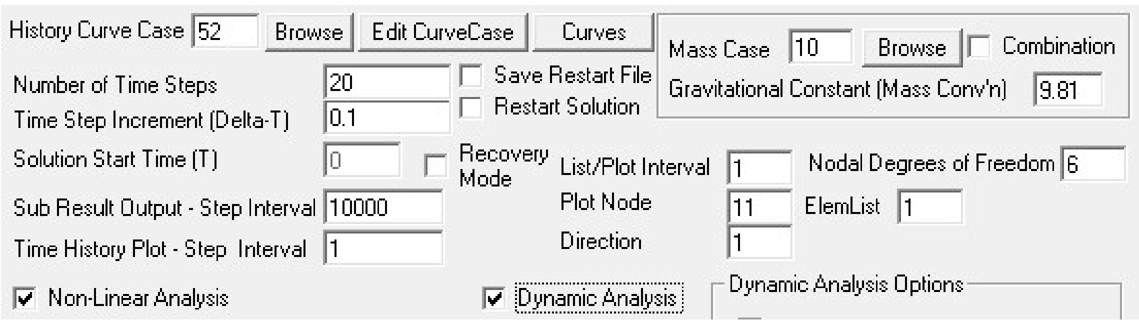
The analysis modules can apply a soft spring (1E-10) to all element degrees of freedom if the **Soft Spring Option** is checked.

The **Lumped Loads** option if active will convert all beam/pipe element loads eg self weight to nodal loads. See Section 6 for more information on large displacement loadings.When the large displacement option is selected the lumped loads option will be activated. This is done because a solution with large body rotations maybe being undertaken and the local distributed element loads may not be appropriate because they do not rotate with the elements. However, if element rotations are not too large it is better that is in not active so that more accurate P-delta formulations that handle span distribute loading are applied.

The **Analysis Options (Run I.D.)** is used to save all solution setting on the form. They can then be recalled for subsequent analysis runs or will be loaded when running the solution in Batch. It is always recommended to use the same load case or load case combination number as the case ID number.

The **Solve** button is used to start the current solution. When this is clicked the setting will be automatically saved and the solution will start. Note that the results case will not be automatically post-processed after solution.

Upper Section



The **History Curve Case** is used to identify the Load Case which contains the Time Curves for the current solution.

> If **C** or **c** in entered it will signify that the load combination is to be solved using the loading sequence of the combination list and apply load factors in the combination as 'load factors' in the same manner as would be done when using the Static Non-linear incremental solution option.
> If a History Curve Case specified then it must contain an ID 1 Curve
> When a single load case is to be analysed it will be assigned to Time Curve 1 in the Curve Case, this eliminates the requirement to use a combination when only a single load case is being used.
> If the History Curve Case is blank a unity ramp curve will be applied.
> If a History Case does not contain time curve commands then a unity ramp curve will be applied.

The **Number of Time Steps** and **Time Step Increment** are used to define the solution duration. See Section 6 for more information.

The **Save Restart File** option will save all model variables relating to the model status i.e. stiffness, displacement, velocities, gap status etc at the end of the last time. Note that the loadings on the model are not saved.The restart file is only saved when a converged solution is complete

The **Restart Solution** option enables the solution to be started, using as initial conditions, the conditions of the model in the restart file. The restart file is not model or case specific it is global, therefore always ensure that the current restart file is the correct one e.g. re-create when changing models. A warning will be given if the restart time history case uses a different Case No from that when the restart file was saved. This warning can normally be ignored. The reason for this warning is that the model loading in a restart time history case should be identical at the start of the restart solution to the loading when the restart file was saved. This requirement can always be satisfied by simply ensuring that different loading time histories are identical up to the time when the restart file is saved/started. Using the save and restart is a way of varying the the time steps in a solution that requires a final small time step.
A dynamic restart is usually preceded by a dynamic solution but it can also be preceded by a static solution.

The **Recovery Mode** option is similar to the **Save Restart File** option but the restart data is saved after the last converged time step. The enables the solution to be restarted even if the the solution fails to fully converge in a subsequent time step. The is only for interactive use and cannot be batched.

The **Mass Case** is use to select the load case that define the mass distribution for dynamic analysis and the **Gravitational Constant (Mass Conversion)** is used to convert y direction forces in a load case to masses. See Mass Definition.

The **Nodal Degrees of Freedom** is used when 2-D and 3-D solid elements are being modeled (set to 2 or 3).

The **List/Plot Interval**, **Plot Node** and **Direction** are used to a identify a degree of freedom that can be monitored as the solution progresses. See Section 4.8.7 for plotting instructions. The output is written to the plot file, **<modelname>.~plot'n'** , where **'n'** is the result case number. Direction numbers 1,2,3,4,5 & 6 reference displacements, 11,12,13,14,15 & 16 reference velocities and 21.22.23.24.25 & 26 reference to accelerations.

The **ElemList** is used to identify an element for which plastic stress and strains are listed. Only for Type6 (7 & 11) pipe elements and solid element plasticity. The list is shown only in the solution window at the end of a solution. In the case of pipe elements, only data relating to Node 1 of the element is listed.

The **Sub Result Output - Step Interval** is used define an time step interval at which result data may be written. A **SubCase** utility (described below) is require to be used to convert the written data to standard FS2000 output results format - see below.

The **Time History Plot - Step Interval** is used define an time step interval at which beam forces and nodal displacements can be written to a plot file for subsequent graph plotting (see Section 4.8.7). This enables the time history of any force action in a model to be plotted. Type 0 beam elements and Type 0 Couple elements do not write such force data, only non-linear elements are capable of writing the force data.

Sub Case Utility

The SubCase utility can only be run in Batch model. The utility converts intermediate time history result cases to standard raw result cases.

SUBCASE C1/C2/C3/

C1 Primary Result Case Number

C2 Time Step Range eg 1-300 (This is a step number interval and not a time interval)

C3 Start Number for the Std FS2000 Results Case

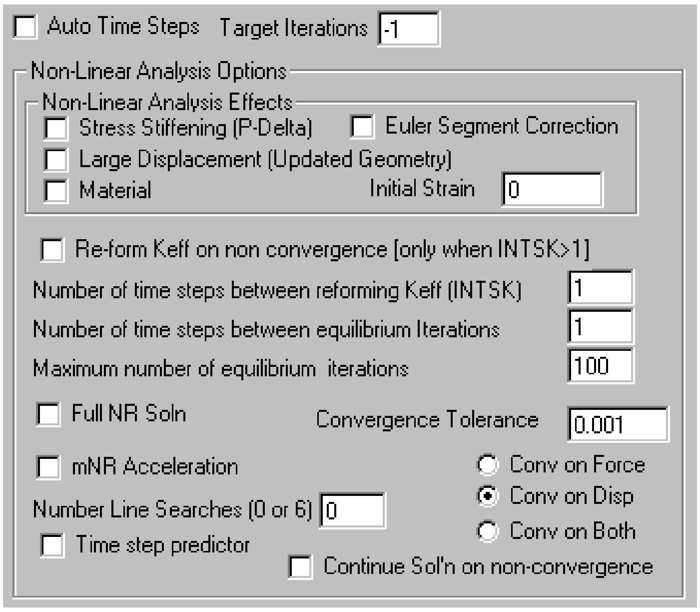
The resulting output from the utility are raw result cases which require to be post-processed in the normal manner.

The Time Step Range can be started at any time step eg 500-1000 would scan that range of time steps from the and if sub case results exist they will be processed. If the **Sub Result Output - Step Interval** was set to 1 and 500 time steps were set, 500 cases would be created. If it was set to a value greater than one eg 2 then 250 cases would be created.

Note that intermediate result cases can only be selected In the FS2000 GUI following post processing.

-o-

Non-Linear Solution Setting



The use of **Auto Time Steps** and **Target Iterations** is described in Section 6.

Non-Linear Solution Options

The **Non-Linear Effects** are described in Section 6.

The **Euler Segment Correction** is used to give a warning if a compressive beam elements has loading greater than the Euler buckling limit. It will also limit the load used to evaluate the P-Delta lateral stiffness to that value. Regard this as a tool for investigating non-convergence in compression buckling. In such cases the element lengths should be reduced.

The **Initial Strain** is used to define an initial tensile axial strain used to evaluate the P-Delta lateral stiffness in a beam element for the first iteration of the initial solution. This can be used to provide initial stability for tension controlled structures.

Solution Strategy

The **Number of time steps between reforming Keff** defines the time step increment for updating the tangent stiffness matrix. For solutions in which frame plasticity is the only non-linear effect, this value should be set to a high value so that the stiffness matrix never reformed (Initial Stiffness Method).

The **Number of time steps between equilibrium Iterations** defines the time step increment for checking that the solution accuracy is within the specified convergence tolerance.

The **Maximum number of equilibrium iterations** sets the number of equilibrium iterations that will be performed. If convergence has not been achieved then the solution will terminate or continue depending upon other control settings.

The **Full NR Soln** selects a full Newton-Raphson solution strategy in which the tangent stiffness is reformed at every iteration and every time step. It is recommended that this option be set for models that include gap elements. When this is not active the solution becomes the modified Newton-Raphson solution **mNR**.

The **Convergence Tolerance** set the convergence limit. The convergence criteria is based on the ratio $\Sigma\Delta D^{**}2 / \Sigma D^{**}2$ being less then $Tol^{**}2$ where D represents forces or displacements or both.

The **Continue Sol'n on non-convergence** option if active will continue the solution to the next time step even if the equilibrium criteria is not satisfied.

The background theory to the solution strategy is described in Section 6..

Convergence Control

Also see iteration control in Section 6

The **mNR Acceleration** option is used to activate Chrisfield's Secant Acceleration Method. This technique can result in fewer iterations when a modified NR solution is being undertaken.

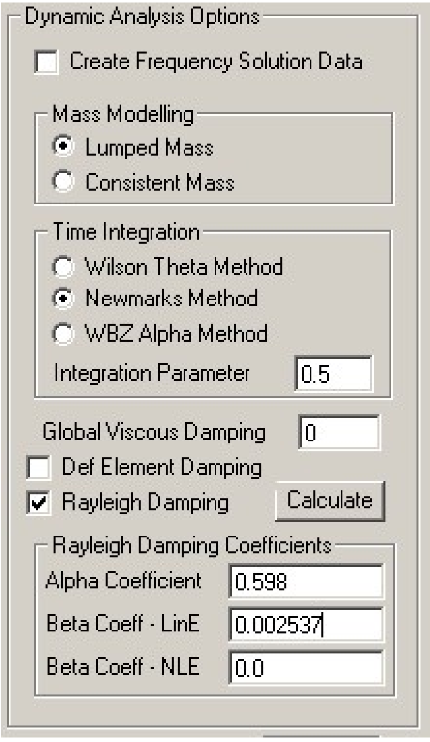
If a **Number of Line Searches** is defined i.e. > 0 it will active the line search option. This would normally be set to its maximum which is equal to 15. The line search option attempts to improve the convergence of a Newton-Raphson (mNR and NR) solution by scaling the displacement increment. It can be very effective in Frame Plasticity when used with **mNR Acceleration.** Not recommended for other type of plasticity (stress-strain)**.**

The **Time step predictor** is a simple predictor which uses the displacement increment from the previous time step to predict the displacements at the start of the next step. The tangent stiffness and NR restoring force are based on this predicted displacement. This can result in fewer equilibrium iterations. Can be used with the above two methods.

Ref: M.A. Crisfield, "Non-linear Finite Element Analysis of Solids and Structures" Wiley 1995

-o-

Dynamic Solution Setting



The **Create Frequency Solution Data** option is used to prepare data for a modal solution using the Frequency Analysis module. This feature enables a frequency solution to be undertaken at any desired loading pattern, time or model configuration. This is would normally used when a restart solution is active. The necessary mass and stiffness data will be created that can be used if the **DyNoFlex Restart Solution (Frequency) option** is active for subsequent modal frequency solutions.

The **Mass Modeling Options** (Lumped or Consistent) selects the type of mass matrix to be formed for beam elements (plate elements always use lumped). If frequency solution is obtained and modal response analysis is to be undertaken then only the Lumped Mass option should be used. See Mass Definition

The **Time Integration** selects the time step algorithm for the prediction accelerations, velocities and displacements. The Newmark method and the Wilson Theta method are currently available. Section 6 gives more information on the solution of the dynamic equations.

The **Integration Parameter** provide a degree of numerical damping. Using high values for these parameter will degrade the dynamic solution. For the Newmark Method and WBZ method the value is generally in the range 0.5 to 0.6 but can be set higher (0.5 has no numerical damping and equates to the Average Acceleration method). By default the minimum value for δ is 0.505 i.e γ =0.005 regardless of the user defined value. For the Wilson Theta method the range should be in the range 1.39 to 2.01.

The **Global Viscous Damping** is used to specify global 'ground' damping to all degrees of freedom in a model. It will be added to any other damping concentrated damping definition. The units are Force/Velocity (M/(m/s) in default units). This can be used to assist convergence in static problems by providing a slow damped dynamic solution. Such an approach can produce a converged solutions for a quasi-static problems that would otherwise not easily converge.

The **Def Element Damping** is used as a solution switch to signify that element damping is specified in Type 3 Couples or Type 15 Beams. This type of damping is inactive during a linear solution. The option indicates that an additional non-diagonal damping matrix is to be employed which increases the in-core memory requirements. Think as this as being a switch to indicate that an additional non-diagonal damping matrix is to be used.

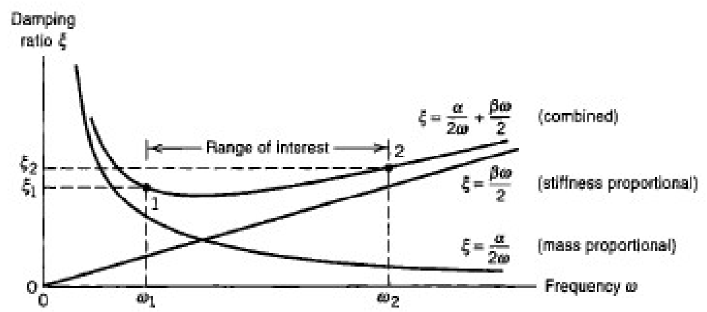
The **Rayleigh Damping** option allows the Rayleigh Damping coefficients α and β to be defined. They relate to the more usual 'damping ratio' γ by the expression $\gamma = (\alpha + \beta\omega^{**}2) / 2\omega$. The α coefficients give mass proportional damping which provides mass to 'ground' damping. The β coefficients give stiffness proportional damping which provides damping between element connected degrees of freedom. Stiffness proportional damping is useful for damping out unwanted high frequency response which is often unrealistic and associated with with discrete FE modeling.

When undertaking an analysis the Stiffness proportional damping is defined using two separate β coefficients.

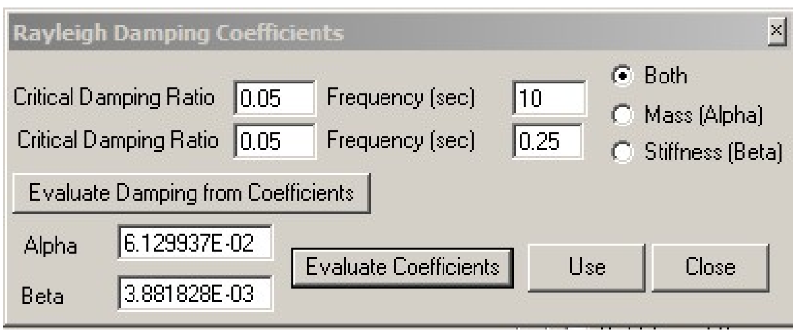
The β coefficient (**Beta Coefficient**) is applied to all linear elements. It is also applied and any non-linear elements that are supported during a linear solution e.g. Type 6 beams. The β coefficient is not applied to non-linear elements during a non-linear solution. The β coefficient is applied to all linear elements in a non- linear solution.

The β_N (**Beta Coeff NL**) is applied to all non-linear elements in non-linear solution. A non zero value for β_N requires additional memory for a non-diagonal damping matrix and increases the in-core memory requirements. It is unlikely that β and β_N would have different values. Think as β_N as being a switch to indicate that an additional non-diagonal damping matrix is to be used.

WARNING - When high β_N damping is applied to non-linear elements, the solution may struggle to converge if their stiffness varies considerably e.g. gap elements and tension only elements.



The **Calculate** button starts a utility that enables the Rayleigh Damping coefficients α and β to be evaluated for specific damping requirements.



-o-

Fatigue Assessments

This section describes how a fatigue assessment of a DyNoFlex time history solution can be undertaken in Batch Mode. This can be achieved by:

- Global Life Assessment - All Elements
- Selective Global Life Assessment - Specific Elements - Uses Elements Groups
- Individual Stress Point Assessment - Specific Element Stress Points

The first two methods use SN curve Stress-Life methods and will be describe here. The latter method is undertake by the FS-Crack module and can employ Stress-Life, Strain Life or Fracture Mechanics approaches

The basic procedure adopted is:

- At each solution time step the stress is evaluated at every element stress point. A pipe has 4 to 12 stress points at each node and non-pipe beams have 4 stress points at each node.
- These stresses are used to produce a stress time history for each stress point based on each time step.
- A Rainflow cycle counting algorithm is then used to convert the irregular loading history into a series of single amplitude cycles (see Section 7.3 in FS-Crack Help file for more info).
- The damage from these single amplitude cycles can be assesses from an SN curve.
- Using the Miner Rule approach the total damage at each stress point for the time history is obtained.

It can only be used with **non-linear beam** type elements. Only these type of elements produce a loading history that can accessed on completion of time history solution. If FS-Graph cannot produce a time history force plot of the element then a fatigue assessment will not be possible because they access the same data.

Time History Life Assessment

This uses two FS2000 fatigue modules which can only be run in Batch mode. These are the same modules used in OffFat but with very different command arguments. Unlike OffFat all intermediate data is always overwritten - only life (damage) date results. The only intermediate data output maybe of user interest is the damage file - <modelname>.D(n). Both fatigue category stresses are listed in this file. See the OffFat Help file for more information.

FATIG1S - The is used to created a <model>.D(n) damage file. This module processes a lot of data with subsequent long run times. Run times can be minimized by the following:

- Use Groups to identify only those elements of interest.
- Increase the DyNoFlex Time History Plot - Step Interval.
- Reduce the number of stress points for pipe elements (Range is 8-12).

FATIG4 - This produces the the final life output in both text and graphical life plot data. The text output from the FATIG4 will be output as Tubular Joint Design Results using the following categories. See Section 4.8 in the OffFat Help file for more information.

Summary Output - Minimum fatigue life at joint.
Stress Output - Fatigue life at all joint stress points.
Individual UR - Damage Ratios sorted by unity ratio (Life=1/UR) - Requires UR sort utility to be run.
Fatigue lives can also be plotted using the UR (Unity Ratio) plot command. The plot show the lowest fatigue life at each end of the element.

Command Line Instructions

FATIG1S C1/C2/C3/C4/C5/C6/C7/C8/C9/C10/C11/C12/C13/C14/

C1 Number of Stress points at member End (Up to 12)
C2 DyNoFlex Results Case Number
C3 DyNoFlex Options Case Number
C4 Fatigue Category
 1 for DS Member Joints (Solid element Principle Stress) or
 2 for VM Member Joints (Solid element Signed VM Stress)
C5 1
C6 -1
C7 Cycle Blocks - See below
 1 – Cycle by Cycle
 >1 No of Stress Blocks
 If C7 is defined as a -ve value history will be reordered (maximums first & last).
C8 Life Factor Default=1, gives life in secs (3600 for Hrs or 3.1536E7 for Yrs)
C9 Additional SCF Default=1.0
C10 Fatigue Curve Default=C
C11 Wall thickness for SN Curve Correction - Optional - Default for pipe is property code wall.
C12 Output Case No for FATIG4 - Optional - Default is C2
C13 Group SET Optional
C14 Restrict to elements in group C14 in Group SET C13 Optional

Note the C1 in FATIG1S is only applied to pipe elements. Beam beam elements only have 4 stress points.

FATIG4 C1/C2/C3/C4/C5/

C1 First DM file number ("d")
C2 Final DM file number ("d")
C3 Fatigue life limit (years) for .JF output text files
C4 RC - Output Results Case Number
C5 Fatigue Category
 1 for Tubular Joints
 2 for DS Member Joints
 3 for VM Member Joints
The follow options are available but group restriction are more efficiently applied in FATIG1S. It can be useful to also duplicate the restrictions here and make use of the C9-Description for Output.
C6 Group SET to be read.l
C7 Restrict output to Group defined in SET C6
C8 Restriction Applies to 0 - Elements 1 - Nodes
C9 Description for Output
If C7 is preceded by a - the output will be sorted into groups up to the group attribute limit defined by C7

Typical command lines to a create global fatigue life assessment excluding group selection for a time history solution would be:

```
FATIG1S 8/121/121/0/1/-1/20/3.1536E7/1/F2/
FATIG4 121/121/0/121/2/
```

Life Factor

The Life Factor is used to evaluate fatigue lives for defined periods of operation. It is effectively a Return Period Factor. If the time history solution was dynamic the return period of the time history would be Δ t*NSteps seconds. If this factor was unity the resulting fatigue life would be in seconds . To convert this to hours a factor of 3600 would be used, to convert it to years a factor of 3.1536E7 would be used.

Wall Thickness Correction

Only pipe element wall thickness SN curve correction will be implemented unless C11 is specified. If C11 is non zero it will be applied to non-pipe elements. If C11 is defined as a -ve value it will overwrite the pipe wall thickness.

Individual Stress Point Assessments - FS-Crack

FS-Crack can be used to undertake more detailed specific assessments in an interactive mode. Stress-Life, Strain Life or Fracture Mechanics approaches can be employed.

Means stress effects and notch elevation can be be a applied to both stress and strain life methods.

The fracture mechanics approach undertakes both fatigue and fracture assessments for surface, embedded and through-thickness flaws. These assessments employ the methods of BS 7910:2005 "Guidance methods for assessing the acceptability of flaws in metallic structures".

To undertake this type of assessment the FS2000 module **FATIG1S** has to initially process the data for the specific element joint end. The command line argument for this module are different to those shown above and are described in Section 7.4 of the FS-Crack Help file.

-o-

Static Stabilisation (Artificial Damping)

This type of stabilisation may in some cases enable a solution in situations where stabilities cause a solution to fail during a static solution. It is an alternative to using a Dynamic solution.

It should be used with caution because it can produce erroneous results if applied too aggressively. If in doubt do not use.

This type of stabilisation adds artificial damping to a static solution. Effectively this adds viscous ground damping to all nodes of the model. The damping uses a pseudo velocity based on the displacement increment and the time increment at the current time step. It has no physical meaning and is purely artificial. It is applied equally to all degrees of freedom of a given node.

This artificial damping degrades the solution and therefore to avoid this dominating the solution, damping energies must be low relative to the energies within the model.

When artificial damping is active the % damping energy will be shown at the end of the solution. Typically, this should be no more that 1-2%.

Because of the somewhat arbitrary nature of the damping a trial and error approach is required to be employed so as to asses the effect of the global damping factor.

The damping applied to each nodal translation is:

D= C.Vol.dx/dt

C is a Global Viscous Damping factor.

Vol is an artificial nodal volume which is evaluated from the weighted mean of the nodal lumped self-weight.

dx is the time step displacement increment.

dt is the time step.

It can be seen from the above that though C is a constant the effective damping applied can vary considerably within a model.

Although this a static solution two dynamic solution parameters must be defined in the DyNoFlex solution options.

1. A gravitational mass case (self-weight) for Volume evaluation.
2. A Global Viscous Damping factor. This must be defined as a negative value.

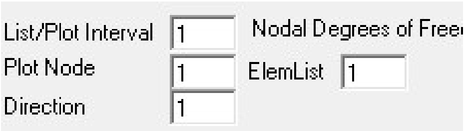
These are entered by making the dynamic parameter visible in the DyNoFlex options form. Once these two are entered the Dynamic Analysis option box should be un-checked so to indicate a static solution. Only if the Global Viscous Damping value is negative value will the artificial damping be applied during the static solution.

-o-

Stress & Strain - End of Solution List

When DyNoFlex solution are undertake there is an option to list stress and strain data for one specific element- **ElemList**. This can be useful for monitoring a solution.

The list only appears when running in interactive mode.



This only applicable to pipe and plastic shell element that use a material stress-strain during a plastic solution.

For solid elements plasticity e.g. Axy-symmetric solids, the **ETABLE** utility must be used.

Plastic Pipe Elements

```
TSTEP    200    200.000      0.4823   19
Writing Results
 Elem  Pt    TotAxStrn   PlAxStrn   AxStres    AccAxPlStrn  AccPlStrn
    1   1      0.6274     0.3555    562.8140      0.3555      0.4257
    1   2      0.5777     0.3067    561.0936      0.3067      0.3676
    1   3      0.3748     0.1081    552.2577      0.1081      0.1302
    1   4      0.0732     0.0000    151.4384      0.0000      0.0000
    1   5     -0.2465    -0.1509   -197.9141      0.1509      0.1829
```

Strain is % strain.

TotAxStrn is the axial component of total strain (elastic + plastic).

PlAxStrn is the axial component of plastic strain.

AXStres is the axial stress (/1E6 - assumes SI units - MPa).

AccAxPlStrn is the accumulative axial plastic strain. If there is no cycling this will equal **PlAxStrn**.

AccPlStrn is the effective accumulative plastic strain. This strain includes the hoop and radial strain due to the effects of pressure difference across the pipe wall.

The **ETABLE** output for pipe stress and strains only outputs **TotAxStrn** or **AccPlStrn.**

Plastic Shell Elements

The list below is for a Type 52 shell element. The Type 51 is similar but only outputs the transverse shear stress at the element centroid.

```
Writing Results
      Restaint Sum
     -0.348E-02    0.586E+06   -0.166E-23
 Elem           1
 LayerDDS          1           1 -2.9525856E+08   9247662.       1460951.
 LayerS            1           1  2.4824367E-09 -6.3743799E-10
 LayerDDS         12           1 -2.9525856E+08   9247662.       1460951.
 LayerS           12           1  2.4824367E-09 -6.3743799E-10
 LayerDDS          1           2 -3.0020970E+08  -433279.9       1158718.
 LayerS            1           2  2.4824367E-09  1.4607427E-09
 LayerDDS         12           2 -3.0020970E+08  -433279.9       1158718.
```

The stresses listed are the mid-layer stress at each of the integration points in the top and bottom layers. (/1E6 - assumes SI units - MPa).

LayerDDS This line shows the the normal Normal Stress X, Normal Stress Y and Shear Stress XY.

LayerS This line shows the Shear Stress YZ and Shear Stress XZ

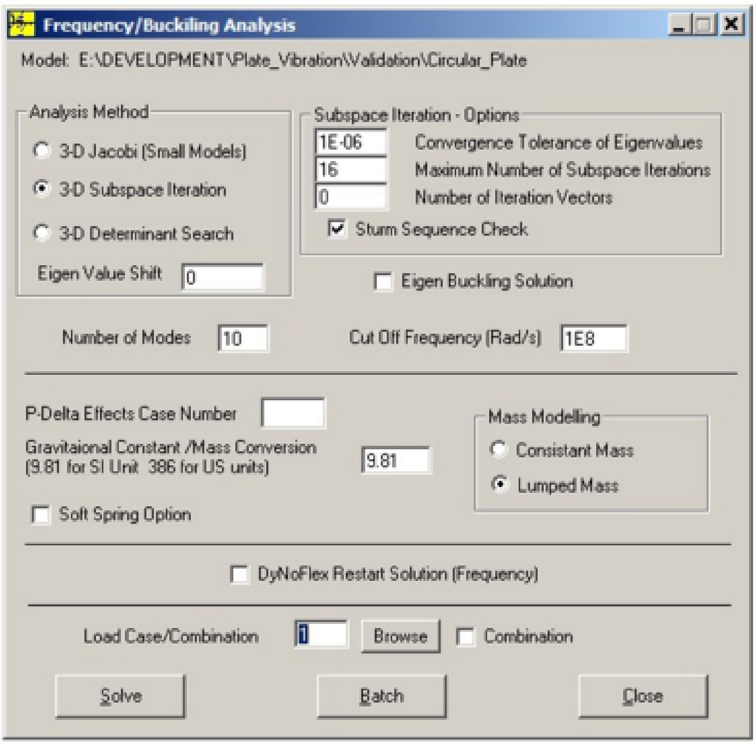
-o-

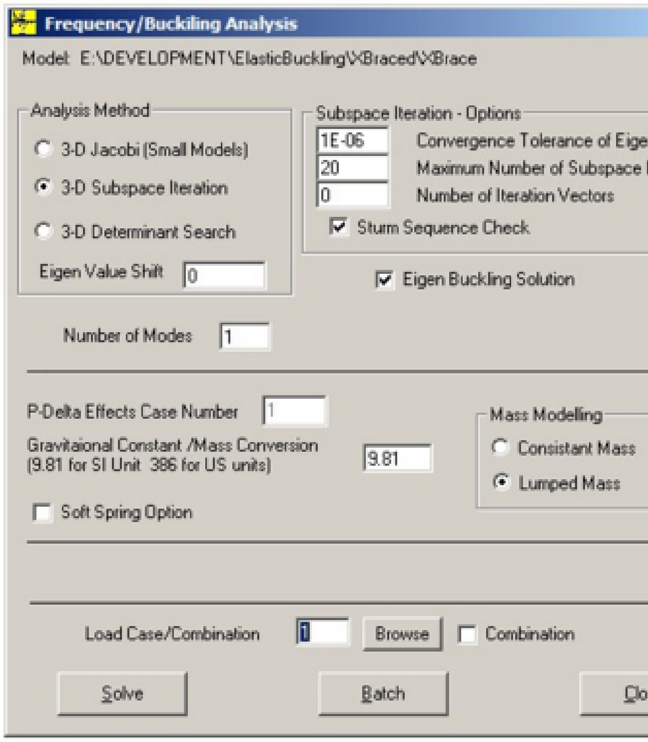
Frequency/Buckling Analysis

The Frequency/Buckling Analysis module is started by selecting the Eigen Frequency-Buckling command from the Solution menu in the Analysis TASK.

When the module has loaded the following input form will become visible. This is used to define either a Frequency Solution or a Buckling Solution.

Note that the saved setting are global model dependent and may require to be modified (see below under batch) if they are require to be load case dependent when running in Batch mode.





The **Analysis Method** options are used to select the Eigen Solution module. The Subspace Iteration method is the recommended method. The 3-D Jacobi method cannot be used for Buckling analysis.

The **Eigen Value Shift** shifts the Eigen solution. It can be used to reduced the likelihood of obtaining negative eigenvalues in a buckling solution (+ve value generally) or enabling the frequency solution of an non restrained model (-ve value generally). The Determinant Search method is often more effective in when a shift is required to be applied in buckling analysis (use minimal No of Modes). In a frequency solution the offset is defined in $(Rad/s)^2$ i.e. frequency squared. In both cases try small values and work up. Similarly try both +ve and -ve values.

The **Eigen Buckling Solution** option is used to activate an eigenvalue buckling analysis. For Buckling analysis either the 3-D Subspace Iteration or Determinant Search method must be used.

The **Subspace Iteration-Options** are used to control the solution in the Subspace Iteration solution option. It is unlikely that the default options will require to be changed.

> **Number of Subspace Iterations** - this can be increased if the solution does not converge.
>
> **Number of Iteration Vectors** when this is zero the default value equal to the lower of 2*NFreq and NFreq+8 will be used. The number used is displayed in output. Increase this value if the solution fails to find all the modes or if it thought that modes are missed (cannot be higher than the mass degrees of freedom) .

The **Sturm Sequence** check will print a warning message if any Eigenvalues are missed within the frequency range (not always). Using more **Iteration Vectors** may find the missed values.

The **Number of Modes** box is used to select the number of modes to be included in the output. When using the Subspace Iteration method do not specify values that are greater than the maximum degrees of freedom. Keep this low especially when undertaking buckling solutions. For the Jacobi method it is possible to specify more than the number of degrees of freedom.

The **Cut-Off Frequency (Rad/s)** is used to limit the output of modes of vibration to those modes whose frequency is below this limit. Does not apply to the Jacobi method.

The **Mass Modeling** options are used to select the type of mass matrix to be used. If modal response analysis is to be undertaken only the Lumped Mass option should be used. See Mass Definition.

The **PDelta Effect Case Number** is the static analysis P-Delta case used to evaluate the force distribution in the structure. This option would be used on structures whose stiffness is dependent upon tension stiffening or compression softening effects. It is first necessary to run the 3-D Standard Analysis with the P-Delta effects active which creates a load definition file. The load distribution from that case will be used to evaluate the P-Delta effects in the frequency or buckling solution. When undertaking buckling analysis the **PDelta** case and the **Load Case** must always be the same case number. The **PDelta** case should be left blank for frequency analysis in which P-Delta effects are not required (normally the case).

The **Gravitational Constant (Mass Conversion)** is used to convert y direction forces in a load case to equivalent masses. See Mass Definition.

The **Soft Spring** option applies a soft spring to all element degrees of freedom (1E-6).

The **Load Case/Combination** box is used to select the load case or combination to be used for the solution i.e. the Mass Case. Load Case Combinations may be used to pre-combine load cases to form the mass case.

The **DyNoFlex Restart Solution (Frequency)** option enables a frequency solution to be undertaken using the stiffness and mass matrices generated when a DyNoFlex solution is is restarted with the DyNoFlex Create Frequency Solution Data option active. The resulting case will be that of the DyNoFlex Mass Case, the case number in this form will not be used but the Load Case must exist (solely for consistency use the mass case no from the DyNoFlex). It make sense for ID purposes to use the DyNoFlex mass case number. Note that the options just above this option switch are not used when this option is active.

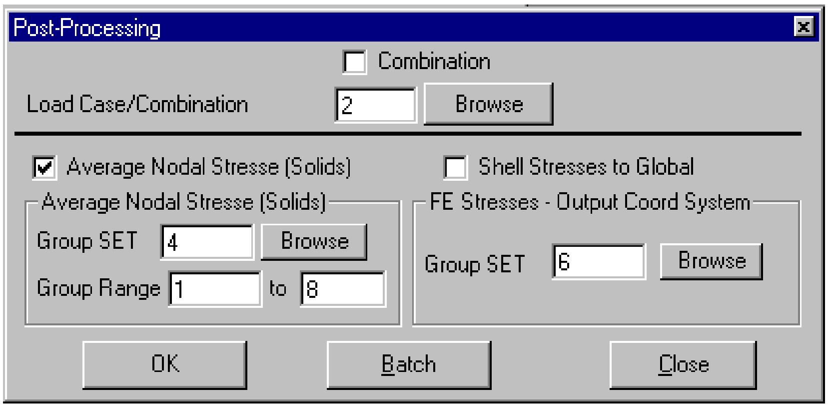
The **Solve** button is used to start the analysis. Note that this solver uses a banded solver so uses the Bandwidth command in the Reseq Menu (Analysis TASK) to renumber (internal) the model for efficient solution.

The **Batch** button will append the current settings to the Frequency Module command line to the <ModelName>.BRM file. The analysis Option file has the model filename with the extension <Modelname>.UFO. This file would need to be copied and renamed in the the batch file if different solution options for different cases were required during a batch run. The current active options are always displayed when the for is opened and only changed when a solution is run or the Batch button is clicked.

-o-

Analysis:Menu:PostPro

This is a command that makes the Post-Processing dialogue box visible.



The analysis solution modules produce raw data results files for each of the load cases analysed. The post -processor is used to convert and optionally combine these Raw Results files into Processed Results Cases. The options below the line are options associated with solid element processing.

A range of cases may be processed by defining a range i.e. 1-9 will process all raw results cases between 1 and 9. C1-9 will process all load case combinations between 1 and 9. If a case does not exist a warning will be given and the process will continue. Combinations may also be combined, CC200 would combine the results cases defined in the combinations listed in C200 using the factors defined in both the participating combinations and the subject combination.

Only Processed Results Cases can be read by the various graphical and non-graphical output modules. Therefore, it is mandatory to post-process prior to any method of result output, even if only one load case is being considered.

Care should be taken to so that Processed Results are not overwritten, e.g. Post-processing Load Case 3 (single case Raw Results) will produce Processed Results Case 3. Post-processing Load Combination C3 (multiple Raw Results) will also produce Processed Results Case 3

Finite Element Stresses - Output Co-ordinate System

By default element stresses are produced in the output co-ordinate system of the element For most elements this is generally the global co-ordinate system but it does vary according to the type of element. Note: Shell elements always use their own coordinate system.

For any element it is possible to output the stresses in a previously defined co-ordinate system. This is done by using the following two options.

If the **Shell Stresses to Global** box is checked all shell stresses in the model will be converted to the global co-ordinate system.

The **Group Set** box is used to select an element Group SET which identifies the element's output stress co-ordinate system. To convert stresses to a co-ordinate systems it is necessary to identify that co-ordinate system using element groups. This means that a group SET requires to be created in which the element groups number corresponds to the co-ordinate system to be used for that element. If the box is left blank or the Group SET does not exist all elements will be output in the element co-ordinate system (or Global Shell if active).

GSh and **CS**(*Set Number*) will be appear on stress plots when the above two options are applied during processing

Averaging Finite Element Stresses

Unlike the beam and pipe elements of FS2000 solid elements do not produce exact results. A coarse mesh of element will produce element stresses that show stress discontinuities at the nodes between adjacent elements. A more detailed mesh will reduce these discontinuities. Note: The magnitude of these discontinuities can be usefully used as a measure of mesh suitability.

To produce more realistic results a common practice is to average the nodal stresses of elements that share common nodes.

When dealing with 2-D and 3-D solids this averaging process can be generally applied globally across the model without any difficulty. In models with geometric and material discontinuities this process when applied globally may not be valid. This is especially true of when shell elements are used. Plate and shell stresses are evaluated in the element's own co-ordinate systems which may be different for each element. This and the fact that the wall thickness of adjacent elements may be different may invalidate any averaging process. In such cases the user has to select which elements are to be included in any averaging process and ensure that output co-ordinate systems are compatible.

Groups are used to identify how elements are to be averaged. The element's group attribute is used to identify which elements can be average with each other.

If the **Average Nodal Stresses (Solid)** is checked the following input options will become visible.

The **Group Set** box is used to select the element Group SET which has been created for averaging control. If the box is left blank all elements will be averaged. If the Group SET does not exist all elements will be averaged.

The **Group Range** is used to identify which elements are to be averaged. Elements not in the Group range will not be averaged. Element in the Group range will be averaged but only with element in the same Group.

Ave and **AS** *Set S1* **-** *Set S2* will be appear on stress plots when the above two options are applied during processing.

-o-

8.16 Output/Results

The Ouput/Results TASK is used to used to interactively inspect results cases in graphical environment. Multiple Viewports may be used to view different parts of the structure and view different results cases.

When the file menu is used to load a results case it will be loaded into the active Viewport. If more than one Viewport is used, more than one results case may be loaded. The title bar of the Viewport will show the results case description currently loaded.

The following menus are available in this task

Plots	Plot results to the screen i.e. bending moment plots
FE-Plots	Produces contour plots for solid elements
Insp	Interactive results interrogation i.e. list forces etc. to the screen
StdOut	Creates formatted reports files for definition data and results data
Design	Activate the optional design modules

-o-

Output/Results:Menu;:Plots

This menu provides the commands to enable analysis results to be plotted on the screen in the from of deflected plots, bending moment diagrams, unity ratios plots etc. Scale factors for the plots can be changed using the Scale Factor command. Scale factors will be set by the program but often they may require to be changed.

Persistent Plot Enables force diagrams to be draw selectively or on all elements.

Basic Geometry Draws the basic geometry cancelling current plot.

Deflections Plots nodal displacements for the whole of the current display.

Vibration Modes Plots fundamental mode shapes from Frequency analysis (Set scale Factors).

Buckling Modes Plots fundamental mode shapes from Frequency analysis (Set scale Factors).

Animate Animates displacement plots and vibration modes.

Time History Displacement animation for a DyNoFlex time history solution.

Animate Results (Scan) Animates any cuurnt plot over a defined number of result cases, starting at the currently loaded one.

Disp Vectors Plotting displacement vectors from a DyNoFlex time history solution.

Stress Contour Plot When active the stresses levels are represented by colour contours.

Axial Force (& Stress) Plots axial force or stress for selected elements.

Shear Force (& Stress) Plots shear force or stresses for selected elements.

Moment (& Stress) Plots bending moments or stresses for selected elements.

Torsion (& Stress) Plots torsional force or stresses for selected elements.

Hoop Stress Plots hoop stress on selected pipe elements.

Von-Mises Stress Plot the maximum VM stress for selected elements.

Couple Forces Vector plots showing couple forces and moments.

Reaction Forces Vector plots showing reaction forces and moments.

UR Unity Ratios Makes visible the Unity Ratio Plots dialogue box.

ETable Values Plots x displacements(useful for pipeline expansion results)
Plots Max stresses or strains for plastic pipes (ETable output)

Plots ETable X data

Settings Makes visible the Line Plot Settings or Contour Settings dialogue box visible

-o-

Persistent Plot

If the Persistent Plot toggle is active then the plots will be drawn for whole of the current display of the model each time the mode is re-drawn. If an element group is active only force actions on those elements will be plotted. Therefore use groups to plot selectively plot action plots

If the Persistent Plot toggle is not active the user is required to select, using the current select method, the nodes or elements for which the plots are required. On large models this enables loading in areas of interest to be clearly shown.

The exception is for nodal displacements for which case whole display is plotted and when the Persistent Plot is active.

Only persistent plots will be sent to the printer when the **Print Graphics** button is pressed.

-o-

Animate

Before using the animate command ensure that only one view on the model is open (close additional view using the Single View command from the Windows menu). When in animated mode the view attributes cannot be changed therefore first set the view options etc. for the displacement or vibration plot so that an acceptable static plot is obtained.

To animate select the animate command. To cease animation re-select the animate command.

When plotting displacement plots it is possible to plot between two different results cases as opposed to using the non-deformed geometry. When this is done the reference load case will replace the results case originally loaded.

-o-

UR Unity Ratio Plots

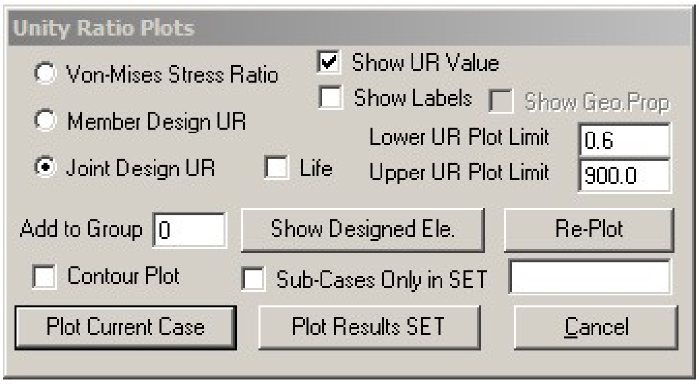
The Unity Ratio Plot form is used to control the plotting of unity ratios (URs) plots. When URs are plotted the element is shown in red. The primary function of the plot routine is to selectively highlight the highest and lowest loaded members by using upper and lower UR plot limits.

Unity Ratios can only be plotted only after the appropriate stress Results Reports have been created. These Results Reports are created in the Output/Results TASK or during Batch runs of the Results/Design modules.

The Von-Mises UR data is created using the Individual Results Format command of the StdOut menu in the Output/Results TASK. The stress output must be active to create the data.

The Member Design and the Joint Design UR data is created by the optional Code Check Modules in the Design menu of the Output/Results TASK.

UR plots are different to standard results plots in that they are not attached to specific Viewports. Only one UR data set can be viewed at a time. This data can be viewed on any active Viewport even if the Results Case in that Viewport is different from the UR case. The UR case is always indicated on the screen.



The **Plot Current Case** button is used plot the URs of the active Results Case. If Sub-Cases exist a list will appear. The appropriate sub-case is that selected by mouse click. Note that the this buttons only becomes active following the opening of a Results Case

The **Plot Results SET** button is used to plot the URs from a number of Results Cases. This option reads the results in the Results SET and plots for each element only the highest UR in the SET. Along side the UR value is the results case that produced this maximum. If Sub-Cases exist they will be included in the plot and be identified by their case number.

The **UR Plot Limits** are used set a upper and lower visibility limit. All elements withURs within these limits will be plotted red.

The **Add to Group** box is used to group all the elements within the UR Limits to the specified Group. This useful when wishing to restrict output to highly loaded members in other output modules e.g. text output.

The **Re-Plot** button is used to re-plot the results following changes in the UR plot parameters e.g. limits.

The **Show Designed Ele** button is used to show those elements that have been designed checked. Elements that have been design checked are highlighted red.

The **Sub-Cases Only in SET** option if active will only plot UR values from the specified Sub-Case.

The **Contour Plot** is used to switch between red line plots or UR contour plots for beam elements. Use the Contour Setting to select the colour scaling.

-o-

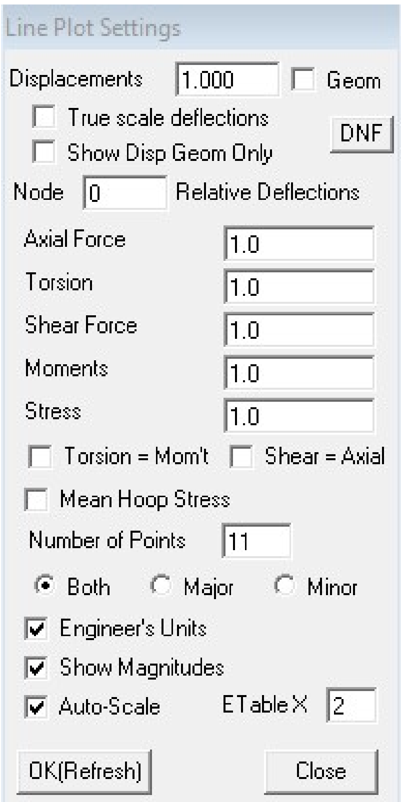
Line Plot Settings

Each time a results case is loaded the default scale factors will be re-evaluated based on that case. This form enables the various factors to be changed.

This form is used to define the parameters which control the contour plots. It is accesses from:

The Results TASK :Plots/Settings/Line Diagram Setting menu command, or the **Res Toolbar** button

(Upper LHS) 



The magnitude of the scaling factors are normalised values which should give a reasonable plot based on the highest loaded elements of the model. Note that these scaling factors are global and not Viewport dependent.

The **True scale deflection** box if checked will draw the deflections using a true scale factor. When plotting True scale deflection the scaling factor becomes an absolute value. When the option is activated the scaling factor will change from the normalised value to the absolute value.

The **Geom** check box is used to plot the force diagrams on the deflected shape. It is generally used when viewing force action plots obtained from DyNoFlex large displacement solutions. Large displacement solutions most always use a unity true scale factor. When this is active the following behavior will be should be expected:

- The node and element labels will be draw on the displaced shape
- The nodes and elements can only be listed ie pick from the plotted position
- When selecting a node for the view rotation centre it will be based on the scaled deflected shape
- Clip plane selection will be based on the non-scaled true defected shape

The **Show Disp Geom Only** is used to display only the deformed shape. If this option is not selected both the deformed and undeformed geometry will be displayed.

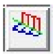
The **Relative Deflection's Node** box is used to plot relative (translational) deflections. The plot will be relative to any non-zero node label.

The **DNF** button is a utility to create: Nodal definition data in which all node co-ordinates of the mode are based on the their non-deformed (geometry + the translational displacement) from the current results case and to create an initial displacements defintion for DyNoFlex based on the current displacements. The resulting node definition file has name **<Modelname>.UDGEOM** and the file containing the INIDISP commands is **<Modelname>.UINIDISP** . The displacements can also be extracted from the Eigenvalue solution mode shape displacements. For Eigen mode shapes the **Displacements** scaling factor can be used to scale the magnitudes (make **True scale deflections** active so as to see the effective factor). The DNF utility can also be run in Batch mode (only for UDGEOM).

The **Torsion = Mom't** and the **Shear = Axial** options are use align the scaling factors so that comparative force plots may be produced.

The **Mean Hoop Stress** if active will use Sh = Δp.(Do-t)/2t for stress evaluation, otherwise Sh = Δp.Do/2t. This is also used for the evaluation of the VM stresses. Listed output data always uses Sh = Δp.Do/2t.

The **Number of Points** is used to define how many points are to be plotted along each element span (2 is the minimum will plot on the nodal values). If active, this will also effects the appearance beam contours plots.

The **Both, Major** and **Minor** option buttons are used to define the axis for which the forces are to be plotted when plotting bending and shear actions. Can also use . If both are set the major axis will overwrite the minor axis for beam contour plots.

If the **Auto-Scale** box is unchecked the scaling factors will not be changed when a new load case is opened. This feature is useful when using different windows for different result cases.

The **Show Magnitudes** will label the force diagram with the magnitudes at the element nodes. Use active groups to make this plot more selective (only active groups are plotted)

The **ETableX** is used to identify an ETableX plot file. The X is the ETable I.D.

-o-

FE-Plots

This menu provides the commands that enable contour plots for solid elements to be drawn.

Shell forces contour plots are also possible if this is activated in the Settings options. When this is active the force menu commands are as below.

The coordinate convention used for shell elements stress output is given in Section 4.3.

Persistent Plot Used for selective contour plots.

Basic Geometry Draws the basic geometry cancelling current plot.

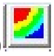
Sx	Direct stress in the x direction	Mx (Sy)
Sy	Direct stress in the y direction	My (Sx)
Sz	Direct stress in the z direction	Mxy
Sxy	Shear stress in the x-y plane	Nx
Syz	Shear stress in the y-z plane	Ny
Sxz	Shear stress in the x-z plane	Nxy
S1	Principle Stress (highest)	Qyz
S2	Principle Stress (mid)	Qxz
S3	Principle Stress (lowest)	Mx* Wood-Armer moments
Stress Intensity	Stress Intensity (TRESCA)	My* Wood-Armer moments
Von-Mises	Von-Mises stress	
Settings	Contour Plot Parameters	

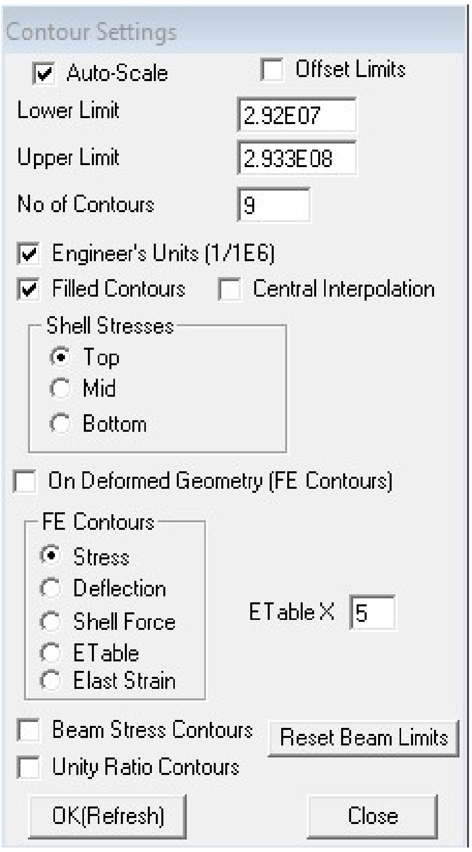
Stress Intensity (TRESCA) = 2 x Maximum Shear Stress

-o-

Contour Settings

This form is used to define the parameters which control the contour plots. It is accesses from:

The Results TASK :Plots/Settings/Contour Setting menu command, or the Results TASK :FEPlots/Contour Setting menu command, or the **Res Toolbar** button (Upper RHS)



The **Auto-Scale** box if checked will apply automatic upper and lower limits based on maximum and minimum values to the any of the contour plots selected from the contouring menu. If the user requires to manually set the limits or if existing limits require to be held constant then this box should be unchecked.

The **Lower Limit** & **Upper Limit** boxes can be used to set the contour limits. Contours are set at equal intervals between these limits.

The **Offset Limits** option if active will set the upper and lower limits to the lower respective contour level. This is useful for showing all stresses above and below specified limits.

The **Engineer's Units** will show the contour legend in non exponent format and divided by a factor of 1E6. This is to show stresses in N/mm2 when the units are in N and m. If this is unchecked the legend will be in exponent format. This would be disbalbed for strain plots.

If the **Filled Contours** box if checked the contours will be filled giving a solid appearance to the elements. If not line contours will be drawn.

The **Central Interpolation** option if active will include an averaged central stress for contour plots. This sometimes eliminates erroneous 'black holes' that sometimes appear in course meshes

The **Shell Stresses** options are used to select the surface for shell output. If the local orientation button is pressed the element orientation Triad will show the surface orientation as the local Z axis points away from the top surface.

The **On Deformed Geometry** if checked will draw the contour on the deflected shape. This is only for solid elements, not for beam elements.

FE Contours

The **Stress** option is used to enable the component stress contours from the various FE-Plots commands to displayed.

The **Deflection Contours** option is used to show the deflected shape. The Sx command refers to the global x direction, Sy to the y direction etc. Solid elements only.

The **Shell Force** option is used to plot shell forces. These forces are based on the element stresses and have the units if stress/unit length. The conventions used are given in Section 4. These loads can also listed in the Sub Case files <model>.ShellF.O'n" for single cases or <model>.ShellF.M'n" for multi cases. Note that any transform coordinate system used in post-processing must have the z axis in the parallel with the transverse plate direction if shell forces are evaluated.

The **Elast Strain** will plot the component elastic strains. Re-load the current case to reset the Auto-Scale.

The **ETable** option is used to plot contours from data generated using the ETABLE utility. Currently this only plots accumulated plastic strains (ETABLE5) for 2D & 3D solids. The OK(Refresh) is used to read the ETABLE data related to the currently opened result case. Any of the FE-Plots component commands can then be used to plot the ETABLE value.Re-load the current case to reset the Auto-Scale.

The **Beam Stress Contours** is used to switch between stress diagram plots or stress contour plots for beam elements.

The **Unity Ratio Contours** is used to switch between red line plots or UR contour plots for beam elements.

The **Reset Beam Limits** is used to set the contour limits between the maximum and minimum values (stresses and URs) for the beams in view (Use OK(Refresh) button to update). When plotting contours the maximum will be shown in the contour legend. Use the Line Plot Settings to select major or minor axis and number of points on span. Note that when plotting stresses, the number of points along the span will effect the display.

The **OK (Refresh)** button will record the settings and re-draw.

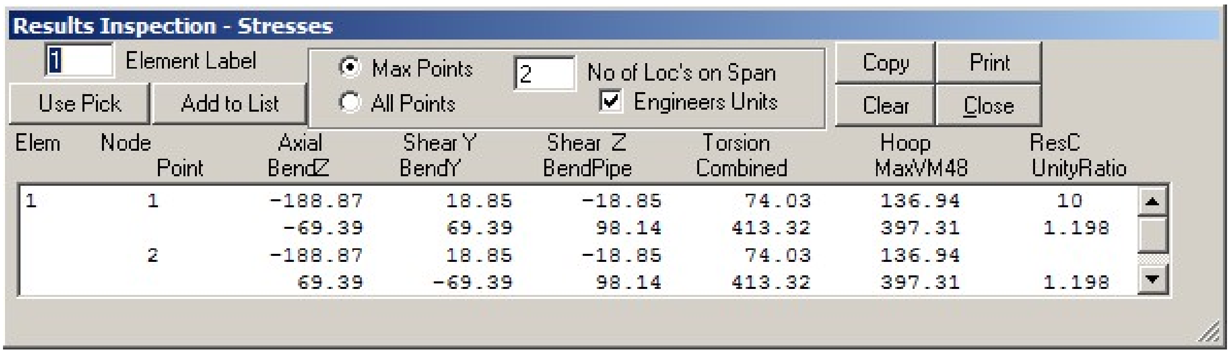
-o-

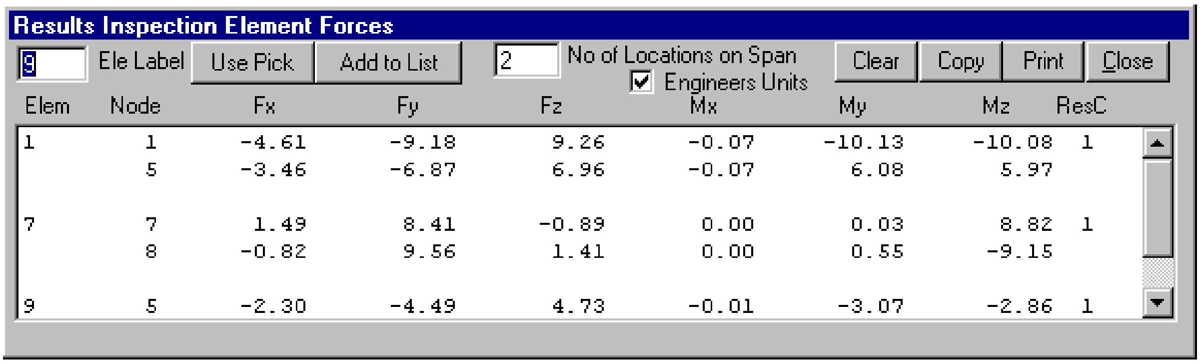
Output/Results:Menu:Insp

This menu provides the commands to enable analysis results that are currently plotted to be interrogated. Nodes and elements are selected by picking or by label ID. Their deflections, forces or stresses may then be listed on the screen in re-sizeable list boxes. The data in the list boxes may be printed or copied to the Windows Clipboard.

Deflections	Makes visible the Results Inspection - Nodal Displacements list box
Forces	Makes visible the Results Inspection - Element Forces list box
Beam Stresses	Makes visible the Results Inspection - Element Stresses list box
FE-Stresses	Makes visible the Results Inspection - FE Stresses list box
Couples	Makes visible the Results Inspection - Element Forces box
Reactions	Makes visible the Results Inspection - Reaction Forces box
Presc Disp	Makes visible the Results Inspection - Nodal Displacements list box

The following shows the Beam Element Stresses & Forces list boxes.





If the list is double clicked the contents will be **cleared**. The size of the form may be changed to extend the range of the visible list.

The **Use Pick** button will add the forces of the element last picked (Element Query button) to the list (see below for FE stress listing). For node related results e.g. deflections the Node Query button is used.

The **Add to List** button is used to add the forces of the element whose label displayed in the **Elem Label** box.

The **ResC** column at the end shows the results case to which the forces belong. The result case used depend upon the result case loaded in the active Viewport.

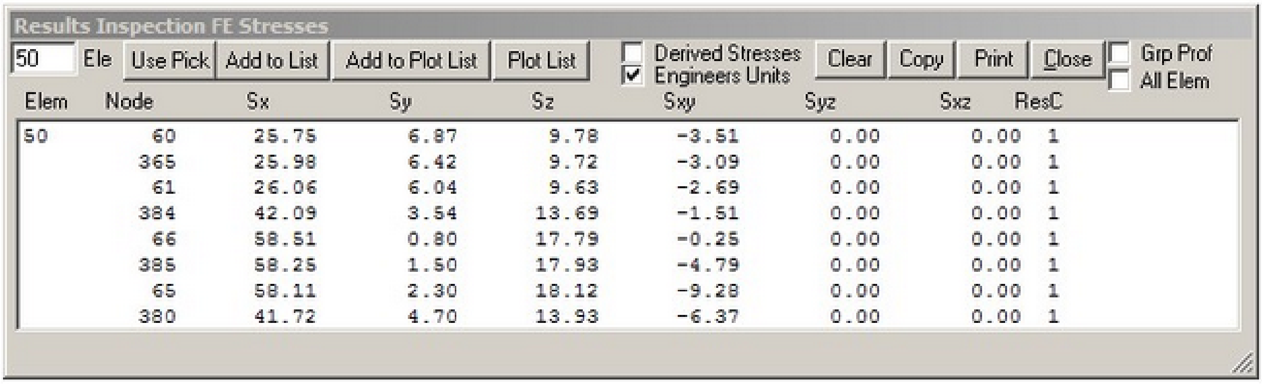
The **Print** button is used to send the current list to a printer. The orientation will be that of the graphics printer settings.

The **Copy** button is used to copy current list to the Windows Clipboard. Use this to paste the text to other windows applications.

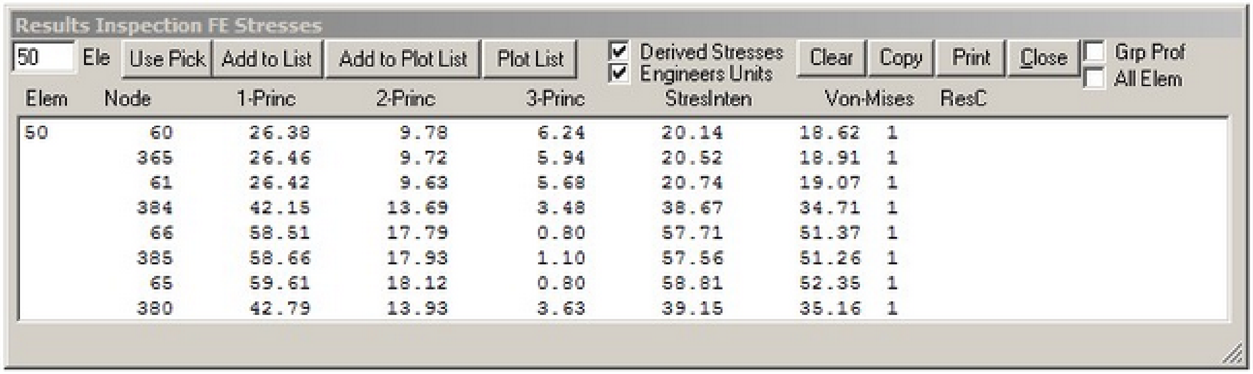
If the **Engineers Units** check box is activated the units for the result data will be formatted in 'SI Engineers Units'. These are mm for deflections, kN for forces and N/mm2 for stresses. If not checked the output will be in exponent format.

Listing FE Stresses

The stress data listed depends upon the data currently plotted when the command is selected. If derived stresses are plotted eg VM stresses the list box will show all derived stresses. Derived stresses will continue to be listed until the command is re-selected during a direct stress plot.



Elem	Node	Sx	Sy	Sz	Sxy	Syz	Sxz	ResC
50	60	25.75	6.87	9.78	-3.51	0.00	0.00	1
	365	25.98	6.42	9.72	-3.09	0.00	0.00	1
	61	26.06	6.04	9.63	-2.69	0.00	0.00	1
	384	42.09	3.54	13.69	-1.51	0.00	0.00	1
	66	58.51	0.80	17.79	-0.25	0.00	0.00	1
	385	58.25	1.50	17.93	-4.79	0.00	0.00	1
	65	58.11	2.30	18.12	-9.28	0.00	0.00	1
	380	41.72	4.70	13.93	-6.37	0.00	0.00	1



Elem	Node	1-Princ	2-Princ	3-Princ	StresInten	Von-Mises	ResC
50	60	26.38	9.78	6.24	20.14	18.62	1
	365	26.46	9.72	5.94	20.52	18.91	1
	61	26.42	9.63	5.68	20.74	19.07	1
	384	42.15	13.69	3.48	38.67	34.71	1
	66	58.51	17.79	0.80	57.71	51.37	1
	385	58.66	17.93	1.10	57.56	51.26	1
	65	59.61	18.12	0.80	58.81	52.35	1
	380	42.79	13.93	3.63	39.15	35.16	1

When listing FE stresses the Element Query and Node Query buttons can also be used to list stresses at only at selected nodes. Clicking the Element Query button using the LH mouse button initiates this mode. When this is done elements and nodes are selected in succession - clicking the **Use Pick** button will add to the list only the stresses at the selected node.

When in the node selection mode is active the **Add to Plot List** button will additionally add the nodal co-ordinates to the list. The purpose of this is to produce date tables that can be used to create stress x-y plots.

The **Plot List** button will automate the ***Stress Linearisation Plots*** described below i.e. run FS-Graph with the plot data loaded.

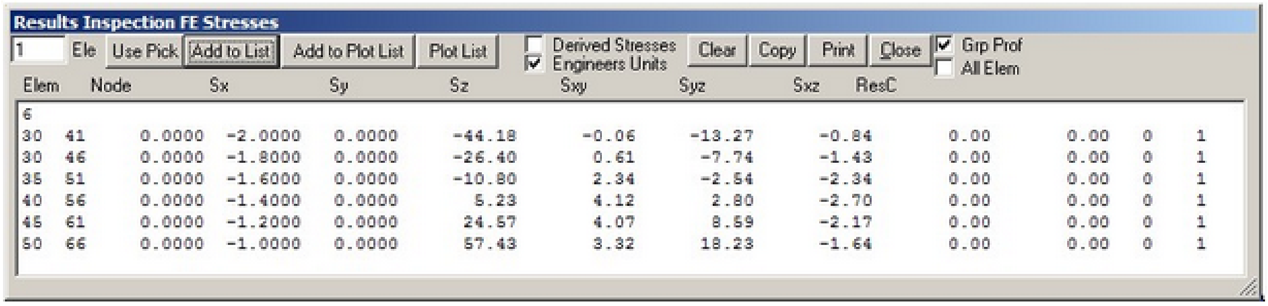
Stress Linearisation Plots

The **GrpProf** and **ALL Elem** are used to create plot profiles from the nodes and elements in the active group. This feature enables stress plot data to be easily created. This data can then be read by FS-Graph which can perform stress linearisations.

If elements and nodes are in the active group and the **GrpProf** is active, clicking the **ADD to Plot List** button will create a plot list of the primary stresses.

GrpProf will add a node only once to the list. The **AllElem** if also active will add the node to the list for each element in the group, this is useful for investigating non-averaged stresses.

The list is sorted based on the dimension (x, y or z) with the largest range. The node coordinate system used will that to which the nodes were assigned when the model was saved.



The steps for obtaining stress linearisation plots are:

1. Put the elements and nodes in the active group
2. Activate the GrpProf option
3. Click the **ADD to Plot List** button to populate the list (Clear first)

The **Plot List** button will undertake step 4-8 bwlow, it will use the <modelname>.~LPROF filename for the stress plot file.

4. Copy the list
5. Start FS-Edit and paste the list into a new page.
6. If the **AllElem** was also active make the first line equal to the last line (no of plot entries)
7. Save the plot data to named file
8. Run FS-Graph and read in the saved plot data file.

Section 8.8.7 described the graph plotting utility.

-o-

Output/Results:Menu:StdOut

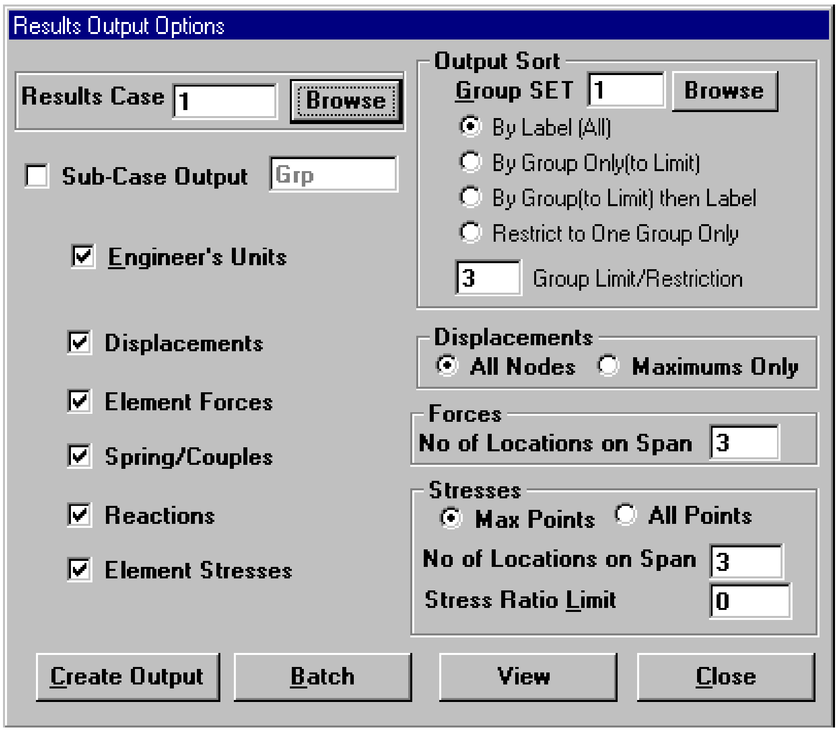
This menu provides the commands to create formatted output data files. These files may be individual Definition Data, Results Cases, multiple Results Cases or sorted UR (unity ratio) files.

Individual Format	This makes visible the Results Output dialogue box for individual Results Case output
Multiple Format	This makes visible the Results Output dialogue box for multiple Results Case output
Sorted UR	This makes visible the Unity Ratio Sort Utility dialogue box for unity file creation

-o-

Individual Results Format

The Individual Format command of the Report Task ResOut menu makes the following form visible. This form is used to create a formatted Results Files.



The **Results File** box is used to select the Results Case to be processed. The Browse button may be used to select a case from a list of existing cases. A range of cases may be processed by defining a range i.e. 1-9 will process all cases between 1 and 9. If a case does not exist a warning will be given and the process will continue.

The **Sub-Case Output** option enables multiple formatted output files for the same Results Case to be created. This may be used if it is desirable to create separate Results files for the different output category options and various sort options available. If the option is checked then the file created will have the file name <Model>.Grp.O<n> where Grp is the name entered in the Sub-Case description box.

If the **Engineers Units** check box is activated the units for the Results files will be formatted in 'SI Engineers Units'. These are mm for deflections, kN for forces and N/mm2 for stresses. If not checked the output will be in pure S.I. exponent format.

The check boxes on the left hand side of the form are used to select which type of output data is to be included in the file.

No of Locations on Span The element forces and element stresses may be evaluated at up to 21 locations along the beam span. The default value is 3 i.e. the end points and mid span. The **No of Locations on Span** defines this value. If a value of 1 is used the output will be restricted to a single end (node) of the element. For an end (node) to be selected the node must be in the same group as the element (nodes or elements not in matched groups will be not be output when 1 is used). When applied to Solid elements stresses only a specified value of 1 is significant otherwise stress at all nodes will output.

Stresses Output

The beam stress evaluation is based on the equations described in the Geometric Property Table Section. When selecting element stresses the user can also select the options that may reduce the stress output to meaningful areas of interest.

Shell stresses are output at top, middle and bottom surfaces. Shell forces listing will be written to a to a Sub case file <model>.ShellF.O'n" if the force option is active and with the stress limit threshold (for multi cases <model>.ShellF.M'n"). Note that any transform coordinate system used in post-processing must have the z axis in the parallel with the transverse plate direction if shell forces are evaluated.

The **Max Points** option will only print the stresses at the point which has the highest Von Mises equivalent stress at each location. The **All Points** option prints all the stresses at all the points across the section. Stress points are defined by the Lx and Lz values (see Section 6.6.1).

The **Stress Ratio Limit** box is used to restrict the stress output to Von Mises stress ratios above the defined value.

The **Batch** button converts the set options to command line switches and appends the module command line to the .BRM batch run file.

The **View** button loads the results case file view form.

Groups

The **Group SET** box is used to define the group SET to be loaded. If a SET is loaded then all node and element labels will be accompanied by their respective group attribute. If this field is left blank or contains the number of a non existent group then only the basic node and element numbers will be used for reference in the lists.

The **By Label (All)** option will output all entities (nodes and elements) in ascending label order.

The **By Group Only (to Limit)** option will output entities in ascending Group order. Entities not assigned to groups or entities assigned to Groups greater than defined by the **Group Limit/Restriction** box will not be output. This is a restricted process option.

The **By Group(to Limit) then Label option** will output entities in ascending Group order. Entities not assigned to groups or entities assigned to Groups greater than defined by the **Group Limit/Restriction** will be output in label order following the sorted groups. All data is processed with this option.

The **Restrict to One Group** option is used to restrict entities to only those entities with the same group number as defined by the **Group Limit/Restriction** box (zero value indicates that all data will be shown). This is a restricted process option.

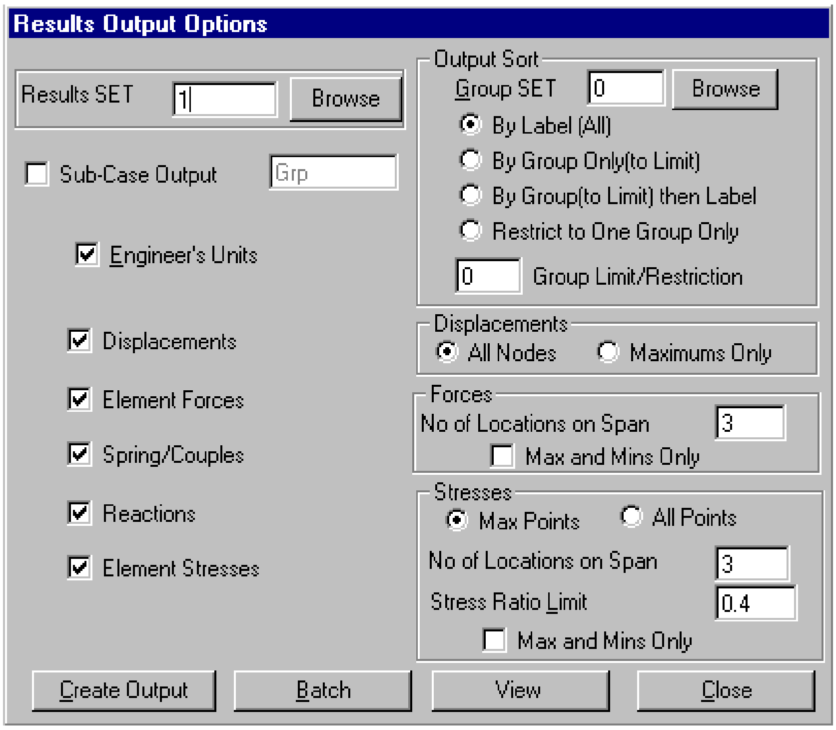
Important Note Stress ratio data created by this module will be limited to the data processed. If restricted process options is used then any Stress Ratio plots or Stress Ratio sorts which use the same results cases will be limited to the processed data. The plot or listed output will indicate if the output is from a restricted process e.g. Von-Mises Restricted.

-o-

Multiple Results Format

The Multiple Format command of the Report Task ResOut menu makes the Results Output form visible. It is used to create a formatted result file for more than one result case. The result cases are selected by defining them in a Result SET.

The maximum number of cases that can be processed in single SET by this module is 502. If this limit is exceeded in a SET, the process will be terminated and a warning will be given.



Its operation is identical to that for Individual Results Format except that instead of loading an individual results case a Results SET is used to load multiple cases.

When a Results SET is used the output created will show for each node or elements the results from all cases within that SET.

It should be noted that this Multiple Results module does not produce Von-Mises unity ratio plot data. The Individual Results module must be used for this purpose.

Selective Output

The following options may be used to selectively reduce the volume of output produced. The second method is very useful for selecting joint loads.

1. If active the **Max and Mins Only** option will only output the maximum and minimum values for each force or stress component. The result cases to which these values correspond are also given.

1. If the **No of Locations on Span** is set to 1, only results at only selected ends (nodes) of a beam element will be given. For a beam end to be selected the node and element must be in the same Group within a Group SET.

-o-

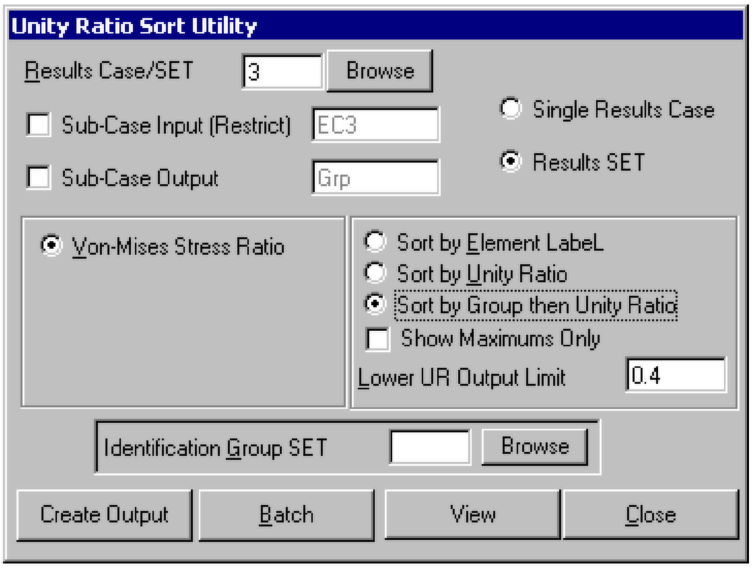
Sorted Unity Ratios

The Sorted UR command of the Report Task ResOut menu makes the following form visible. It is used to create a result file that shows the Unity Ratios (URs) for individual or multiple result cases.

The Unity Ration Sort Utility does not generate the UR data it merely sorts and formats it. The source data requires to be created using the appropriate output module.

The Von-Mises Stress ratio data is created using the Individual Format command of the ResOut menu in the Report TASK. The stress output must be active to create the data.

The Member Design and the Joint Design unity ratio files are created by the various Code Checkers.



The **Sub-Case Input (Restrict)** option . If active this will restrict the UR that are read to only those from the specified sub-case results. When this is active the Sub-Case output would normally be set active and to set to create the same sub-case output.

The **Sub-Case Output** option enables multiple formatted output files for the same Results Case to be created. This may be used if it is desirable to create separate Results files for the different sort options available and Group SETs. If the option is checked then the file created will have the file name <Model>.Grp.MTM where Grp is the name entered in the Sub-Case description box.

The **Single Results Case** option is used to process a single Results Case. The **Results SET** option is used to process a Results SET containing multiple cases. If Sub-Cases exist they will be included in the output for their respective case unless the Sub-Case Input option is active.

The Von-Mises Stress Ratio options etc. are used to select the type of results data to be processed. Note that each type creates its own file type.

The **Results Case/SET** box is used to select the appropriate case. The browse button may be used to select the case from a list of existing cases that match the selection options.

The **Sort by Element** option is used to sort the data by element label. In the case of multiple cases the URs for each case will be sorted in ascending order for each element label.

The **Sort By Unity Ratio** option is used to sort the data in ascending order by UR value. In the case of multiple cases the sort will be based on the maximum UR for that element.

The **Sort by Group the Unity Ratio** option is used to sort the data in ascending order of group attribute. Within each group the data is sorted by Unity Ratio. The **Show Maximums** is also available for this option. If this is active only the maximum Unity Ratio for each group is printed.

The **Lower UR Output Limit** is used define a lower below which URs will not be listed. This reduces the size of the list to the highest loaded elements.

The **Identification Group SET** is used to optionally define a Group SET to be used for secondary identification or group sorting. If the SET does not exist or a zero value is used the group ID will be ignored.

The **Batch** button converts the set options to command line switches and appends the module command line to the .BRM batch run file.

The **View** button loads the results case file view form.

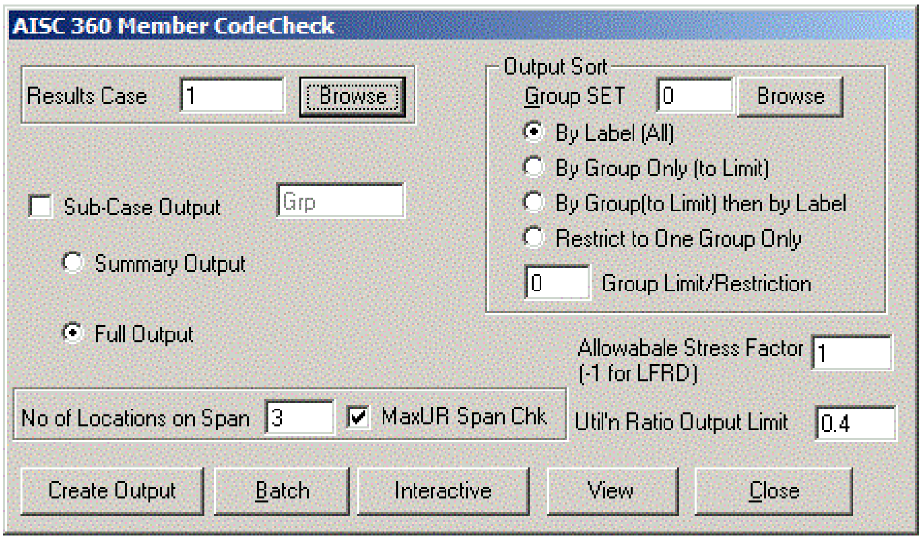
-o-

Output/Results:Menu:Design

This menu provides the command to activate the optional design modules and view/print the output from these modules.

These modules each have their own Help files which can be activated when the module is operated in it's **Interactive** mode.

The form below shows a typical codecheck input form from the FS2000 GUI.



Results Case is used to define the Processed Results Case to be code checked. The **Browse** button is used to select from a list.

A range of cases may be processed by defining a range e.g. 1-9 will process results cases between 1 and 9. If a case does not exist a warning will be given and the process will continue.

The **Sub-Case Output** option enables multiple formatted output files for the same Results Case to be created. This may be used if it is desirable to create separate Results files for the different output category options and various sort options available. If the option is checked then the file created will have the file name <Model>.Grp.* where Grp is the name entered in the Sub-Case description box.

Summary Output option produces an output listing that shows only the design unity ratio.

Full Output option produces an output listing that shows the actual and allowable loading..

Number of Locations on Span defines the number of points along an element at which the code checks are to be carried out. At each point the loading will be output (Full Output listing). Up to 21 points along the length of the element may be specified. If the **MaxUR SpanChk** is active the element will be check at 21 points on the span but only ends and the mid point with the maximum UR will be listed. When this is active the number of locations will always be set to 3.

Stress Ratio Limit is used to restrict the output to the elements whose maximum unity ratios are greater than that specified. The default value is zero. This facility is extremely useful since it reduces the output from the program and identifies critical elements more quickly.

The **Create Output** button is used to run the cod check with the currently shown settings.

The **Batch** button converts the set options to command line switches and appends the option's command line to the .BRM batch run file.

The **Interactive Mode** button activates Interactive operation of the CodeChecker

The **View** button loads the results case file view form.

Groups

The **Group SET** box is used to define the group SET to be loaded. If a SET is loaded then all node and element labels will be accompanied by their respective group attribute. If this field is left blank or contains the number of a non existent group then only the basic node and element numbers will be used for reference in the lists.

The **By Label (All)** option will output all entities (nodes and elements) in ascending label order.

The **By Group Only (to Limit)** option will output entities in ascending Group order. Entities not assigned to groups or entities assigned to Groups greater than defined by the **Group Limit/Restriction** box will not be output. This is a restricted process option.

The **By Group(to Limit) then Label option** will output entities in ascending Group order. Entities not assigned to groups or entities assigned to Groups greater than defined by the **Group Limit/Restriction** will be output in label order following the sorted groups. All data is processed with this option.

The **Restrict to One Group** option is used to restrict entities to only those entities with the same group number as defined by the **Group Limit/Restriction** box (zero value indicates that all data will be shown). This is a restricted process option.

Important Note: Stress ratio data created by this module will be limited to the data processed. If restricted process options is used then any Stress Ratio plots or Stress Ratio sorts that use the same result case will be limited to the processed data. The plot or listed output will indicate if the output is from a restricted process e.g. Restricted.

-o-

9 Command Line Definition

9.1 Command Line Definition Instructions

Command line model definition is a alternative method of creating model and load case data. It would probably only be used by an experienced analyst in special cases. It can be very useful when the source of the model data is text based i.e. from a Spread Sheet or when converting from a different model format. To use the command reference in its most convenient form it is recommended that they be printed out using the Help print button.

Command Line Instructions - Summary

NODE Definition Commands

ELEMENT Definition Commands

PROPERTY Definition Commands

RESTRAINT Definition Commands

LOAD Definition Commands

GROUPS Group SETs and Active Groups

Model Data Definition - An Overview of Command Line Operation

When a model is saved in the Model Definition TASK the basic model data (geometry, restraints and properties) is saved to a definition text file (.MDL file) and a number of binary files (XFiles). The XFiles are model system files read by the various system modules. When FS2000 opens a standard model it only reads these Xfiles.

If the user does not wish to define the model by command line definition than the existence of the MDL file is of no real significance.

When a model is Archived, the MDL file is merged with other definition files e.g. load cases to form the MOD file, a single definition file. When a model is opened from Archive format the MDL file is extracted and interpreted and the model saved in Xfile format.

The user can use FS2000's interpreter to interpret both single line commands entered directly at the keyboard or any text file containing valid commands. The Interpret commands in the File Menu in the Model Definition TASK are used to initiate the interpreter. Interpretation does not update the XFiles these are only updated when the model is saved. It is possible to interpret and save the model in Batch mode using Dynamic Interpretation. This allows the model to be modified between the solution of different load cases.

Note: If command files are given the model name and the file extension .UM* they will always be archived with the model i.e merged into the MOD file.

If the user wishes to define the model by direct editing of the MDL file using command line instructions then this file must be interpreted using the Interpret File command in the File Menu in the Model Definition TASK.

When the model is saved in the Model Definition TASK two modes of save are available. The default mode is to reformat (Reformat Mode) the MDL file with all model data written sequentially and comment lines, if any, omitted. The alternative mode is to preserve (Preserve Mode) the existing MDL file format including comment lines and to append any additional definition to the bottom of the file in a sequential format. Existing data if, modified will still maintain its original location in the file. In this mode comment lines indicating model (and dates & time) will be inserted into the file. The default Reformat Mode is active when the first line of the MDL reads ' REFORMAT'. Simply delete this line to set the Preserve Mode file format.

If generation commands are used to generate node or element data then only the data created from these commands will not be saved with the MDL file i.e. the MDL file written does not preserve generation commands.

If generation commands are to be preserved then they should be located in separate file (s). Use the file extension .UM* so as the always archive them.

Load Case Data File - An Overview of Command Line Operation

The .L files are used for load case data definition. The .L file is an interpreted file and may be created by the user in a text editor or interactively in the Load Definition TASK. The command syntax for the file is the same for both cases. If the user does not wish to define the model by command line definition then the actual contents of the .L file is not of any real significance to the user.

If the user wishes to create the load case definition file using text editing methods then the load should still be loaded and re-saved in the Load Definition TASK. This is necessary so that the load case is registered as existing. If a load case is not registered it will not appear in the load case selection lists. Unregistered load case files that do not appear in the selection list can be retrieved by simply entering the load case number.

When a load case is saved in the load definition module the .L file will is rewritten. The default mode when saving load case files is to re-sequence the file and omit any user defined comment lines. The alternative mode is to preserve the existing load file format and include existing comment lines and append any additional data to the end of the file in a sequential format. The default mode is active when the first line of the .L file reads 'REFORMAT'. Simply delete this line to preserve existing file format.

When load case data is saved in the Load Definition TASK the file date and time are always updated and data relating to the total load summation and Centre of Force is generated and appended to the .L file. The Centre of Force information is only generated in this save process therefore if this data is of significance then the re-saving of load cases should follow any significant model changes. If the C of F is not significant but the user does no wish to have potentiality incorrect data in the formatted output simply delete the lines from the definition file and the data will not be used.

-o-

9.2 Command Line Instructions - Summary

The following sections describe the Model Definition Commands and their associated arguments.

Only single commas may be used as delimiters for Command Arguments

If arguments are omitted the default values will be used. If a line is not complete the arguments not entered will be taken to be the default values. If a space(s) are entered between the commas or the commas are adjacent to each other the missing argument will be taken by the default value.

Comment lines may be inserted into command line files by using the command line

COM, *comment.*

To retain comment lines the **REFORMAT** line, if present, must be deleted from the top of the file.

Command Summary - Model Definition

NAME, *Model Name*

TITLE, *Model Description*

UNIT*, Units Description*

UDEF*, Unit Type Definition 0 - SI 1 - US Units*

DATE,*Current Date of Model*

TIME, *Current time of Model*

REF*, Any description*

DESC*, Any description*

Co-ordinate Systems

CSYS, *No, TX, TY ,TZ, RX, RY, RZ, P1, P2, N3*

Node Related

N, *Node, X, Y ,Z*

NR, *Node, X, Y ,Z*

NGEN*, N1, N2, NS, NF, NINC*

NCOPY, *N1, N2, NINC, NTIME, STNODE, DX, DY, DZ*

NMAX, *MAXNNUM*

NARC, *N1,N2, NS, NF, NINC, RAD, N3, ROT*

NTRANS, *N1,N2, NINC, TX, TY, TX, RX, RY, RZ*

NCORDCON, *N1,N2, NINC*

NGROUP*, Gno*

NMAX*, Label*

Element Related

E, *Elem, N1, N2, N3, ROT, GEOM, MAT, RELZ, RELY,TAPER,TYPE,CO*

EN, *Elem, X, Y, Z, N2*

ENR, *Elem, X, Y, Z, N2*

EGEN*, N1,N2, NINC, E1, EINC*

ECOPY*, E1,E2, EINC, NTIMES, EST, NINC*

ENCOPY*, E1,E2, EINC, NTIMES, EST, NST , DX, DY, DZ*

EMOD*, E1,E2, EINC, Attribute, AttValue*

EOF, *Elem, NOEF, EREF1, x, y,z, EREF2, x, y, z*

PDATA, *Elem, Type, NodeEnd, KFlex, SIFI, SIFO, K1, K2*

S, *Elem, Type, NN,N1,----NN, GEOM, MAT, ROT, SOpt*

SCOPY, *E1,E2, EINC, NTIMES, EST, NINC*

SNCOPY, *E1,E2, EINC, NTIMES, EST, NST , DX, DY, DZ*

EMAX, *Label*

EGROUP, *Gno*

SC, *Elem, N1 ,N2, RefElem, ROT, SPCONST, SCCSys*

SCCOPY, *E1,E2, EINC, NTIMES, EST, NINC*

SCDEL, *E1,E2, EINC*

Property Data

GTAB1, *Code, Type, Name, Designation*

GTAB2, *Code, C1 to C6*

GTAB3, *Code, C7to C11*

GTAB4, *Code, C12 to C15*

GTAB5, *Code, C 16 fo C20*

GTABP, *Code, CorrAll, MillTol, ContDen, InsulT, InsulDen, LiningT, LiningDen*

PIPE, *OD, WALL, Code*

SECT, *Lib, Designation, Code*

MTAB, *Code, E, G, POIS, DENS, ALPHA, YIELD,MatNam, ULT*

MTABP, *Code, UltSt, ColdAllSt, QualFact, PressCoeff*

MTABT, *Code, Pt,Temp, ,ALPHA ,E ,AllStress*

STAB, *Code, k1, k2, k3, k4, k5, k6,TYPE,CO*

IC, *Code, IC0, to IC6*

RC, *Code, RCx1, RCy2 to RXx7, Rcy7*

Groups

EGROUP, *GrpNo*

GRPSET, *GrpSET*

NGROUP, *GNo*

Active Constants

ACTCSYS, *Value*

ACTN, *Value*

ACTN3 , *Value*

ACTROT, *Value*

ACTGEOM, *Value*

ACTMAT, *Value*

ACTRELZ , *Value*

ACTRELY,*Value*

ACTTYPE, *Value*

ACTCON, *Value*

Restraints

REST*, NODE, X, Y, Z, RX, RY, RZ*

RESTCOPY*, NODE, N1 ,N2, NINC*

Load Definition

LDESC, *Load Case Description*

LDATE,*Current Date of Load Case*

LTIME, *Current time of Load Case*

ACCEL, *Gx, Gy, Gz*

ND*, Node, Tx , Ty, Tz, Rx, Ry, Rz*

NF*, Node, Fx , Fy, Fz, Mx, My, Mz, Nmass*

NT*, Node, Temperature*

ED, *Elem, COORD, SLength, FLength, GDX1, GDX2, GDY1, GDY2, GDZ1, GDZ2*

EP, *Elem, COORD, Length, Fx , Fy, Fz, Mx, My, Mz*

FP, *Elem, Face, P1, P2, P3, P4*

TEPR, *Elem, TEMP, PRESS, TEMP2*

AMBT, *Ambient Temperature*

ACOM, *Comment Line*

ETPR, *Node,EGroup,ProfileNo*

PPRESS, *Code, PRESS*

PTEMP, *Code, TEMP*

PUDL, *Code, Dir, LOAD*

ESTR, *Elem, Value*

SEFO, *Elem, Value*

CDISP, *Couple, Freedon, Value*

CFACT, *Couple, Time Curve*

Descriptive Model Data

NAME, *Model Name*

TITLE, *Model Description*

UNIT*, Units Description*

DATE,*Current Date of Model*

TIME, *Current time of Model*

REF*, Any description*

DESC*, Any description*

-o-

9.3 Node Definition

CSYS, *No, Type, TX, TY, TX, RX, RY, RZ, P1, P2, N3*

Creates a user defined co-ordinate system

N0	Co-ordinate System Number
Type	System Type (0-Cartesian, 1-Cylindrical, 2-Spherical, 3 - Conical)
T1, T2, T3	Co-ordinates of origin in Global Cartesian or 3 nodes that define the x-y plane if *N3* = -1
RX, RY, RZ	Rotational orientation (degrees) Not used if *N3* = -1
P1, P2	Parameters For conical systems P1 is the Radius at Z=0 and P2 is the cone angle (from Z axis). These two parameters make the R and Z coordinates interdependent i.e. define a conical surface.
N3	If *N3* = -1 then *T1, T2, T3* are used to identify 3 nodes that define the x-y plane

N, *Node, X, Y ,Z, CSys*

Defines a node in the active co-ordinate system

Node Node number to be assigned. A previous defined node will be re-defined

(Default = No of Nodes +1)

X, Y , Z Node location in global co-ordinate system. If a co-ordinate value is omitted the node co-ordinate will be taken to be the same as that of the last node defined.

Exp.	N,6,1,0,3	(Node 6 at x = 1;y = 0;z = 3)
	N,8,,9,6	(Node 8 at x = 1;y = 9;z = 6)
	N,9,3	(Node 9 at x = 3;y = 9;z = 6)

CSys Co-ordinate system number (default = ACTCSYS)

NR, *Node, X, Y ,Z*

Defines a node in the active co-ordinate system where X, Y & Z are relative to the last node defined. Use **ACTN** to re-define reference node.

Node Node number to be assigned. A previous defined node will be re-defined

(Default = No of Nodes +1)

X, Y , Z Node location in global co-ordinate system relative to the co-ordinates used to define the previous node. If a co-ordinate value is omitted the node co-ordinate will be taken to be the same as that of the last node defined.

Exp.	N,6,1,0,3	(Node 6 at x = 1;y = 0;z = 3)
	N,8,,9,6	(Node 8 at x = 1;y = 9;z = 6)
	N,9,3	(Node 9 at x = 3;y = 9;z = 6)

NGEN, *N1, N2, NS, NF, NINC*

Generates one or more equally spaced nodes in a straight line between two previously defined nodes.

N1 First existing node

N2 Second existing node

NS Node number of first node to be generated (default = No of Nodes + 1)

NF Node number of last node to be generated (default = NS i.e. single node generation)

NINC Node increment for generated nodes (default =1)

NCOPY, *N1, N2, NINC, NTIME, STNODE, DX, DY, DZ*

The NCOPY command copies a range of node to a location defined by co-ordinate increments. The original node label pattern is preserved in newly defined nodes.

N1 First existing node to be copied (Last node defined)

N2 Final existing node to be copied (default = N1)

NINC Node increment of nodes to be copied (default = 1)

NTIME Number of copies required (default = 1)

STNODE Start node number for new nodes

DX, DY, DZ Co-ordinate increment between copies (Default = 0)

NARC, *N1,N2, NS, NF, NINC, RAD, N3, ROT*

Generates one or more equally spaced nodes in a circular arc between two previously defined nodes.

N1 First existing node

N2 Second existing node

NS Node number of first node to be generated (default = No of Nodes + 1)

NF Node number of last node to be generated (default = NS i.e. single node generation)

NINC Node increment for generated nodes (default =1)

RAD Arc radius

N3 Third node for arc orientation (optional)

ROT Rotation angle for arc orientation (optional)

NTRANS, *N1,N2, NINC, TX, TY, TX, RX, RY, RZ,Origin*

Translates a set of defined nodes relative to their co-ordinate system.

N1 First node in set (no default) or Group Number

N2 Final node in set (default = N1)

NINC Node increment for set (default =1)

T1, T2, T3 Linear translation

RX, RY, RZ Rotational translations

Origin Define rotation origin by node association (defautl = 0 ie global origin)

Using Group If N1 is defined as a negative value (define N2 and NINC=0) then it will be interpreted as a Group No in the current Group SET. See GRPSET command.

NCORDCON, *N1,N2, NINC*

Converts and assigns a set of specified nodes to the active co-ordinate system.

N1 First node in set (no default) or Group Number

N2 Final node in set (default = N)

NINC Node increment for set (default =1)

Using Group If N1 is defined as a negative value (define N2 and NINC=0) then it will be interpreted as a Group No in the current Group SET. See GRPSET command.

NGROUP, *GNo*

The **NGROUP** command is used set the current group to which nodes will assigned

NMAX, *Label*

The **NMAX** command is udes to define the maximum node label in the model.

-o-

9.4 Element Definition

Element Active Constants

Active contents are used to set defaults to be used in element definition. They remain active until they are re-defined by the **ACT** command or by the element definition command **E.**

ACTCSYS,*Value* Defines active C-ordinate System (default = 0-Cartesian)

ACTN,*Value* Used to re-defines reference node for relative node definition

ACTN3 , *Value* Third node for local element rotation definition (Default = 0)

ACTROT, *Value* Local element rotation angle (default = 0)

ACTGEOM, *Value* Geometrical property table code (default = 1)

ACTMAT, *Value* Material Property table code (default = 1)

ACTRELZ , *Value* Hinge Definition - local z axis (0, 1,2, or 3) (default = 0)

ACTRELY, *Value* Hinge Definition - local y axis (0, 1,2, or 3) (default = 0)

ACTTYPE, *Value* Element type (default = 0)

ACTCON, *Value* Element CO constant (default = 0)

E, *Elem, N1, N2, N3, ROT, GEOM, MAT, RELZ, RELY, TAPER, TYPE, CO, BendRad*

The E command defines individual elements.

Elem The element number to be assigned. Previous will be re-defined.

(Default = No of EL +1)

N1 Start node of element (default = previous N2 node or last node defined)

N2 End node of element (default = N1+ 1)

N3 Third node of element used to define local rotation (default = ACTN3)

ROT Local element rotation angle (default = ACTROT)

GEOM Geometrical property table code (default = ACTGEOM)

GEOM = -1 identifies a rigid link

MAT Material Property table code (default = ACTMAT)

RELZ Hinge Definition - local z axis (0, 1,2, or 3) (default = ACTRELZ)

RELY Hinge Definition - local y axis (0, 1,2, or 3) (default = ACTRELY)

TAPER This is used to identify tapered beams by defining the geom property codeat the end node of an element. A +ve values signifies a Type A Taper a -ve value signifies a Type B Taper (default =0 no taper)

TYPE Element Type

CO Additional CO constant

BendRad Bend radius for Type 2 and Type 3 elements

EN, *Elem, x, y, z, N2*

The EN command is used to define an individual element and its end node. The start node of the element is the previously defined end node or the node defined by the **ACTN** command. This command is very useful for defining string elements since it creates node and elements on a 'from' to basis.

Elem The element number to be assigned. Previous will be re-defined.

(default = No of EL +1)

X, Y , Z Node location in global co-ordinate system. If a co-ordinate value is omitted the node co-ordinate will be taken to be the same as that of the last node defined.

N2 Node number for final node (default = No of Nodes + 1)

ENR, *Elem, x, y, z, N2*

This is identical to **EN** but the node co-ordinates are relative to the last node defined.

EGEN, *N1,N2, NINC, E1, EINC*

The **EGEN** command is used to generate line elements using a node pattern of existing line nodes. Use the ACT command to select property codes etc.

N1 First existing node (no default)

N2 Second existing node (no default)

NINC Increment of existing node pattern (default = 1)

E1 First in element set to be generated (default = No of El +1)

EINC Increment for element to be generated (default = 1)

ECOPY, *E1,E2, EINC, NTIMES, EST, NINC*

The **ECOPY** command is used to copy an existing pattern of line elements using a pattern of existing nodes. The element numbering pattern will be preserved

E1 First element in existing element pattern (default=1)

E2 Last element in existing element pattern (default = E1)

EINC Element increment in existing element pattern (default = 1)

NTIMES No of copies required (1)

EST First element (default = No of EL + 1)

NINC Node increment between existing node sets (default = 1)

ENCOPY, *E1,E2, EINC, NTIMES, EST, NST,DX DY,DZ*

The **ENCOPY** command is used to copy an existing pattern of line elements and create the nodes for the new elements. The element and node numbering pattern will be preserved

E1 First element in existing element pattern (default=1)

E2 Last element in existing element pattern (default = E1)

EINC Element increment in existing element pattern (default = 1)

NTIMES No of copies required

EST First element (default = No of EL + 1)

NST Start node number for new elements (default = No of Nodes + 1)

DX, DY, DZ Co-ordinate increment between copies (Default = 0)

EDEL, *E1,E2, EINC*

The **EDEL** command is used to delete an existing pattern of line elements. The element numbering pattern will be preserved

E1 First element in existing element pattern (no default) or Group No

E2 Last element in existing element pattern (default = E1)

EINC Element increment in existing element pattern (default = 1)

Using Groups If E1 is defined as a negative value (define E2 and EINC=0) then it will be interpreted as a Group No in the current Group SET. See GRPSET command.

EMOD, *E1,E2, EINC, Attribute, AttValue*

The **EMOD** command is used to define specific element properties.

E1 First element in existing element pattern (no default) or Group No

E2 Last element in existing element pattern (default = E1)

EINC Element increment in existing element pattern (default = 1)

Attribute This identifies the property to be re-defined

1 - Geometric Property Code

2 - Material Property Code

3 - Element Type

4 - CO Constant

AttValue Specifies the value of the attribute.

Using Groups If E1 is defined as a negative value (define E2 and EINC=0) then it will be interpreted as a Group No in the current Group SET. See GRPSET command.

EOF, *Elem, NOEF, EREF1, x, y,z, EREF2, x, y, z*

This command is used to define rigid end offsets for specific beam elements. All default are zero unless specified.

Elem The element number to be assigned. Previous will be re-defined.

(default = Max EL Label)

NOEF Element End ID code 1 - First Node End Only

2 - Second Nodes End Only

3 - Both Ends Element Ends

EREF1 Ref Element for Coord sytem for offsets x, y, z for First Node

(0 is global)

EREF2 Ref Elements for Coord sytem for offsets x, y, z for Second Node

(0 is global)

X Y, Z Element end offsets

PPARAM, *Elem, Type, NodeEnd, KFlex, SIFI, SIFO, K1, K2*

The **PPARAM** command is used to define element flexibility factors and stress intensification factors for pipe elements. The Type, K1 and K2 variables are only used for reference purposes.

Elem Elem Label

Type Pipe Fitting ID Code - Reference Data Only (see below)

NodeEnd = 1 Apply SIF to fore end

= 2 Apply SIF to aft end

= 3 Apply SIF to both ends

KFlex Flexibility Factor Default=1

SIFI In plane Stress Intensification Factor Default=1

SIFO Out of plane Stress Intensification Factor Default=1

K1 Constant depending on Type - Ref. Data Only (see table below)

K2 Constant depending on Type - Ref. Data Only (see table below)

BEND K1 = Bend Radius

1 Welding Elbow or Pipe Bend

	2	Bend with one flanged end	
	3	Bend with two flanged ends	
	4	Mitre Bend	K2 = Mitre Angle
	5	Socket Welded Elbow	
TEE			
	6	Welding Tee per ANSI B16.9	
	7	Reinforced fabricated tee with pad or saddle	K1 = Pad Thickness
	8	Unreinforced fabricated tee	
	9	Extruded welding tee	K2 = Tee Radius
	10	Welded-in contour insert	
	11	Branch welded on fitting (integrally reinforced)	
CONN			
	12	Buttwelded joint, reducer or weld neck flange	
	13	Double-welded slip-on flange	
	14	Fillet-welded joint or socket weld flange	
	15	Lap joint flange (with ANSI B16.9 lap joint stub)	
	16	Screwed pipe joint or screwed flange	
	17	Corrugated straight pipe or corrugated or creased bend	
	18	User Defined	

S, *Elem, Type, NN,N1,----NN, GEOM, MAT, ROT, Sopt, Offset, CO*

The S command defines individual soild elements.

Elem	The element number to be assigned. Previous will be re-defined. (Default = No of EL +1)
Type	The element type 30 - Plane 2-D Solid 40 - Axisymmetric Solid 50 - Thin Shell 51 - Thick Shell 60 - Membrane 70 - 3-D Solid
NN	Number of nodes in element
N1	Start node of element (default = previous N2 node or last node defined)
NN	End node of element (default = N1+ 1)
ROT	Local element rotation angle (default = ACTROT)
GEOM	Geometrical property table code (default = ACTGEOM) GEOM = -1 identifies a rigid link
MAT	Material Property table code (default = ACTMAT)
ROT	Local element rotation angle (default = ACTROT)
SOpt	Solution option for element (default = 0)
Offset	Shell offset in local z direction (default = 0)

CO	Defines the Winkler support stiffness (default = 0)

SCOPY, *E1,E2, EINC, NTIMES, EST, NINC*

The **SCOPY** command is used to copy an existing pattern of solid elements using a pattern of existing nodes. The element numbering pattern will be preserved

E1	First element in existing element pattern (default=1)
E2	Last element in existing element pattern (default = E1)
EINC	Element increment in existing element pattern (default = 1)
NTIMES	No of copies required (1)
EST	First element (default = No of EL + 1)
NINC	Node increment between existing node sets (default = 1)

SNCOPY, *E1,E2, EINC, NTIMES, EST, NST , DX, DY, DZ*

The **SNCOPY** command is used to copy an existing pattern of solid elements and create the nodes for the new elements. The element and node numbering pattern will be preserved

E1	First element in existing element pattern (default=1)
E2	Last element in existing element pattern (default = E1)
EINC	Element increment in existing element pattern (default = 1)
NTIMES	No of copies required
EST	First element (default = No of EL + 1)
NST	Start node number for new elements (default = No of Nodes + 1)
DX, DY, DZ	Co-ordinate increment between copies (Default = 0)

EGROUP, *GNo*

The **EGROUP** command is used set the current group to which elements will assigned

EMAX, *Label*

The **EMAX** command is used set the maximum element label in the model.

SC, *Elem, N1 ,N2,ROT, RefElem, SPCONST, SCCSys*

The **SC** command defines individual spring/couple elements.

Elem	The element number to be assigned. Previous will be re-defined.
	(Default = No of SC +1)
N1	Start node of element (default = previous N2 node or last node defined)
N2	End node of element (default = N1+ 1
ROT	Local element rotation angle (default = ACTROT))
RefElem	Orientation - Ref. element for local co-ordinate system (default =0)
SPCONST	Spring Constant Property table code (default = ACTMAT)
SCCSys	Orientation - Reference Coordinate System (Default=0)

SCCOPY, *E1,E2, EINC, NTIMES, EST, NINC*

The **SCCOPY** command is used to copy an existing pattern of spring/couples using a pattern of existing nodes. The element numbering pattern will be preserved

E1	First spring element in existing element pattern (default=1)
E2	Last spring element in existing element pattern (default = E1)

EINC Element increment in existing element pattern (default = 1)

NTIMES No of copies required

EST First spring element (default = No of SC + 1)

NINC Node increment between existing node sets (default = 1)

SCMOD, *E1,E2, EINC, Attribute, AttValue*

The **SMOD** command is used to define specific element properties.

E1 First element in existing element pattern (no default) or Group No

E2 Last element in existing element pattern (default = E1)

EINC Element increment in existing element pattern (default = 1)

Attribute Identifies the property to be re-defined

1 - Couple Property Code

2 - Couple Coord Reference element

3 - Couple Coordinate System

AttValue Specifies the value of the attribute.

Using Groups If E1 is defined as a negative value (define E2 and EINC=0) then it will be interpreted as a Group No in the current Group SET. See GRPSET command.

SCDEL, *E1,E2, EINC*

The **EDEL** command is used to delete an existing pattern of spring/couples. The element numbering pattern will be preserved

E1 First element in existing element pattern (no default) or Group No

E2 Last element in existing element pattern (default = E1)

EINC Element increment in existing element pattern (default = 1)

Using Groups If E1 is defined as a negative value (define E2 and EINC=0) then it will be interpreted as a Group No in the current Group SET. See GRPSET command.

-o-

9.5 Property Definition

The **GTAB** commands are used to define the geometric properties corresponding to the property codes referenced in element definition. For convenience the data has been divided into five lines. Only **GTAB2** is essential for stiffness analysis. Always enter the code number unless the previously entered code is require to be used.

Note that the **PIPE** & **SECT** commands will enter all appropriate GTAB data.

GTAB1, *Code, Type, Name, Designation, GType, GOFY, GOFZ*

This optional command is used to define the geometric property type, its name and its designation.

Code Code number used by element definition to reference properties.

(default = max code label+1)

Type Geometric Type (Non-Linear) Typically

10 = Compression only (non-linear)

11 = Tension only (non-linear)

Search for Geometric Type for more info.

Name 3 Character Name

Designation Up to 9 numeric characters

GType Beam type for Virtual views (optional) (I, B, C, T or A)

GOFY Graphic offset for virtual views (optional)

GOFZ Graphic offset for virtual view (optional)

GTAB2, *Code, C1 to C6*

All default values are zero. If Cl and C2 are specified then GTAB related data will be generated.

. *Code* Code number used by element definition to reference properties.

(default = max code label)

C1 to C6 Property Constants

C1 = Pipe OD For Pipe Data

C2 = Pipe wall thickens

(All property data will evaluated from pipe data if specified)

C3 = Area For Beam Data

C4 = Izz

`C5 = Iyy

C6 = J

GTAB3, *Code, C7 to C11*

Optional data depending upon analysis requirements

. Code Code number used by element definition to reference properties.

(default = max code label)

C7 = Ay - Shear area

C8 = Az - Shear area

C9 = Plastic Modulus zz axis

C10= Plastic Modulus yy axis

C11= Torsional Modulus

GTAB4, *Code, C12 to C15*

Optional data depending upon analysis requirements.

Code Code number used by element definition to reference properties.

(default = max code label)

C12 = Stress point 1 y co-ordinate

C13 = Stress point 1 z co-ordinate

C14 = Stress point 2 y co-ordinate

Cl5 = Stress point 2 z co-ordinate

GTAB5, *Code, C16 to C20*

Optional data depending upon analysis requirements

Code Code number used by element definition to reference properties.

(default = max code label)

C16 = Stress point 3 y co-ordinate

C17 = Stress point 3 z co-ordinate

C18= Stress point 4 y co-ordinate

Cl9 = Stress point 4 y co-ordinate

C20 = Torsional modulus

GTABP, *Code, CorrAll, MillTol, ContDen, InsulT, InsulDen, LiningT, LiningDen*

The GTABP is used to defined data relating to pipe elements

Code Code number used by element definition to reference properties.

(default = max code label)

CorrAll = Corrosion Tolerance

*MillTo*l = Mill Tolerance %

ContDen = Contents Density

InsulT = Contents Density

InsulDen = Insulation Density

LiningT = Internal Lining Thickness

LiningDen = Internal Lining Density

PIPE, *OD, WALL, Code*

This generation command (GTAB data) is combines the **GTAB** and **ACTGEOM** command for the definition of pipe elements. If Code is omitted the existing property codes will be scanned and if one exists with the same parameters it will be made active. If it one does not exist one will be created and then made active. If Code is specified than that code will be replaced.

OD Pipe outside diameter

WALL Pipe wall thickness

Code Code number used by element definition to reference the properties

Note that when the model is saved only generated GTAB commands will be added into the model

MDL file.

SECT, *LIB, DESIG, Code*

This generation command (GTAB data) is used to get the geometric properties from a standard section library e.g. **UB** would be used for the **UB.PRI** library. If Code is omitted the default will be used. When a code is entered it will be made the active code (ACTGEOM).
Note that only libraries in the user's FS2000 folder can be accessed.

LIB Library Name

DESIG Section Designation I.D. used in the library

Code Code number used by element definition to reference the properties (default = No of Codes +1)

Note that when the model is saved only generated GTAB commands will be added into the model MDL file.

MTAB, *Code, E, G, POIS, DENS, ALPHA, YIELD, MatNam, ULT*

This command is used to define the material properties corresponding to the material table codes used in element definition. All default values are typical values for steel.

Code	Code number used by element definition to reference properties.
	(Default = No of Codes +1)
E	Modulus of Elasticity (Default = 205E9)
G	Modulus of Rigidity (Default based on POIS)
POIS	Poisson's ratio (Default = .3)
DENS	Density (Default = 7850)
ALPHA	Thermal Coefficient of Expansion (Default = 1.1E-5)
YIELD	Yield Strength (Default = 250E6)
MatNam	Material Description (up to 8 characters)
ULT	Ultimate Strength (Default = 250E6)

MTABP, *Code, UltStr, ColdAllSt, QualFact, PressCoeff*

This command is used to define the additional material properties relating to pipework design

Code	Code number used by element definition to reference properties.
	(Default = No of Codes +1)
UltStr	Ultimate Tensile Strength
ColdAllSt	Cold Allowable Stress
QualFact	Quality/Joint Factor
PressCoeff	Pressure Coefficient

MTABT, *Code, Pt,Temp, ,ALPHA ,E ,AllStress*

This command is used to define the additional material properties relating to temperature dependancy.

Code	Code number used by element definition to reference properties.
	(Default = No of Codes +1)

Pt	Curve Point (Up to 15 point) Points most be successive.
ALPHA	Thermal Coefficient of Expansion
E	Modulus of Elasticity
AllStress	Allowable Stress

STAB*, Code, k1, k2, k3, k4, k5, k6, TYPE, CO*

The SPCONST command is used to define the couple element constants used by spring/couple elements.

k1 to k6	Spring stiffness for linear couple element TYPE 0
TYPE	Element Type - Default =0 (Standard linear couple)
CO	Additional CO constant

Gap Elements are defined by the following.

Type 10	Compression Gap		
Type 11	Tension Gap		
k1	Gap stiffness - Normal direction		
k2	Gap stiffness - Tangential direction		
k3	Sliding Friction Coefficient - Coulomb friction		
k4	Sliding Friction Coefficient Local Z		
k5	Gap Size		
k6	Initial Gap Status	0	Open
		1	Closed

IC*, Code, IC0, to IC6*

The IC command is used to define additional integer constants used by some elements.

Code	Code number used by element definition to reference properties. (Default = No of Codes +1)
IC0 to IC6	Integer constants.

RC*, Code, RCx1, RCy2 to RXx7, Rcy7*

The RC command is used to define additional real constants used by some elements. .

Code	Code number used by element definition to reference properties. (Default = No of Codes +1)
RCx1, Rcy1 *To* *RCx7, Rcy7*	Real constants.

-o-

9.6 Restraints

REST, *NODE, X, Y, Z, RX, RY, RZ*

Defines the restraint of a node in the global co-ordinate system

Node Node number to be restrained. A previous defined node will be re-defined (Default = 0)

X, Y , Z Translation restraint direction in global co-ordinate system.

RX, RY , RZ Rotational restraint direction in global co-ordinate system.

0 signifies free

1 signifies fixed

RESTCOPY, *NODE, N1 ,N2, NINC*

The **RESTOPY** command is used to copy a node restraint to a pattern of existing nodes.

NODE Node to be copied (no default)

N1 First node in existing node pattern (no default)

N2 Last node in existing node pattern (default = N1)

NINC Node increment (default = 1)

-o-

9.7 Load Definition

LDESC, *Load Case Description*

LDATE,*Current Date of Load Case*

LTIME, *Current time of Load Case*

ACCEL, *Gx, Gy, Gz*

The **ACCEL** command is used to define the acceleration constants in the global co-ordinate system

ND*, Node, Tx , Ty, Tz, Rx, Ry, Rz*

The **ND** command is used to prescribe nodal displacements. Default /zero values are considered free. Definition is in the global co-ordinate system.

Node	Node label
Tx, --	Node translations
Rx,--	Node rotations

NF*, Node, Fx , Fy, Fz, Mx, My, Mz, NMass*

The **NF** command is used to define nodal (concentrated) loads. All defaults are zero. Definition is in the global co-ordinate system.

Node	Node label
Fx, --	Concentrated force
Mx,--	Concentrated moment (couple)
NMas	*Concentrated nodal mass*

NT*, Node, Temp*

The **NT** command is used to define nodal temperatures on solid elements for use in thermal expansion solutions. All defaults are zero. Definition is in the global co-ordinate system.

Node	Node label
Temp	Temperature at node

UDL, *Elem, UDX, UDY, UDZ*

The **UDL** command is used to define uniformly distributed element loads

Elem	Element label
UDX	Load intensity in global X
UDY	Load intensity in global Y
UDZ	Load intensity in global Z

ED, *Elem, COORD, SLength, FLength, GDX1, GDX2, GDY1, GDY2, GDZ1, GDZ2*

The **ED** command is used to define non-uniformly distributed element loads

Elem	Element label
COORD	Co-ordinate system (default = global) 1 Global 2 Local Element 3 Projected Global
SLength	Length from SNode of element to start of load (default =0)
FLength	Length from SNode of element to end of load (default = Element Length = 0)
GDX1 etc	Load intensity in X, Y or Z direction at pt 1 (SLenght)

GDX2 etc Load intensity in X, Y or Z direction at pt 2 (Flenght*) (default = GD'-'1)*

EP, *Elem, COORD, Length, Fx , Fy, Fz, Mx, My, Mz*

The **EP** command is used to define concentrated mid span element loads

Elem Element label

COORD Co-ordinate system (default = global) 1 Global 2 Local Element 3 Projected Global

Length Length from SNode of element to start of load (default =Element length/2)

Fx, -- Concentrated force

Mx,-- Concentrated moment (couple)

ETPR, *Node,EGroup,ProfileNo*

The ETPT command is used to generate a temperature profile on line elements defined by an element group. This is a wizard command which should be used interactively in the Load Definition TASK.The routine generates TEPR load commands and should only be used once using the same parameters in a specific load case. The profile is applied by linear interpolation from a profile curve defined by a number of discrete points.

Node Start node for the profile (attached element must be in group)

EGroup Identifying element group for the profile - must be in current SET

ProfileNo Profile ID number. The profile is defined a user defined text file **.UTP(N)** where N is the profile ID number. The format for the file is:

Number of points

Distance1, Temperature1

….. , ……

DistanceN,TemperatureN

If the element group in not continuous along the line element, the profile definition will terminate. If a branch is encountered it will follow the grouping. If groupings are common at a branch it follow the element with the lowest label. The profile can be plotted using the TP button located in the Pipework Toolbar.

FP, *Elem, Face, Dir, P1, P2, P3, P4*

The **FP** command is used to define face pressures and edge loads on solid elements. It is also used to define thermal loading for heat transfer solutions.

Elem Element label

Face Face Number - See Element Technical Description for detail

Dir The direction for edge loads

1-Normal - In-plane

2- Tangential - In-plane

3 - Tangential - Out of Plane

P1, -- P4 Pressure or Edge load intensity

Heat Transfer Solutions

Dir Boundary Type

1- Temperature-Heat Transfer Coefficient

P1 = Tf P2 = h

P1 = Ts P2 = 0

2- Heat Flux

P1 = Heat Flux

3 - Heat Generation

P1 = Element Heat Generation

PPRESS, *Code, PRESS*

The **PPRESS** command is used to define hydro static internal pressure loading on pipe elements by geometric property code reference. All default are zero.

Code Element Geometric Property Code Number

PRESS Differential internal pipe pressure

PTEMP, *Code, TEMP*

The **PTEMP** command is used to define thermal expansive loading on beams type elements by geometric property code reference. All default are zero.

Code Element Geometric Property Code Number

TEMP Differential element temperature (relative to ambient)

PUDL, *Code, Dir, LOAD, Coord*

The **PUDL** command is used to define global distributed loading on beam/pipe elements by geometric property code reference. All default are zero.

Code Element Geometric Property Code Number

Dir Load direction (global)

1 - X Direction

2 - Y Direction

3 - Z Direction

LOAD Load magnitude

COORD Co-ordinate system (default = global) 1 Global 3 Projected Global

TEPR, *Elem, TEMP1, PRESS, TEMP2*

The **TEPR** command is used to define thermal expansive loading on beam type elements and hydro static pressure loading on pipe elements. All default are zero unless stated.

Elem Element Label

TEMP1 Differential temperature on upper surface

PRESS Differential pressure

TEMP2 Differential temperature on lower surface (default = *TEMP*1 ie uniform)

AMBT, *Ambient Temperature*

The AMBT command is used to define the ambient temperature when evaluating differential expansion. It is also used for the evaluation of thermal properties for pipe design.

ACOM, *Comment Line*

Comment Line Comment line to be added to the bottom of the formatted loada case output

Additional Load Commands

The following additional load command can only be used with 3-D NL & DyNoFlex . These commands are very useful when used with spar (Type 15) elements for simulating winch wires where large strains can be use represent wire movement or where tension is required to be defined. These element commands can only be added to a load case by using text input or interpreted command input.

ESTR, *Elem, Value*

The **ESTR** command is used to define element axial strain in non-linear beam element type. This command can be used with the large displacement option ie the strain rotates with the element.

Elem Element label

Value Strain value

SEFO, *Elem, Value*

The **SEFO** command is used to define element axial loads with Type 15 elements. This command can be used with the large displacement option ie the force rotates with the element.

The axial stiffness of the element should be zero or an extremely small value to ensure convergence. This can be done setting the E or Area to near zero. Setting the E value to zero has the advantage that the evaluated stresses will be low.

Elem Element label

Value Axial Force

CDISP, *Couple, Freedom, Value*

The **CDISP** command is used to define relative movement between the coupled degrees of freedom of a Type 7 couple element. It effectively applies prescribed displacements between connected nodes in a similar manner to which nodal prescribed displacements do between a node and the boundary. Note that Type 7 couple is large displacement couple when associated with a large displacement element.

Couple Couple element label

Freedom Coupled degree of freedom in local couple axis (1 to 6)

Value Displacement magnitude (angles are in radians)

If an element is also subjected to a CFACT command then the CFACT command must also be included in the same load case as the CDISP command.

CFACT, *Couple, Time Curve*

The **CFACT** command is used to factor the stiffness (all components) of a Type 7, Type 4 and Type 20 couple element. Its main use is to add or remove couple elements (birth & death) during a time history solution. The stiffness of the couple is directly factored by the value of the associated Time Curve at the current time step.

Couple Couple element label

Time Curve Time Curve number

The command can be added to any one or more of the load cases in a time history combination. The CFACT command is not affected by the Time Curve assigned to the containing load case because only the Time Curve in the CFACT command defines it behaviour.

The CDISP and CFACT command can be used on the same element but there is one important restriction. The load case that contains the CDISP command must also contain the CFACT command.

Generated Information

If the load case is saved in the model Definition Module some additional information, described below, will appear in the definition file.

MODEL, Model Name File information only

TITLE, Model Description File information only

LCASE, Load Case Number File Information only (File extension defines load case)

SCASE, Special Load Case Used to identify special Load Cases e.g. merged load cases or generated load cases.

The following data is interpreted for the formatted output files if the lines are present.

LSUM, Total Loads on Model Optional Data also generated prior and during analysis

COF, Center of Force Data Optional Data

-o-

9.8 Groups

EGROUP, *GrpNo*

The **EGROUP** command is used set the active group,

GRPSET, *GrpSET*

The **GRPSET** command is used open an existing Group SET

NGROUP, *GNo*

The **NGROUP** command is used set the active group,

-o-

9.10 Time Curves - Command Line Definition

Examples of commonly used Time Curves are given in Section 5

The **TCURVE** command is used to define a time/amplitude curve. This enables the amplitude of the excitation to be function of time. The amplitude value is used to factor the amplitude of the excitation force. The default maximum number of data points is 300. This may be extended by including the MAXTIME command in the time curve case. Note that during the solution the curve solution, the time step is used to interpolate the the curve data points therefore data pints can be maintained to the minimum to define the curve i.e linear sections only require two definition points. For simple ramps the **SCURVE2** and **RCURVE2** are possibly better options.

TCURVE, *No, NoPoints, Type*

Time1, Amplitude1

Time2, Amplitude2

to

TimeN, AmplitudeN where N is NoPoints

Define the time history in terms of a defined curve of amplitude verses time. The maximum number of point is 1000. This may be extended by using the MAXTIME command.

No	Curve Reference Number	
NoPoints	Number of points on curve	
Type	Curve type	
	1	Time by value
	2	Time by increment
TimeN	Time (s)	
AmplitudeN	Amplitude Factor	

In some cases it may be more convenient to read the data from an external file. This can be done using the **TFILE** command.

The **SCURVE** command is used to define a ramped profile that flattens off. The ramped portion starts with a zero slope and finishes with a zero slope. The flat sections continues to t=1E6. The ramped potion is based on the equation Δ = FACT.(1 - cos(π.t/T) / 2 . The **SCURVE2** command can also be used for this purpose and is the recommended option. It is exactly the same but does not require the *NoPoints* to be defined and does not involve secondary curve interpolation (curve points match solution time step).

SCURVE, *No, FACT, T, St, NoPoints*

SCURVE2, *No, FACT, T, St*

Define the time history in terms of a defined curve of amplitude verses time. The default maximum number of point is 1000. This may be extended by using the MAXTIME command.

No	Curve Reference Number
FACT	Curve factor at end of ramp time - Final amplitude factor
T	Ramp rise time(s)
St	Time at which the ramp starts
NoPoints	Number of points on curved portions

The **RCURVE** command is used to define a ramped profile that starts with a zero slope and finishes with a constant slope. The gradual rise time until the constant slope is obtained is defined by T. The ramped portion is based on the equation: $\Delta^2/\Delta t^2$ = S/T(1 - cos(2π.t/T). The constant slope section continues to t=1E6. The **RCURVE2** command can also be used for this purpose and is the recommended option. It is exactly the same but does not require the *NoPoints* to be defined and does not involve secondary curve interpolation (curve points match solution time step).

RCURVE, *No, S, St, T, NoPoints*

RCURVE2, *No, S, St, T*

Define the time history in terms of a defined curve of amplitude verses time. The default maximum number of point is 1000. This may be extended by using the MAXTIME command.

No	Curve Reference Number
S	Final slope (Δ/Δt)
St	Time at which the ramp starts
T	Time from amp start to Final Slope Point
NoPoints	Number of points on curved portion

TFILE, *No, Type, FileName*

Defines the time history in terms of a defined curve of amplitude verses time. This is similar to the TCURVE command but the data is read from a defined text file. The default maximum number of point is 300. This may be extended by using the MAXTIME command.

No Curve Reference Number

Type Curve type

1 Time by value

2 Time by increment

FileName The name of the file containing the time history data. Note that the full file specification must be defined i.e. name and path. If file specification is entered as MODEL, then the default model name will be assumed (this makes the command portable i.e. model independent - which is highly desirable when copying models - use a U prefix in the file extension to ensure data is archived with the model e.g MODEL.UDataID).
The file format is as below. The number of points will be interpreted form the number of records in the file.

Time1, Amplitude1

Time2, Amplitude2

to

TimeN, AmplitudeN

The **HCURVE** command will generate a time history based on the following equation for a damped harmonic signal. It does by generating an internal TFILE which is then interpolated during the solution.

F(t)=(C1.sin(C2.t+C3) + C4.cos(C5.t+C6)).EXP(C7.t)

HCURVE, *No, C1, C2, C3, C4, C5, C6, C7, C8, C9, C10, C11*

This command will generate a time history based on the following equation for a damped harmonic signal. If the End time is larger than the Curve time the amplitude at points beyond the Curve is taken to be zero i.e. the curve is cut off.

No	Curve Reference Number
C8	Curve start time
C9	Curve time
C10	End time
C11	Number of Points

C2, C3, C5 & C6 are in radians

F(t)=(C1.sin(C2.t+C3) + C4.cos(C5.t+C6)).Exp(C7.t)

The **HCURVE2** command will generate a time history based on the following equation for a ramped sinusoidal signal. It does by generating an internal TFILE which is then interpolated during the solution. See also **RSCURVE**.

F(t)=(C1.sin(C2.t+C3)).C4.t If C4.t>C5 C4.t = C5

HCURVE2*, No, C1, C2, C3, C4, C5, C6, C7, C8, C9*

This command will generate a time history based on the following equation for a ramped sinusoidal signal. If the End time is larger than the Curve time the amplitude at points beyond the Curve is taken to be zero i.e. the curve is cut off.

No Curve Reference Number

C6 Curve start time

C7 Curve time

C8 End time

C9 Number of Points

C2 & C3 are in radians

The **RSCURVE** command will generate a time history based on the following equation for a ramped sinusoidal signal. This is similar to **HCURVE2** but does not utilise an internal TFILE and thus requires less parameters.

F(t)=(C1.sin(C2.t+C3)).t/C4 If t/C4>1 then t/C4 = 1

RSCURVE*, No, C1, C2, C3, C4, C5*

This command will generate a time history based on the following equation for a ramped sinusoidal signal.

No Curve Reference Number
C4 Ramp rise time
C5 Curve start time

C2 & C3 are in radians

This **MAXTIME** command will change the default maximum number of points that can be defined in the TCURVE command.

MAXTIME, *MaxPts*

MaxPts Maximum number of points

-o-

10 Batch Operation

10.1 Batch Operation

FS2000 uses a number of distinct processes during the course of analysis. The analysis of a load case or the creation of formatted results data are typical examples of such processes. The activation and control of these processes is often achieved by Direct User Interaction within FS2000 i.e. using the various menu options and related data selection forms.

For large models or models with numerous load cases or load case combinations it is often more convenient and almost essential to activate and control of these processes using Batch Mode command line instructions.

Command line instructions can be incorporated into batch script files (Batch Control Files) to enable the user to set up repeatable multiple module processing that eliminates user intervention.

In FS2000 batch files are identified using the <modelname>**.BRid** where id is the specific batch file identifier. The default batch file is **BRM.**

The FS2000 Batch Process Control Module reads and actions the command line instructions in the Batch Control Files. This is a separate Windows application and can be used to run different models to FS2000.

Whenever a batch file is opened in the Batch Process Control Module that batch file becomes the **Active Batch** file.

-o-

10.2 Batch Control Files

Command line instructions are used to initiate the different analysis processes.

A Batch Control File is simply a text file that lists the Command line instructions. The following is a simple batch file for generating hydrodynamic wave loading on a structure. The batch generates the wave load cases and solves and post-processes 15 cases.

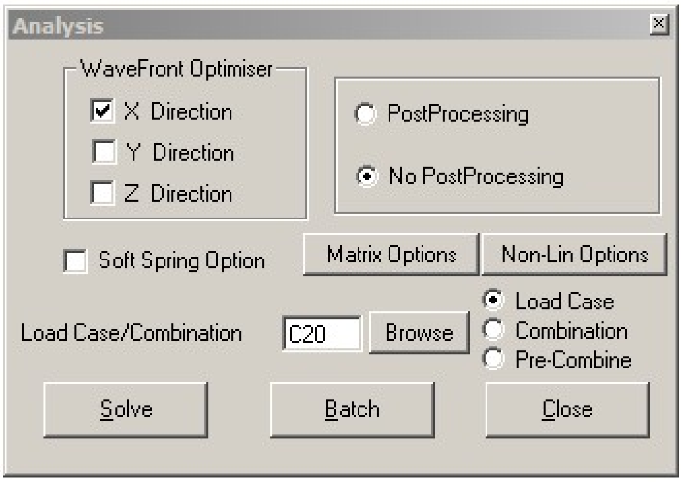
```
WAVELOAD WAV
WAVESORT Y
LOADA C201/0/0/
OFRAME
POST6 C101-116
MOUT6 -1/0/0/3/0/1/1/8/0/1/1/0/
ISO19902 101-116/2/8/1/0/1/1/0/None/0/
```

The file may be created by one of the following methods

- Append Method. Appending the commands from the module control forms of FS2000.
- Using the line edit buttons of the Batch Process Control box.
- Using a text editor. Not recommended, error prone and requires knowledge of commands but good for line movement.

The Append method is the recommended method. Using this method there is no requirement to fully understand the command line instructions

When using FS2000 interactively it will be seen that some input boxes have a Batch button. The boxes with batch buttons indicate processes that may be initiated using command line instructions. The following shows an analysis form.



By clicking the Batch button prior to clicking the interactive process button e.g. Solve button, the interactive procedure may be directly copied to the BRM file.

When the Batch button is clicked then the current options are converted to command line switches and the command line appended to the **<ModelName>.BRM** file. Appended commands are always appended to the **<ModelName>.BRM** file or the last control file opened in the FS2000 Batch Process Control i.e. the active batch file.

The Batch button in the above form would append the following command lines to the batch file.

```
WAVESORT X

LOADA C20/0/0/
```

```
OFRAME
```

A batch control file can use any file name but to unsure that the batch file is archived with the model it must have the following name **<ModelName>.BR"id"** where **"id"** is identifier. File extensions such as **<ModelName>.<Name>.BR"id"** are permissible.

-o-

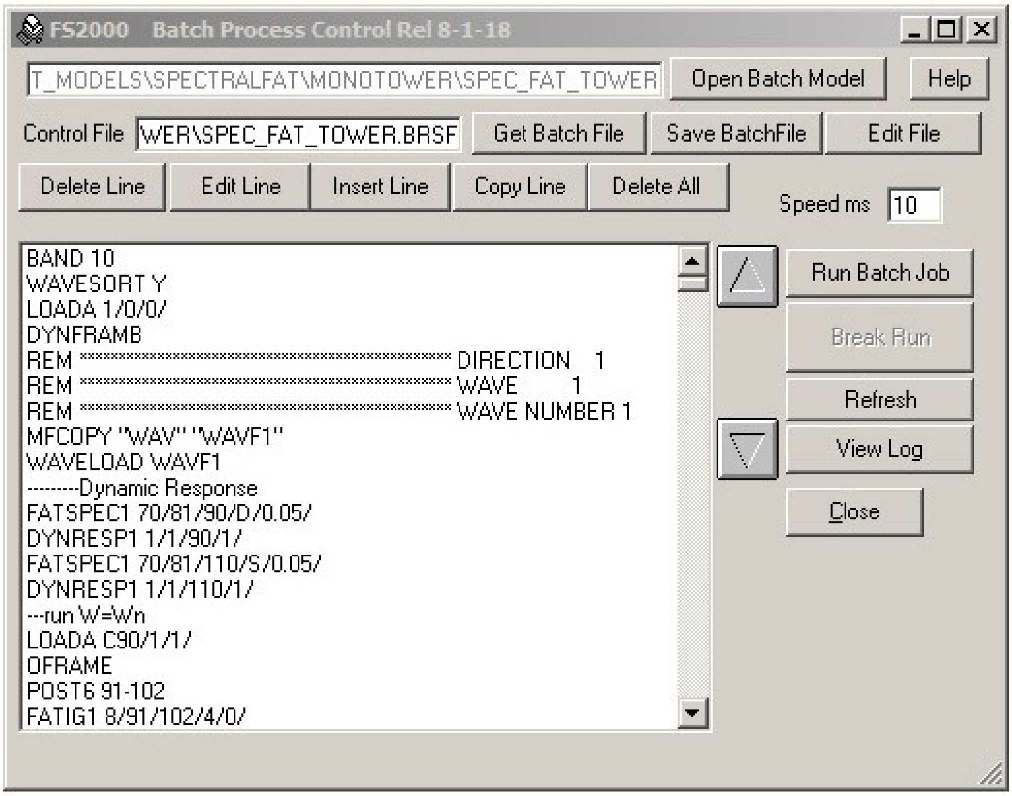
10.3 Batch Process Module

The Batch icon in the FS2000 program group or the Batch button in FS2000 is used to start the Batch Process Application.

The application can also be started and be made to run the current batch file by using the following Window's command line and command line argument '**BATCH** RUN'.

This application is used to interpret the batch list and initiate module processing. See Command Line Instructions for module command option switches.

When the application starts the following box will become visible.



Open Batch Model is used to select the default batch model. This model need not be the same as the Interactive FS2000 default model.

Get Batch File is used to load an existing control file into the Process Control List.

Save to File is used to save the current list to the file entered in the Control File box

Edit File will start FS-Edit and load the current batch file for editing

Speed ms is used to set the time period between command line initiation. The default is 500 but the may be reduce to a minimum value of 1 depending upon the host machine speed.

Run Batch Job is used to submit the Control List for processing. The process will start at the line shown highlighted. The progress of the job can been seen in the list. When a process is completed it will be preceded by a *.

Break Run is used to halt the batch run. It will stop when the current process is complete. Run Batch button may be used to restart the batch processing.

Refresh is use to remove all the * markings from previously processed lines. This enables the commands to be reprocessed.

The buttons at the bottom of the list box provide list editing functions.

Delete Line Deletes the current line

Edit Line Loads the current line into the Command Line Editor

Insert Line Inserts a line before current line and loads the Line Editor

Copy Line Copies current line and loads the Line Editor

Delete All Remove all lines from the command list

The Arrow buttons on the RHS of the list are used to move lines within the list.

Changes to the list will not be saved unless the Save to File option is used.

The **View Log** button is used to to view batch log file. This log shows any run time errors.

-o-

10.4 Run Time Error - Log File

When running in batch mode any recognisable errors are written (appended) to a log file called **<ModelName>.LOG** located in the model directory.

The log file also shows when the process was started and when it was finished.. Use the **View Log** button to view the current log files.

-o-

10.5 Command Line Instructions

This section 10.5 summarises the command line instructions used by the most commonly used modules processing modules. Note that the command are not case sensitive.

The Help files for the various FS2000 modules list the details of the batch command lines for that module.

General Control Commands

The commands listed below are used to provide additional model control during processing.

Stop Causes the processing to halt when it reaches the stop command

Model <ModelName> Changes the default batch model. <ModelName> must include path.

Insert <ControlFileName> Inserts a control file into the control list. Include full path and extension. see example below

BInsert <ControlFileExtension> Inserts a control file into the control list from the current model. e.g. **binsert brm2**

Clear Removes all Command Line instructions from the list that precede the Clear command.

MFCopy Copies model files C1 to C2. Where C1 and C2 are the model file extensions. e.g **mfcopy "UFO1" "UFO"** could be used to reset the Frequency/Buckling solution's global model options.

MFRen Renames model files C1 to C2. Where C1 and C2 are the model file extensions. e.g **mfren "UPN" "UPN_temp"** could be used to temporally disassociate pile springs from pile elements.

MFDel Deletes model file C1. Where C1 is the model file extensions. e.g **mfdel upt** could be used to delete the analysis option file. It supports the use of multiple-character (*****). e.g. **mfdel UWData*** would delete all DyNoFlex time history wave load association files.

Copy Copies file C1 to file C2. C1 and C1 must be in quotes and separated a space character and be valid file specifications e.g **copy "C1" "C2"**. A typical file specification would be c:\projects\fs2000_models\modelname.l4

Run Executes any Windows executable (.exe) file.

RunBat <Batch File Ext> Will start a new Batch Control Process and start running (concurrent) the batch file.e.g. **runbat .brmx** Note that a decimal point character (**.**) is required to precede the batch file ext. It is generally not possible to run concurrent batch processes because of open file conflicts.

Exit Closes the Batch Process Control Application

Two examples of batch file use to run successive models are:

> **model** c:\exp7\modelA
>
> **clear**
>
> **insert** c:\exp7\modelA.brm
>
> **model** C:\models\modelB
>
> **clear**
>
> **BInsert** br1

If the clear commands are omitted the final control file will include all command previously activated.

To merge files use the **insert** and **stop** commands and then manually save the combined control list.

-o-

Bandwidth/Wavefront Optimisation

a) Bandwidth Optimiser

BAND C1\C2\----\C8

C1 to C8 defines start nodes (usually only one corner node is defined).

b) Wavefront Optimiser

WAVESORT C1

C1 is the sort direction ie X, Y or Z

-o-

Analysis

Solution Preparation - LOADA

The solution commands must be preceded the LOADA command. The LOADA module processes the load case(s) and creates the secondary definition used by the solution modules.

LOADA C1/C2/C3/C4/C5/

C1 Load Case or Load Case Combination to be analysed

If the number is preceded by a **C** or **c** the analysis module will :
Solve multiple load cases in combination in succession (3-D Standard Solution Only) or,
Solve combinations as a time history solution (Non-linear solutions)

If the number is preceded with a **P** or **p** then the load cases in the combination will be merged (including all load factors) and solved in one case.

If the number is preceded with a **CP** or c**p** then the cases in the combination will be interpreted as combinations will be merged (including all load factors) and solved in one case.

In a frequency solution or DyNoFlex dynamic analysis the case represent the the Mass Case.

C2 Soft Spring Option

0 Inactive   1- Active

C3 Only Used by the *3-D Standard Solution* and *Eigen Frequency & Buckling Solutions*

3-D Standard Solution

0 No restart   1- Save restart file   2- Read Restart file

Eigen Frequency & Buckling Solutions

P-Delta Case Number

C4 Element Loads – Loads applied as nodal loads (DyNoFlex Option)

0 - Inactive 1- Active

C5 Solution Analysis Type

0 - Linear Analysis 1 - Static Non-linear analysis   ID - DyNoFlex Analysis Option ID Number

******* WARNING******* -** If C5 is not correctly set for the appropriate solver then the strain effects in beam type elements will be incorrectly processed.

If C2 to C5 are omitted default values of 0 will be used.

Section 6.2 present an overview of the operation of different solution options.

Solution

The different analysis modules are started by entering the appropriate analysis command. The ones listed below are the most commonly used solution modules.

OFRAME 3-D Standard (Primary Linear Solver) Solution (A global option file is employed for non-linear capability settings (**<modelname.UPT**))

PILE3D Static 3-D Incremental Non-linear Solution

DYNFLEX FS-DyNoFlex Dynamic Incremental Non-linear Solution (Case dependent option file is also employed)

DYNFRAMB Eigen Frequency/Buckling Solution (a global option file is also employed for the solution settings (**<modelname.UFO**))

The command lines for other solvers e.g. modal analysis can be found by using the Batch button and inspecting the subsequent batch file entry.

A typical entry for linear analysis would be:

```
LOADA 1/0/0/
OFRAME
```

A typical entry for static incremental non-linear analysis would be:

```
LOADA C100/0/0/0/1/
PILE3D
```

A typical entry for static DyNoFlex Non-linear analysis would be:

```
NLPARAM 10
LOADA C10/0/0/0/10/
DYNFLEX
```

A typical entry for dynamic DyNoFlex Non-linear analysis would be:

```
NLPARAM 10
LOADA 1
LOADA C10/0/0/0/10/
DYNFLEX
```

A typical entry for an Eigen analysis including P-Delta effects would be:

```
LOADA 1/0/1/0/
DYNFRAMB
```

-o-

Dynamic Interpretation

Dynamic interpretation is when FS2000 is used to modify a model using batch control. Its purpose is to enable the uses to modify models between load case solution runs. It does this by re-interpreting the MDL model definition files and re-saving the model. The following restriction/recommendations apply when using dynamic interpretation.

Do NOT Change Model Size

The model size parameter generally should NOT be change i.e. the maximum number of nodes, elements, couples and restraints must be maintained constant since these values are used to index the results files.

Note: Model sizes maybe changed providing result data is only accessed when the model is in the same size as when the result cases were created. UR Plot data can be accessed with changed model sizes. However, it is ***strongly*** recommended not to change the model size as this will ovoid data access problems if the FS2000 GUI (graphical interactive interface) has not been re-opened with model in its current size status.

Always Maintain Model Size

Any number of Elements and Spring Couples can be deleted providing they are not the last labeled entity in the model. If they are the last their deletion would change the model size. Use the EDEL and SCDEL commands to remove elements.

Property data can be changed without restriction (do not remove any).

Restraints cannot be changed. If the restraint condition of the model is required to be changed use Node to Ground Couples to restrain the model and then modify the stiffness properties of the couple or delete the couple

Note that only MDL files can be interpreted.

Warning - When using Dynamic Interpretation the model in the FS2000 GUI (graphical interactive interface) may not show the current status of the model following the changes undergone in Batch. To view the model it its current status simply re-open the model using either the Open command or select from recent model list (Primary TASK:File menu).

Linking Result Cases to the the Model Condition

When viewing output results it is essential that the result cases can be linked to a model status. All text result case output shows:

- Date & Time the model was saved
- The model **Title**, the model **Ref** and the model **Desc**

The **Title, Ref** and **Desc** are command line instructions and can be included the the UM file. These can therefore be used to identify the model condition.

- The REF descriptor appear in all output so this would be a good choice
- The DESC descriptor appears in all text output, but not in any graphic plots
- The TITLE descriptor appear in all output except graphical results i.e. printing graphics from the Output TASK

The model data could also be formatted after each WINFRAM command but this is not essential because the Batch file and the UM file detail the model changes.

Post-Processing

It recommended that all post-processing of result cases be undertaken when the model is the same state as when the solution was undertaken.

Interpretation Commands

The following command line will interpret the mdl file and save the model. Note that results are not deleted during the save operation

WINFRAM I (or i)

This command line would normally be preceded with the following commands which runs the Merge Model utility. The Merge Model utility will copy model files or merge two command files to form an MDL file. The two files to be merged must have the same filename as the model. It is also recommended that the file extensions used are UM"n" types since these will be archived with the model.

MODMERGE C1/C2/C3/

C1 Process Action

- C Copies File C2 to File C3 (for C3 use file extension starting with B when backing MDL files)
- M Merges File C2 with File C3 to produce a MDL file

C2 File extension of command file 1 e.g. **um1**

C3 File extension of command file 2 e.g. **um2**

Typical batch files using the above command is shown below.

Two examples of using batch file to modify the model are given below. Of the two, **Strategy B** is strongly recommended.

Creating Model file for Interpretation (UM files)

The easiest way to create a UM file is the save the model in the required condition and then use FS-Edit to copy the sections of the MDL file that are required to make the changes to the model to the UM file.

Testing a UM Command File

The effect of UM command file can be easily checked by interpreting them in the Model Defintion TASK using the **File:Interpret File** command.

Strategy A

Strategy A is the easiest one to set up as the basis approach is to always backup the base model mdl file to a backup file called ubak (note that ubak contains the filename therefore re-save after a model Save-as). The modifications to the model are then implemented by UM files that simple make the required changes to model based on the model being in a specific state. This approach makes the modifying UM file simpler because they only have to have the commands that make the current changes.

The disadvantages of this approach are:

- The batch WINFRAM commands must always be run in succession to get the model to a specific state
- If the model is not always backed up it in its initial state, model data can be very easily lost

LOADA 1/0/0/
OFRAME
POST6 1

Copies the mdl file to bak

modmerge c/mdl/ubak/

Appends file um1 to file mdl

modmerge m/mdl/um1/

Interpretes the mdl file & saves model

winfram I
LOADA 2/0/0/
OFRAME
POST6 2

Merges um3 & um4 to produce a mdl

modmerge m/um3/um4/

Interpretes the mdl file & saves model

winfram I

LOADA 2/0/0
OFRAME
POST6 2

Returns the model to its original condition

modmerge c/ubak/mdl/

Interpretes the mdl file & saves model

winfram I

Strategy B

Strategy B is of similar construction but the use of a backup file is eliminated The modifications to the model are again implemented by UM files but in this strategy they not only make the modifications to the model they also remove the modification made by all other WINFRAM commands in the batch. This approach makes the modifying UM files more complex because they not only have to make the current changes but have to remove ALL other temporary modifications to the model.

The advantages of this approach are:

- No back up file required - less likely to lose model data
- The analysis can be stated at any point in the batch file
- The current model status can be changed by simply running the appropriate section of the batch file and re-opening the model

LOADA 1/0/0/
OFRAME
POST6 1

Appends/Merges file 'um_LIFT' to the 'mdl' file.

modmerge m/mdl/um_LIFT/

Interpretes the mdl file & saves model

winfram I
LOADA 2/0/0/
OFRAME
POST6 2

An optional STOP command can be use to see the current model status and results in the GUI ie re-open the model to see the current condition.

STOP

Appends/Merges file 'um_IN_PLACE' to the 'mdl' file.

modmerge m/udl/um_IN_PLACE/

Interpretes the mdl file & saves model

winfram I
LOADA 2/0/0
OFRAME
POST6 2

-o-

Dynamic Load Case Merging

Load cases can be merged from within the Load Definition TASK in FS2000. In this TASK load cases may be merged by individual selection or by reference to a load case combination.

Dynamic Load Case Merging is when a load case is formed from a load case combination using command line operation.

The reason this would be done is so that precise management and QA check on complex load cases can be achieved by splitting a load case into a number of component load cases.

This has the advantage that the merged load case can be updated at analysis time when running in batch i.e. if a component load cases is changed this change will be incorporated into the merged case at analysis time.

Note that running a Pre-Combined load case will achieve the same objective as Dynamic Load Merging.

Load Case Factors are used in the merge process

WINFRAM LC/C1/C2/C3

C1 Load Case Combination Number

C2 Recipient Load Case

C3 Load Case Title (Optional - Default is Comb No C1)

-o-

Post Processing

POST6 C1/C2/C3/C4/C5/C6/C7/

C1 is the raw results file or combination (prefix combinations with the letter C).

e.g. 2 for Load Case 2 or C6 for Load Case Combination 6.

Ranges may be processed e.g. 1-10 would process Load Cases 1 to 10 and C7-15 would process combination C7 to C15.

If cases do not exist when processing ranges a warning will be given and the process will continue to the next case.

Load Case combination can be combined using an additional C prefix. When this is done the subject conbinations must contain the combinations to be combined not load cases

e.g. CC200 would combine the results cases defined in the combinations listed in C200 using the factors defined in both the participating combinations and the subject combination.

C2 to C5 are optional parameters associated with FE stress averaging

C2 Average stress switch. 0 is Off :1 is On

C3 Group SET for identifying averaging group

C4 Lowest group number to be averaged

C5 Highest group number to be averaged

C6 to 7 are optional parameters associated with FE stress output coordinate systems

C6 Shell Stress to Global. 0 is Off :1 is On

C7 Group SET for identifying output co-ordinate systems

-o-

Results Output - Single Case Mode

OUT6 C1/C2/C3/C4/C5/C6/C7/C8/C9/C10/C11/C12/C13/

C1 is the processed results case ID number.

Ranges may be processed e.g. 1-10 would process Results Cases 1 to 10 . If cases do not exist when processing ranges a warning will be given and the process will continue to the next case.

C2 Displacements 0-off, 1-on

C3 Element Forces 0-off, 1-on

C4 Number of locations on span for forces . 0-default is 3

C5 Spring Forces 0-off, 1-on

C6 Reactions 0-off, 1-on

C7 Stress Output 1-Max Point, 2-All points, , 0-no stress output

C8 Number of locations on span for stress output . 0-default is 3

C9 Stress limit for printing output stresses. Default 0.4

C10 Group SET to read

C11 Group Limit/Restriction

C12 Groups Only switch

C13 Subcase Name

If C1 is preceeded by **-** the output will be in Engineers Units.

For C10-C12 see note on using Groups for output at end of section

-o-

Results Output - Multiple Case Mode

MOUT6 C1/C2/C3/C4/C5/C6/C7/C8/C9/C10/C11/C12/C13/

C1 is the Results SET number

C2 Displacements 0-off, 1-on

C3 Element Forces 0-off, 1-on

C4 Number of locations on span for forces. 0-default is 3

C5 Spring Forces 0-off, 1-on

C6 Reactions 0-off, 1-on

C7 Stress Output 1-Max Point, 2-All points, , 0-no stress output

C8 Number of locations on span for stress output. 0-default is 3

C9 Stress limit for printing output stresses. Default 0.4

C10 Group SET to read

C11 Group Limit/Restriction

C12 Groups Only switch

C13 Subcase name

If C1 is preceeded by **-** the output will be in Engineers Units.

If C4 or C8 are preceeded by **-** the output will be restricted to the maximum and minimum values for forces or stresses

For C10-C12 see note on using Groups for output at end of section

-o-

Unit Ratio Sort Utility

URSORT C1/C2/C3/C4/C5/C6/C7/

C1 is the unity ratio type (VM, MR or JR)

C2 is the Results Case or Results SET number. (prefix with S for SET number)

C3 is type of type sort required.

0 - By Element

1 - By Unity Ratio value

2 - By Group and then by Unity Ratio value

3 - By Group with only the maximum UR per group listed

C4 is the unity ratio output limit

C5 is the element group SET

C6 is the OUTPUT Subcase name - Optional

C7 is the INPUT Subcase name - Optional

-o-

Definition Data Formatting

IN C1/C2--------/C16/

Because of the length of the command line it is reccomended that this line be inserted from FS2000 using the Batch button in the Format Defionition data form.

-o-

Displaced Geometry - DNF

The **DNF** utility creates nodal definition data in which all node co-ordinates of the model are based on the their (non-deformed geometry + the translational displacement) from the current results case.

The DNF utility can also be run in interactive mode. The displacements can also be extracted from the Eigenvalue solution mode shape displacements but only in interactive mode.

DNF C1/C2/

C1 is the result case number
C2 is the output file ID characters

The geometry file created will be, <modelname>.UDGEOM'C2'

-o-

Pile Print Utility

FSPRINT C1/C2/C3/C4/C5/C6/C7/

C1 Full Filename for file to be printed

C2 Section description for Page Headers

C3 Sub-Section Number

C4 Start Page Number

C5 Merge Flag - Always set to zero (System Flag)

C6 Use B for batch. When printing all headers etc will refer to the batch default model

C7 Print to File - Output will be printed to a file textfile

-o-

Member Design Code Checking

AISC C1/C2/C3/C4/C5/C6/C7/C8/C9/C10/

C1 is the processed results case ID number.

Ranges may be processed e.g. 1-10 would process Results Cases 1 to 10 . If cases do not exist when processing ranges a warning will be given and the process will continue to the next case.

C2 Text file output format 2-Summary Report, 3-Short Report

C3 Number of location on span for code check (Setting to -3 will activate 21pt check)

C4 Allowable stress increase factor.

C5 Unity check ratio limit for output.

C6 Group SET to read

C7 Group Limit/Restriction

C8 Groups Only switch

C9 Subcase name

C10 Depth of water above model origin (Hydro collapse check - if code applicable)

For C6-C8 see note on using Groups for output at end of section.

The above shows the command line to run the API RP2A/AISC code checker. For other design codes use the following command arguments.

Command	Code
AISC	API RP2A/AISC - ASD
AISC5	AISC A360
CODE	BS 5950
APILRFD	API RP2A LRFD
ISO19902	ISO 19902
EC3	EC3 Euro Code

-o-

Tubular Joint Design Code Checking

PUNCH C1/C2/C3/C4/C5/C6/C7/C8/C9/C10/

C1 is the processed results case ID number.

Ranges may be processed e.g. 1-10 would process Results Cases 1 to 10 . If cases do not exist when processing ranges a warning will be given and the process will continue to the next case.

C2 1- Full Report 2 - Summary Report

C3 Unity check ratio limit for output

C4 Allowable stress increase factor

C5 Code to be used. Enter API or DOE

C6 Operating condition E for extreme, O for Operational

E or O is only used in DEn codecheck.

C7 Group SET to read

C8 Group Limit/Restriction

C9 Groups Only switch

C10 Subcase name

For C7-C9 see note on using Groups for output at the end of section.

-o-

Using Groups to Sort Output

G1 Group SET to read

G2 Group Limit/Restriction

G3 Groups Only switch

G1 defines the group SET to be loaded. If a SET is loaded then all node and element labels will be accompanied by their respective group attribute.

G2 defined the Group Limit\Restriction used by the folowing options. If **G2** is positive then output will be restricted to only those entities with the same group number as defined the **G2** (zero value indicates that all data will be shown). This is a restricted process option.

It **G2** is negative the Grouped output will be sorted by group up to the group limit defined by **G2.**

If **G3=1** and **G2 is negative** then entities not assigned to groups or entities assigned to Groups greater than defined by G2 will not be output. This is a restricted process option.

If **G3=0** and **G2 is negative** then entities not assigned to groups or entities assigned to Groups greater than defined by the **Group Limit/Restriction** will be output in label order following the sorted groups. All data is processed with this option.

-o-

11 Tutorials and Verification Examples

11.1 The Basic Analysis Procedure

A considerable number of menu commands are available for the different stages of analysis. To provide logical access to these commands the analysis process has been divided into separate processes i.e. distinct stages of analysis. These processes are termed TASKS.

Tasks are initiated and terminated by command selection in the TASK menu.The main title bar indicates the current task.

For each task, a different set of menu commands become available. Some menu commands are available for all tasks. These are termed global menus or commands. The menus to the left of the TASK menu in the top menu bar are Global Menus. Those that appear to the right are TASK Related Menus (except Window & Help).

When quitting the definition TASKS data should be saved. It is essential that any changes to the model data be saved otherwise the data will be lost as memory is reallocated as the program returns to the Primary TASK. Definition TASKS are:

- Model Definition
- Load definition
- Design Parameters

The TASK menu and the basic functions of the tasks are:

Primary
Used for general model management i.e. opening, archiving, deleting models etc.

Model Definition
Provides all the commands used to define a model.

Load Definition
Provides all the commands used to define load cases on a model.

Design Parameters
Enables design parameters associated with code checkers to be defined e.g. effective lengths. Not required to be used if the optional code checking modules are not used.

Analysis
Used to submit a model for solution

Output/Results
Used to interactively inspect results cases in graphical environment. Multiple Viewports may be used to view different parts of the structure and view different results cases. Creates formatted reports files for definition data and results data. Activate the optional design modules.

A Simple Example

This worked example is given in A Simple Worked Example to demonstrate some basic program procedures by running an existing model.

-o-

11.2 A Simple Worked Example

This worked example is intended to demonstrate some basic program procedures by running an existing model. It does not demonstrate how to define model, its purpose is to show a typical procedural approach to be followed when using the program interactively.

The example to be used is one of the worked examples EXP3D2 described in Appendix 3.

To invoke the Help file press the F1 key. This will show context sensitive help that will explain the process currently active. Use this feature to find out what the program controls do and how they operate.

Start FS2000

Step 1 Open a model from Archive format

*From the **File** menu select **Open**. The Open box will appear*

Change the List of files type from Std Model to Archive Models. Use the lists to open the model EXP3D2.MOD in the fs2000\examples (default location is c:\Profrajm Files\FS2000) directory. When selected the Archive Utility - Model Recovery box will appear.

*Select a suitable directory for the model. Do not use FS2000 or its sub-directories, use a different directory. If the recipient directory does not exist it will be created. Click the **Recover** button.*

The model will be drawn on the screen.

Step 2 Run the Analysis (Solution)

*From the **TASK** menu select **Analysis**.*

*From the **Solution** menu select **3-D Light** or **3-D Standard**. The Analysis form will appear.*

*Press the **Browse** button and select Load Case 1*

*Press the **Solve** button. The solution will now be completed and the results case post-processed. Press the Close button.*

Step 3 View the results interactively

*From the **Task** menu select **Output/Results**.*

*From the **File** menu select **Open Proc. Results**. Select Results Case 1. The results are now loaded into View 1.*

*From the **Plots** menu select **Bending Stress**. The bending stress plots will now be drawn.*

*From the **Insp** menu select **Stresses**. The stress Inspection box will appear.*

*Use the Element Query button **[E?]** and click the centre of an element. When selected the element data will appear at the bottom of the screen.*

*Press the **Use Pick** button to list the element stresses.*

Close the Results Inspection

Step 4 Create a Results Output File

*From the **StdOut** menu select - **Individual Results Format**. The Results Option Output will appear.*

*Use the **Browse** button to select the Results Case. Change the stress ratio limit to 0. and then press the **Create Outpu**t button.*

*Press the **View** button to load the results file viewer and click the **Load/View File** button to view the formatted results file.*

Close the File viewer.

Step 4 Format the Definition Data

*From the **Data** menu click the **Format Definition Data** command. Accept the defaults and click the **Format Data** button. When complete click the **View** button to see the formatted data.*

Close the file viewer.

Step 5 Print out a Report

*From the **Data** menu click the **Report Collation - Dat**a Selection command. The Collate Report form will appear. Click the **Add Def'n Data** button. This will add the previously formed formatted definition data file to the **Print List**. Now click **All Results to List** button. This will add Results Case 1 to the list and any other results case, if they exist.*

Nn TASK or create and solve a different load case etc.

More Detailed Instructions

The Help system contains Instruction Cards that demonstrate the basic operational features in more detail than the above example. To start the cards click the Help command from the FS2000 menu bar.

This will start the Help file. From the contents page of the Help click Instruction Cards. Close the main Help and follow the instructions.ow go back and experiment with different features of the program, modify the model in the Model Definition TASK

-o-

11.3 Instruction Cards

This Help file will demonstrate the use of the program by displaying Instruction Cards for various analysis related tasks and topics.

-o-

11.4 Verification Examples

To demonstrate the use of the program and provide a degree of validation, structural models of the examples described in this section have been included. The Help files for the optional FS2000 modules have validation examples relating to that module.

The example listed in this Help file only require the capabilities of the Core Module.

The examples relating to all the Help files are in archive format i.e. MOD files and can be found in the Examples sub-directory of FS2000 System Directory (default location if C:Program Files\FS2000).

Sample solutions (spot checks) for each model are stated along with a reference solution obtained from a different solution source.

Only the model definition files are on the disc i.e. in archived form. Archive files do not include analysis results files. To view analysis results requires that the model be re-analysed. If you are new to computer aided structural analysis it is suggested that the input data from the model be printed out. The input data may then be inspected to see how the structures have been modeled.

A useful exercise is to modify the models in some manner and re-analyse to assess the effect. Modifications such as restraint condition or type of loading would be illustrative.

Verification Document

In addition the to above described examples, a "stand alone" verification document that contains a wide range verification examples is available from AES.

Many of examples in the verification require use of the non-linear solvers.

Batch Operation

Included with each of the examples is a batch file for running the models in batch mode. The batch files illustrate the flexibility and use of batch operation.

-o-

11.4.1 Verification Examples - Beam Models - Beam Models

The EXP2* modelS were originally created for use in a 2-D solver that is not now obsolete. Because of this some of the model may require the Soft Spring option to be active for successful solution. The batch fileS do have this option active. This of course can be avoided if the zero stiffness propertiy entries are replaced with non-zero entries.

EXP2D1 Built-in Beam
This model is a built-in beam subjected to global linearly varying distributed loading and global mid element point loads. The model has two load cases for each load type. Shear deflections are ignored. The reference solution is from the std formula in the CONSTRADO Steel Designers Manual.
A = 7.58E-3 m2I = 6.09E-5 m4E = 205E9 N/m2μ = 0.3

	End Moment	Mid Span Moment	Mid Span Defln
Load Case 1	35.156 kN	6.51 kN	3.129 mm
(Ref Soln	35.156 kN	6.51 kN	3.129 mm)
Load Case 2	27.5 kN	12.5 kN	4.272 mm
(Ref Soln	27.5 kN	12.5 kN	4.272 mm)

EXP2D2 Continuous Beam
In this model the effect of shear (shear area ratio = 1.2) deflections are included. The loads are applied in one load case. The reference solution is from the user manual of SDRC's FRAME program.
Elements 1 & 2 A = 80 ins2I = 1000 ins4
Elements 3 A = 115 ins2I = 1000 ins4
E = 1000μ = 0.25
Deflection at Node 2 = -1.571E-1 (Ref Soln = -1.571E-1)
Moment at Node 4 = -2.273E2 (Ref Soln = -2.273E2)

EXP2D3 Beam with Elastic Supports
The spring supports were modelled using beam elements. In these elements only the beam area was specified (e.g. a spar element with k=AE/l). Shear deflection effects are included. An end moment release was used to model the hinged column to beam connection. Ref Soln is from the COSMOS user manual. This model used the Soft Spring Option (originally a 2-D model)
Beam A = 0.125 ft2 I = 0.263 ft4
Column A = 0.175 ft2 I = 0.193 ft4
E = 4.32E6 μ = 0.3
Deflection at Node 5 = 1.079E-3 in x and -4.82E-3 in y
(Ref Soln at Node 5 = 1.079E-3 in x and -4.83E-3 in y)
Reactions at Node 4 Mz = 30 Fy = 11.36
(Ref Soln at Node 4 Mz = 30 Fy = 11.36

EXP2D4 Portal Frame
The loads are applied as two load cases and combined to obtain the required factored load combination. Two methods are used. One is to pre-combine load cases 1 and 2 to produce loads case 3. The other is to combine the results of load cases 1 and 2 in the post-processor. Ref Soln is from the COSMOS user manual.
LH Column A = 3.5 I = 0.643
Beam A=4.4 I = 1.286
RH Column A = 2.79 I = 0.3215
E = 4.32E6 μ = 0.3
Moment at Node 2 28.32 (Ref Soln 28.32)
Moment at Node 4 10.67 (Ref Soln 10.67)

EXP2D5 Pitched Frame with Wind Load
This example is used in the Steel Designers Manual (pages 205 to 210) to illustrate the area moment method. Load case 1 is dead load. Load case 2 is snow load. Load case 3 is wind load. The solutions obtained from the computer are slightly different to those obtained by the less accurate hand solution. There appears to be an error in the in the example in the application of the wind loads (ambiguity in use of internal pressure).
A = 3.23E-3 m2 I = 2.348E-5 m4

E = 207E9 N/m2 μ = 0.3

Deadweight moment at Node 2	44.05(Ref Soln 44.1)
Deadweight moment at Node 3	16.34(Ref Soln 16.25)
Snow moment at Node 2	62.96(Ref Soln 63.1)
Snow moment at Node 3	23.36(Ref Soln 23.2)
Wind load moment at Node 3	4.52(Ref Soln 4.4)

EXP2D6 Thermal Expansion

This example models three wires supporting a 4000 lbs load. Two wires are copper and one is steel.The wires are subjected to a change in temperature of 10 F. The result of the analysis are the tensions in the wires. The reference solution (and problem) for this example is from Timoshenko's "Strength of Materials" Part 1, 1955.

A = 0.1 ins2 I ≅ 0

αc = 9.2E-6 ins/insFEc = 16E6 lbs/ins2
αs = 7.0E-6 ins/insFEs = 30E6 lbs/ins2

Load in steel wire	1970(Ref Soln 1970)
Load is copper wire	1015(Ref Soln 1015)

EXP2D7 Planar Frame

This example models a planar frame. The reference solution is from SDRC's FRAME user manual. The model illustrates moment releases.

Moment in E3 at N4	14.92(Ref Soln 14.91)
Axial Force in E14	1.413(Ref Soln 1.412)

EXP2D8 Mechanism with Prescribed Displacement

This example models a planar structure which includes a slideway. The slideway is modelled by defining E3 with zero axial stiffness ie area=0.The loading on the model is a prescribed nodal displacement. The reference solution is from SDRC's FRAME user manual.

Element 3 only A = 0 ins2 I = 21 ins4
Other elements A = 5 ins2 I = 10 ins4
E = 30E6 μ = 0.25

Vertical displacement of N3	-0.2968(Ref Soln -0.2983)
Moment at N5	9.639E4(Ref Soln 9.69E4)

EXP2D9 Vierendeel Girder

This example is taken from the "Steel Designer's Manual" (Page 267) where it is used to illustrate moment distribution. The results from the computer solution are not identical because of the comparison with the less accurate moment solution.

A = 1 I = 1.152E-3 E = 207E9 N/m2 μ = 0.3

Y Deflection at N3	-19.78E-3 Reactions Identical to ref. 833.33 & 666.67 kN
Moment in E2 at N2	570E3(Ref Soln 590.5E3)

EXP2D10 Pressurised Pipe

This example models internal pressure in a piece of straight pipe. The object of the model is to illustrate how the program considers endcap pressure. Two load cases are applied. Load case 1 allows free expansion of the pipe under internal pressure. Load case 2 is identical apart from a prescribed displacement at the free end which is made almost zero so as to effectively restrain the pipe and prevent longitudinal expansion. The reference solution is from Roark Table 32 1b.

This example also shows that the axial force output from FS200 in a pressurised pipe is the effective axial force and the axial stress is the true wall stress.

Free end expansion(LC1)	3.469E-4(Theory 3.469E-4)
Eff. Axial Force =0.0	True Axial Stress = 88.89E6
End Restraining Force(LC2)	1.002E5(Theory 1.005E5)
Eff. Axial Force = 1.002E5	True Axial Stress = 53.44E6

EXP3D1 Built-in Beam
This identical to EXP2D1 but the beam is inclined in both the 'y' and 'z' axis.The loading on the element is applied in the local element axis. The resulting loads in the elements are identical to those in EXP2D1. The deflections are different in the output since the deflections are always in the global axis.

EXP3D2 Stresses in an Offset Cantilever
This example illustrates the convention adopted for the evaluation of element stresses at defined stress points (defined by Ly and Lz) at a section in the span.The example is a simple determinate structure and may be easily verified. This example is useful for illustrating the effect of local rotation.

EXP3D3 Oblique Restraints
This example illustrates the use of spring/couples to provide restraints in directions other than the global directions. Node 1 is a restraint similar to a tube within a tube in which the node can slide in and rotate about its own axis. This is modelled by using two coincident nodes. One is connected to the structure and the other is fully restrained. These nodes are connected to each other using a stiff spring (K1 and K4 = 1E12) orientated to element 1. The reference solution is from SDRC's FRAME user manual.
Z Deflection at Node 2 = -.1283(Ref Soln -.1283)
Force in spring = -248.8(Ref Soln -248.8)

EXP3D4 Portal Frame
This is identical to EXP2D4 but the frame is inclined and the loads are applied in the local element axis. The resulting loads in the elements are identical to those in EXP2D4.Deflections are the components deflections in the respective axis.
Moment at Node 2 28.32 (Ref Soln 28.32)
Moment at Node 4 10.67 (Ref Soln 10.67)

EXP3D5 Simple 3D Tubular Frame
This is a model of a simple tubular frame. Four load cases have been analysed. Load case 1 is gravitation loading applied in the three principle axes. Load case 2 is nodal loads. Load case 3 is also nodal loads. Load case 4 has nodal masses with G forces that produce the same effective nodal loads as in load case 3.A reference solution was obtained using COSMOS for load cases 1 & 2.
X Deflection (LC1) at N8 1.333E-3(Ref Soln 1.333E-3)
X Deflection (LC2) at N12 7.586E-3(Ref Soln 7.586E-3)

EXP3D6 Plane Grillage
The supports are fully fixed. Two load cases are analysed. The final solution is a factored combination of case 1 and 2. The reference solution is from SDRC's FRAME user manual.
I = 1000 ins4 J = 2000 ins4 E = 30E6µ = 0.25
Defection at Node 5 -0.385(Ref Soln -0.385)
V Reaction at Node 1 98.38(Ref Soln 98.39)

EXP3D7 Planar Frame
This example is the same as EXP2D7 but instead of using end moment releases for the hinge, the hinge is modelled using spring/couples.

EXP3D8 Schwedler Dome
This is classical example of 3D Truss analysis. The reference solution is from SDRC's FRAME user manual.Beam end moment releases are applied to all elements of the model since the mode is analysed in a 3-D rigid beam model. To ensure that the model solution does not fail the soft spring option is activated.
A = 12.4 E = 29E6
X Deflection at Node 6 8.085E-4(Ref Soln 8.084E-4)
Force in Element 5 543.3(Ref Soln -543.3)

EXP3D9 Mechanism with Prescribed Displacement
This model is identical to EXP2D8 except for the slideway modelling. In EXP2D8 the slideway is modelled using an element of zero area, whereas in this example the slideway is modelled using couples orientated to the axis of the slideway.

EXP3D10 Pipework System
This example is taken from SDRC's FRAME user manual. It demonstrates how pipework systems may be analysed. In the reference solution the effects of pressure has not been included within the stiffness

analysis. This is often done in pipework analysis since many piping design codes do not require longitudinal effects of pressure to be taken into account. In FS2000 the pressure is required to be included so that hoop stress is included in the post-processor. The comparisons below are for loadcases 1 and 2 which exclude hoop stress. Load cases 3 and 4 include pressure effects.

The bends have been modelled with 4 straight elements., Generally In pipe bends (and curved beams) the most significant effect is the loss of stiffness due to the curvature. Bend or beam elements do not include this effect but the effect is may be included by using flexibility factors. In this example a flexibility factor 5.778 is applicable and is applied to the bends. Model EXP3D10_BE replaces the segmented bends using a single pipe bend elements and give similar results.
The following coefficients are used in the analysis

Bend Flexibility coefficient		5.778		
Bend in-plane Stress Intensification Factor		1.73		
Bend out-plane Stress Intensification Factor		2.076		
At Node 1 LC1	Fx=-5.482E3	Mz=4.793E5		
(Ref Soln	Fx=-5.47E3	Mz-4.79E5)		
At Node 1 LC2	Fx=-6.504E2	Mz=4.286E4		
(Ref Soln	Fx=-6.497E2	Mz-4.285E4)		
Force in Spring LC1	3.652E3		LC2	-2.157E2
(Ref Soln	3.646E3		LC2	-2.165E2)

EXPA1 Rigid Link & Tapered Beams
The example illustrates the use of tapered beam and rigid links in a simple portal frame. The reference solution is taken from SDRC's FRAME user manual.
Deflection at Node 4 6.855E-2(Ref Soln 6.496E-2)
Bending Stress at Node 2 in El 5 2.505E3(Ref Soln 2.507E3)
Bending Stress at Node 4 in El 6 2.113E3(Ref Soln 2.116E3)

EXPA2 Rigid Links & Springs
This example illustrates rigid links and spring to model a spring supported rigid base. The rigid links model the spatial offsets of the base. The reference solution is taken from SDRC's FRAME user manual.
Deflection at Node 76.267E-3(Ref Soln 6.267E-3)

EXPA3 Beam Offsets
This example demonstrates the use of beam offsets on a 2-D K braced frame. The results have been validated against a model which uses additional nodes and elements to model the offsets (EXPA3V) braces.

EXPA3		EXPA3V	
X Deflection at N4	2.116E3	X Deflection at N4	2.116E3
Moment in E1 at N3	1.047E3	Moment in E8 at N7	1.047E3
Moment in E5 at N3	8.595E3	Moment in E5 at N6	8.595E3

EXPBU_1 Eigen Frame Buckling
This example demonstrates the Eigen buckling analysis of a fixed portal frame. In Load Case 1 both columns are loaded. In Load Case 2 only one column is loaded. The reference solution is taken from 'Elastic Beams and Frames' (J D Renton 1967)

	Case 1	Case 2
FS2000	7.476	14.771
Reference Soln	7.477	14.772 (x/π/π*10)

-o-

11.4.1 Verificaton Examples - Solids (FE)

FEExp1 Simply Supported Rectangular Plate

This model is a simply supported rectangular plate 1 m square by 25 mm thick. The model uses 4 Node Type 50-0 Thin shell elements.
Properties E = 205E9 v = 0.3 t = 25 mm L = 1 m
Load Case 1 15 kN
Load Case 2 Uniform normal pressure 5 bar
Due to double symmetry in both geometry and loading only a quarter of the plate is modelled.
Reference Solution: Timoshenko and Woinowsky-Krieger , "Theory of Plates and Shells" McGraw_Hill Book Co

		Theory	FS2000
Load Case 1	Deflection	2.373 mm	2.38 mm
Load case 2	Deflection	6.92 mm	6.78 mm
Max Bending St		230 N/mm2	231 N/mm2

FEExp2 Cantilever Beam

This model is a short cantilever 0.5 m long, 0.1 m deep and 50 mm thick. Two models have been used. One uses 4 node elements and the other uses 8 node elements. The 4-Node model uses 2-D plane stress (Type 30-0) solid elements. (FEExp2-8).
The loading is a 10 kN shear end load.
Properties E = 205E9 v = 0.3 d = 0.1 m t = 50 mm L = 0.5 m
Reference Solution: Engineers beam theory

	Theory	FS2000 (4 Node)	FS2000 (8 Node)
Tip Deflection	0.4878 mm	0.496 mm	0.493 mm
Max .Bending Stress	60 N/mm2	54 N/mm2	60 N/mm2
Shear Stress	2 N/mm2	2 N/mm2	2 N/mm2

Comment: The beam is modelled using only 5 elements. This mesh would be considered too coarse for practical FE analysis when using linear elements (4N). However even with such a coarse mesh the accuracy of the 4 node model is excellent (90% on stress). The 8 node element has a linear strain distribution and produces an exact stress match. The 4 node uses a constant strain and the stress is the stress at the element centroid i.e.60/5*4.5.

FEExp3 Bending of a T Section Cantilever

In this example a T section cantilever is modelled using three distinct techniques.
Model FEExp3 uses shell elements and beam elements with offsets and demonstrates a useful technique to model a stiffened panel. In this model the flange is modelled using shell elements and the stem is modelled by beam elements. The beam elements have offsets of 30 mm which represents the sum of the offsets of the centroid of flange and the centroid of the stem from the neutral axis.
Model FEExp3A uses shell elements and beam elements and is similar to FEExp3 in that it uses offsets but in this example the 30 mm offset is applied to the shell element .
Model FEExp3B uses beam elements and represents the reference solution.
Model FEEXp3-8 uses 8 node shell (Type 50-0) elements to represent the T section.
The loading is a 1 kN tip shear load.
Properties: E=205E9 v = 0.3 w = 200 mm d = 60 mm t = 10 mm

Reference solution: Engineers beam theory

	Theory	FEExp3	FEExp3A	FEExp3B	FEExp3-8
Tip Deflection	27.1	27.2	27.2	27.1	28
Flange Stress	45.83	51.15	51.2	45.83	44.7
Stem Stress	204	214**	214**	204	199

** Bending + direct stress in the beam element

FEExp4 Internal Pressure on Thick Cylinder

In this example a hollow thick cylinder is subject to a uniform internal pressure. The model uses five 8 node axi-symmetric (Type 40) solid elements. The radial direction is in the x direction (requirement for axi-symmetric elements).
Properties: E=205E9 v = 0.3 Internal r = 100 mm External r = 250 mm
Internal pressure = 1400 bar

Reference solution: Thick cylinder theory (Lames Eqns)

	Theory	FS2000
External radial deflection	6.5E-5 m	6.5E-5 m
Max Tangential Stress (z)	193.3 N/mm2	191 N/mm2
Max Radial Stress (x)	140 N/mm2	137.6 N/mm2

FEExp5 Cylindrical Shell Roof

In this example a cylindrical roof shell is loaded vertically by its self-weight and supported by a rigid diaphragm at each end. The problem is to evaluate the vertical deflection at the mid-point of the free edge. Due to double symmetry in both geometry and loading only a quarter of the plate is modeled. A 5 x 5 mesh of 4 node thin shell (Type 50-0) elements are used.
Properties: E=4.32E8 v=0.0 R=25 L=50 w= 80 deg t=0.25
Loading: Shell weight = 90 per unit area
Reference Solution: COSMOS The model is commonly used model in FE literature for the testing of finite element accuracy.

	Quoted Exact	FS2000
Max vertical Deflection	0.3024	0.3075

FEExp6 Pinched Cylinder

In this example a cylinder is subjected to two equal and opposite radial point loads. The problem is to evaluate the radial deflections at the load points.
Due to double symmetry in both geometry and loading only a quarter of the plate is modelled. The model use a 8 x 8 mesh of 4 node shell elements (Type 52-0).
Properties: E=3.0E6 v=0.3 R=300 L=300 t=3.0
Loading: Unit point load.
Reference Solution: Cook (ANSYS)

	Reference	FS2000
Max vertical Deflection	0.1139	0.115

FEExp7 Hemispherical Shell

In this example a hemispherical shell is subjected to anti-symmetrical radial point loads. The problem is to evaluate the radial deflection at the load point.
Due to double symmetry in geometry only a quarter of the plate is modelled. A 6 x 6 mesh of 4 node thin shell (Type 50-0) elements are used.
Properties: E=6.825E7 v=0.3 R=10 t=0.04
Loading: Unit point load.
Reference Solution: The model is commonly used model in FE literature for the testing of finite element accuracy.

	Quoted Exact	FS2000
Max Radial Deflection	0.0924	0.091

FEExp8 Internal Pressure on Thick Cylinder

In this example a hollow thick cylinder is subject to a uniform internal pressure (same as FEExp4). The model uses five a 5 x 5 mesh 8 node 2-D plane stress (Type 30-0) solid elements. A cylindrical coordinate (CSys 1) system is used to define the model and to output the element stresses
Properties: E=205E9 v = 0.3 Internal r = 100 mm External r = 250 mm
Internal pressure = 1400 bar
Reference solution: Thick cylinder theory (Lames Eqns)

	Theory	FS2000
External radial deflection	6.5E-5 m	6.5E-5 m
Max Tangential Stress	193.3 N/mm2	193.7 N/mm2
Max Radial Stress	140 N/mm2	140.1 N/mm2

FEExp9 Internal Pressure on Thick Cylinder

In this example a hollow thick cylinder is subject to a uniform internal pressure (same as FEExp4). The model uses five a 6 x 9 mesh 8 node 3-D (Type 70-0) solid elements. A cylindrical coordinate (CSys 1) system is used to define the model and to output the element stresses
Properties: E=205E9 v = 0.3 Internal r = 100 mm External r = 250 mm
Internal pressure = 1400 bar
Reference solution: Thick cylinder theory (Lames Eqns)

	Theory	FS2000

External radial deflection	6.5E-5 m	6.48E-5 m
Max Tangential Stress	193.3 N/mm2	192.1 N/mm2
Max Radial Stress	140 N/mm2	130.1 N/mm2

FEExp10 Thermal Exp Stresses Thick Cylinder

In this example a hollow thick cylinder is subject to a logarithmic radial temperature distribution. The model uses five a 5 x 5 mesh 8 node 2-D plane strain (Type 30-0) solid elements. A cylindrical coordinate (CSys 1) system is used to define the model and to output the element stresses.
Load Case 1 defines Ti = 100 and To = 0. A heat transfer solution is used to establish the nodal temperature distribution.
Load Case 2 is the nodal temperature distribution resulting from load case 1 (<model>.UNT1 command file)
Properties: E=205E9 v = 0.3 Internal r = 100 mm External r = 250 mm Thickness=0 (Plain strain)
Coef of Exp=1.1E-5 Any non zero value for k will produce the same result.
Reference solution: P232, Timoshenko, Strength of Materials Part2 , Von Nostrand, 3rd Ed

	Theory	FS2000
Internal Tangential (Hoop) Stress	207 N/mm2	204 N/mm2
External Tangential (Hoop) Stress	114 N/mm2	117 N/mm2

Axial Stress Outer 34.8 to inner -285.8

FEExp11 Heat Transfer & Thermal Exp Stresses Thick Cylinder

This is the same problem as **FEExp10** The model uses 5 8-node 2-D Axisymetric elements (Type 40-0). Stiff couple elements have been used to ensure that the displacement in the Y direction (axial) remain plane. This represent the case of the away from the ends in a long pipe.
Load Case 1 defines Ti = 100 and To = 0. A heat transfer solution is used to establish the nodal temperature distribution.
Load Case 2 is the nodal temperature distribution resulting from load case 1 (<model>.UNT1 command file)
Load Case 3 is the same as load case 2 but with prescribed displacements in the Y set to 1E12 ie fully restrained - gives the axial stress distribution as FEExp10
Properties: E=205E9 v = 0.3 Internal r = 100 mm External r = 250 mm Thickness=0 (Plain strain)
Coef ofExp=1.1E-5 Ant non zero value for k will produce the same result.
Reference solution: P232, Timoshenko, Strength of Materials Part2 , Von Nostrand, 3rd Ed
Load Cases 2 and 3 give the same results for the hoop stress and are in close agreement to FEExp10 (hoop stress in independent of plain strain magnitude). The axial stress at the inner and outer agree well with theory in that the hoop and axial stresses are same at the surfaces of the cylinder.

	Theory	FS2000
Internal Tangential (Hoop) Stress	207 N/mm2	201 N/mm2
External Tangential (Hoop) Stress	114 N/mm2	115 N/mm2
Internal Axial Stress	207 N/mm2	205 N/mm2
External Axial Stress	114 N/mm2	114 N/mm2

Load case 3 give the a similar axial stress distribution to that in FEExp10 i.e. when the ends of the cylinder are axially restrained. Outer 34.8 to inner -285.8

FEExp12 Heat Transfer with Heat Generation

This model uses 2-D Axisymetric elements (Type 40 -0) to establish the temperature in a heat generating wire.
Wire Radius 0.03125 ft
k = 13 Btu/hr-ft-F h = 5 Btu/hr-ft2-F Ta = 70 F E=1 (not used but must be non-zero)
Heat generation q = 111312 Btu/hr-ft3
Refernce Solution W.M Rohsenow, et al , Heat Mass and Momentum Transfer, Prentice-Hall, 1963 page 106 (ANSYS validation)
Target Solution Tc = 419.9 Ts = 417.9
FS2000 Tc = 420.5 Ts = 418.4

-o-

11.5 Tutorials

The Tutorials folder in the User's FS2000 folder contains a number of tutorials on various topics relating to the use of FS2000.

The **List_Tutorials.PDF** gives an overview of those currently available.

Each of the tutorials are in a zip file which contains a pdf of the tutorial topic text and the associated FS2000 models (MOD files).

-o-

12 Program QA

12.1 Program Verification

Program Verification

The program has been validated against the analytical solutions, analysis solution in the public domain and other software packages.

Included in Section 11 are a number of validation examples. The main purpose for the inclusion of these examples is to demonstrate some of the analysis features of FS2000. They also provide evidence of program verification. These examples may be recreated by a new user to verify their understanding of the operation of the program. Sample solution extracts for each of the models are given so that the user may check their solution. Also given for validation purposes are solutions from either analytical solutions or computer based solutions from other software.

In addition to the above mentioned examples AES have produced a " stand alone" verification documents that contains a wide range of examples. This document along with the subject models is available from AES.

It is important that the end user verifies their understanding of the operation of the program as well as the analytical correctness of the solutions. AES consider that some verification documentation should be created by the user to suit their particular analysis requirements and over their range of application. This is a task that should be done by any user during the initial familiarisation process. The above mentioned examples or the verification document may assist in this.

If any user has specific requirements for verification documentation/examples then AES will consider the provision of such.

Program Development

During the development of the program modules of FS2000 every effort has been made to ensure the correctness of any analysis results. The quality system adopted during program development is that all program developments are thoroughly verified through worked examples. This verification is generally based on small models for which analytical solutions are available or computer based solution using other software packages.

Ongoing program verification following program modification and maintenance is accomplished using a number of benchmark models. These models are re-analysed and their results are compared with results from previous versions of the programs.

-o-

12.2 Operational - Model Traceability

Model files, load case files and analysis results files are totally separate files therefore the possibility of having the results files not related to the current model does exist. This however, can only occur if model file is changed and previous result files are not re-analysed with the new model. By default all results files are purged when a model is saved. The user can override this default

To provide a check that files results file are sequential all definition related files have an identification date and time stamp associated with them. These ID stamps are echoed on the formatted output data. These stamps and their functions are described below.

Formatted Input Data

- Model File This formatted input data file shows the ID stamp when the model was last saved. When recovering the model from archive the file date is not changed. The will prevent archived files from appearing to have changed.

- Load Case Files The formatted input data file shows the ID stamp when the file was last saved in FS2000. Note that if the file is manually edited the stamp will not be updated unless the file is loaded and re-saved in FS2000.

It is not important that the above files are sequential since the loads definition is only uniquely matched to the model file at the time of analysis. The input data echo shows the load summation both at the time of saving the load case and the time of analysis. The latter always takes precedence.

-o-

12.3 Maintenance of Released Software

To ensure that any serious operational malfunctions or errors are detected and corrected in future releases the following system is used.

- **Reportage** - If a user detects an error or serious malfunction then they should contact AES and provide a program infomation report (PIR)
- **Remedial** - On receipt of the PIR form (or whenever a malfunction is detected) appropriate remedial action is taken to rectify the error.
- **Notification** - Depending upon the severity of the malfunction AES will log an Error Notification Report (ENR). The ENR defines what errors exist and what remedial action or work around is required.

It is essential that all users notify AES the details of their contact representative.

-o-

Copywrite/Software Licence Agreement

Software Licence Agreement & Limited Warranty

This package contains your FS2000 software. Please read this agreement before using the software. If you do not agree with the terms of the agreement, return this package as is and Analytical Engineering Services Ltd. will, upon receipt, refund your purchase price. No refund will be made for packages returned with missing or damaged components.

1. LICENSE: Analytical Engineering Services Ltd. (AES) grants you a nonexclusive right to use the enclosed software. The enclosed software may be transferred from one computer to another, but may be used only on one computer at a time, by a single user. If you have purchased the network version, it may be used on a single network, by a single business entity, or organisation, by up to the number of authorised simultaneous users. If the enclosed software is supplied with a hardware copy protection device, that device remains the property of AES. This license may not be transferred without written authorisation from AES. You may not: (a) distribute copies of the software or user documentation to others; (b) provide, sell, or sublicense access to, or use of, the software to other third parties (c) modify, use, copy, reverse assemble, reverse compile or transfer this software or user documentation, or any copy thereof without prior written consent. You may: make copies of this software solely for backup purposes. Such "backup copies" must be so labelled. Some of the software may be licensed on a beta or evaluation basis, which is not intended for commercial or professional use· Condition of Payment. The license granted herein is conditioned upon payment in full for the software in advance of your use of the software or as previously agreed with AES.

2. COPYRIGHT: This software and user documentation is protected by a copyright which is owned by AES. You may not copy the software or user documentation except for the purpose of backup, as stated above, and to load the software into a computer for executing the software. All other copies of the software and user documentation, except as stated herein are in violation of this agreement and AES's copyright.

3. TERM: The term of this license is effective until terminated by your failure to comply with this agreement and AES's copyright or any lease conditions which may be effective. You may terminate the license by destroying the software, user documentation, and all copies thereof, and returning any security devices along with written notice to AES of such termination.

4. LIMITED WARRANTY: (a) AES warrants that the media upon which the enclosed software is recorded is free from defects in materials and workmanship when used under normal conditions, and (b) AES warrants that the software will perform substantially as described in the User Documentation. AES does not warrant that the software will function properly in every hardware/software environment.

5. DISCLAIMER: AES, its agents, employees and distributors shall not be liable for technical, editorial or other errors or omissions which may be contained in, or the negligent preparation of this material. AES hereby disclaims any express or implied warranty, that the enclosed software, documentation or other materials are fit for any particular purpose. AES does not guarantee the accuracy and/or usefulness of the results or solutions, even if performed in accordance with the procedures, commands and theories contained in the enclosed material. AES, its agents, employees and distributors shall not be liable for compensatory, punitive, or other damages of any nature, arising from, or allegedly arising from any breach of the Limited Warranty above, nor shall they be responsible for claims for lost profits or revenues, lost data, or recreating lost data, substitute programs, nor any claims of third parties. This disclaimer shall apply regardless of any notice to AES of the possibility of such damages.

7. EXCLUSIVE REMEDIES: Upon breach of the Limited Warranty, stated above, AES shall at its option: (1) provide corrected software to you, or (2) provide a corrected User Documentation, or (3) refund the fee paid for the software, without charge to you. In no case shall AES's liability exceed the amount of the license fee paid, or £50.00 (U.K Stirling.), whichever is greater. These are the exclusive remedies agreed in case of breach of the Limited Warranty.

8. ACKNOWLEDGEMENT: You acknowledge that you have read this agreement and are bound by its terms, that this agreement is the complete and exclusive agreement of the parties hereto, and that it supersedes any and all communications, proposals, representations, and agreements, oral or written, previously or subsequently made between the parties.

9. NOTICES: All notices required by this agreement, questions pertaining thereto or concerning the enclosed software and User Documentation should be directed to:

Analytical Engineering Services Ltd
Bucklerburn House, School Road, Peterculter
Aberdeen AB14 0NP

-o-

Index

- 1 -

- 3 -

- A -

- B -

- C -

- D -

- E -

- F -

- G -

- H -

- I -

- K -

- L -

- M -

- N -

- O -

- P -

- Q -

- R -

- S -

- T -

- U -

- V -

- W -