FS2000

Modal Dynamic Response

Advanced Structural Analysis
for Windows
(c) A.E.S. Ltd 1988,2020

Email:support@aes-uk.com

Table of Contents

CONTENTS

-o-

1.0 Introduction

FS-Dynamic is general description of the dynamic response analysis modules of FS2000. This manual describes the capabilities and operation of the various dynamic analysis modules.

The Frequency Analysis module is part of the standard FS2000 system. This module is used to evaluate the frequencies and mode shapes of the fundamental modes of vibration of linear structural models. The module is started form the Solution menu in the Analysis TASK.

The optional Dynamic Response modules are used for the solution of dynamic response problems. These modules are based on the normal mode method which utilises the output results of the Frequency Analysis module.

The Dynamic Response modules provide solutions to the following type of vibration problem:

- Steady State Harmonic
- Time History
- Response Spectra (Seismic) Analysis

The Steady State Harmonic and Time History module is started form the Solution menu in the Analysis TASK.

The Response Spectra (Seismic) Analysis is started from the FS2000 Windows Start menu.

-o-

2.0 Frequency Analysis

The standard dynamic analysis modules provide the capability to undertake frequency and mode shape analysis of 2-D and 3-D structures. Two basic analysis methods are used by FS2000 - the Jacobi method and the Subspace Iteration method. Ref. K-J Bathe, "Solution Methods for Large Genaralized Eigenvalue Problems in Structural Engineering" Report UC SESM 71-20 Civil Eng Dept Univ California Berkeley 1971)

The module is started form the Solution menu in the Analysis TASK.

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 (less than 200 degrees of freedom).

The solutions are only applicabe to beam element structures solid elements are ingnored.

The effects stress stiffening or softening ie P-Delta effects can be included. If the 3-D Standard solver is run with PDelta effects active then an element force file will be created that can be used to include P-Delta effects into the frequency solution.

The Subspace Iteration method will only evaluate a few of the lowest frequency modes. Its memory requirements are considerably less enabling it to handle large models. The standard 3-D module will handle 6000 degrees of freedom. It should be noted that the methods ability to only evaluate the lowest modes is not a disadvantage since generally only the lowest modes are of significance in real engineering systems.

Note that frequency analysis uses a banded technique so use the Bandwidth command in the Reseq Menu (Analysis TASK) to renumber (internal) the model for efficient solution.

-o-

2.1 Mass Modelling

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. See Sect 2.3 - Mass Definition

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-

2.3 Definition of Mass

The Load Definition TASK is used to create mass load cases that are used in the dynamic analysis. The mass case is defined and saved in an identical way to that of a static load case.

The use of a dynamic gravitational conversion constant enables standard load cases to be used a mass definition cases. Loading that is defined in the units of force will be converted to mass using the constant.

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 this mass will be included in the analysis. Note that the gravitational constant conversion constant should be equal to the value otherwise the mass due to self weight will be factored.

Distributed Mass

Distributed mass can be applied to elements by applying them as Y direction distributed loads. The mass is evaluated by dividing the load by the dynamic gravitational conversion constant. In the case of patch loads the total load from the patch will be distributed over the full length of the element. This is a requirement since only 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.

Element Concentrated Mass

Concentrated element mass cannot applied in dynamic analysis. To enable element load cases to be converted to mass load cases, element loads can be converted to element masses. Element point loads applied in the Y direction will be converted to mass. The mass is evaluated by dividing the load by the dynamic gravitational conversion constant. The mass is then applied as though it were uniformly distributed over the length of the element.

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 load by the dynamic gravitational conversion constant.

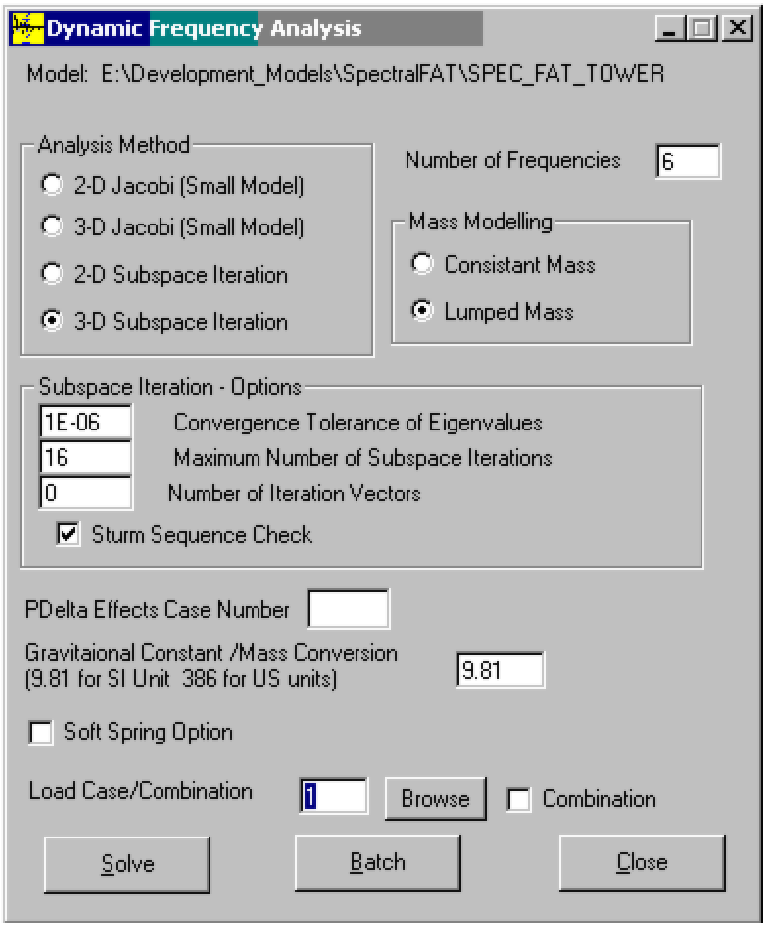
Nodal inertia (rotational mass) in the global direction can be applied as moments in the corresponding direction. The inertia is evaluated by dividing the load by the dynamic gravitational conversion constant.

-o-

2.3 Frequency Analysis - Operation.

The Frequency Analysis module is started by selecting the Frequency command from the Solution menu in the Analysis TASK.

When the module has loaded the following input form will become visible.



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 **Analysis Method** options are used to select the frequency analysis module. The Subspace Iteration methods are by far the most efficient. See Sect 2.0 - Frequency Analysis.

The **Number of Frequencies** 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. For the Jacobi method it is possible to specify more than the number of degrees of freedom.

The **Mass Modelling** options are used to select the type of mass matrix to be used. See Sect 2.2 - Mass Modelling.

The **Subspace Iteration** options are used to control the solution in the Subspace Iteration module. 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 default value equal to the number of frequences will be used. Increase this value if the solution fails to find all the modes . **Sturm Sequence** check. The check will print a warning message if any Eigenvalues are missed within the frequency range. Using more **Iteration Vectors** may find the missed values.

The **PDelta Effect Case Number** is the static analysis case used to evaluate the force distribution in the structure. This option would be used on structures whose stiffness is depedent upon tension stiffening effects. It is first necessary to run the 3-D Standard Analysis with the P-Delta effects active. The load distribution from that case will be used to evaluate the P-Delta effects in the frequency solution.

The **Gravitational Constant (Mass Conversion)** is used to convert y direction forces in a load case to masses. See 2.3 Definition of Mass.

The **Soft Spring** option applies a soft spring to all element degrees of freedom. Same use as in static analysis.

The **Load Case/Combination** box is used to select the load case that defines the mass distribution in the model. Load Case Combinations may be used to pre-combine load cases.

The **Batch** button will append the current settings to the Frequency Module command line to the <ModelName>.BRM file. (See Sect 2.5)

-o-

2.4 Frequency Analysis Output

The results from the Frequency Analysis Module are the fundamental modes of vibration. These 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 vibration plot may be animated by selecting the Animate command. Animation is stopped by re-selecting the animate command.

-o-

2.4 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

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	Static PDelta Resu;st Case Number (DYNFRAM only)

and one of the following

DYNFRA2	2-D Jacobi (Small Model)
DYNFRA2B	3-D Jacobi (Small Model)
DYNFRAMB	2-D Subspace Iteration
DYNFRAM	3-D Subspace Iteration

-o-

2.5 Files used/Created

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 **'.UFO'**

The Frequency Results files have the model filename with the extension **'.Hn'** 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.

If PDelta effects are included an element force file **'.RAFn'** from the static solver is used.

The output file showing the modal results is a **'.FREQ.On'** text file.

-o-

3.0 General Response Analysis - Introduction

The Dynamic Response Module of FS2000 uses the Normal Mode method to provide dynamic response solution to the following types of vibration problems. Ref: Clough & Penzien, Dynamics of Structures McGraw-Hill 1975

- Harmonic
- Transient(Time History)
- Response Spectra (see Section 4)

The program uses a command line mode of operation. These command are grouped together in a load case file so as to produce a Dynamic Response Load Case. The command instructions used are given in Appendix 2. The Dynamic Response Module has an interface which enables most of the command to be entered in an interactive mode. The operation of this module is described in Sect 3.4.

The module is started form the Solution menu in the Analysis TASK.

Before any response analysis can be undertaken a Frequency analysis of the model using an appropriate mass case must have been completed. The Frequency analysis determines the vibration modes shapes. See Section 2 for a description of Frequency analysis.

Note: When undertaking the Frequency analysis only lumped mass models option should be used when modal response analysis is to be undertaken.

With the Normal Mode method is it not necessary to include all modes of vibration in the solution. In most cases it is only necessary to include a fraction of the total number of modes (modal degrees of freedom). In most real engineering problems it is generally only first few modes that are of significance. The modes which are included in the response analysis are determined from the number of modes selected in the frequency analysis. The response output results show modal mass participation which can in in most models indicate the insignificance of higher mode inclusion, but not always.

Modal Damping

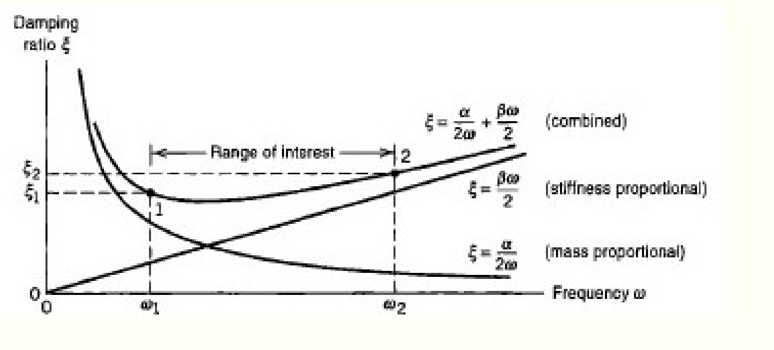
Structural damping is applied in FS-Dynamic by defining a critical damping ratio for each mode. This is theoretically incorrect as this type of damping couples the de-coupled modes in modal analysis. However, the error introduced will be insignificant for the levels of damping in a real structural systems . Timoshenko quotes that this approximation will produce realistic results for damping ratios of up to 0.2 critical.

The critical damping ratio may be specified for each mode using the **[DAMP,*Mode,Value*]** command. If damping is only specified for the first mode then all other modes will be assigned this value unless otherwise defined.

By default a critical damping ratio of 0.05 is applied to the first mode which will then be applied to all other modes. To change the damping, the damping ratio for mode one should be specified as appropriate e.g. a small value (1E-12) will remove all damping.

Rayleigh Damping

Defining the Rayleigh Damping coefficients α and β enables modal damping ratios to be based on Rayleigh damping. The **[RDAMP, α, β]** command is used to define the coefficients. They relate to the more usual 'damping ratio' γ by the expression $\gamma = (\alpha + \beta\omega^{**}2) / 2\omega$. The β coefficients give stiffness proportional damping which provides damping between element connected degrees of freedom. The α coefficients give mass proportional damping which provides mass to 'ground' damping.


Damping ratio ξ
$\xi = \frac{\alpha}{2\omega} + \frac{\beta\omega}{2}$ (combined)
Range of interest
$\xi = \frac{\beta\omega}{2}$ (stiffness proportional)
$\xi = \frac{\alpha}{2\omega}$ (mass proportional)
ξ_2
ξ_1
1
2
0
ω_1
ω_2
Frequency ω

-o-

3.1 Results Output

The results of a response analysis can take two basic forms.

- **Selective Graphical/Tabular**
- **Standard Results Case**
- **Standard Load Case**

The form of output required is defined by command line instructions in the Dynamic Response Load Case. The Selective output command would be generally used to identify critical area due to the type of excitation. The Standard Results Case method may be used to establish loading within the whole structure at specific instances.

Selective Graphical/Tabular

The displacement response output is defined using the **[NDOUT]** command. This command defines a node and its freedom direction at which the displacement response is required. In each Dynamic Response Load Case the response at up to four locations may be defined.

The force response output is defined using the [EFOUT] command. This command defines an element and freedom direction (local element co-ordinate system) at which the response is required. In each Dynamic Response Load Case the response at up to four locations may be defined.

The output for the NDOUT and EFOUT command will be in both tabular and graphical form. The tabular form will be written to a Standard Analysis Results case output file, the case number will be the same as the Dynamic Response Load Case. The graphical form may be plotted from the Dynamic Response Module.

Standard Results Case

The [FCASE] and [TCASE] commands can be used to create standard result cases for a specific response point. These are standard results case which must be post-processed before they can be accessed by FS2000.

The FCASE command is used for harmonic response. It will create a results case at a specified time for a specified frequency of excitation.

The TCASE command is used for transient (time history) analysis. It will create a results case at a specified time.

The [FNCASE] command can be used to create standard result cases for a specified fundamental mode shapes.

Standard Load Case

The [INFORCE] command is used for harmonic analysis. It will create a series of equivalent load cases over a defined time period and increment for a specific frequency.

-o-

3.2 Harmonic Response

Harmonic analysis evaluates the response due to steady state harmonic loading. This section described the type of harmonic loading available and the commands used in the Dynamic Response Load Case.

The name given in [ ] is the command used to define the loading. The underlined commands must be entered in the Dynamic Response Case using a test editor.

Harmonic Response - Types of Excitation

Constant Amplitude Nodal Forces

Nodal Harmonic excitation due to a concentrated nodal force or moment **[HFORCE]**. The excitation may be applied to any number of nodes. Note that all excitations are applied at the same frequency in harmonic analysis.

Rotating Unbalance

Nodal excitation due to an unbalanced rotating mass **[RUFORCE]**. The user defined a rotating mass, its radius of rotation and its frequency. Any number of unbalanced forces may be applied to any number of nodal freedom directions but they must be at the same frequency.

Support Motion

This excitation is in the form of induced movement (displacement, velocity or acceleration) of the support points of the structure. Linear **[LSUPP]** or rotational motion **[RSUPP]** of the supports may be defined. When rational support is defined the centre of rotation is required to be defined . This is defined relative to the model origin.

Support motion may be applied in the form of displacement, velocity or acceleration amplitude using the **[SMOTION]** command. Support motion cannot be applied with any other form of excitation. Support motion takes precedence.

Harmonic Response - Frequency/Time Definition

Frequency Response Range

The **[FREQRESP]** command is used to define a number of separate excitation frequency ranges. The range is define by specifying an initial frequency, a final frequency and number frequency points. The output will be the maximums at the appropriate frequencies.

Frequency Response/Varying Amplitude Range

The **[FCURVE]** command is used to define a frequency/amplitude curve. This enables the amplitude of the excitation frequency to be different at each excitation frequency. The amplitude value is used to factor the amplitude of the excitation force.

Harmonic Time History Response

The **[TIMERESP]** command is used to define the response period over which the output is require. The output will be a harmonic curve. The period is define by specifying an initial time, a final time and number of points. The forcing frequency for the time curve is defined using the **[FREQ]** command.

-o-

3.3 Transient (Time History) Response

Transient analysis evaluates dynamic response due to excitation that varies with time. This section described the type of transient loading available and the commands used in the Dynamic Response Load Case.

The name given in [ ] is the command used to define the loading. The underlined commands must be entered in the Dynamic Response Case using a test editor

Apart from impulse [IMPULSE] analysis the response uses a closed form solution of the convolution integral which will produce accurate and always stable solutions with relatively large time steps since the solution assumes that the slope of the excitation is constant between points. The IMPULSE command is applicable to undamped systems and uses an exact explicit form of solution analysis

Transient Response - Types of Excitation

Initial Conditions

The initial conditions at any node may be defined in terms of the initial displacement **[INIDISP]** or initial velocity **[INIVEL]**

Non-Periodic Nodal Forces

Nodal excitation due to a concentrated nodal force or moment **[TFORCE]**. Any number of nodal forces may be applied to the model. The nodal force is associated with a time history which define an amplitude factor as a function of time. This history may be define using the TCURVE or the HCURVE command.

The time co-ordinates (time steps) of all time histories must be the same. They may be of different duration. Dummy histories with zero amplitude may be used to extends the response period.

Impulsive Forces

Excitation due to shock/impact on an undamped system can be applied using the **[IMPULSE]** command. The solution uses an explicit solution that always produces a theoretically exact solutions.

The default response period is taken as 4 time the pulse duration. The TIMERESP command may be used to change the time intervals of the output response . IMPULSE cannot be applied with TFORCE. IMPULSE takes precedence over TFORCE. IMPULSE cannot be applied to base excitation.

Any number of impulsive forces may be applied to any number of nodal freedom directions.

Non-Periodic Support Motion

This excitation is in the form of induced accelerations of the support points of the structure. Linear **[LSUPP]** or rotational motion **[RSUPP]** of the supports mat be defined. When rational support is defined the centre of rotation is required to be defined . This is defined relative to the model origin.

The support motion is always associated with a time history which define an amplitude factor as a function of time. This history may be define using the TCURVE or the HCURVE command and is always associated with Curve Reference No 1.

Support motion cannot be applied with any other form of excitation. Support motion takes precedence.

Transient Response - Time History Definition

Time/Amplitude Curve

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 maximum default number of data points is 1000. This may be increased using the **MAXTIME** command.

The data associated with the **TCURVE** command is included in the Dynamic Response Case. In many cases it may be more convenient to read the data from an external file. This can be done using the

[TFILE] command.

Time/Amplitude Curve - Damped Harmonic

The **[HCURVE]** command will generate a time history based on the following equation for a damped harmonic signal.

F(t)=(C1.sin(C2.t+C3) + C4.cos(C5.t+C6)).EXP(C7.t)

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.

Transient Time History Response

The **[TIMERESP]** command is used to define the response period over which the output is require. The period is define by specifying an initial time, a final time and number of points.

When defined it will interpolate time curves so as to enable time steps to be used in the response analysis than those used to define the time curves.

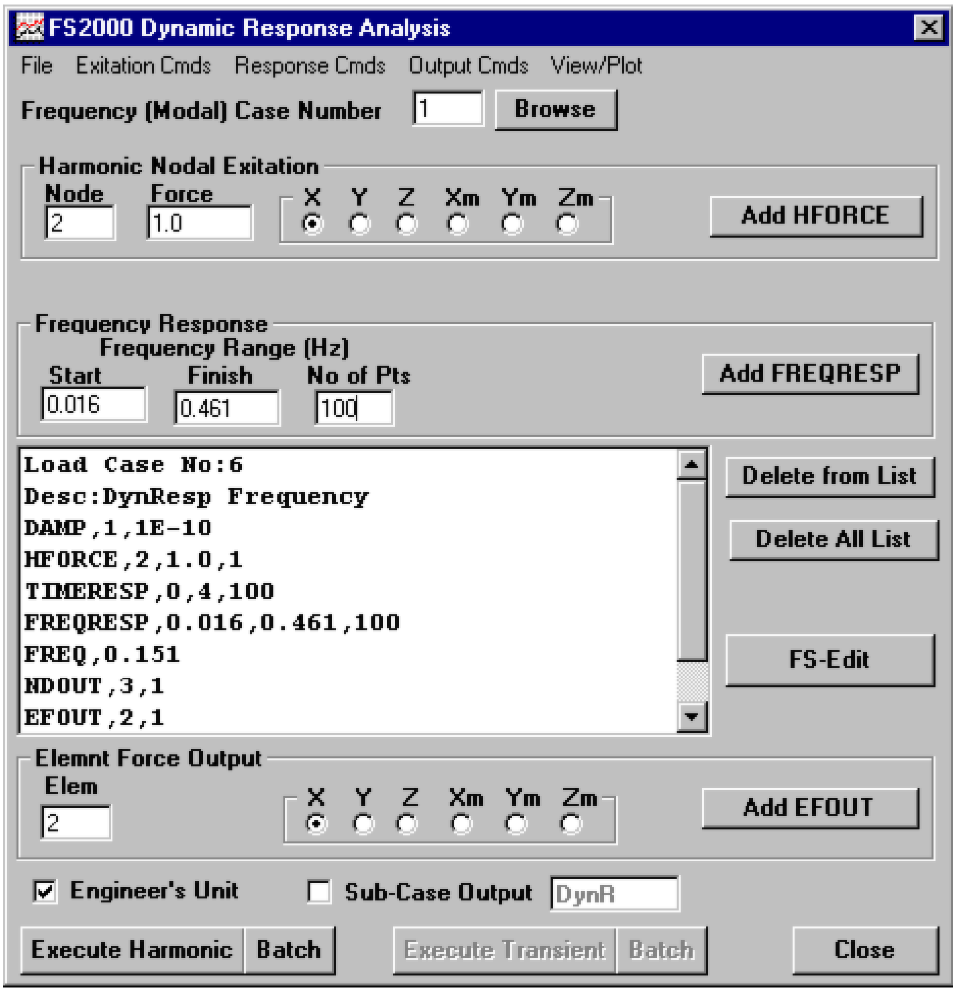
This command can also be used to obtain the time response when only initial conditions are specified or can be used to extent the response time when using impulse excitation.

-o-

3.4 Response Analysis Operation

The Response Analysis module is started by selecting the Dynamic Response command from the Solution menu in the Analysis TASK.

When the module starts it will load the current default model and the following window will become visible.



The central text box shows the Response Command List in current memory. The commands always have to be saved as a Response Load case before an analysis can be undertaken. A Response Load Case is a text file containing the command line instructions. They can be created in a test editor by following the command structure described in Appendix 3. Text files must be loaded and re-saved for them to exist in FS2000 as a registered load case.

Commands are appended to the list by entering data in an appropriate command frame (e.g. **Harmonic Nodal Excitation**) and then clicking its Add (e.g. **Add HFORCE**) button to add the command to the list. The command frames are made visible by selecting the command type form the appropriate menu option.

Menu Commands

File - Saves and opens the Response Load Cases.

Excitation Cmds - Makes visible the excitation data frames. These frames appear uppermost.

Response Cmds - Used to enter the Damping value for mode 1 and make visible the harmonic response data frames. These frame appear just above the command list box.

Output Cmds - Used to select the type of output required. These frames appear below the command list box.

View/Plot - Used to view the or plot the response results of the current response case.

The **Frequency (Modal) Mass Case Number** box is used to enter the Frequency Results Case.

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>.DynR.O<.n> where DynR 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 **Execute** buttons execute the response analysis. The **Batch** button converts the set options to command line switches and appends the module command line to the current .BR<id> batch run file.

The **FS-Edit** button will start FS-Edit and load the current response case i.e. the saved response case, not the displayed response list. Therefore save the list before editing.

-o-

3.5 Batch Command Line Operation

Providing that the Frequency Results and Dynamic Response cases exists the module may be operated using command line (Batch) operation.

The command line is

DYNRESP1 C1/C2/C3/C4/

C1	Analysis Type : 1-Frequency 2-Time History
C2	Mass (Frequency Results) Case
C3	Dynamic Response Case Number
C4	Engineers Units 1 or 0 1 - Activates 0-Exponent Format
C5	Sub-Case Description

-o-

4.0 Response Spectra (Seismic) Analysis - Introduction

The response spectra analysis is based on the Normal Mode Seismic Response Method (RSM). It evaluates the maximum structural response due to rigid base motion defined by a Response Spectra Curve.

Response Spectra Curves may be defined in terms of acceleration, velocity or displacement versus Period (secs).

Ref: Clough & Penzien, Dynamics of Structures McGraw-Hill 1975

Before a response analysis can be undertaken the Frequency module must be run so as to create the necessary data on natural frequencies and mode shapes of the structure.

The Response Spectra (Seismic) Analysis is started from the FS2000 program group icon.

Translational base excitation and/or rotational base excitation spectra may be defined. The rotational base spectra is evaluated in accordance with Eurcode 8.

The principal direction of the excitation can be in any direction relative to the model.

To ensure that sufficient modes are used, the modal participating mass for each mode. The modal mass percentage of the total mass is also shown. Missing mass modes can be often used to compensate for lack of mass participation.

The basic input to the module are the results of the frequency module and a defined Response Spectra.

The output from the module is a standard results case that gives the maximum absolute values of displacement and force. It is however very different from a standard equilibrium solution and only nodal values should be considered i.e. mid span force actions should not be evaluated because the mid span evaluations will be incorrect. This restriction also applies when spectra cases are post combined with other cases, only use nodal values.

Modal combination methods are selected from the SRSS, CQC, 10 Percent Rule, DSM or ABS Sum methods.

Spatial combinations use the SRSS or ABS Sum methods.

A Response Case options file is used for each response case. The module may be run in interactive mode of in batch mode.

Unsigned Results (All Positive)

The resulting deflections, element force and reactions from a response spectrum analysis are unsigned i.e. all postive. This means that the force actions will not look the same as a conventional equilibrium solution case. When combining with other load case the response spectra result should be combined in two combinations, one positive and one negative.

It is possible to obtain a signed response using the signs obtained for a single dominant mode. This would applicable to cases were the modal displacements for the combines modes case is similar in shape to the that of a dominant mode. The dominant mode is usually the mode with a the highest mass participation.

Individual Modes

Results for any individual mode solution may also be obtained. The results from a single mode solution will be a signed response.

Evaluation of Mid Span Forces

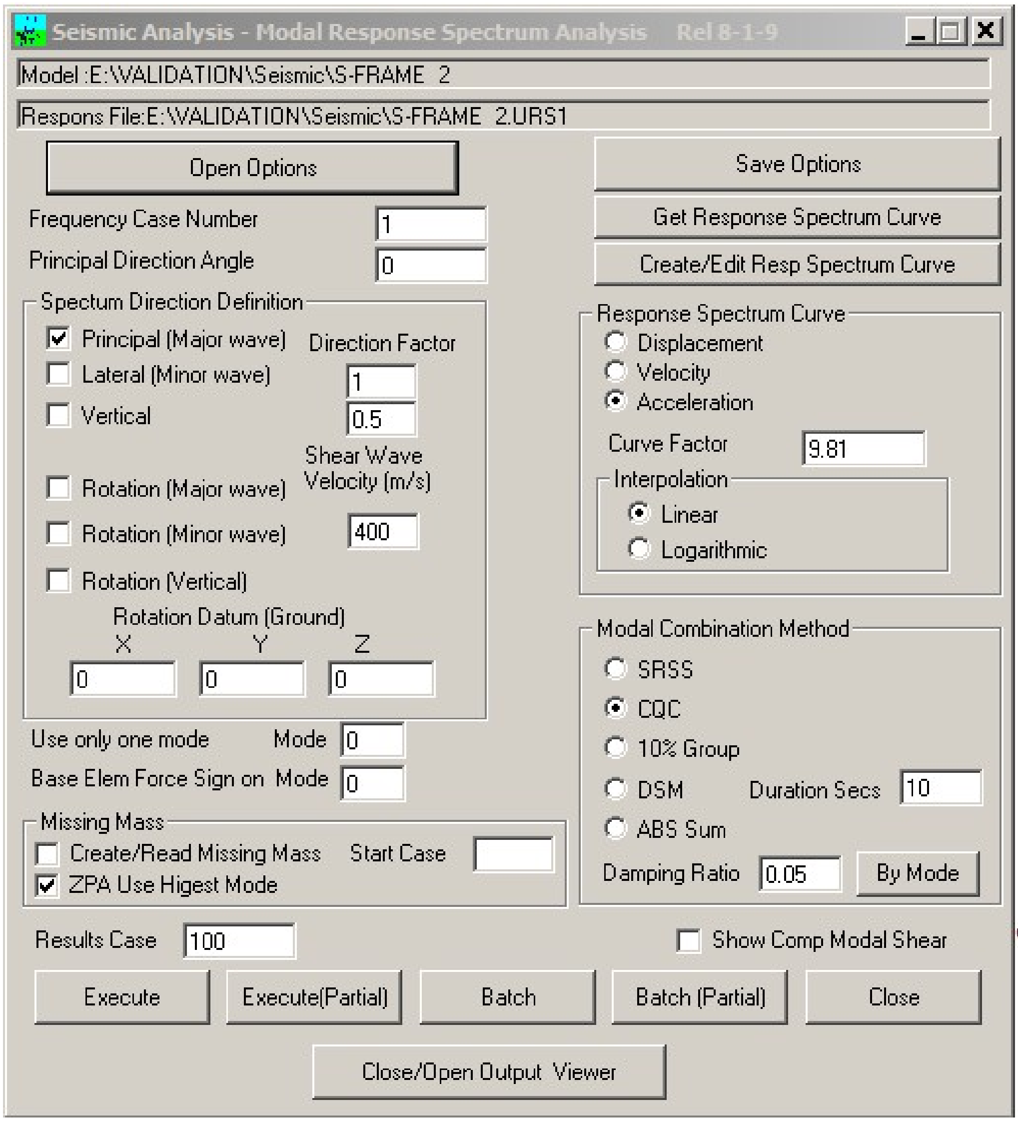
When plotting force actions always ensure the the Number of Points (Line Plot Settings) is set to 2. Similarly only undertake output processing at the nodes i.e. make No Locations on Span = 2.

-o-

4.1 Operation - Data Input

The module is started by using the Seismic Response icon on the FS2000 program group. When the module start it will load the current default mode and the following windows will become visible.

IMPORTANT NOTE The output from response spectra analysis is very different from a standard equilibrium solution and only nodal values should be considered i.e. mid span force actions should not be evaluated because the mid span evaluations will be incorrect. This restriction also applies when spectra cases are post combined with other cases - only use nodal values.



The **Open Options** and **Save Option** buttons are used to save and retrieve the current form settings. Option files are saved using a numeric id.. Prior to execution the Options must be saved.

The **Frequency Case** is used to identify the frequency input data. The case must be an existing results case from the Frequency Analysis Module.

The **Principal Direction** angle is used define the direction of the earthquake. The convention used is shown in Appendix 1.

The **Get Response Spectrum Curve** button is used to load an existing curve file. Appendix 2 describes the file format and the naming conventions.

The **Create\Edit Response Spectrum Curve** will load the user preferred editor (See FS2000 Run Editor:File command) and thus enable any spectrum file to be created/viewed or edited.

The **Response Spectrum Curve** box contains the spectrum parameter options. The parameters are described in more detail in Section 4.2.1

The **Spectrum Direction Definition** box is used to define the directional properties of the earthquake. The property options are described in Section 4.2.2

The **Use only one mode** option will evaluate loading based on the **Defined Mode** number. If this value is zero all modes will be used. The force actions etc. are signed when only one mode is processed.

The **Base Elem Force Sign on Mode** can be used to base the signage of the results on a specific dominant mode.

The **Modal Combination** method is used to select the method by which the modal contributions are combined. See Section 4.2.3

The **Missing Mass** entries are used to include the effects of high frequency rigid modes response See Section 4.2.4.

The **Show Comp Modal Shear** option if active will list the component mass participation shear loads. See Section 4.2.

The **Results Case** box defined the results case to be created. The case created requires to be post-processed before the results can be accessed.

The **Execute** and **Execute(Partial)** active the solution - see Section 4.2

The **Batch** button will append the current settings to the Seismic Response Module command line to the <ModelName>.BRM file. (see Sect 3.3)

The **Batch(Partial)** button will append the current settings for missing mass load evaluatiion to the Seismic Response Module command line to the <ModelName>.BRM

-o-

4.2 Analysis Method - Input & Results

The input to the module is described in Section 4.1. This section provides additional background to how these parameters are used and what results they will give.

The Response Spectra (Seismic) Analysis will create a load case definition file and a results case.

The analysis is done in two basic processes (after the frequency solution is undertaken):

- Evaluation of nodal displacements based on mass participation and modal displacements due to the earthquake excitation.This stage, if required, also generates the Missing Mass cases.
- Evaluation the maximum forces and displacements within the structure from each of the displacements (modal) and undertake modal and spatial combinations. This stage produces the raw Results Case that can then be Post-processed.

The **Execute(Partial)** button will only execute the first process.

The **Execute** button will evaluate both processes i.e. perform the complete solution.

Note that the load case created is a special generated load case and cannot be analysed as a standard load case. It can only be analysed using the Execute button (or by Batch Operation) using this module.

The output from the first stage of analysis is printed to an output viewer. This data is also echoed to the load case definition. This echoed data shows all relevant input data and shows the participating mass contribution of each mode.

The output from the second stage of analysis is a standard raw results case. The case requires can be post-processed in isolation or in a combination like any other results case.

Component Modal Shears

The output from the **Execute(Partial)** can also shows the modal shears in the direction of the excitations due to the effective modal mass and mode combination method. In some 3-D models (with diagonal modes) this may not be agreement with the shears obtained from the complete solution because only component shears may be shown. Only in cases where the principle modal shapes are aligned with the principle seismic excitation direction will there always be agreement. This is always achieved for 2-D solutions (no diagonal modes). Diagonal modes can be eliminated in double symmetric tower type structures by fractionally changing stiffness properties or model geometry by the very smallest amount to eliminate primary diagonal modes. The SRSS combination is not the best choice for models that posses diagonal modes. The CQC method is better because of it's ability to combine closely spaced modes and is generally recommended for 3-D analysis.

Unsigned Results

The displacements and forces etc. are different from a conventional results case in the that the resulting values are unsigned i.e. they are all positive since modal combinations methods can only use absolute values. This has implications that are not associated with conventional equilibrium result case. These are:

- **Combined Stress Evaluation**

Evaluated stresses at stress points due to combined axial and bending will invalid because the sign are not taken into account

- **Result Combinations**

If response spectrum results are to be combined with other load cases then at least two case for the same response case will have to be included i.e. +ve and -ve. This ensures that a maximum case is considered

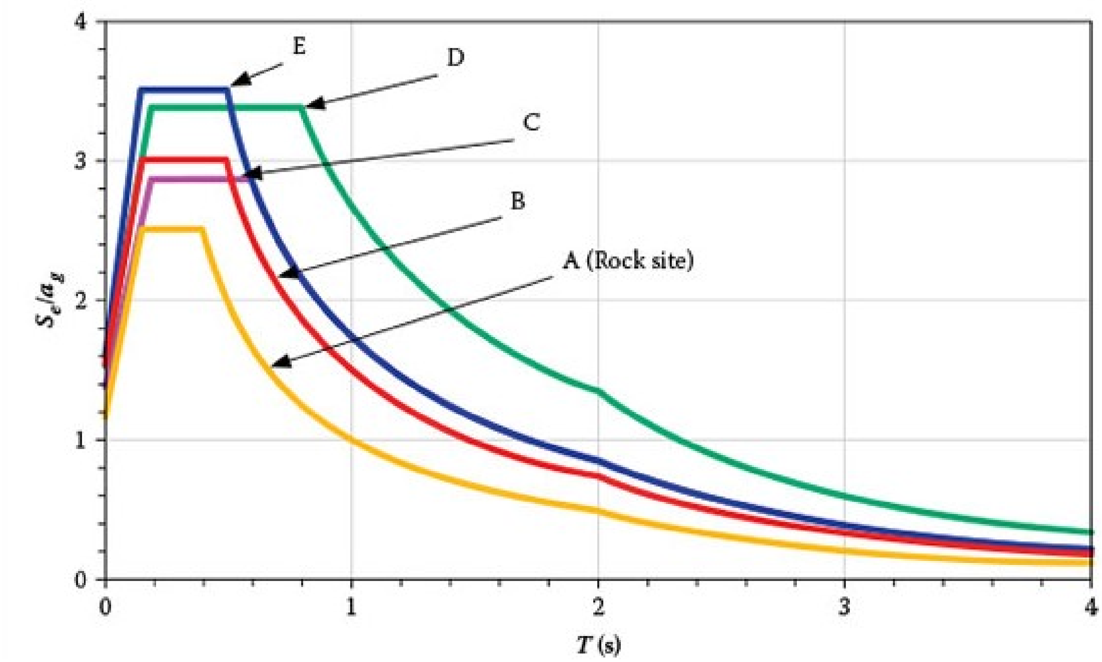
- **Important - Mid Span Forces and Moments**

Only forces at the node location are significant i.e. accurate. Attempting to evaluate mid span forces will produce invalid (wrong) results at the mid span. This can be clearly see if an attempt to draw bending moments along the length of an element. It therefore recommended that when creating output files or code checking the number of point on the span should be set to 2 i.e. only output the data at the node points, unless of course, careful consideration of mid span loading is undertaken.

Note that single mode cases will produce signed result cases.

-o-

4.2.1 Response Spectrum Curve



The response spectrum curve describes the Response Value as function of Period (Seconds). Descriptions of the file format etc. of the response spectrum data files are given in Appendix 2.

The Response Spectrum Curve option box is used to define the parameters that determine how the curve is interpreted.

The **Displacement**, **Velocity** and **Acceleration** option button are used to define the units of the response value. The modal displacement using is obtained from: $Sd = Sv / \omega$ or Sa / ω^2

The **Curve Factor** is used to factor the curve. This factor is often used to convert normalised (g force) acceleration curves to absolute acceleration curves. It can also be used to include seismic zone factors.

The response curve may be linearly or logarithmically interpolated according the users preference.

-o-

4.2.2 Spectrum Direction Definition - Directional Excitation

Rigid Ground Translation

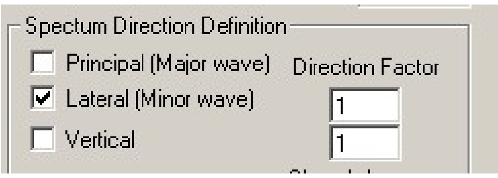
The Principal (Major) direction of the earthquake relative to the global axis of the model is defined by the **Principal Direction Angle**. The default is to align the Major axis with the x-axis and Lateral with the z-axis. If this is re-defined then the shear force is not shown after a partial solution and is only evaluate during a full solution.

Note: Vertical must always be in the global Y direction. See Appendix 1

Simultaneous ground motion in the other two transitional directions can be included by using the direction check boxes.

The Lateral (Minor) and Vertical direction each have a corresponding **Direction Factor**. This factor is used to produce the respective direction spectrum as a factor of the Principal Direction.

The entry shown below would apply the full seismic excitation at 90 degrees to the principle direction.



Rigid Ground Rotation

Rotational ground motion may be also be included but this is only applicable to tower like structures i.e. a structure in which the mass lateral motions due to rotation can be related to one specific location. The forces due to rotation are based on the translations evaluated using the offset of the masses from the centre of rotation. Only lumped translational effects are considered i.e mass rotational inertia is not considered.

The ground rotation is evaluated using the method specified in Eurocode 8 Part 3 2005 (Towers, masts and chimneys) Annex A.

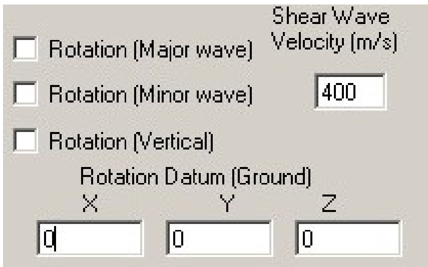
Horizontal Rotational Response Spectra = $1.7*\pi*S_d / (T*v_{SV})$

Vertical Rotational Response Spectra = $2*\pi*S_d / (T*v_{SV})$

If T < Tc then T = 0.4 (Taken from earlier code guidance)

Subsoil class	v_{SV}
A	800
B	580
C	270
D	150

A seismic wave moving in the Major axis direction would produce rotation about the Minor axis. If the Principal Direction Angle is zero this would be the x-direction and result in rotation about the z-axis.



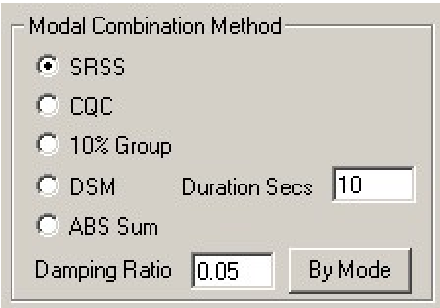
When including rotational movement the centre of the rotation of the ground relative to the model co-ordinate system must be defined. If the y origin is at ground level this need not be defined.

Spatial components are combined using the SRSS or ABSS methods. See also 4.2.3 Modal & Directional Combination Methods.

-o-

4.2.3 Modal & Directional Combination Methods

4.2.3.1. Modal Summation



Duration Secs is t_D used by Double Sum Method

The **Damping Ratio** define the damping for all modes.

The **By Mode** button cam be used to create/edit an optional modal file that define damping for individual modes used by the QCQ and DSM methods.

The following method can be used to combine the peak response from the individual modes.

SRSS

This method is not a recommended method for 3-D analysis.

$$(R^i)_k^{max} = \sqrt{\sum_\alpha \left((R^i_\alpha)_k^{max}\right)^2}.$$

QCQ - Modal Damping for individual modes is defined using the file **<modelname>.USD'm'** when **m** is the Frequency Case Number.

$$(R^i)_k^{max} = \sqrt{\sum_\alpha \sum_\beta (R^i_\alpha)_k^{max} \rho_{\alpha\beta} (R^i_\beta)_k^{max}},$$

$$\rho_{\alpha\beta} = \frac{8\sqrt{\xi_\alpha \xi_\beta}(\xi_\alpha + r_{\beta\alpha}\xi_\beta) r_{\beta\alpha}^{3/2}}{(1 - r_{\beta\alpha}^2)^2 + 4\xi_\alpha \xi_\beta r_{\beta\alpha}(1 + r_{\beta\alpha}^2) + 4(\xi_\alpha^2 + \xi_\beta^2) r_{\beta\alpha}^2},$$

10% Rule Regulatory Guide 1.92 (1976)

$$(R^i)_k^{max} = \sqrt{\sum_\alpha \left((R^i_\alpha)_k^{max}\right)^2 + 2\sum_{\alpha<\beta} \left|(R^i_\alpha)_k^{max} (R^i_\beta)_k^{max}\right|}.$$

$$\frac{\omega_\beta - \omega_\alpha}{\omega_\beta} \le 0.1, \quad \alpha < \beta.$$

Double Sum Method - Modal Damping for individual modes is defined using the file **<modelname>.USD'm'** when **m** is the Frequency Case Number.

$$(R^i)_k^{max} = \sqrt{\sum_\alpha \sum_\beta (R^i_\alpha)_k^{max} c_{\alpha\beta} (R^i_\beta)_k^{max}},$$

$$c_{\alpha\beta} = \frac{1}{1 + \left(\frac{\omega_{\alpha'} - \omega_{\beta'}}{\xi_{\alpha'}\omega_\alpha + \xi_{\beta'}\omega_\beta}\right)^2}, \qquad \omega_{\alpha'} = \omega_\alpha \sqrt{1 - \xi_\alpha^2}, \qquad \xi_{\alpha'} = \xi_\alpha + \frac{2}{t_D \omega_\alpha}.$$

ABS Sum

$$(R^i)_k^{max} = \sqrt{\sum_\alpha \left((R^i_\alpha)_k^{max}\right)^2}.$$

4.2.3.2 Directional Summation

Directional peak combination use the SRSS method apart from when the ABS Sum method for modal combination is specified. If the ABS Sum method is specified the Direction summation also uses an ABS sum method.

Running individual directions and using Load Case Combination in the post processor may be use to use other direction combination rules e.g. the 40%

$$(R^i)^{max} = \pm[(R^i)_1^{max} \pm 0.4(R^i)_2^{max} \pm 0.4(R^i)_3^{max}],$$

$$(R^i)^{max} = \pm[(R^i)_2^{max} \pm 0.4(R^i)_1^{max} \pm 0.4(R^i)_3^{max}],$$

$$(R^i)^{max} = \pm[(R^i)_3^{max} \pm 0.4(R^i)_2^{max} \pm 0.4(R^i)_1^{max}].$$

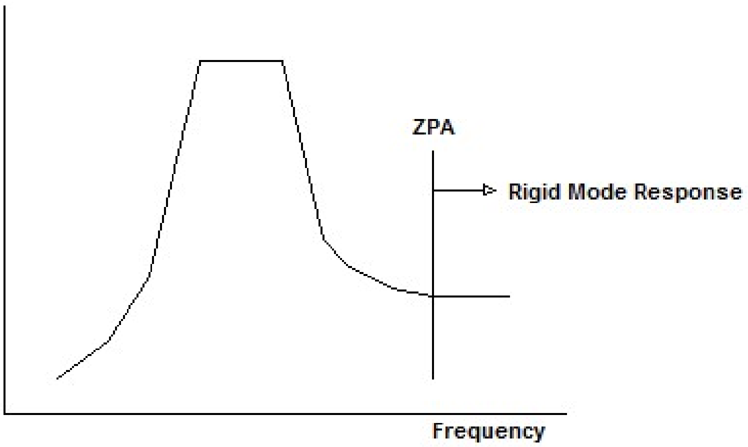
-o-

4.2.4 Missing Mass

In most cases, when using the response spectrum method, it is not practical to calculate all mode shapes and frequencies. Most building design codes have a requirement that mass participation be in the region of 90%.

For very large models with many degrees of freedom or for structures that have eigenvalues in the high-frequency range (for example, piping systems), these requirements may be difficult to meet. In such cases, the Missing Mass method may be used. In this method the missing mass is included in the seismic analysis as an extra pseudo-mode which represents the effect of the missing mass. The modal shape of this mode is evaluated by the solution of a static equivalent load case.

In this method, sometimes called the Rigid or Residual Mode Method, the static loading is evaluated using the zero period acceleration (ZPA) and accounts for the missing modes above the specified ZPA.

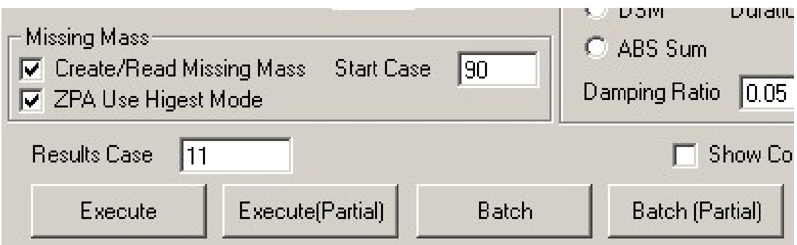


The ZPA is the acceleration used to evaluate the missing mass loads and can be defined in two ways:

- The acceleration corresponding the highest model frequency evaluated in the modal solution (**ZPA Use Highest Mode - Active**), or
- The acceleration corresponding the high frequency cut-off in the response spectrum (**ZPA Use Highest Mode - Inactive**).

The ZPA based on the highest model frequency will always produce a conservative result result providing that it is higher than that at the peak response frequency. A warning is given if the ZPA is too low.

The inclusion of the missing mass loads is a two part process in that the seismic response program has to be run twice, once to evaluate the the loading for the static solution and once to include the missing mass displacements as extra modes. A static solution of the the equivalent missing mass load has undertaken between these tow processes.



Stage 1- Missing Mass Load Evaluation

This stage is executed by clicking the **Execute (Partial Button)**. The **Create/read Missing Mass** option must be active and a **Start Case** be defined.

Up to 3 load cases will be created for each of the excitation directions. Allow for 3 load cases starting at the **Start Case** e.g. 90, 91 & 92.

A Load Case Combination (always C0) is also created that will contain active load cases.

This combination (C0) has to be run using the Std 3-D static solver prior to executing the second stage . Note that no post-processing is required, only the raw results are used by the response program.

The **Batch(Partial)** will add the following to the batch file for future batch operation.

```
SeisResp 10/11/1/
LOADA C0
OFRAME
```

Stage 2 - Response Spectrum Solution

Once the static solution of the missing mass loading has been completed the **Execute** button can be used to evaluate the complete response solution.
The additional modes due to the missing mass are combined using the active combination method e.g. QCQ.

The **Batch** will add the following to the batch file for future batch operation.

```
SeisResp 10/11/2/
```

Stage 1 and Stage 2 in Batch File

The batch commands for then complete solution including missing mass would be :

```
SeisResp 10/11/1/
LOADA C0
OFRAME
SeisResp 10/11/2/
```

-o-

4.3 Batch Command Line Operation

Providing that an appropriate <modelname>.USM(C1) option file exists the module may be operated using command line (Batch) operation.

The Response Spectra File and Frequency Case are defined in the Response Option file.

The command line is:

SEISRESP C1/C2/C3/

C1 Option File Number

C2 Results Case Number

C3 Missing Mass (1 - Missing Mass Load Case Generation, 2 - Missing Mass Displacement Inclusion)

Typical batch when missing mass is not included

```
SeisResp 10/11/
```

Typical batch when missing mass is included

```
SeisResp 10/11/1/
LOADA C0
OFRAME
SeisResp 10/11/2/
```

-o-

4.4 Files Used/Created

The only model definition files unique to the Seismic Response module are:

- Response Option
- Response Spectra
- Modal Damping Ratio

The Response Option files have the model filename with the extension **'.USMn',** where **n** is an ID number.

The Response Spectra can have any filename. If using the model name it is recommended that the default file extension **'.URSn'** be used. See Appendix 2

The Modal Damping files MUST have the model filename with the extension **'.USDm',** where **m** is the Frequency Case Number. See Appendix 2

Output Files

The output from the program is standard results case.

Load Case Files

Note: The load case definition files created are special cases. They only echo the response definition data which can be seen when the data is formatted. They should NOT be loaded into the loads modules, re-saved or otherwise be updated. Unlike other generated load case e.g. FS-Wind or FS-Seismic they cannot be analysed as standard load case.

-o-

Appendices

APPENDIX 1

Direction Angle shown is in the negative direction.

-o-

Appendix 2

Response Spectra Curves - File Formats

Text files are used for the definition of Response Spectra Curves. **<modelname>.URS(n)** where **n** is a numeric I.D.

The file format use is shown in the examples below. The frequency is defined using the period in seconds .

The first column is the spectra period in seconds.   The second column contains the corresponding data values.   Commas or spaces can be used as delimiters, do not use TABS

0.4,   1

0.2 ,   2.5

0.4,   2.5

0.5,   2.1

0.75,   1.175

1.0,   1.05

1.25,   0.82

1.5 ,   0.7

2,   0.5

2.5,   0.4

3,   0.33

-1

These data values may be displacement, velocity or acceleration (Ensure that Response Spectrum Option buttons reflect the type used).

Any number of points may be used but the file must include a -1 termination character.

Any valid filename may be used for the data.   If the response file is model dependent it is recommended the default naming convention be used i.e. <modelname>.UR(n) where n is a numeric I.D.. This will ensure the data file is archived with the model.

The Spectra data is always echoed in the Load definition case.

Modal Damping Ratios - File Formats

Text files are used for the optional definition of individual Modal Damping Ratios. The damping is defined by reference to the mode number obtained from the frequency solution. They are only used for combining spectral modal response.

Modal Damping for individual modes is defined using the file **<modelname>.USD'm'** when **m** is the Frequency Case Number

The file format use is shown in the examples below:

The first column is the mode number.   The second column contains the damping ratio.   Commas or spaces can be used as delimiters, do not use TABS

1, .04

3, .03

-1

The first column is the spectra period in seconds. The second column contains the corresponding data values. Commas or spaces can be used as delimiters, do not use TABS.

The defined Damping Ration will be used for modes that are not specifically defined.

-o-

Appendix 3

Response Analysis - Command Line Instructions

The following sections describe the Model Definition Commands and their associated arguments.

Only single commas may be used as delimiters for Command Arguments

All command arguments MUST be defined there are no default values.

The underlined commands are command that can only be entered into the Response Case using a text editor. These command are shown below.

FCURVE *Defines a frequency/amplitude curve*

INIDISP *Initial condition for transient analysis*

INIVEL *Initial condition for transient analysis*

TCURVE *Defines a time history*

TFILE *Specifies a file defining a time history*

MAXTIME *Changes the maximum default(1000) number of point for a time history curve*

HCURVE *Defines a time history in terms of a damped harmonic curve*

The command are divided into the following sections

Harmonic Response Analysis

Transient Response Analysis

-o-

Harmonic Response Analysis

DAMP, *Mode, Value*

Defines the critical damping ratio for each vibration mode. If no damping values are defined a default damping ratio of 0.05 is used for all modes (use a small value 1E-12 to exclude damping). If the damping ratio for only the first mode is defined then all other modes will be assigned this value.

Mode Vibration mode

Value Critical damping ratio

RDAMP, *Alpha, Beta*

Defines the Rayleigh Damping coefficients α and β. If damping ratio is not defined for a specific mode then it value will be evaluated using the specified coefficients.

Apha α

Beta β They relate to the more usual 'damping ratio' γ by the expression $\gamma = (\alpha + \beta\omega**2) / 2\omega$

Harmonic Excitation Commands

HFORCE, *Node, Force, Phase, Direction*

Harmonic excitation due to a concentrated nodal force or moment.

Node Node to which excitation is applied

Force Applied Force (N)

Phase Phase angle in degrees

Direction Global Freedom Direction (1 to 6)

RUFORCE, *Node, Mass, Radius, Phase, Direction*

Harmonic excitation due to an unbalanced rotating mass producing a concentrated nodal force.

Node Node to which excitation is applied

Mass Unbalanced Mass (kg)

Radius Radius at which the mass rotates (m)

Phase Phase angle in degrees

Direction Global Freedom Direction (1 to 3)

LSUPP, *XValue, YValue, Zvalue*

Harmonic excitation due to linear movement of the of the restraint points (base excitation). Motion may be displacement, velocity or acceleration (see SMOTION). The default is displacement. Base excitation cannot be applied with nodal excitation. Base excitation takes precedence.

XValue Translation in global x direction in m

YValue Translation in global y direction in m

ZValue Translation in global z direction in m

RSUPP, *XValue, YValue, ZValue, XDat, YDat, ZDat*

Harmonic excitation due to rotational movement of the of the restraint points (base excitation). Motion may be displacement, velocity or acceleration (see SMOTION). The default is displacement. Base excitation cannot be applied with nodal excitation. Base excitation takes precedence.

XValue Rotation in global x direction in Radians

YValue Rotation in global y direction in Radians

ZValue Rotation in global z direction in Radians

XDat *Centre of x* rotation relative to model origin

YDat *Centre of y* rotation relative to model origin

ZDat *Centre of z* rotation relative to model origin

SMOTION, *Value*

Define the type of motion used by support motion excitation

Value 0 Displacement - Default value - No command req'd

1 Velocity

2 Acceleration

Output - Harmonic Response

NDOUT, *Node, Direction*

Plots and list displacements for a selected node and freedom direction.

Node Node at which output is required

Direction Global Freedom Direction (1 to 6)

EFOUT, *Node, Direction* Force/Moment Output Mode

Plots and list force actions for a selected element and freedom direction.

Element Element at which output is required

Node Node at which output is required

Direction Local Freedom Direction (1 to 6)

EFOUT, *Node, -Direction, Point* Stress Output Mode

Plots and list stresses for a selected element and freedom direction. This is the same command as the force command but a - before the Direction signifies that a stress output is required.

Element Node at which output is required

Node Node at which output is required

Direction Local Freedom Direction (1 to 6) ***Must be negative***

Point Point on section where the stresses are required (only effects bending)

For beams

Point = 1 to 4 the standard stress points

For pipes

Point =1 to 12 (15 deg increments), Point 1 is the top (local y direction)

FREQRESP, *StartFreq, EndFreq, NoPoints*

Define the frequency range (Hz) over which output is required. This command takes precedence over the FCURVE command.

StartFreq	Start frequency
EndFreq	End frequency
NoPoints	No of points

FCURVE, *No, NoPoints, Type*

Freq1, Amplitude1

Freq2, Amplitude2

to

FreqN, AmplitudeN where N is NoPoints

Define the frequency range (Hz) over which output is required in terms of a defined curve.

No	Curve Reference Number	
NoPoints	Number of points on curve	
Type	Curve type	
	1	Frequency by value
	2	Frequency by increment
FreqN	Frequency Hertz (cycles/s)	
AmplitudeN	Amplitude Factor	

TIMERESP, *StartTime, EndTime, NoPoints*

Define the time response period over which output is required for a given excitation frequency defined by the **FREQ** command .

StartTime	Start time
EndTime	End time
NoPoints	No of points

FREQ, *Value*

Value	Forcing frequency in Hertz (cycles/s)

FCASE, *CaseNo, Time, Freq*

This command will create a results case at a specific time (phase) and frequency. The results case created is a standard results case must be post-processed in FS2000 like any other result case.

CaseNo	Results Case Number
Time	Time (Phase) in seconds
Freq	Frequency (Hz)

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

INFORCE, *StartCaseNo*

This command will create a series of load case files containing dynamic equivalent static loads for a specific frequency at times (phases) specified by the FREQ and TIMERESP commands . For each point in the TIMERESP command a load case will be created. An increment of one is used for each load case. The load cases created are standard load cases containing nodal forces and can be run using the linear solver.

StartCaseNo	Results Case Number

-o-

Transient Response Analysis

DAMP, *Mode, Value*

Defines the critical damping ratio for each vibration mode. If no damping values are defined a default damping ratio of 0.05 is used for all modes (use a small value 1E-12 to exclude damping). If the damping ratio for only the first mode is defined then all other modes will be assigned this value.

Mode	Vibration mode
Value	Critical damping ratio

RDAMP, *Alpha, Beta*

Defines the Rayleigh Damping coefficients α and β. If damping ratio is not defined for a specific mode then it value will be evaluated using the specified coefficients.

Apha α

Beta β They relate to the more usual 'damping ratio' γ by the expression $\gamma = (\alpha + \beta\omega**2) / 2\omega$

Transient Excitation Commands

TFORCE, *Node, Force, History, Direction*

Excitation due to a non-periodic nodal force. The accuracy of this is a function of the time step in time history. The solution uses a closed form of solution with no integration therefore accurate stable results can be obtained for large time steps since the solution assumes that the slope of the excitation is constant between points.

Only one excitation per degree of freedom is allowed. The time co-ordinates (time steps) of all time histories must be the same. They may be of different duration.

Node	Node to which excitation is applied
Force	Applied Force (N)
History	History curve reference number see TCURVE and HCURVE commands
Direction	Global Freedom Direction (1 to 6)

IMPULSE, *Type, T1, T2, NoPts, Node, Force, Direction*

Excitation due to shock/impact on an undamped system. The default response period is taken as 2 time the pulse duration. The TIMERESP command may be used to change the time intervals of the output response . IMPULSE cannot be applied with TFORCE. IMPULSE takes precedence. IMPULSE cannot be applied to base excitation.

Type	Shock Curve Type		
	1	Rectangular pulse	T1= Total pulse duration
	2	Triangular pulse	T1= Fall time = Total pulse duration
	3	Triangular pulse	T1= Rise time T2 =Total pulse duration
	4	Half sine pulse	T1=Total pulse duration
T1	Pule duration		
T2	Pulse duration - Only used with Type 3 pulse (use 0 for others)		
NoPts	Number of points on the output time curve		

Force	Applied Force (N)
Direction	Global Freedom Direction (1 to 6)

LSUPP, *XValue, YValue, Zvalue*

Transient excitation due to linear displacement of the of the restraint points (base excitation). Base excitation cannot be applied with nodal excitation. Base excitation takes precedence. This command always uses Curve Ref No 1.

XValue	Translation in global x direction in m
YValue	Translation in global y direction in m
ZValue	Translation in global z direction in m

RSUPP, *XValue, YValue, ZValue, XDat, YDat, ZDat*

Transient excitation due to rotational displacement of the of the restraint points (base excitation). Base excitation cannot be applied with nodal excitation. Base excitation takes precedence. This command always uses Curve Ref No 1.

XValue	Rotation in global x direction in Radians
YValue	Rotation in global y direction in Radians
ZValue	Rotation in global z direction in Radians
XDat	*Centre of x* rotation relative to model origin
YDat	*Centre of y* rotation relative to model origin
ZDat	*Centre of z* rotation relative to model origin

INIDISP, *Node, Displacement, Direction*

Node	Node to which condition is applied
Displacement	Applied displacement (m)
Direction	Global Freedom Direction (1 to 6)

INIVEL, *Node, Velocity, Direction*

Initial velocity condition for a defined node

Node	Node to which condition is applied
Velocity	Applied velocity (m/s)
Direction	Global Freedom Direction (1 to 6)

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

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 maximum number of point is 1000. 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 .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

MAXTIME, *MaxPts*

This command will change the default (1000) maximum number of points that can be defined in the TCURVE command.

MaxPts	Maximum number of points

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

$$F(t)=(C1.\sin(C2.t+C3) \quad + \quad C4.\cos(C5.t+C6)).\mathrm{Exp}(C7.t)$$

Output - Transient Response

The output for transient response is defined explicitly by the time history excitation. The exception is for shock/impact response on an undamped system which can optionally use the TIMERESP command.

NDOUT, *Node, Direction*

Plots and list displacements for a selected node and freedom direction.

Node Node at which output is required

Direction Global Freedom Direction (1 to 6)

EFOUT, *Node, Direction* Force/Moment Output Mode

Plots and list force actions for a selected element and freedom direction.

Element Element at which output is required

Node Node at which output is required

Direction Local Freedom Direction (1 to 6)

EFOUT, *Node, -Direction, Point* Stress Output Mode

Plots and list stresses for a selected element and freedom direction. This is the same command as the force command but a - before the Direction signifies that a stress output is required.

Element Node at which output is required

Node Node at which output is required

Direction Local Freedom Direction (1 to 7) ***Must be negative***

7 will give the combined direct + bending stress

Point Point on section where the stresses are required (only effects bending)

For beams

Point = 1 to 4 the standard stress points.

If Point = 0 then maximum (axial+ bending) will be given.

For pipes

Point =1 to 12 (15 deg increments), Point 1 is the top (local y direction)

If Point = 0 Then maximum (axial+ bending) will be given.

TIMERESP, *StartTime, EndTime, NoPoints*

Define the time response period over which output is required for Shock/Impulse response (IMPULSE command). This is an optional command which extends the time period for the output response. It must follow the IMPULSE command.

It also defines the response period when only initial conditions are specified (INIDISP or INIVEL commands.

StartTime Start time

EndTime End time

NoPoints No of point

TCASE, *CaseNo, Time*

This command will create a results case at a specific time on the response curve. The results case created is a standard results case must be post-processed in FS2000 like any other results case.

CaseNo	Results Case Number
Time	Time

-o-

Appendix 4

Validation Examples

DynFreqExp1 Natural frequency of a Two-Mass spring system.

Ref: Thomson W.T. Vibration Theory and Applications, Prentice Hall

M1=1 M2=0.5 k=200 Kc=800

Truss elements with zero density are used as springs. The model uses the 2-D frequency solution.

		Theory	FS2000
Results	Node 1	2.581 Hz	2.581 Hz
	Mode 2	8.326 Hz	8.326 Hz

DynFreqExp2 Natural frequency of a Cantilever Beams

Evaluate the first 3 modes.

Ref: Thomson W.T. Vibration Theory and Application, Prentice Hall

L=80 E=30E6 I=1.3333 A=4 Density=0.7272E-3

The model uses the 2-D frequency solution. Mass case is Load Case 1

		Theory	FS2000(Lumped)	FS2000(Consistent)
Results	Node 1	10.25 Hz	10.24 Hz	10.25 Hz
	Mode 2	64.25 Hz	63.97 Hz	64.25 Hz
	Mode 3	179.91	178.6 Hz	179.90 Hz

DynFreqExp3 Natural frequency of a 3-D Beam Structure

Evaluate the first 5 modes.

Ref: COSMOS Validation /ASME Pressure Vessel and Piping 1972, Programs Verification, Pub I-24.

The model uses the 3-D frequency solution. Mass case is Load Case 1

		Ref Hz	FS2000 Hz
Results	Node 1	111.2	111.2
	Mode 2	115.8	115.8
	Mode 3	137.1	137.1
	Mode 4	215.7	215.7
	Mode 5	404.2	404.2

DynRespExp1 Dynamic Response - Rotating unbalance

A flywheel of mass 180 kg rotates at 1200 rpm at the end of a massless cantilever. The CofG of the flywheel is offset 2.5 mm from the centre of rotation. Damping is 0.2 critical in all modes.

L=2.5m A=6.11E-3 m2 I=3.74E-5 m4

What are the delections at the end of the flywheel.

The basic command used in the response modules are

RUFORCE,2,180.0,0.0025,0,2	Unbalance in the y direction
RUFORCE,2,180.0,0.0025,-90.0,1	Unbalance in the y direction (90 out of phase)
DAMP,1,.2	Damping
NDOUT,2,2	Displacement at Node 2 in y direction
NDOUT,2,1	Displacement at Node 2 in y direction
FREQRESP,10.0,25.0,25	Frequency range for deflections output
FREQ,20.0	Frequency and periods for time response
TIMERESP,0,0.05,50	ie 0 to .05 secs at 50 intervals at 20 Hz

Results

Flexural mode 14.393 Hz

Axial mode 265.5 Hz

Maximum Vertical deflection = 4.452 mm

Maximum horizontal deflection = 0.01426 mm

The results from FS2000 are in exact agreement with theory.

DynRespExp2 Dynamic Response - Pile Driving Impact

Ref: Clough & Penzien, Dynamics of Structures McGraw-Hill 1975

A concrete pile with a rigidly support tip is subjected to a half sine pulse at the head. Evaluate the stress in the pile at t=0.0104 s and t=0.0133 s.

L=100 ft A=4000 ins2 E=3E5 psi Density=150 pcf

Half sine pulse 600 kips t=.005

The basic command used in the response modules are

```
IMPULSE,4,0.005,0,20,2,-6.000E05,2
TCASE,10,0.0104
TCASE,11,0.0133
TIMERESP,0.0,0.02,200
NDOUT,2,2
EFOUT,51,52,1
EFOUT,1,1,1
```

Result Case 10 & 11 show the axial stresses distribution in the pile at t=0.0104 and t=0.0133 respectively. At t=0.0104 the stress wave reaches the tip, At t=0.0133 the wave is being reflected from the tip of the pile.

	Ref	FS2000
Stress at t=0.0104	-1.5	-1.5
Stress at t=0.0133	-2.91	-2.95

DynRespExp3 - Blast Loading on SDOF Building

Ref: Clough & Penzien, Dynamics of Structures McGraw-Hill 1975 - Converted to SI units

A building is subject to a triangular blast pulse. What is the maximum deflection and shear load.

Transverse stiffness = 1750 MN/m

Effective mass = 272.16 tonne

Blast pulse 4448 kN pulse length=0.05s

The solution is obtained using two distinct methods. Load Case2 uses the IMPULSE command. Load Case 3 using the TFORCE command.

Load Case 2

The basic commands used in the response modules are:

```
IMPULSE,2,0.05,0,40,5,4448.0E3,1
EFOUT,1,1,2
NDOUT,5,1
TCASE,10,0.0325
```

This gives the maximum deflection at 0.0325 s with a maximum shear force of 5947 kN

Load Case 3

The basic commands used in the response modules are:

```
DAMP,1,1E-10
TFORCE,5,1.0,1,1
TCURVE,1,9,1
0,4448E3
.007143 ,3812.6E3
.01428  ,3177.1E3
.021428 ,2541.71E3
.02857  ,1906.2E3
.0325   .1556.8E3
.03571  ,1270.85E3
.042857 ,635.42E3
.05     ,0
EFOUT,1,1,2
NDOUT,5,1
```

This gives the maximum deflection of 3.385 mm at 0.0325 s with a maximum shear force of 5921 kN

Load Case 4

This is the same as case 3 but uses the TFILE command.

```
DAMP,1,1E-10
TFORCE,5,1.0,1,1
TFILE,1,1,model.UTCURVE
EFOUT,1,1,2
NDOUT,5,1
```

DynRespExp4 - Blast Loading on SDOF Tower

Ref: Clough & Penzien, Dynamics of Structures McGraw-Hill 1975 Page 104

A water is subject to a triangular blast pulse. What is the maximum deflection and shear load.

Transverse stiffness = 2700 kips/ft

Effective mass = 96.6 kips

Blast pulse 96.6 kips pulse rise=0.025 s pulse fall=0.025 s

The solution is obtained using two distinct methods. Load Case2 uses the IMPULSE command. Load Case 3 & 4 using the TFORCE command.

Load Case 2 Undamped

The basic commands used in the response modules are:

```
IMPULSE,3,0.025,0.05,20,1,96.6e3,1
EFOUT,4,1,1
NDOUT,1,1
TCASE,20,0.05
```

At t=0.05 s

	Ref	FS2000
Deflection	0.0176	0.0174
Shear	47.6	47.11

Load Case 3 Undamped

The basic commands used in the response modules are:

```
DAMP,1,1E-10
NDOUT,1,1
EFOUT,4,1,1
TFORCE,1,1.0,1,1
TCURVE,1,41,1
0,0
.005, 19.32e3
.01,  38.64e3
.015, 57.96e3
.02,  77.28e3
.025, 96.6e3
.03,  77.28e3
.035, 57.96e3
.04,  38.64e3
.045, 19.32e3
.05,  0
```

At t=0.05 s

	Ref	FS2000
Deflection	0.0176	0.0174
Shear	47.6	47.11

Load Case 3 Undamped

At t=0.05 s

	Ref	FS2000

Deflection	0.0176	0.0174
Shear	47.6	47.11

Load Case 4 5% Damped

The basic commands used in the response modules are as above but with

```
DAMP,0.05
```

At t=0.05 s

	Ref	FS2000
Deflection	0.0169	0.0167
Shear	45.7	45.17

DynRespExp5 Frequency response of a two mass system

Determine the frequency response of a symmetric two mass system when excited by a harmonic force at mass M2.

Ref: Thomson W.T. Vibration Theory and Application, Prentice Hall

K=200 N/m M=0.5 kg

F=2sin(wt) N

The basic commands used in the response modules are:

```
DAMP,1,1E-10
HFORCE,2,2.0,0,1
FREQRESP,0.0,7.5,31
FREQ,6.5
TIMERESP,0.0,0.1538,50
NDOUT,2,1
NDOUT,3,1
```

The model uses spar elements to model the springs.

The results are in exact agreement with the reference solution.

Freq	X2	Phase	X3	Phase
1.5	8.227	0	4.627	0
4	5.115	180	12.153	180
6.5	5.851	180	2.697	0

DynRespExp6 Frequency response due to support movement

Determine the natural frequency and error of a vibrometer when being operated at 22.4353 rad/s

Ref: ANSY/Thomson W.T. Vibration Theory and Application, Prentice Hall

K=9.8696 M=1

Support motion =sin(wt)

The basic commands used in the response modules are:

```
LSUPP,0.0,1.0,0
FREQRESP,0,3.57,100
NDOUT,2,2
```

The model uses spar elements to model the springs.

The results are in exact agreement with the reference solution.

Nat Freq = 0.5 Hz   Measured Movement = 1.02   Error is 2%

DynSeismExp1 Seismic Response of a Three-Story Frame

Determine the maximum RMS displacements and shear forces

Ref: COSMOS/Biggs, J.M

E=30E6   A=10   L=144

I1=12442   I2=8294   I3=4147

Response Spectrum Curve (Saved as MODEL.URS1)

Period (s)	Displacement
0.2	0.1
0.5	1.6
2.5	8
100.0	8

The Response case options are saved as Options Case 1

RMS Displacement

Node	Biggs	FS2000	COSMOS
1	1.5	1.470	1.4666
2	3.24	3.113	3.111
3	5.03	4.557	4.554

RMS Shear Force

Ele	Biggs	FS2000	COSMOS
1	2250	2205	2199
2	1740	1713	1709
3	895	885	874

DynSeismExp2 Seismic Response of a Five-Story Frame

Determine the maximum RMS forces when subjected to vertical base motion.

Ref: COSMOS/Salmonte, A. J.

E=2E10   A 1 to 4 =1.6 m2   A5 = 80 m2   L=4 m

M1=100 tonne   M2 to 4= 160 tonne   M5=800 tonne

Response Spectrum Curve (Saved as MODEL.URS1)

Period (s)	Acceleration (m/s2)
0.001	1.0
0.04	1.0
0.05	2.5
1.00	2.5

The Response case options are saved as Options Case 1

RMS Axial Force kN

Ele	Ref	FS2000
1	321	321
2	784	784
3	1135	1135
4	1327	1327
5	1548	1548

DynSeismExp3 Seismic Response of a SS Beam

Determine the fundamental displacement and corresponding bending stress.

Ref: ANSYS/Biggs

E=10E6 m=0.2

A=273.97 ins2 I=333.33 ins4 L=4 m d=14 ins

Response Spectrum Curve (Saved as MODEL.URS1)

Period (s)	Displacement (ins)
0.1	0.44
10	0.44

The Response case options are saved as Options Case 1

	Ref	FS2000
F(Hz)	6.097	6.096
Defl	0.553	0.553
Stress	20156	20160

-o-