Overview of ACS SASSI NQA V4.3 Application to Seismic SSI Analysis of Safety-Related NPP Buildings

Ghiocel Predictive Technologies Inc.

Dr. Dan M. Ghiocel

Member of ASCE 4 & 43 Standards

Email: dan.ghiocel@ghiocel-tech.com

Ghiocel Predictive Technologies Inc.

http://www.ghiocel-tech.com

Part 3: ACS SASSI NQA V4.3 Software Description and Use

GP Technologies, Inc., Rochester, New York

October 2021

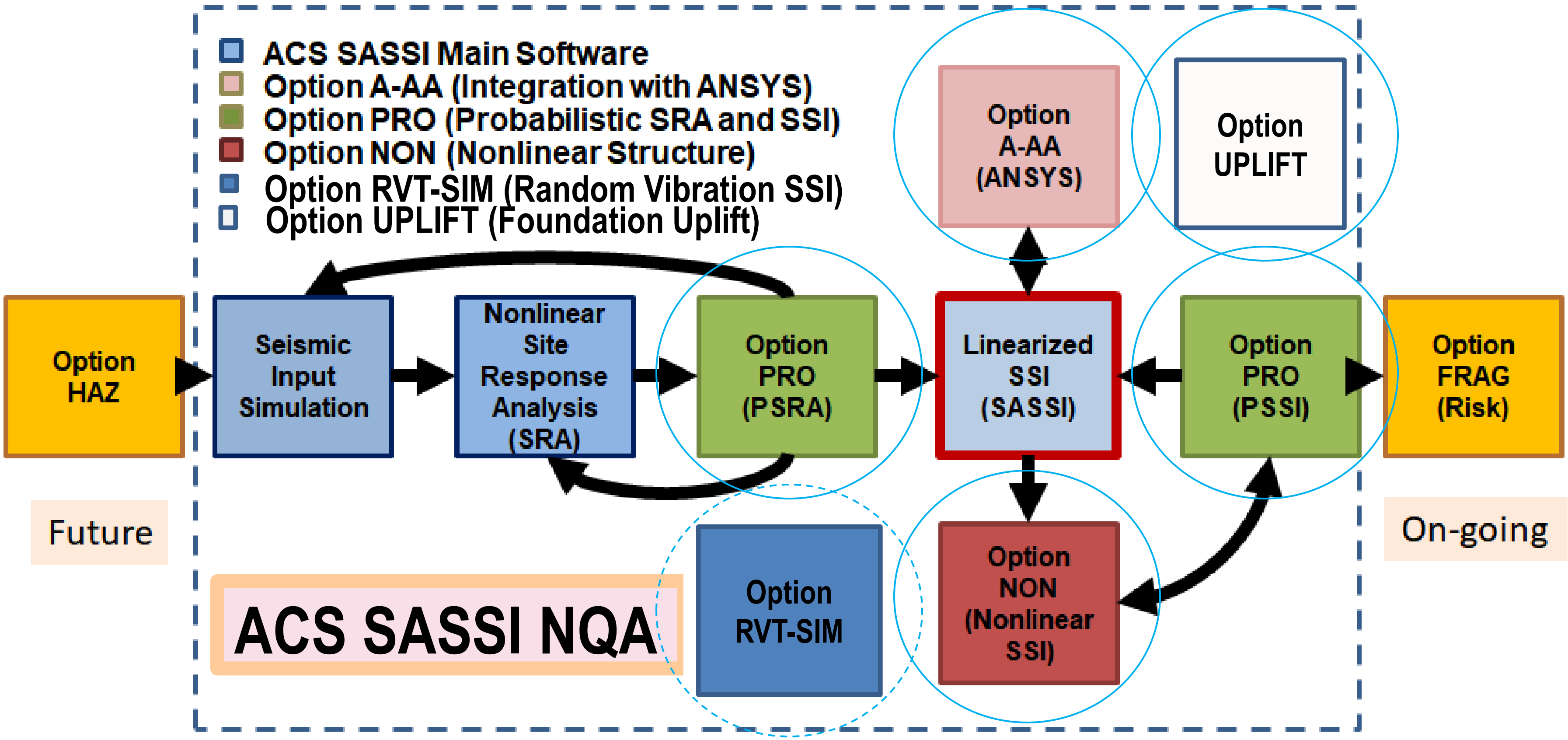
Part 3: Presentation Content:

1. Main Software Toolboxes and FE Modeling Basics
2. Main Software Modular Configuration
3. User Interface (UI) Input Dialog for Selecting Analysis Options for SSI Modules
4. Description of SSI Response Text Files and Frames (for Animations)
5. Main UI Menus for Model Inputs, Performing SSI Analysis and Plotting Results
6. UI General Commands
7. UI Commands for Generating and Checking FE Models
8. Batch Run Mode Commands for SSI Modules and Auxiliary Programs
9. Post-Processing Using UI Plotting Commands and Binary Databases
10. Automatic Post-Processing Using Multilevel UI Command Macros
11. Section-Cuts Capabilities for Shell and Solid Elements
12. UI Parametric Language Commands for Building SSI FE Models
13. Brief Review of Demo Contents
14. Verification and Validation

1. Main Software Toolboxes and FE Modeling Basics

ACS SASSI NQA V4.3 Toolboxes

Present/Options A-AA, NON and PRO and UPLIFT
Future/Options HAZ and FRAG



2021 ACS SASSI V4 Software Toolboxes

1) **Main Software.** Include advance pre-post processing, nonlinear soil modeling, motion incoherency, others. Plus, includes seismic motion simulation and site response capabilities.

2) **Option A-AA.** Integration with ANSYS. The ANSYS structure FE models can be used directly for the 1st step of the overall SSI analysis (Option AA, AA-R), and/or in the 2nd step for the detailed stress analysis using the SSI responses as input BCs (Option A)

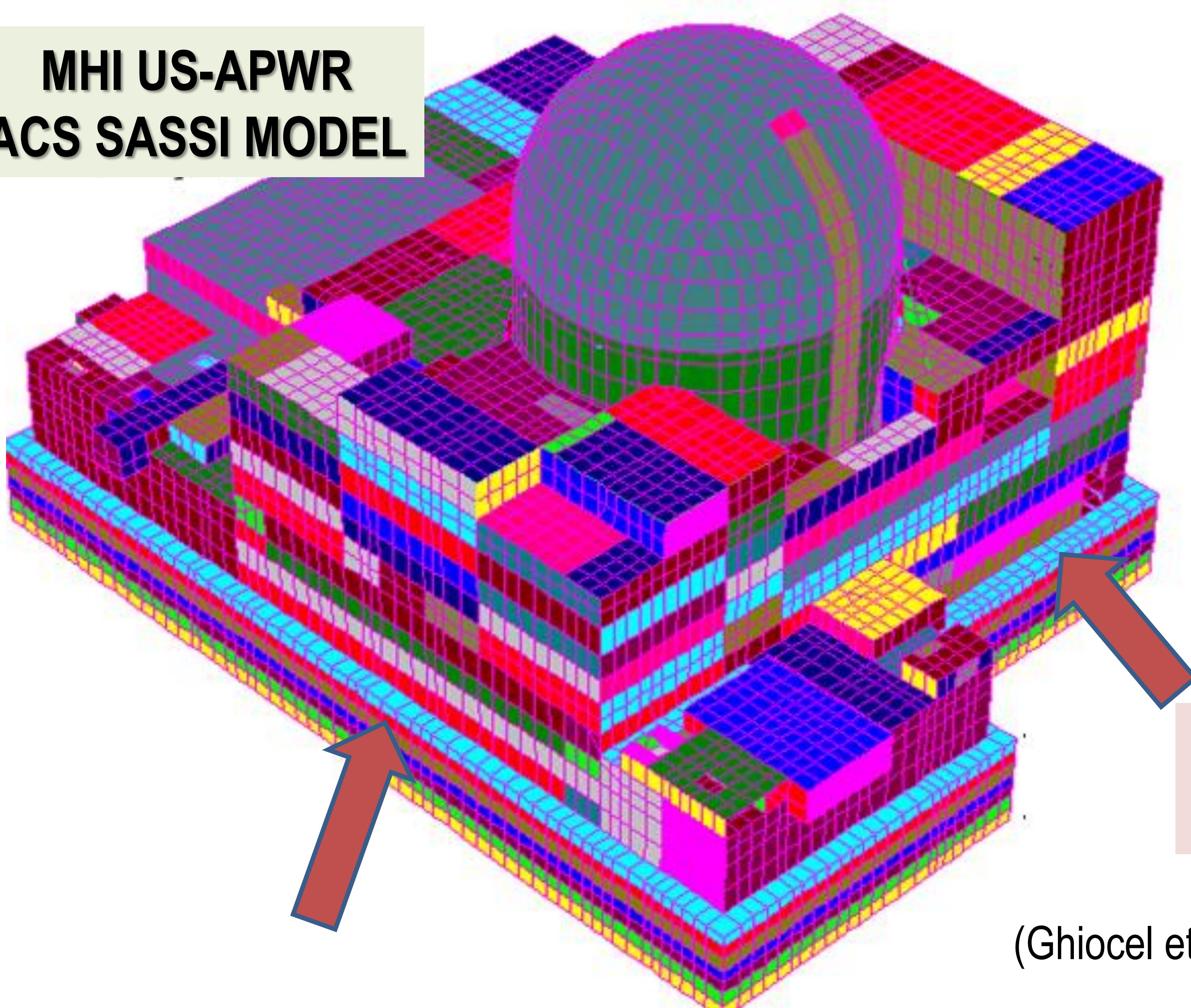
3) **Option NON.** Nonlinear structure, applicable to concrete structures and base-isolation using iterative scheme (ASCE 4-16, ACI-318, and JEAC 4601-2015).

4) **Option PRO.** Probabilistic SRA and SSI analyses (ASCE 4-16 Sections 2 and 5.5, and RG 1.208 E)

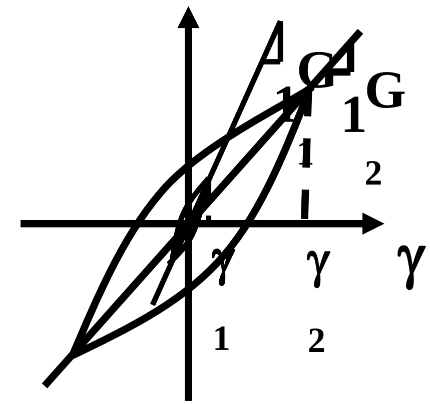
5) **Option RVT-SIM.** No input time histories are required.

ACS SASSI Main Software Modeling for Linearized SSI Analysis (Optionally with Nonlinear Soil)

MHI US-APWR ACS SASSI MODEL



Iterative EQL



Nonlinear soil solid elements

(Ghiocel et. al., 2013, SMIRT22)

ACS SASSI V4 Finite Element Library

The FE library allows the use of the following linear element types:

3D solid elements	type SOLID
3D beam elements	type BEAMS
3D plate / thin shell elements	type SHELL
3D plate / thick shell elements	*type TSHELL*
2D plane strain elements	type PLANE
3D spring elements	type SPRING
3D stiffness/mass generalized elements	type GENERAL (SE)
3D highly viscous damper elements	type HVD

The excavated soil is modeled using the following element types:

SOLID for 3D FEM
PLANE for 2D FEM

Element Node Degrees of Freedom (Dofs)

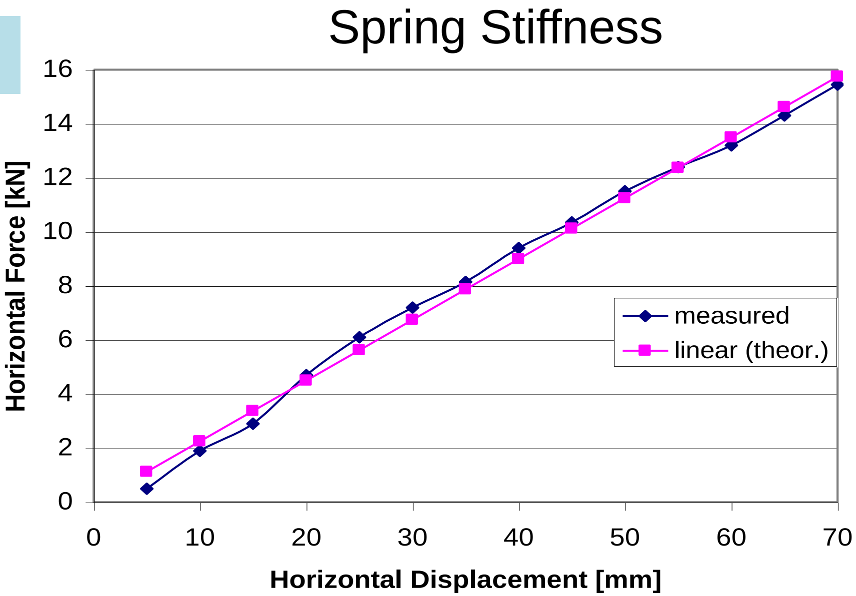
Element	Node DOFs					
Type	X	Y	Z	XX	YY	ZZ
SOLID	•	•	•			
BEAM	•	•	•	•	•	•
SHELL/ TSHELL	•	•	•	•	•	•
PLANE	•		•			
SPRING	•	•	•	•	•	•
HVD	•	•	•			
GENERAL	•	•	•	•	•	•

Element Group Names

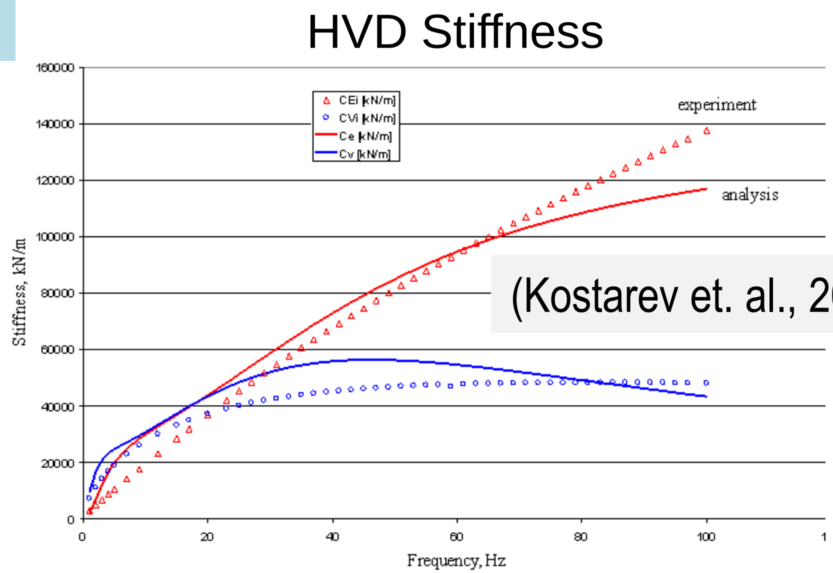
Group Type (number)	Group Type (string)	Number of Nodes	Description
1	SOLID	8, or 7, or 6, or 5	3D solid element
2	BEAMS	3	3D beam elements
3	SHELL	4 or 3	3D plate/shell elements
5	TSHELL	4 or 3	3D plate/shell elements
4	PLANE	4 or 3	2D plane strain solid elements
6	HVD	3	3D High Viscosity Damper
7	SPRING	2	3D spring elements (translation or rotation)
9	GENERAL	3 (local axes) or 2 (global axes)	3D stiffness/mass generalized element

New 3D HVD Elements Simulate BCS Isolators

3D Springs



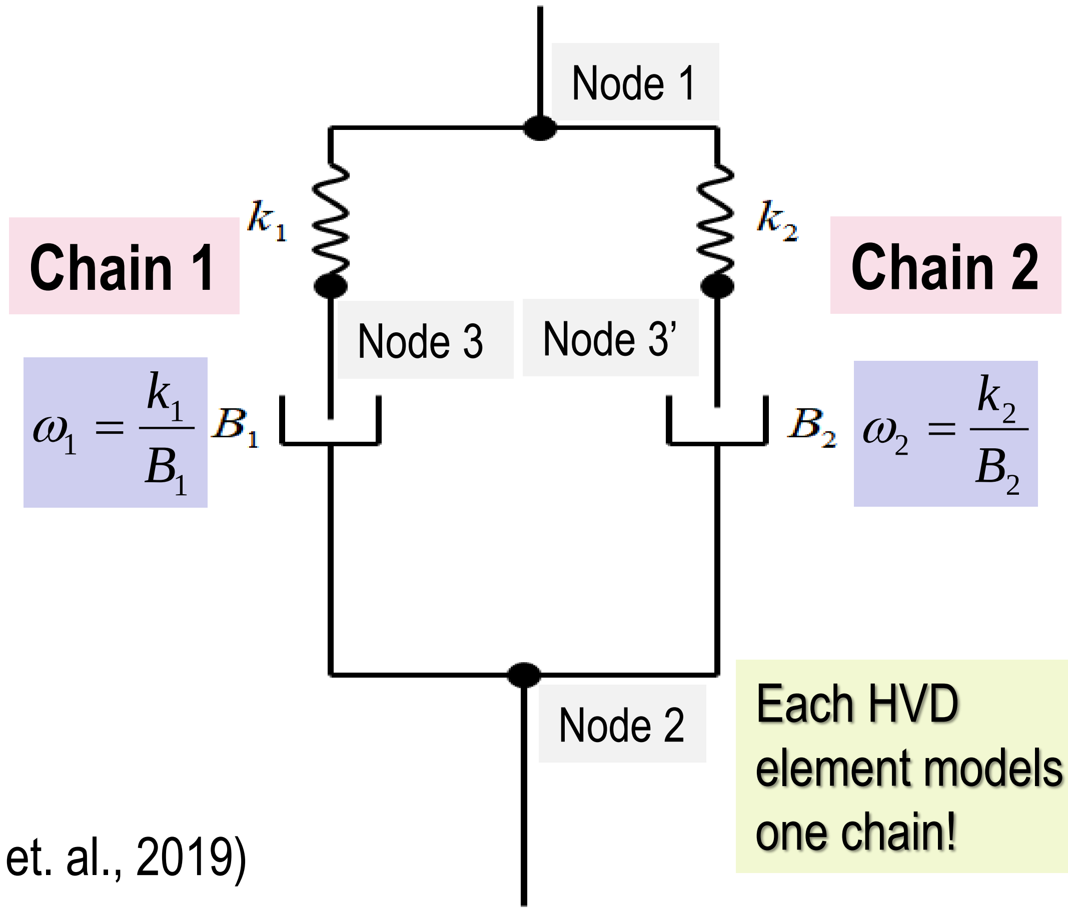
3D HV Dampers



(Kostarev et. al., 2018)

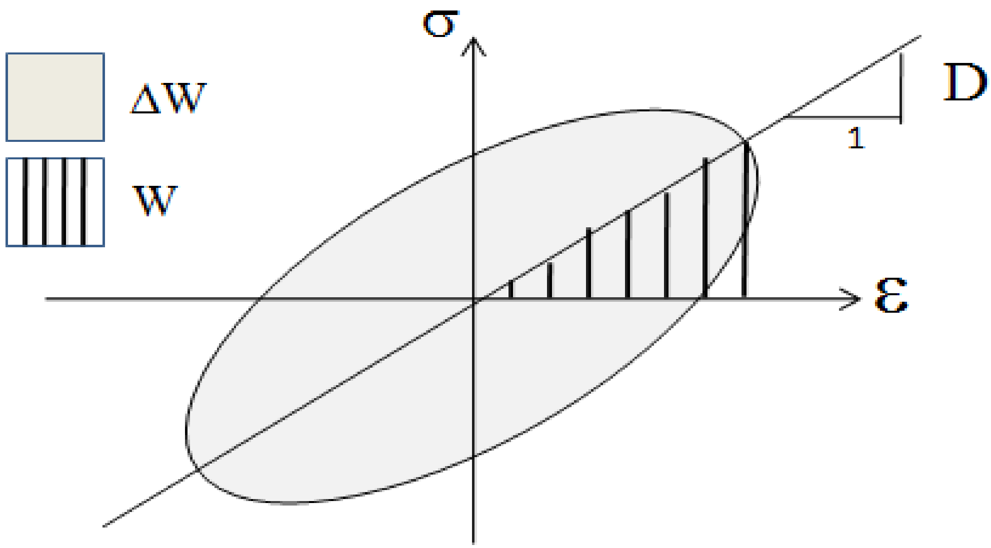
3-Node HVD Element is Based on 4-Parameter Maxwell Model (K1, B1, K2, B2)

(Kostarev et. al., 2019)



Each HVD element models one chain!

Complex Linearized Hysteretic and Viscous Models



Damping (Imaginary Part)

Hysteretic Model (Frequency-Independent); Soil, Structure & LRB, FPB

$$\tan\delta = \frac{\mathrm{Imag}(D^*)}{\mathrm{Real}(D^*)} = \frac{1}{2\pi}\frac{\Delta W}{W}$$

Viscous Model (Frequency-Dependent); HVD

$$\tan\delta = \frac{\mathrm{Imag}(D^*)}{\mathrm{Real}(D^*)} = \frac{c(\omega)\omega}{\mathrm{Real}(D^*)}$$

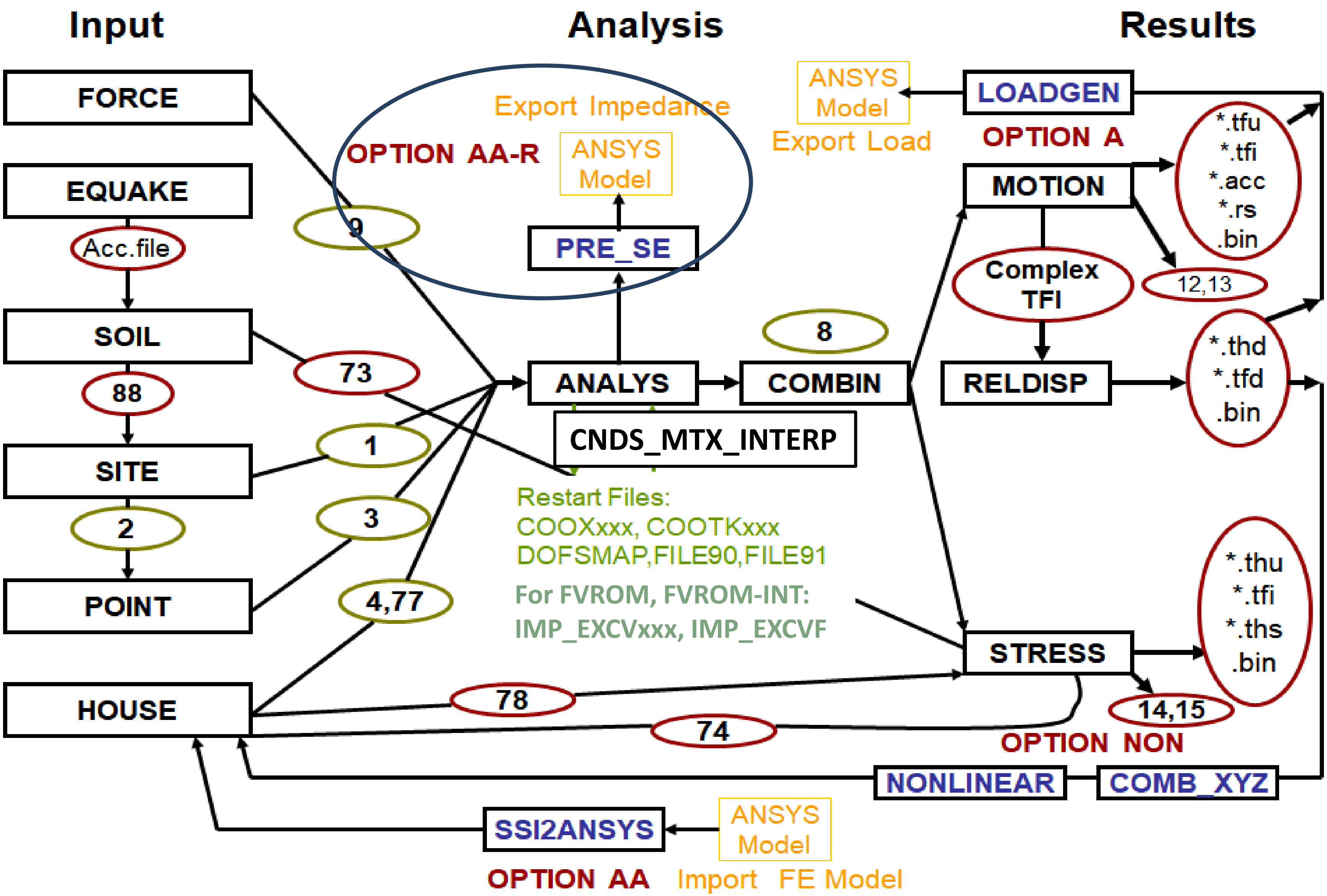
(Not mentioned in ASCE 4-16)

2. Main Software Modular Configuration

ACS SASSI Main Software SSI Modules

1. EQUAKE – Generates Spectrum Compatible Control Motion
2. SOIL – Compute Equivalent Soil Properties and Free-Field Motions
3. SITE – Compute Site Layering Behavior Under Different Wave Types
4. POINT – Compute Soil Layering Flexibilities Under Point Loads
5. FORCE – Define Nodal Loads (Forces and Moments)
6. HOUSE – Defines the Structure and Near-Field Soil and Incoherence
7. ANALYS – Compute Impedances & Solves SSI Problem (ATF solution)
8. MOTION – Computes Accelerations, RS in Structure/Near-Soil
9. RELDISP - Computes Relative Displacements
10. STRESS – Computes Stresses/Strains in Structure and Near-Soil
11. COMBIN – Combine ANALYS Solutions with Different Frequencies

ACS SASSI V4.3.2 Modular Configuration (w/o Advanced Options than A-AA and NON Simple)



ACS SASSI V4.3.2 with Options Includes 38 SSI Modules (Plus GUI Shell, and 6 DOS Auxiliary Programs)

MAIN Software: 1) EQUAKE, 2) SOIL, 3) SITE, 4) POINT, 5) FORCE, 6) HOUSE, 7) ANALYS, 8) MOTION, 9) RELDISP, 10) STRESS, 11) COMBIN, 12) CDNS_MTX_INTERP only for FVROM-INT approach

Option A-AA: 1) LOADGEN,2) SSI2ANSYS, 3) PRE_SE, 4) ans_frs2file8 (plus ANSYS macros)

Option UPLIFT: 1) UPLIFT_3DFEM, 2) UPLIFT_JEAC_4601_2015, 3) GLOBAL_IMP

Option NON: 1) NONLINEAR, 2) Section_Cuts_for_BBC, 3) BBC_JEAC_4601_ACI_318, 4) Create_Flange_Materials,
5) COMB_Shear_Bend

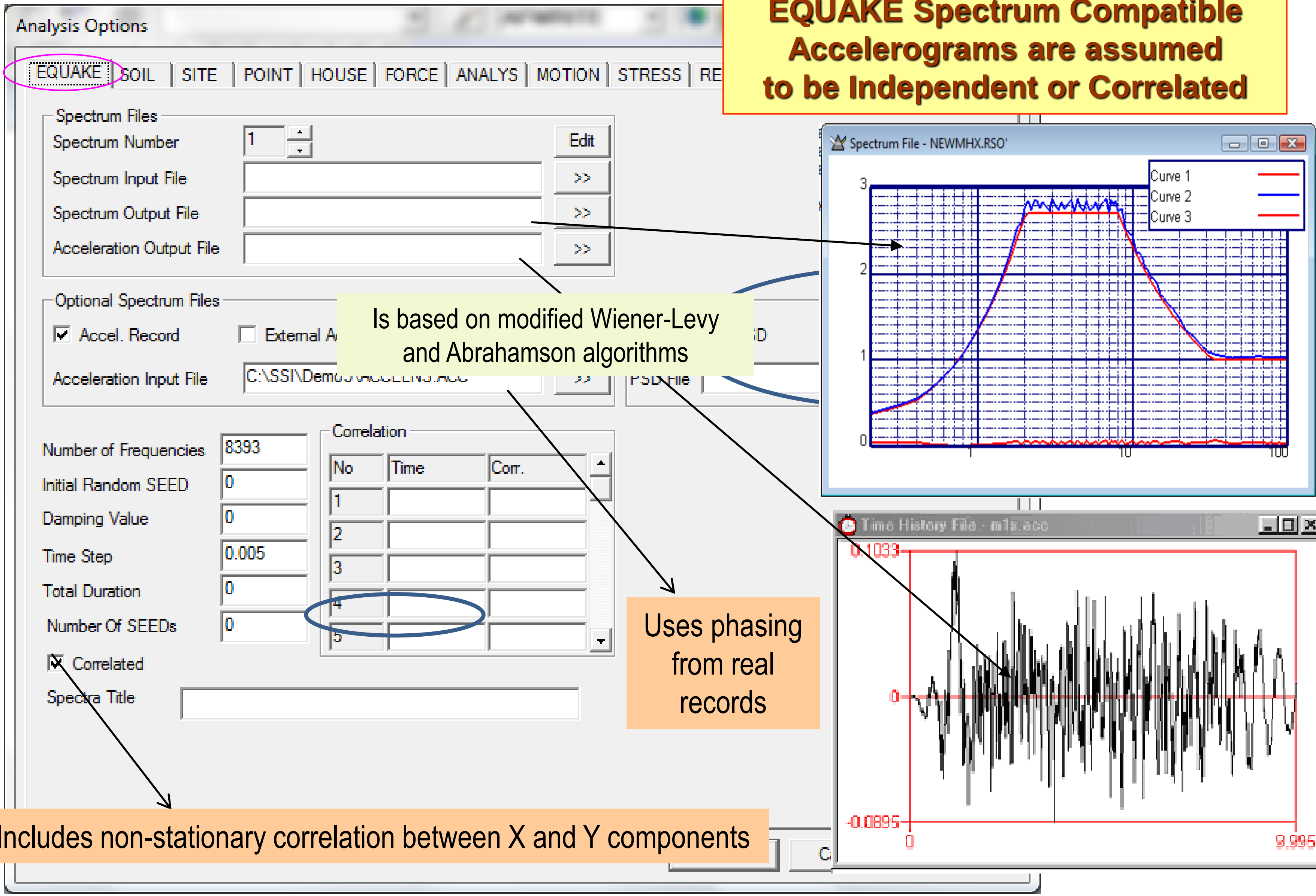
Option RVT-SIM: 1) RVT

Option PRO: 1)ProEQUAKE, 2)ProSOIL, 3)ProSITE, 4)ProHOUSE, 5)ProMOTION, 6)ProSTRESS, 7)ProNON, 8)ProSRSS and 10)ProRESPONSE, plus auxiliary modules 11)SITEPro, 12)Check_SITE_Output & 13)Write_SITE_Output

3. User Interface (UI) Input Dialog for Selecting Analysis Options for SSI Modules

Simulation of Input Control Motion (EQUAKE)

EQUAKE Spectrum Compatible Accelerograms are assumed to be Independent or Correlated



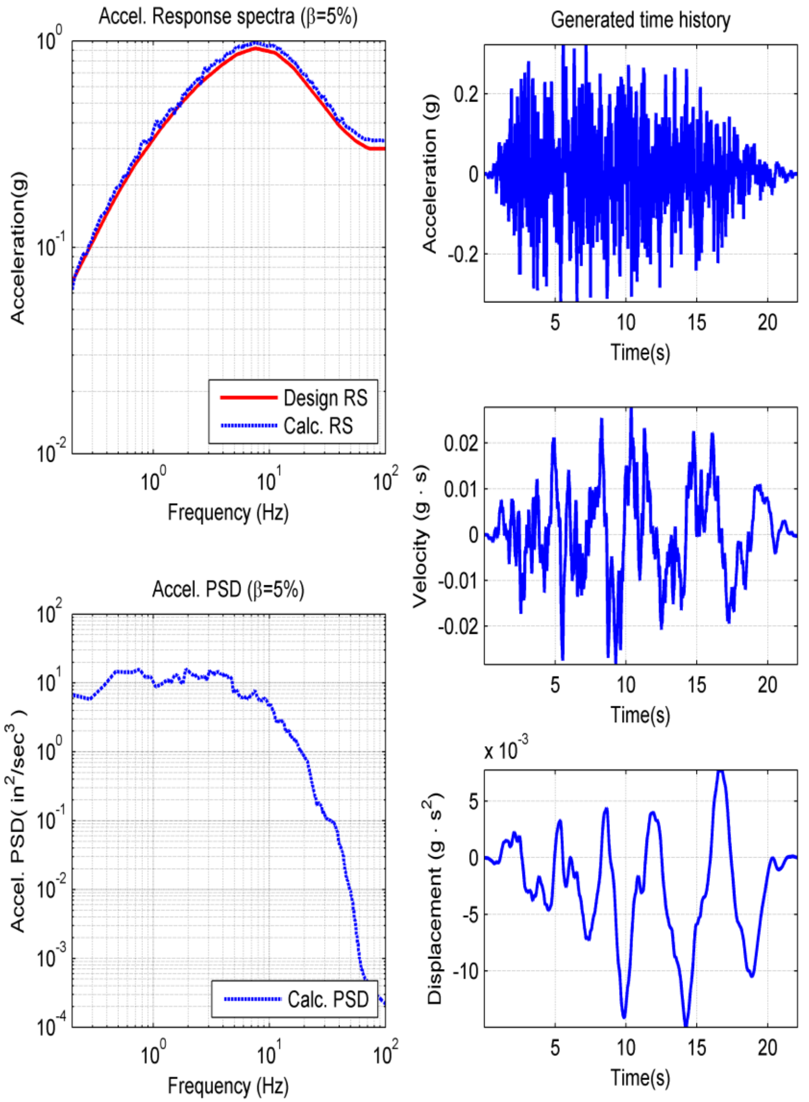
Is based on modified Wiener-Levy and Abrahamson algorithms

Uses phasing from real records

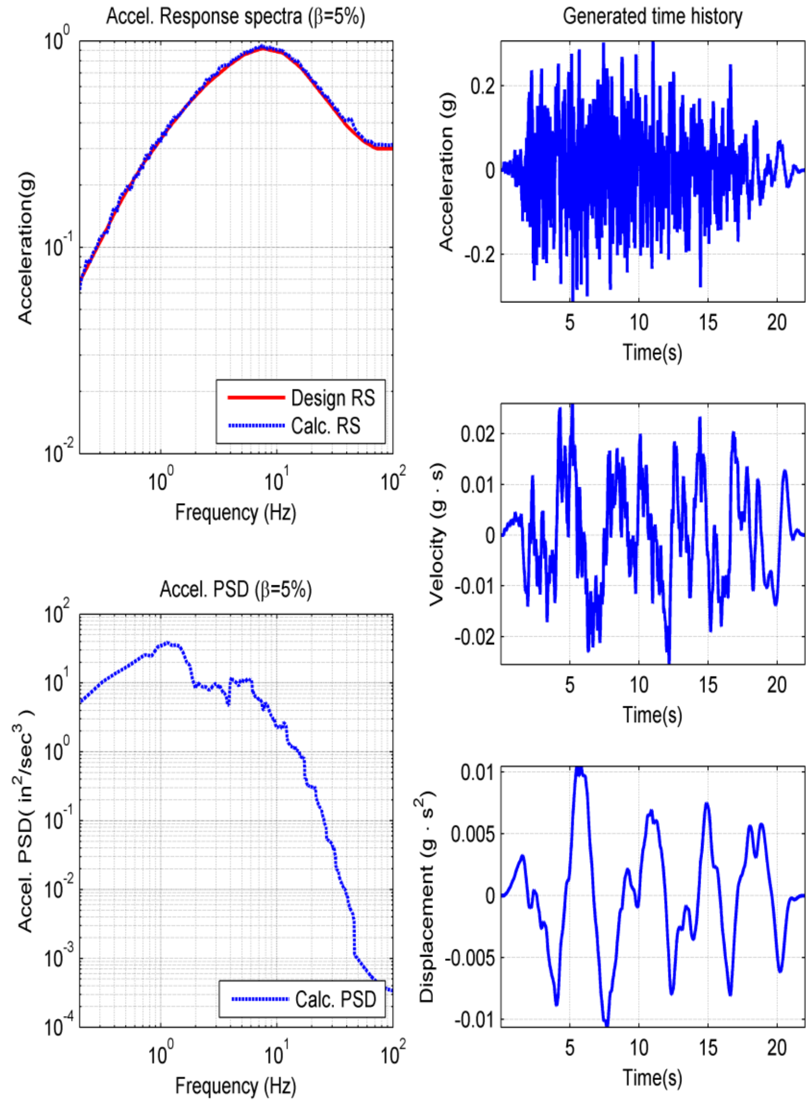
Includes non-stationary correlation between X and Y components

EQUAKE Module Capabilities – Firm Soil Site

WITHOUT SEED RECORDS

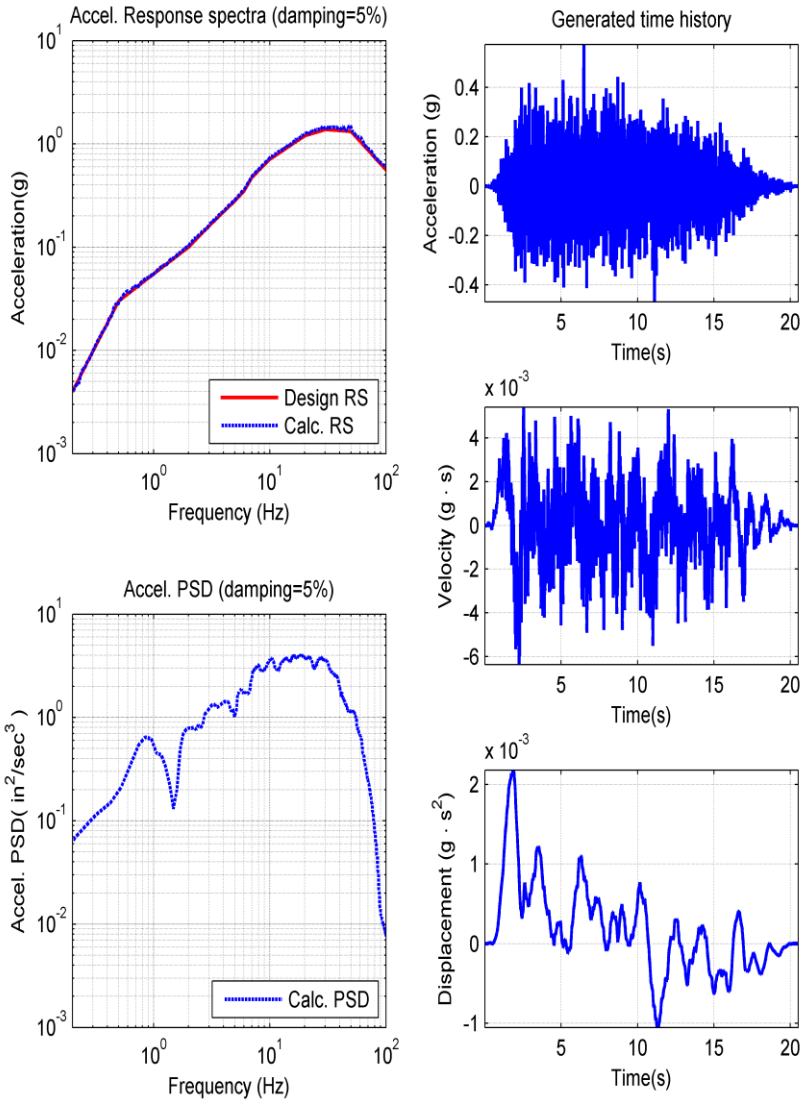


WITH SEED RECORDS

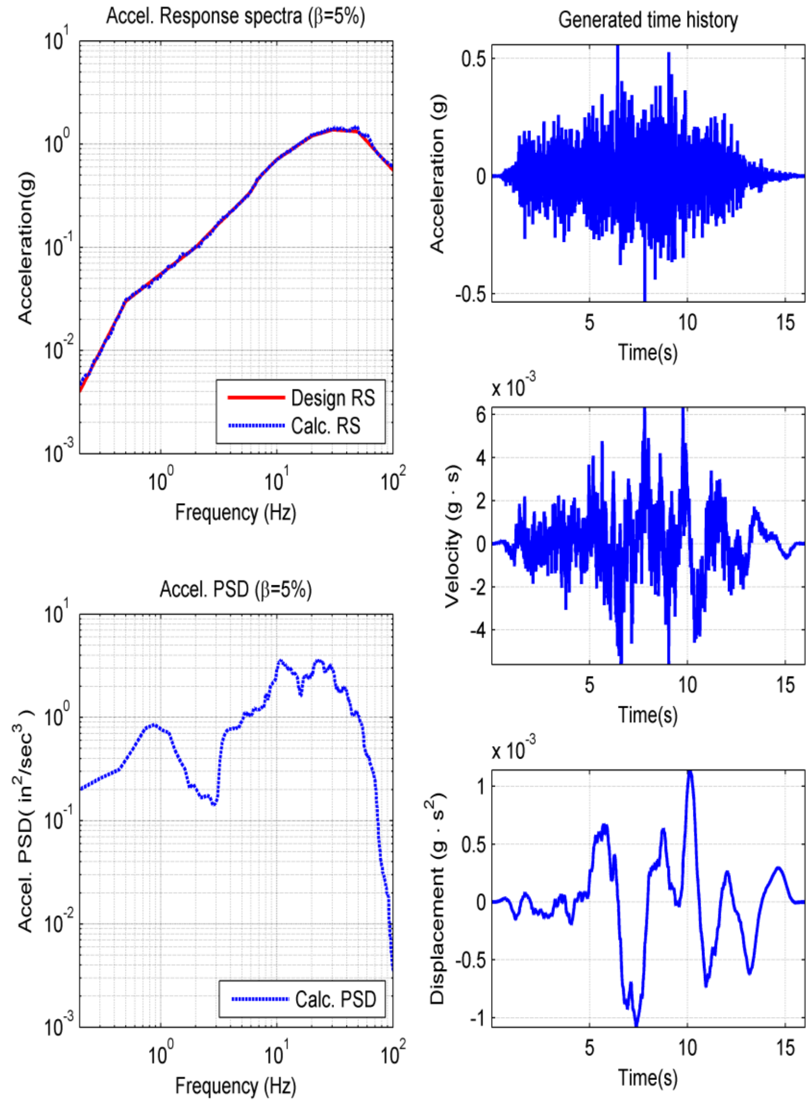


EQUAKE Module Capabilities – Rock Soil Site

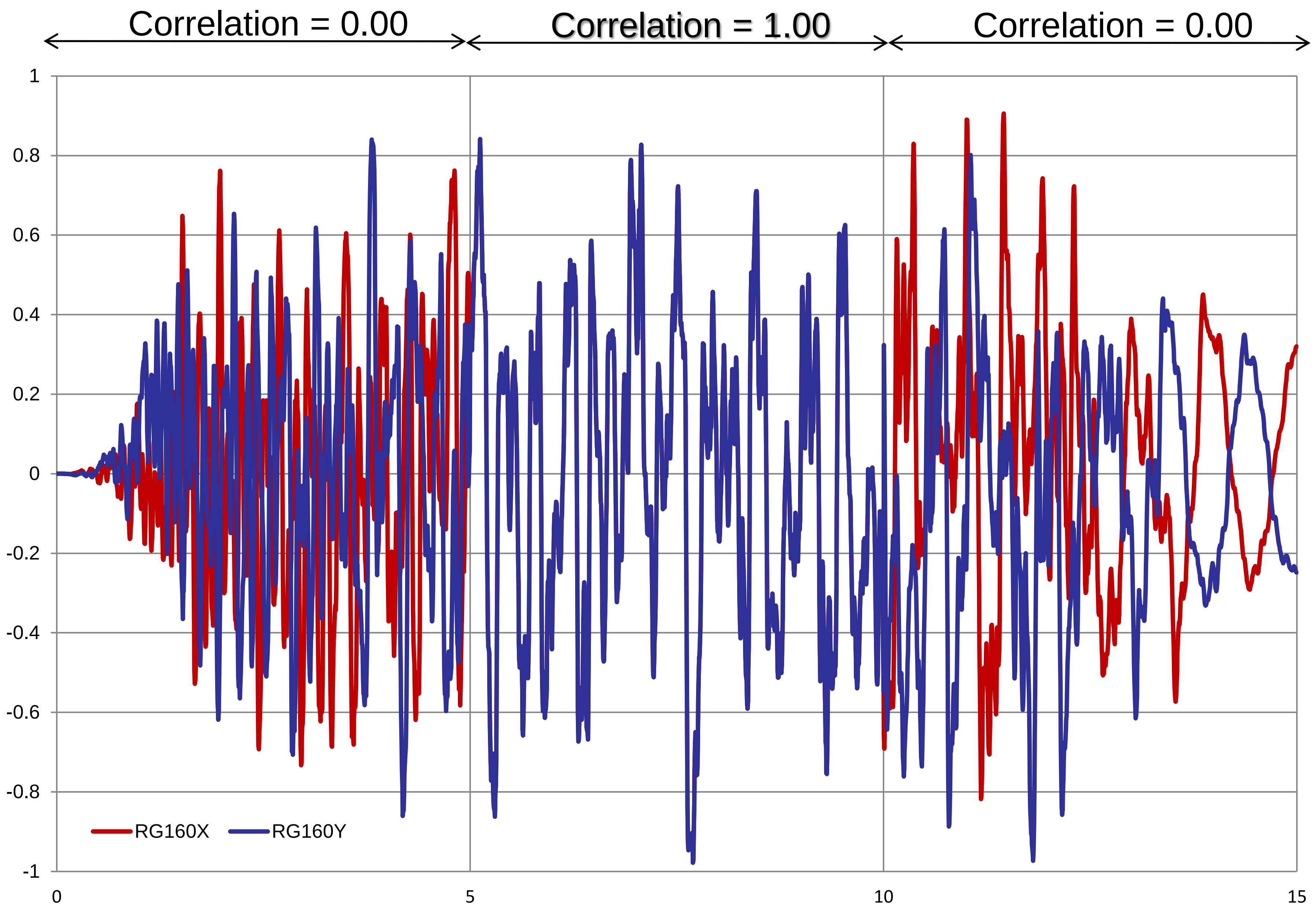
WITHOUT SEED RECORDS



WITH SEED RECORDS

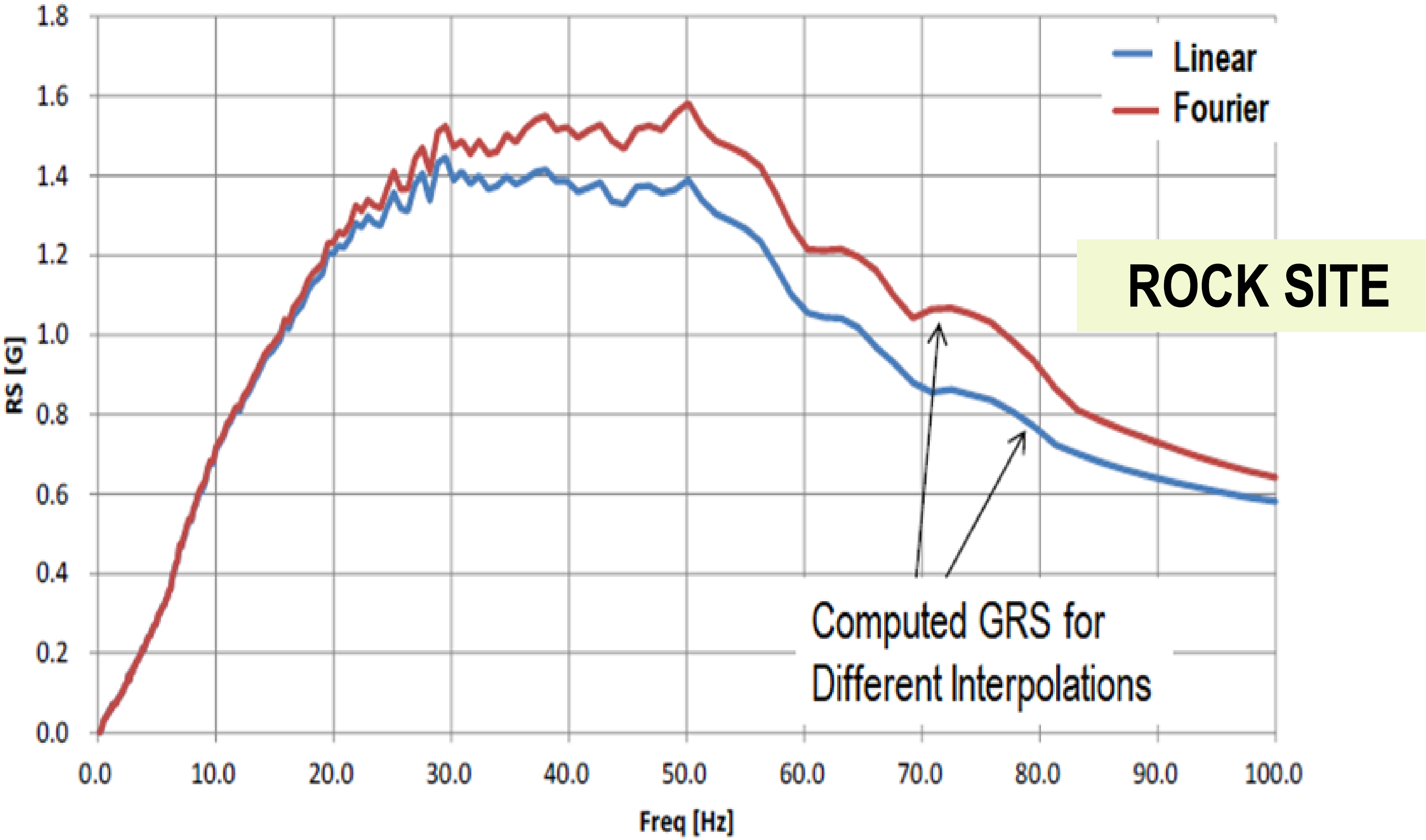


Spectrum Compatible Accelerograms with Nonstationary Correlation

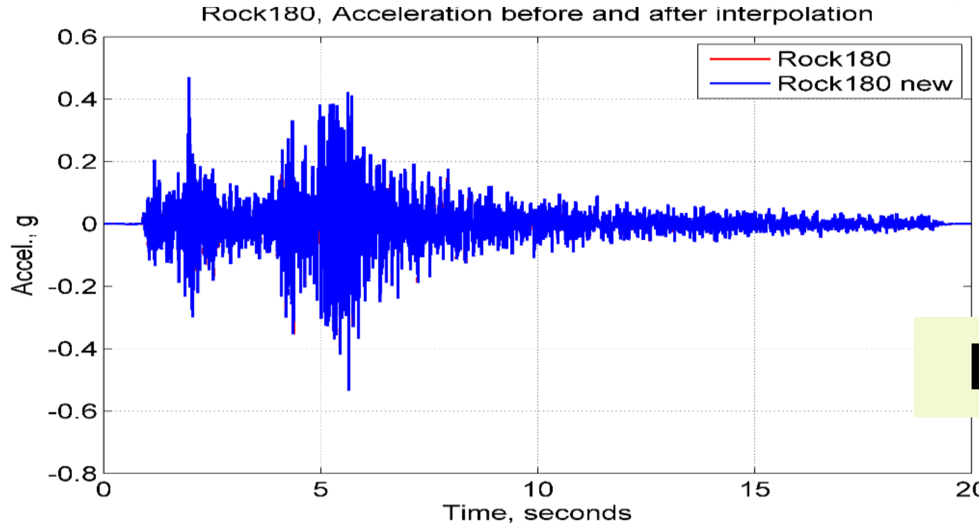


Fourier Interpolation (FZPI) for High-Frequency

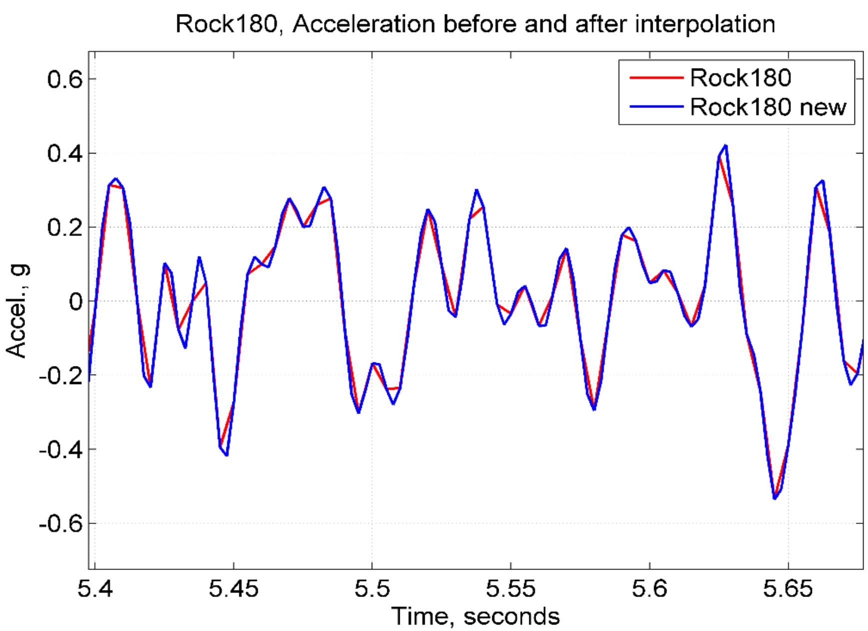
ACS SASSI V4 EQUAKE module includes the *Fourier zero-padding interpolation for acceleration histories* for computing the response spectra in the high-frequency range. *ASCE 43-19 requirement.*

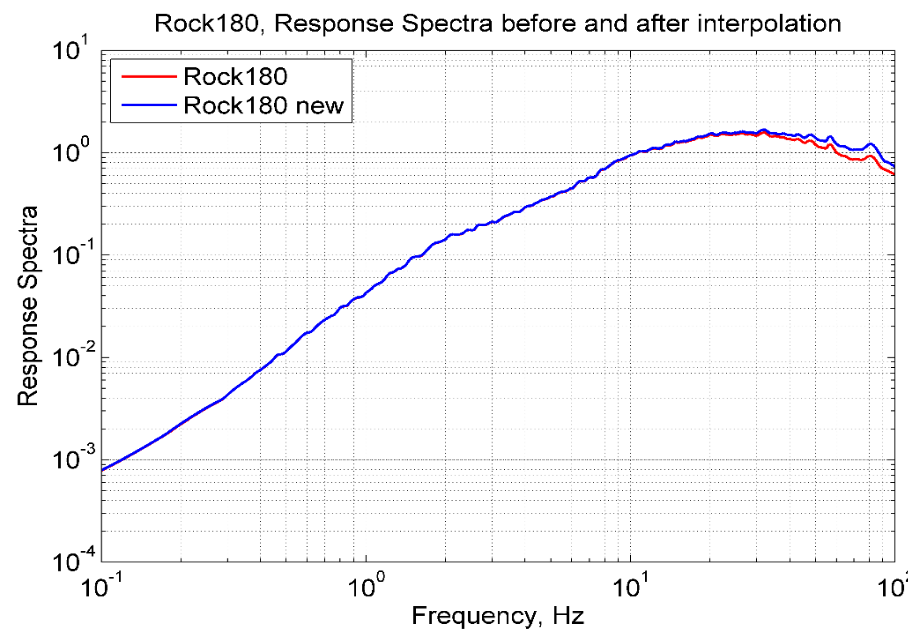


Computing High-Frequency RS Using Fourier Zero-Padding Interpolation (FZPI) Example

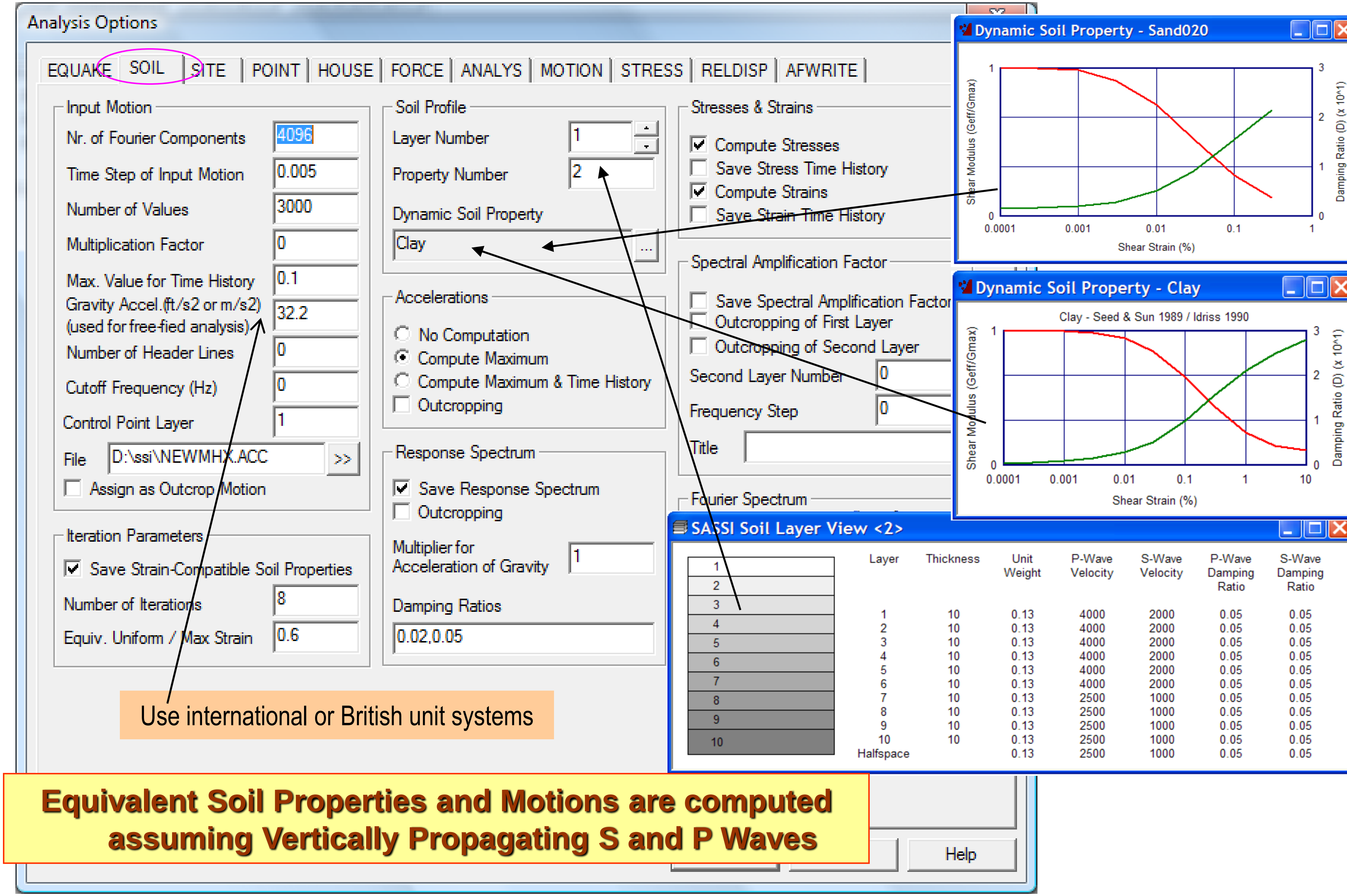


ROCK SITE



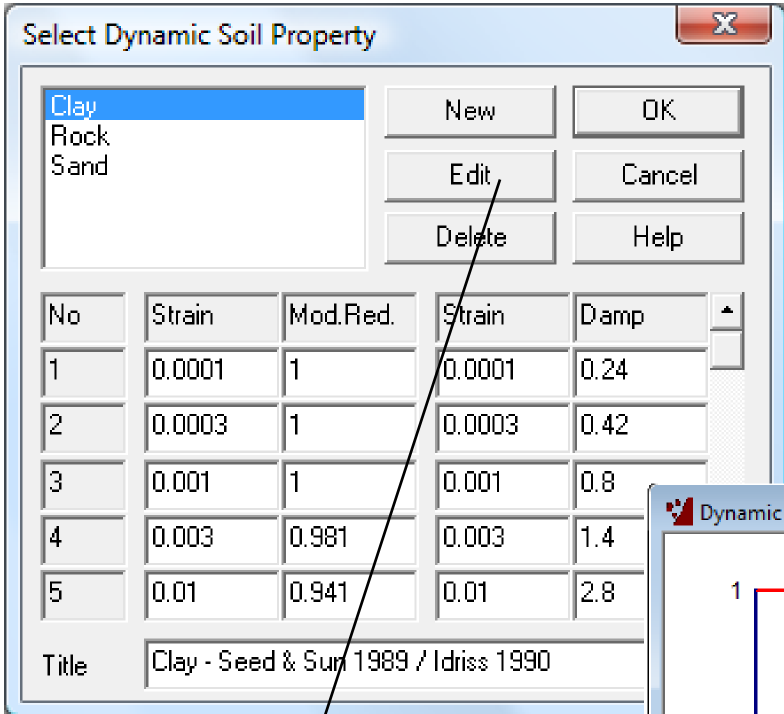


Site Response Via SHAKE Methodology (SOIL)



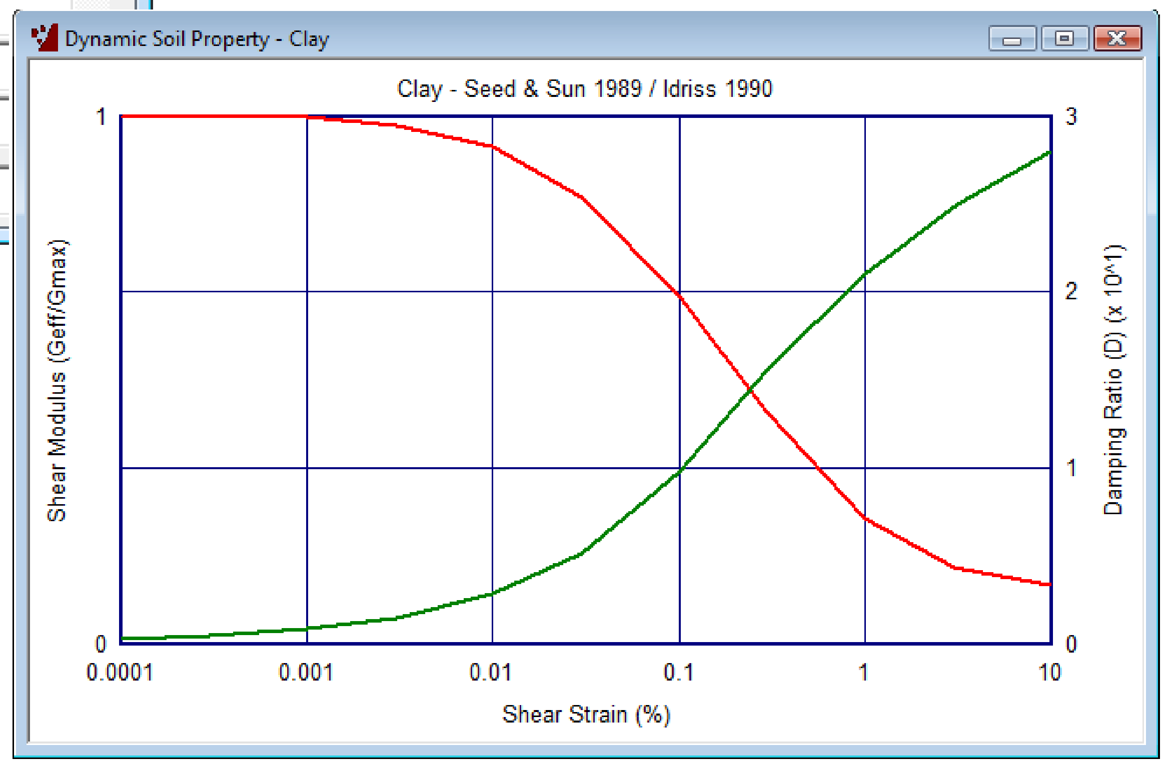
Layer	Thickness	Unit Weight	P-Wave Velocity	S-Wave Velocity	P-Wave Damping Ratio	S-Wave Damping Ratio
1	10	0.13	4000	2000	0.05	0.05
2	10	0.13	4000	2000	0.05	0.05
3	10	0.13	4000	2000	0.05	0.05
4	10	0.13	4000	2000	0.05	0.05
5	10	0.13	4000	2000	0.05	0.05
6	10	0.13	4000	2000	0.05	0.05
7	10	0.13	2500	1000	0.05	0.05
8	10	0.13	2500	1000	0.05	0.05
9	10	0.13	2500	1000	0.05	0.05
10	10	0.13	2500	1000	0.05	0.05
Halfspace		0.13	2500	1000	0.05	0.05

Equivalent Soil Properties and Motions are computed assuming Vertically Propagating S and P Waves



Nonlinear soil material curve database; Shear modulus and damping ration as functions of soil shear strain

User can add edit, delete, and included new curves



Computation of Equivalent Soil Properties

Input Acceleration Time History

```
MAXIMUM ACCELERATION =    .85600
AT TIME              =   4.37 SEC
THE VALUES WILL BE MULTIPLIED BY A FACTOR =    .350
TO GIVE NEW MAXIMUM ACCELERATION          =    .30000
MEAN SQUARE FREQUENCY =         6.87 C/SEC.
***  CONTROL MOTION LAYER ***
 **  MOTION OF LAYER NUMBER      1  OUTCROPPING

***  STRAIN COMPATIBLE SOIL PROPERTIES ***
 MAXIMUM NUMBER OF ITERATIONS     =      8
 STRAIN FACTOR IN TIME DOMAIN     =    .60

    EARTHQUAKE    -             C:\ACS_C\NEWMHX.ACC
```

SOIL Module (based on SHAKE approach) Computes Equivalent Soil Properties Using Seed-Idriss Equivalent Linear Model

Initial Soil Layering Properties

```
***  SOIL PROFILE DESCRIPTION ***
NEW SOIL PROFILE NO.    1       IDENTIFICATION
NUMBER OF LAYERS                     5          DEPTH TO BEDROCK         40.00
```

NO.	TYPE	THICKNESS (ft)	DEPTH (ft)	Tot. PRESS. (ksf)	MODULUS (ksf)	DAMPING	UNIT WT. (kcf)	SHEAR VEL (fps)
1	1	10.00	5.00	.65	4037.	.050	.130	1000.0
2	1	10.00	15.00	1.64	4037.	.050	.130	1000.0
3	1	10.00	25.00	2.31	4037.	.050	.130	1000.0
4	1	10.00	35.00	2.99	4037.	.050	.130	1000.0
5	BASE				4037.	.050	.130	1000.0

PERIOD =   .16 FOR AVERAGE SHEAR VELOCITY =   1000.

Final Soil Layering Properties

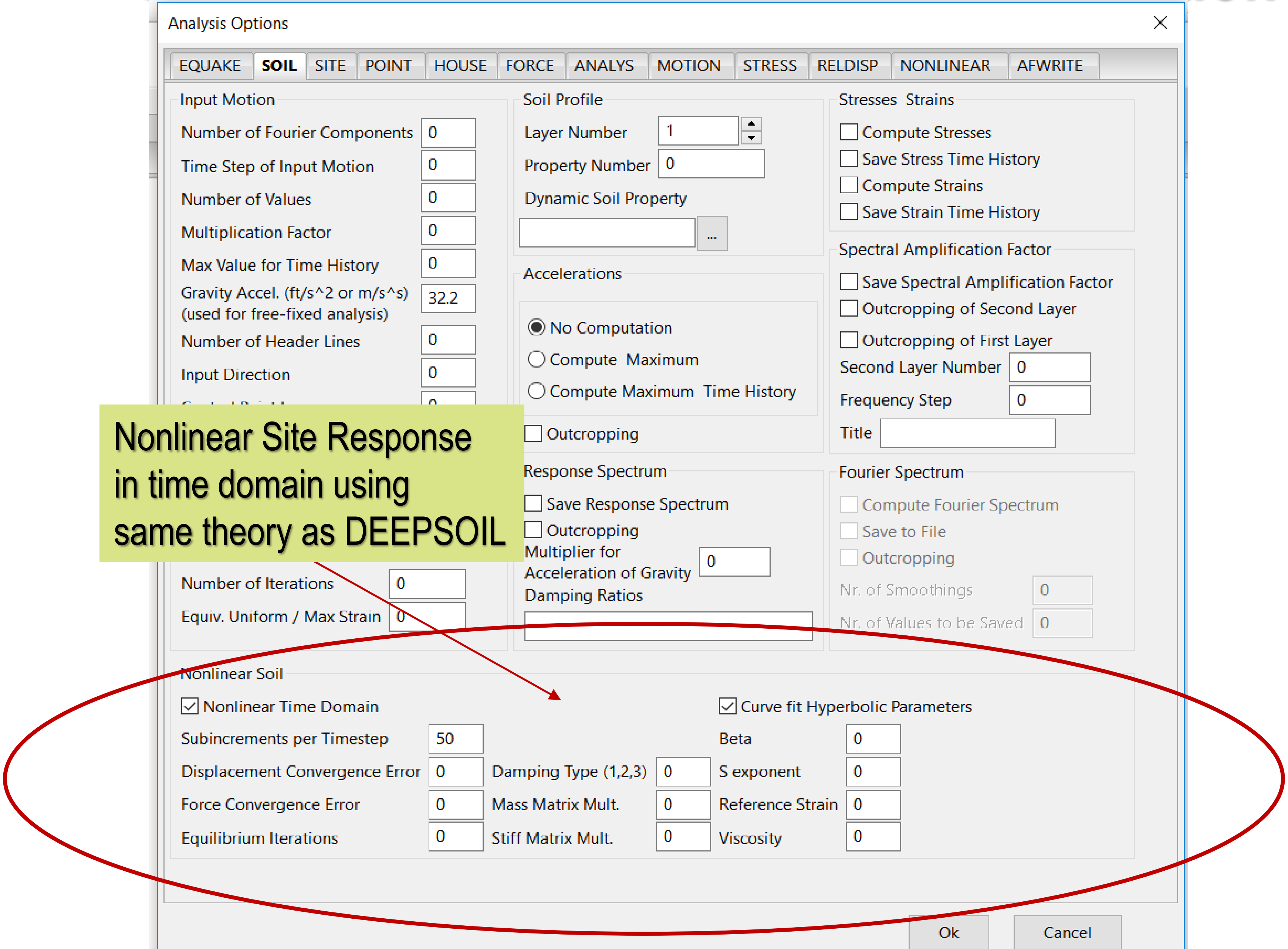
ITERATION NUMBER   8

VALUES IN TIME DOMAIN

NO	TYPE	DEPTH (FT)	UNIFRM. STRAIN	<---- DAMPING NEW	USED	----> ERROR	<---- SHEAR MODULUS NEW	USED	-----> ERROR	G/Go RATIO
1	1	5.0	.00296	.014	.014	.0	3877.2	3877.2	.0	.960
2	1	15.0	.00909	.027	.027	0.0	3466.9	3466.9	.0	.859
3	1	25.0	.01629	.038	.038	0.0	3055.3	3055.3	0.0	.757
4	1	35.0	.02485	.047	.047	0.0	2729.1	2729.2	0.0	.676

PERIOD =   .18 FOR AVERAGE SHEAR VELOCITY =   900.

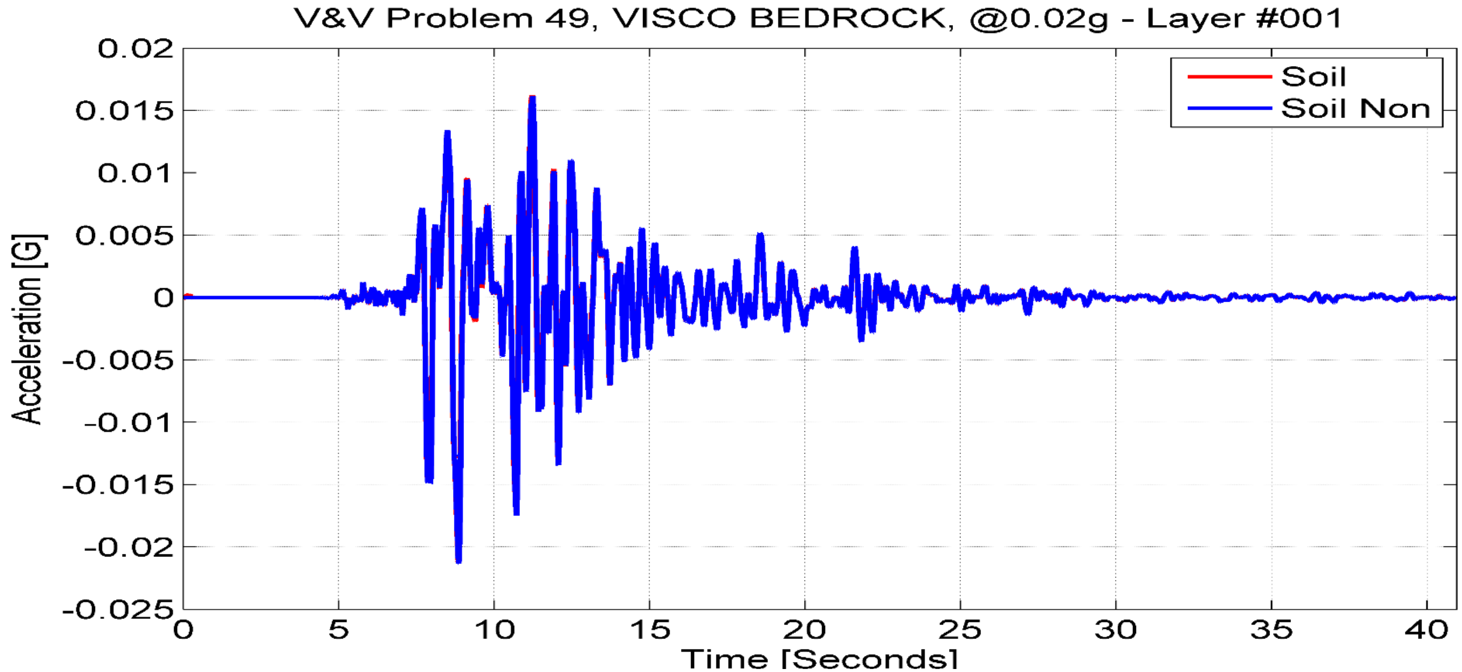
SOIL Module Including DEEPSOIL Option

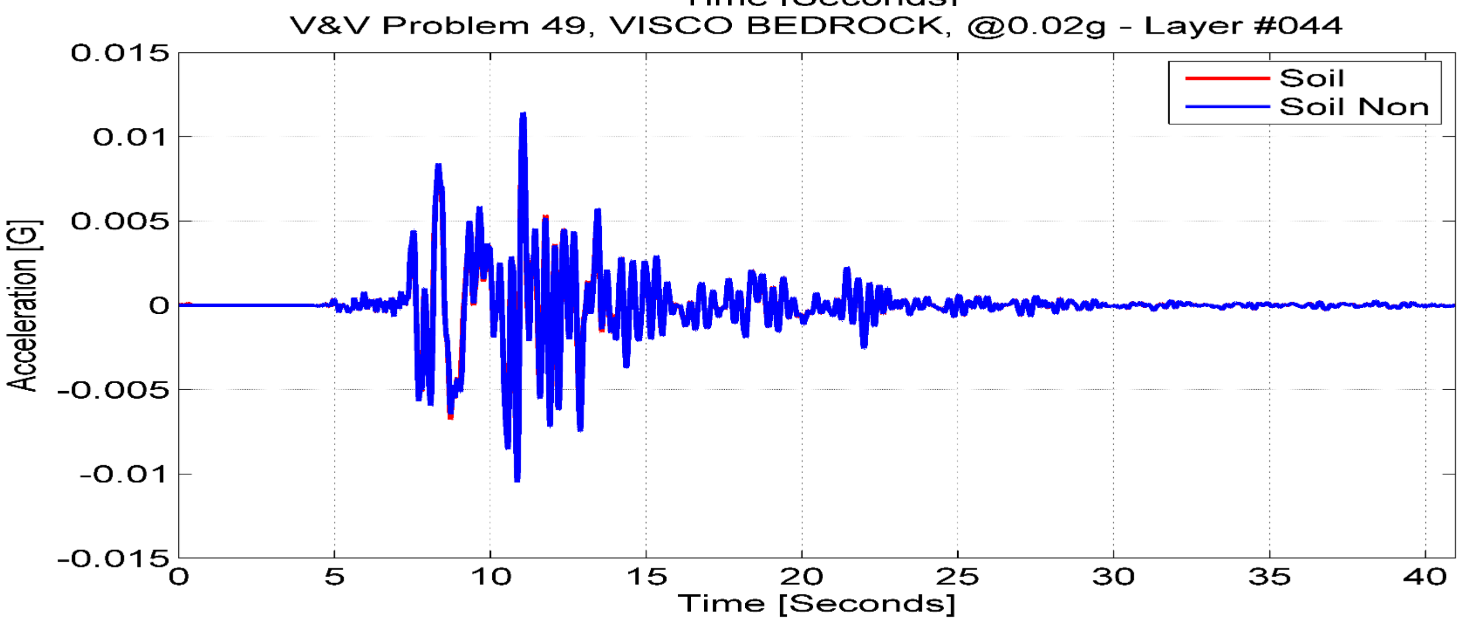


Nonlinear Site Response in time domain using same theory as DEEPSOIL

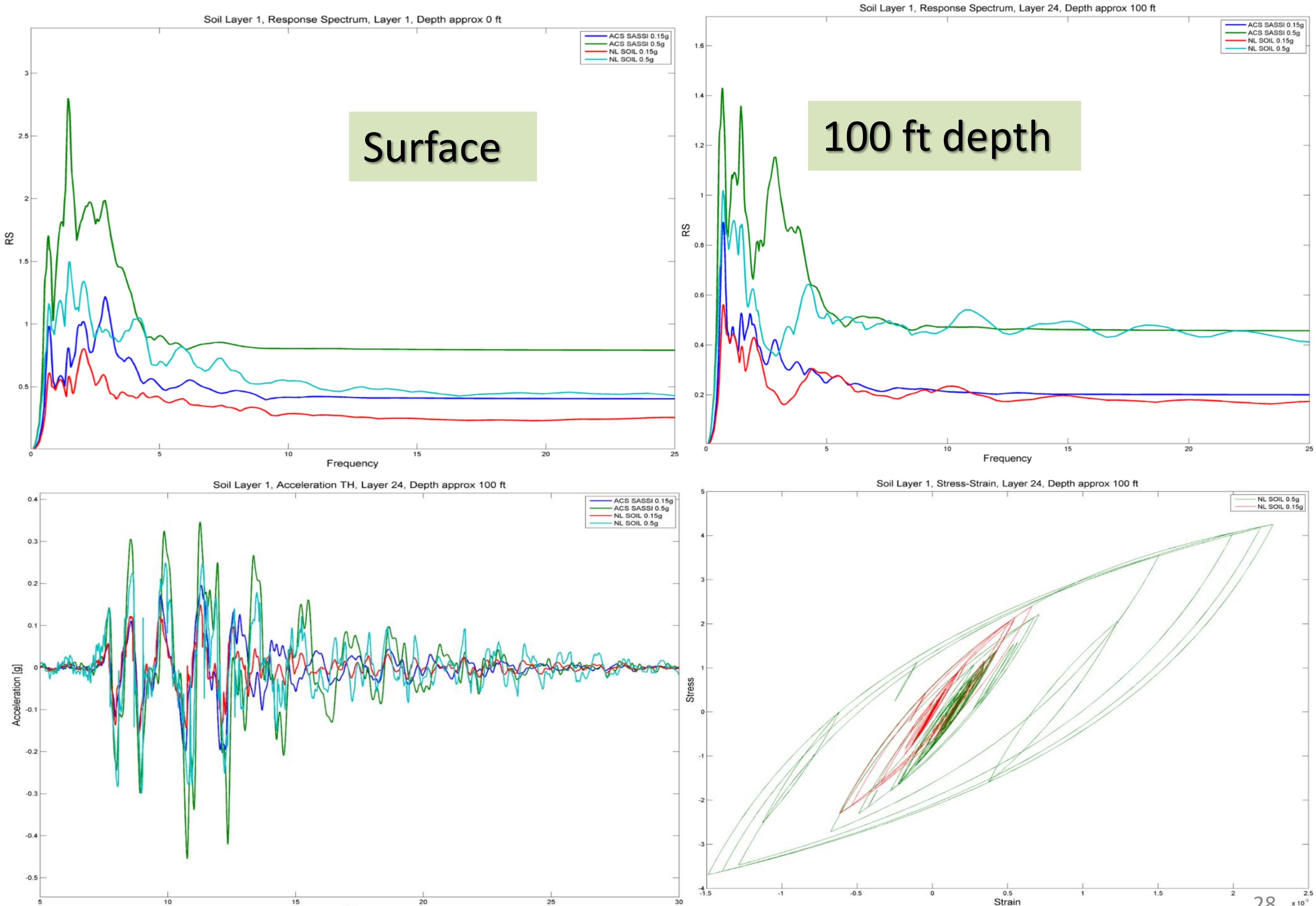
26

SOIL-NON DEEPSOIL vs. SOIL SHAKE Methods

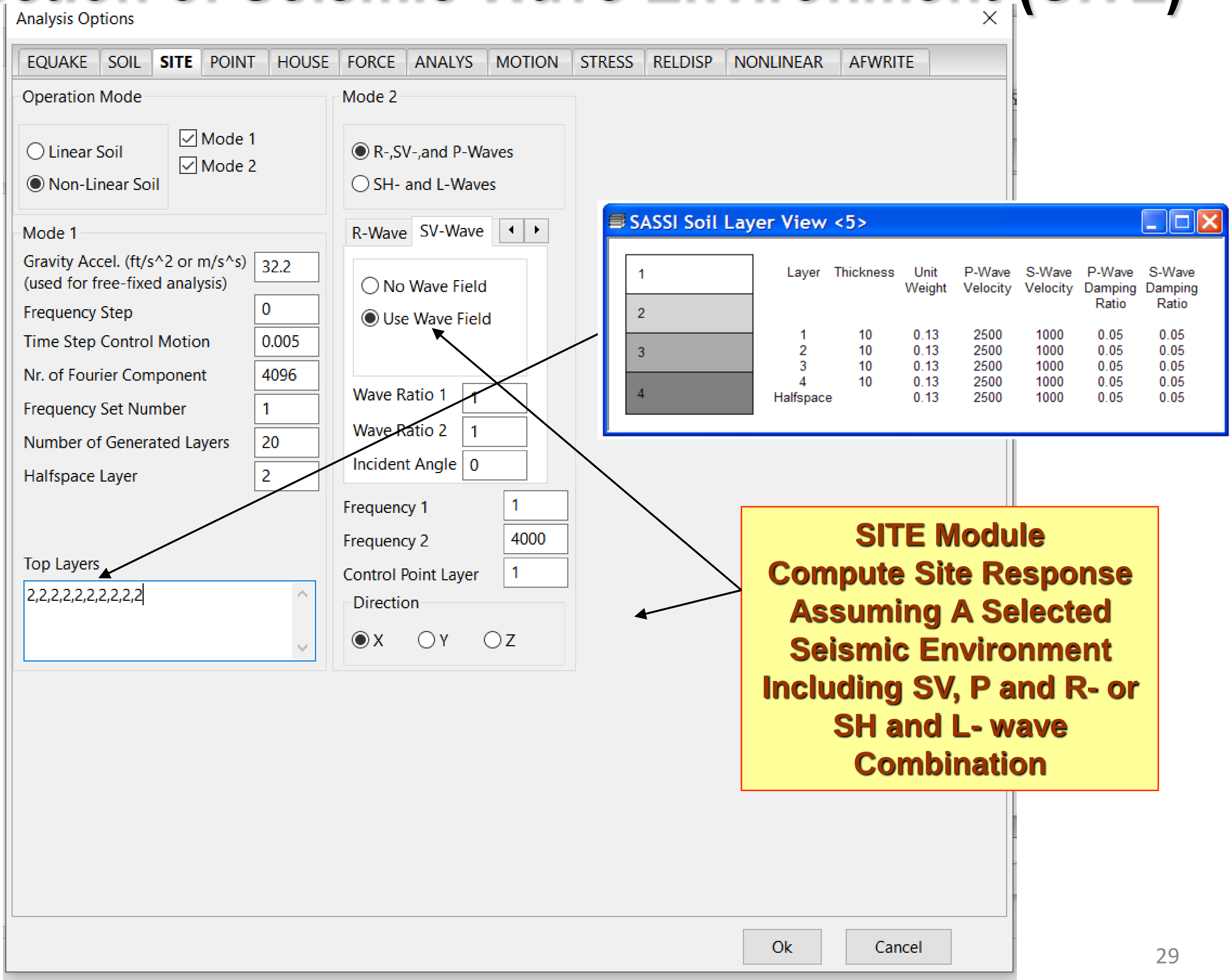




Equivalent-Linear vs. Nonlinear Site

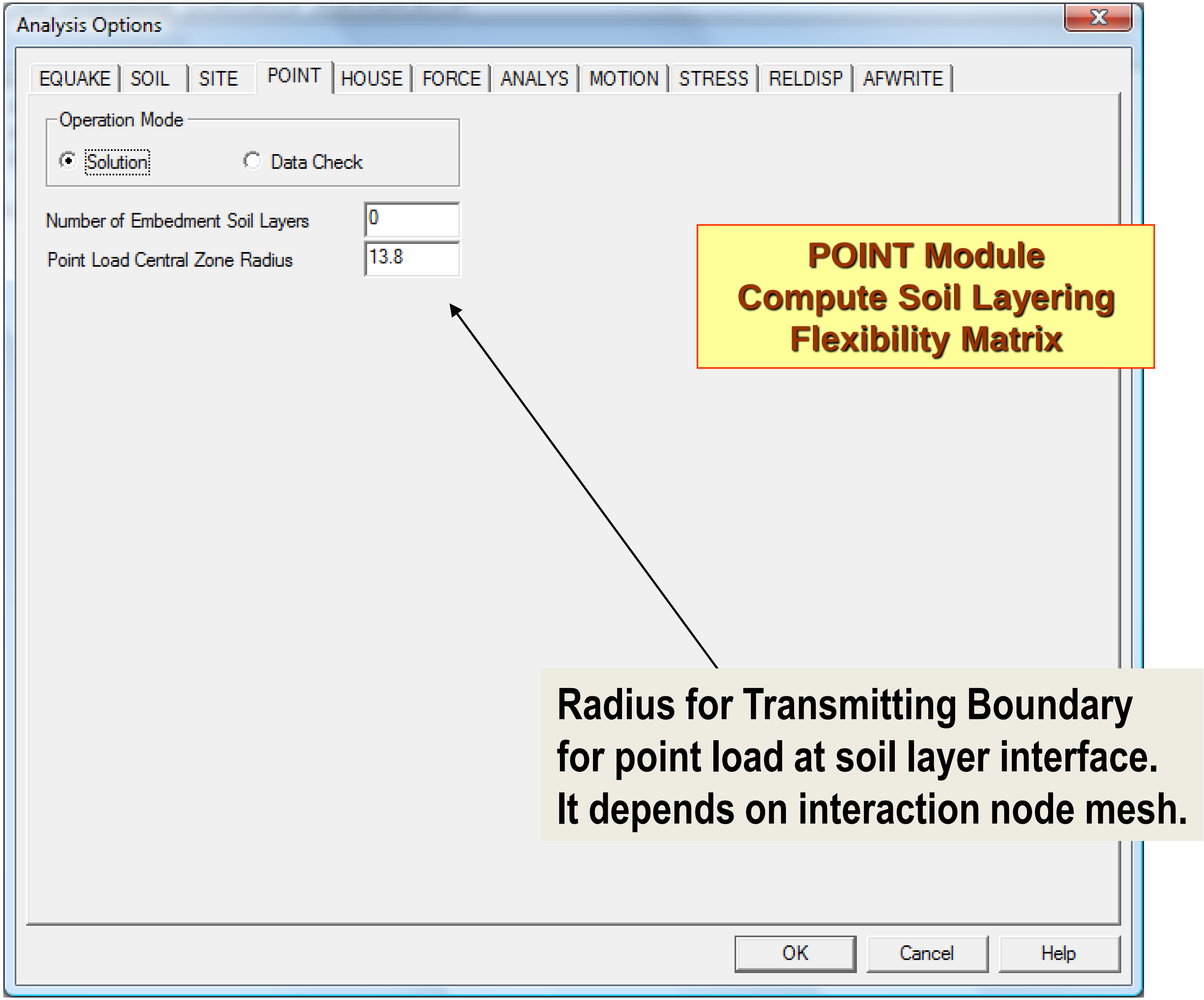


Selection of Seismic Wave Environment (SITE)

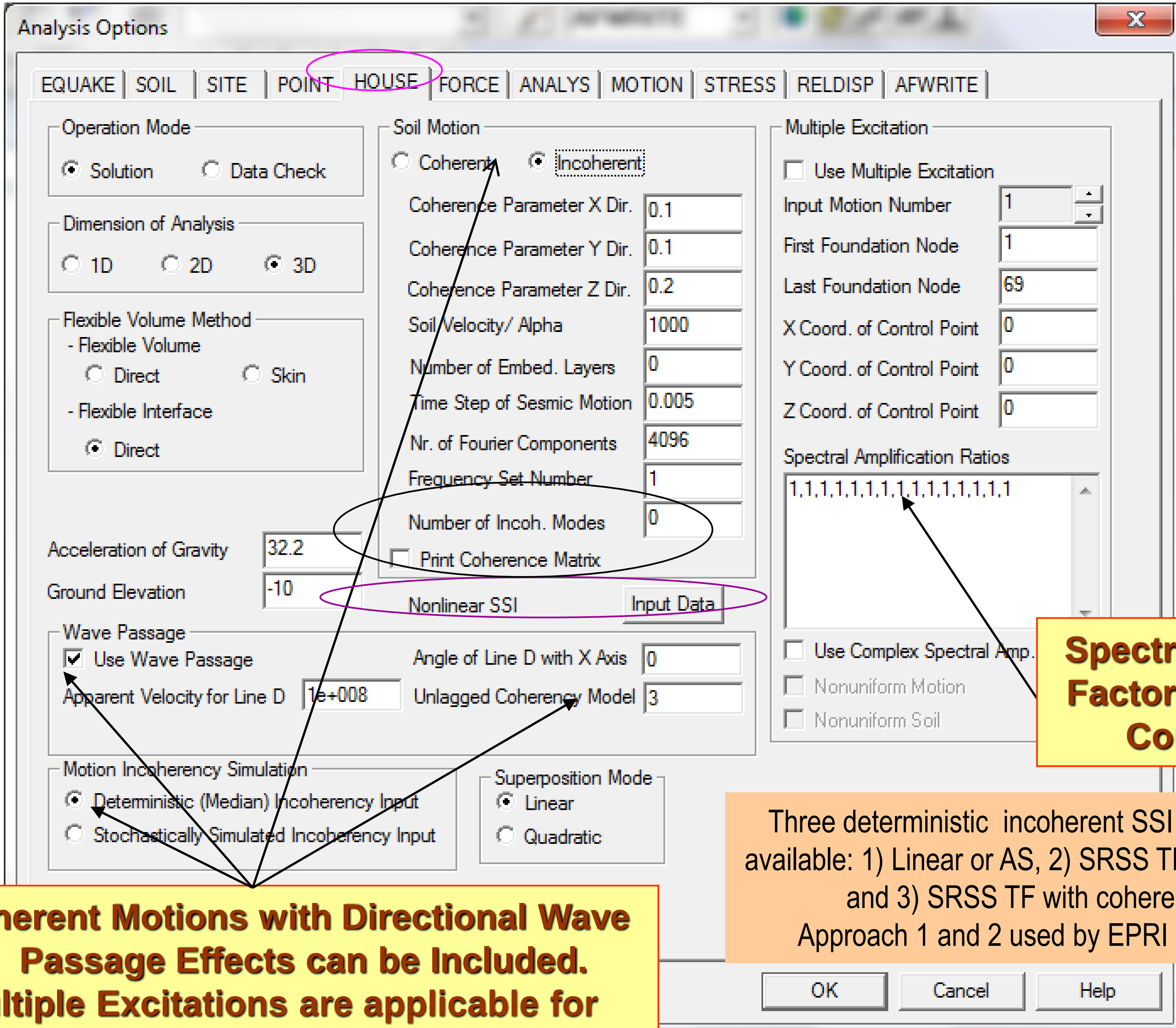


SITE Module
Compute Site Response Assuming A Selected Seismic Environment Including SV, P and R- or SH and L- wave Combination

Input for Computing Soil Flexibility Matrix (POINT)



Inputs for Coherent and Incoherent SSI (HOUSE)



HOUSE include also a (hidden) node numbering optimization algorithm (1 in col 1 line of .hou)

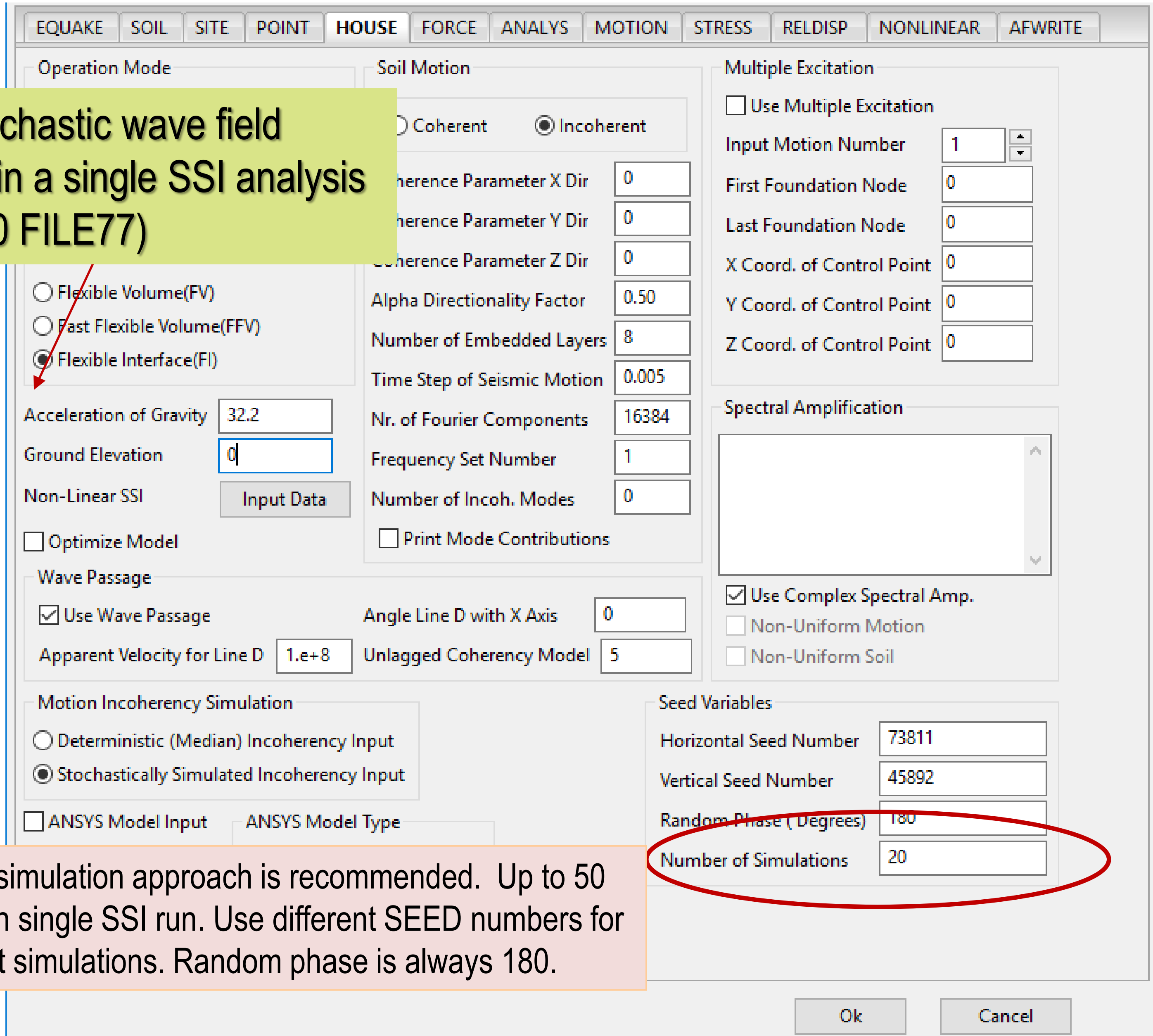
Spectral Amplification Factors are applied to Control Motion

Three deterministic incoherent SSI approaches are available: 1) Linear or AS, 2) SRSS TF with zero phase, and 3) SRSS TF with coherent phase.
Approach 1 and 2 used by EPRI (TR 1015111)

Incoherent Motions with Directional Wave Passage Effects can be Included.
Multiple Excitations are applicable for Separate Isolated Foundations.

HOUSE Module for Incoherent SSI

Up to 50 stochastic wave field simulations in a single SSI analysis run (up to 50 FILE77)



Stochastic simulation approach is recommended. Up to 50 simulations in single SSI run. Use different SEED numbers for different simulations. Random phase is always 180.

HOUSE Incoherent SSI Capabilities

There are several plane-wave incoherency models (with wave passage effects):

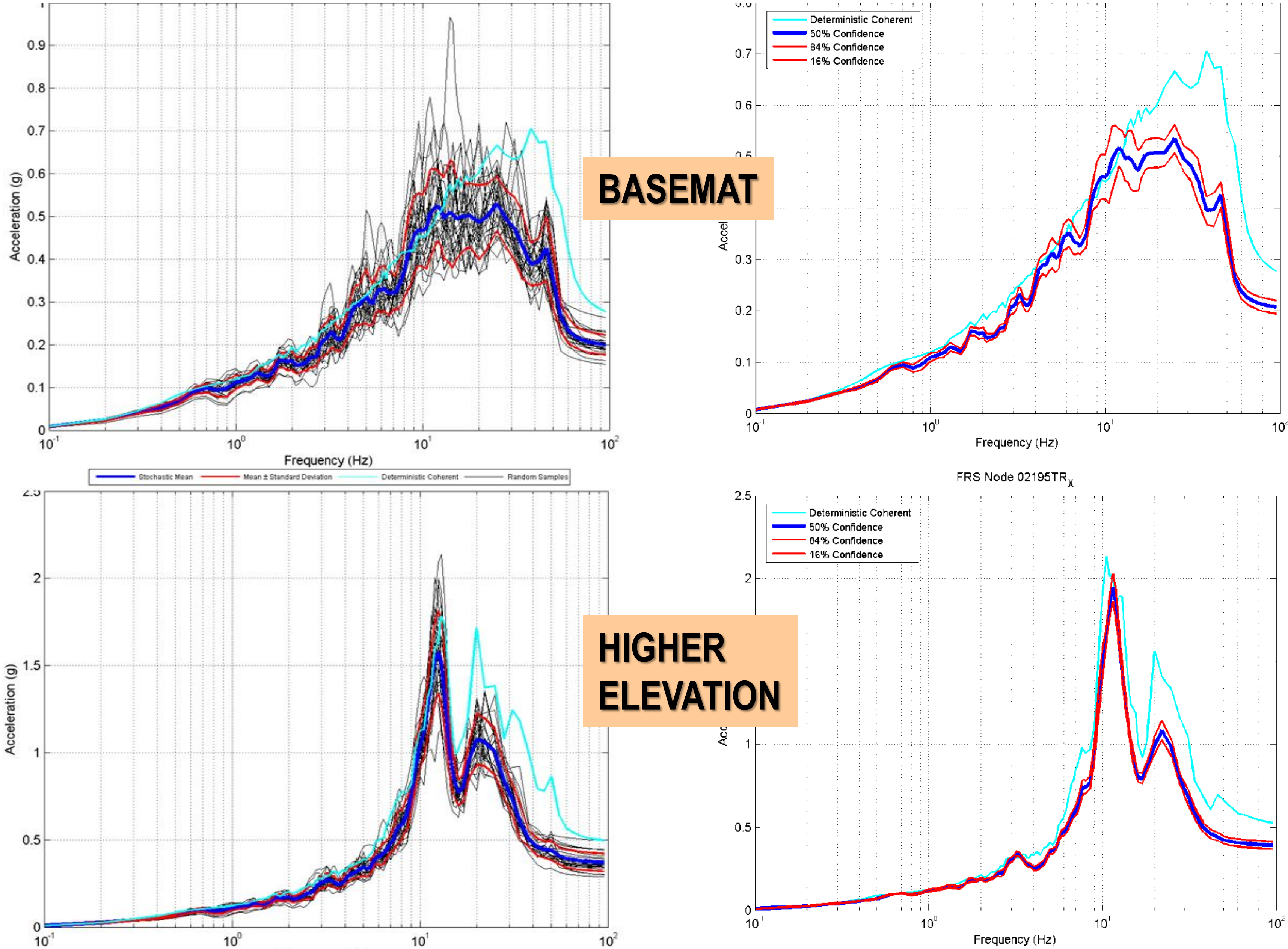
1) 1986 Luco-Wong model (theoretical, unvalidated, geom anisotropic)
2) 1993 Abrahamson model for all sites and surface foundations
3) 2005 Abrahamson model for all sites and surface foundations
4) 2006 Abrahamson model for all sites and embedded foundations
5) 2007 Abrahamson model for hard-rock sites and all foundations (NRC)
6) 2007 Abrahamson model for soil sites and surface foundations
7) *User-Defined Plane-Wave Coherency Functions for X, Y and Z.*

REMARKS:

1) Also includes *directional* Abrahamson or user-defined coherency models.
2) For general, more complex situations, can include *nonuniform motion in horizontal plane* by both amplitude and phase changes at different interaction nodes;
3) Analyst can include *different coherent functions at different depth levels* in the free-field using HOUSE create FILE77 for each node layers of interaction nodes, and append all FILE77 files together for all interaction nodes. Need expert support.

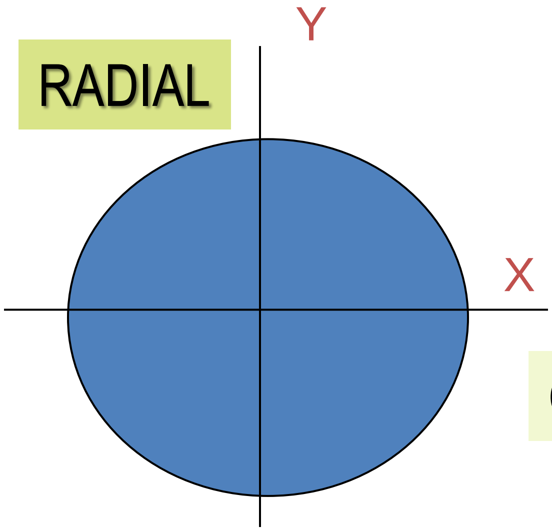
NOTE: To include automatic checking for the incoherent mode shapes

Stochastic Simulation Incoherent SSI Approach



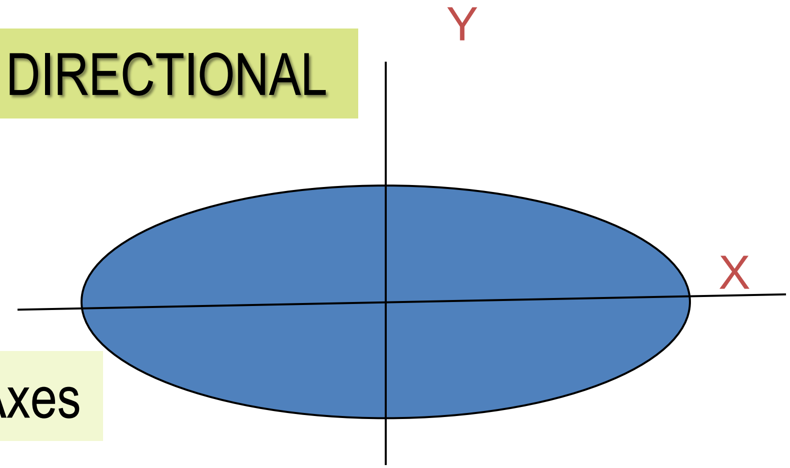
Radial vs. Directional Motion Coherency Models

RADIAL



$$\alpha = 0.5$$

DIRECTIONAL

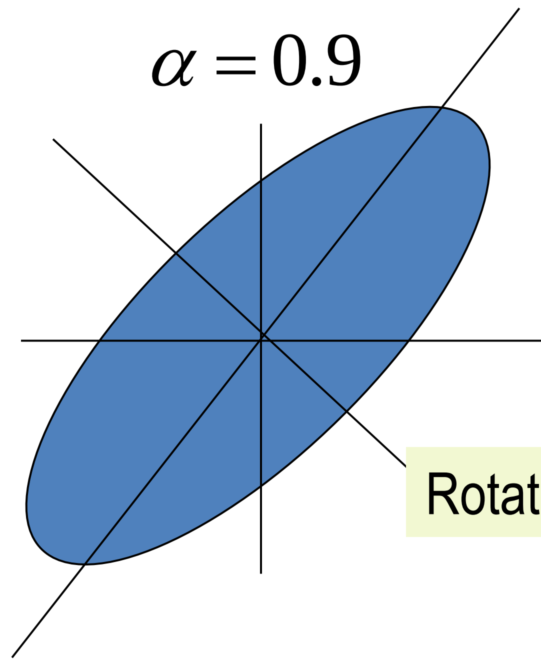


Global Axes

$$\alpha = 0.9$$

$$D^{\wedge}2 = 2[(1-\alpha)Dx^{\wedge}2 + \alpha Dy^{\wedge}2]$$

Applicable to generic, Abrahamson models and user-defined, site-specific coherency models



Rotated Axes

Deterministic Incoherent SSI Approaches (NOT RECOMMENDED)

ACS SASSI uses simplified superposition rules for combining incoherency modes or their random SSI modal effects:

i) Linear superposition of motion incoherency modes scaled with their standard deviation to simulate the free-field motion (AS in EPRI studies) – *single* SSI analysis

ii) Quadratic superposition of incoherency modal amplitude responses, applicable for the computed ATF or RS modal responses (SRSS in EPRI studies) – *multiple* SSI analysis

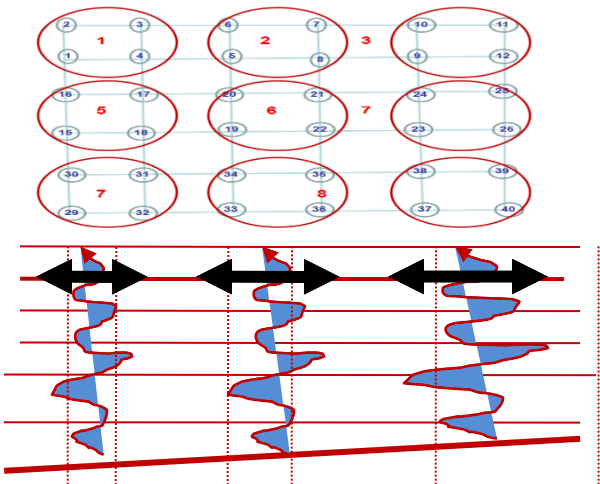
Five deterministic incoherent SSI approaches could be used:

1) Linear/algebraic summation (AS) w/ phase adjustment *(EPRI TR#1015111)*
2) Linear/algebraic summation (AS) w/o phase adjustment *
3) SRSS of ATF Amplitude w/ zero-phase *(EPRI TR#1015111)*
4) SRSS of ATF Amplitude w/ non-zero phase *
5) SRSS of RS (used in 1997 EPRI TR#102631, but not validated in 2007 EPRI TR#1015111) *

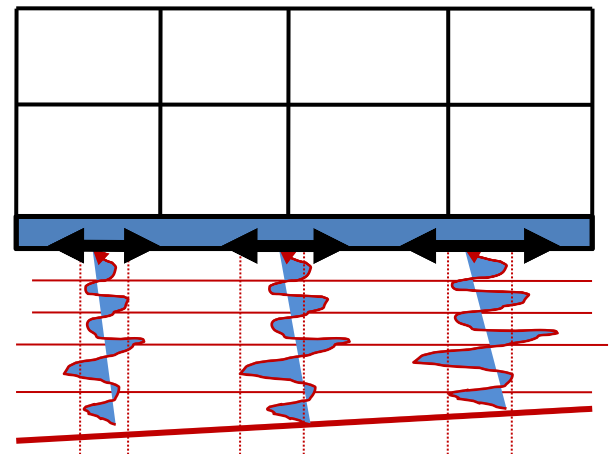
* Note: Not considered in the 2006-2007 EPRI studies (EPRI TR# 1015111)

Nonuniform Input Motion in Horizontal Plane

Multiple Soil Column Response Analyses



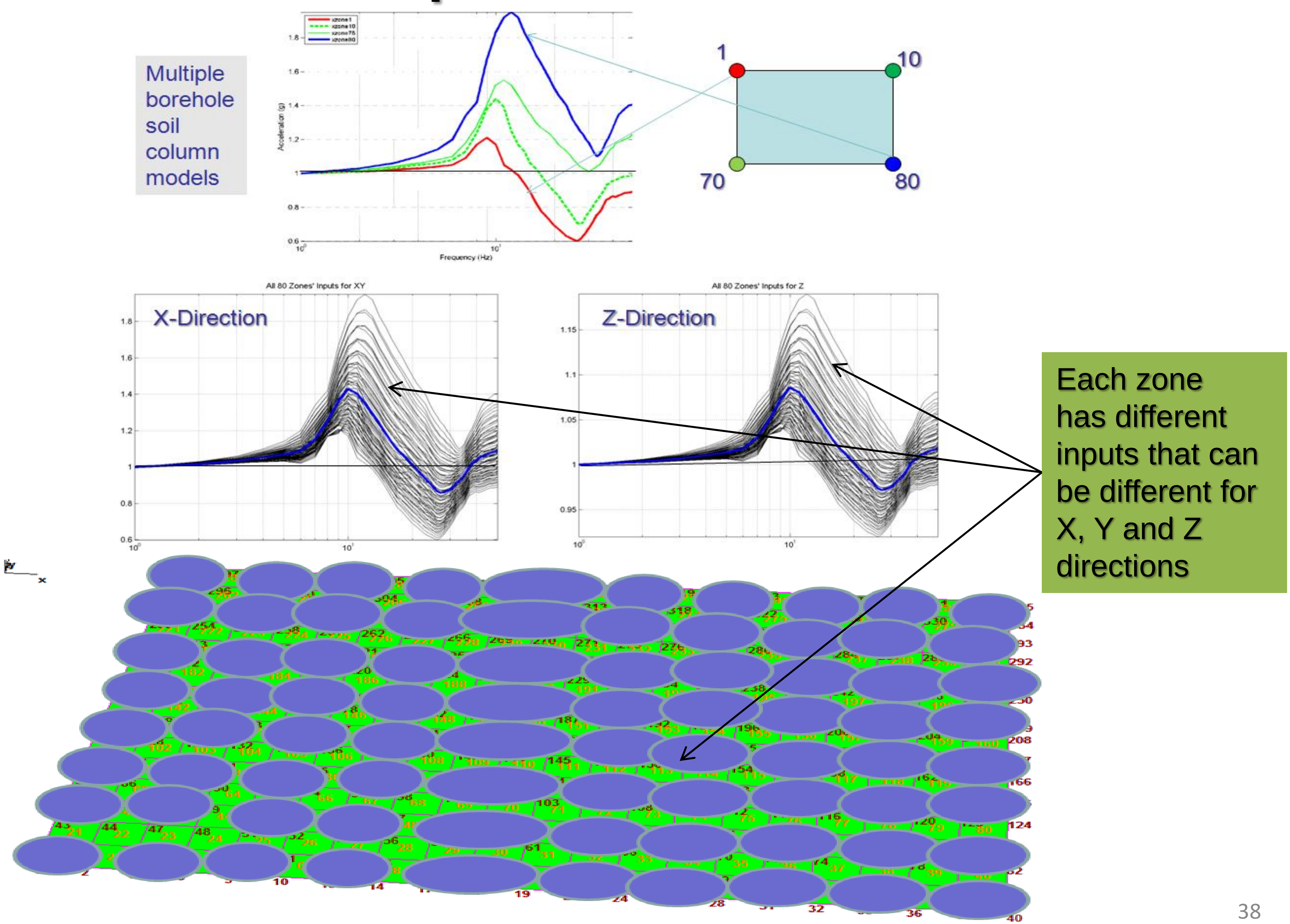
Non-Uniform Excitation and Soil Stiffness



ACS SASSI Version 4 has the capability to consider deterministic spatial variation patterns for differential input motions in the horizontal plane (HOUSE input).

These deterministic spatial variation effects can be combined with the effects of motion incoherency and wave passage to create more realistic seismic inputs for SSI analysis of NPP structures, especially for those that have large foundation sizes.

Nonuniform Input Motion in Horizontal Plane



Near-Field Soil Input for Nonlinear SSI

By clicking the “Nonlinear SSI” Input Data in HOUSE a text file is opened for editing.

This file has extension .pin and needs to input in a free-format:

1st line: Number of nonlinear soil element groups, effective strain factor, number of soil material curves defined in SOIL (soil constitutive model);

2nd line: Number of the nonlinear soil element group, number of materials (could be equal with the number of layers or not) in the group and number of solid elements in the group

3rd line and after define a loop over the number of soil materials, with each line including: The initial shear modulus reduction factor (1.00 indicates same shear modulus as in free-field), the initial damping ratio factor (1.00 indicates the same damping as in free-field) and the soil material curve order number.

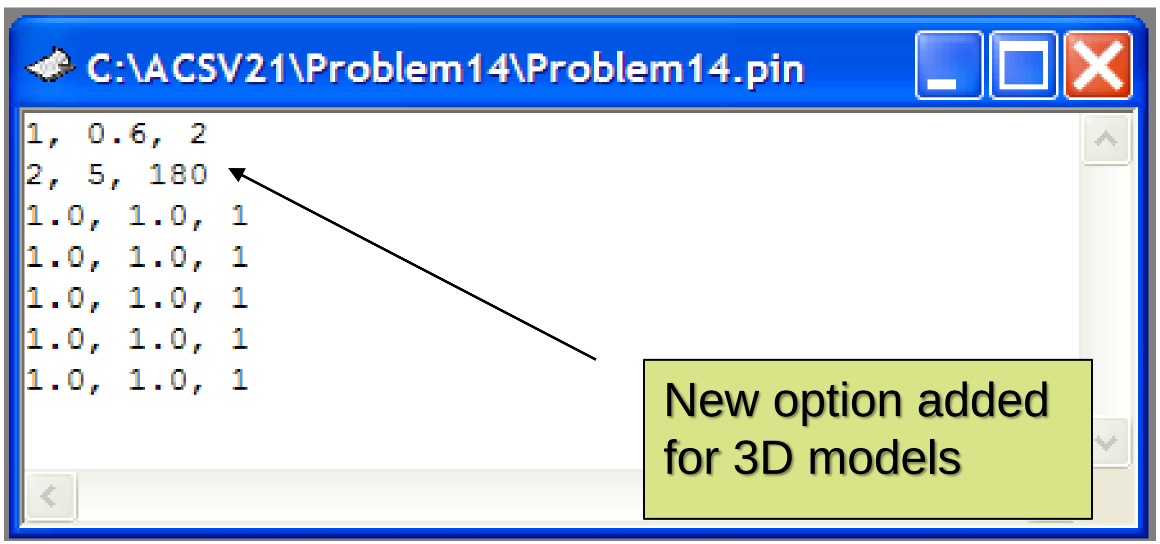
The block of lines after 1st line, needs to be input for all nonlinear soil element groups.

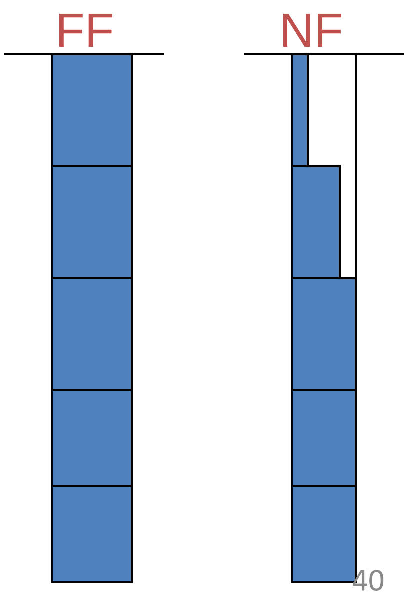
Near-Field Soil Input for Nonlinear SSI (cont.)

Example with a single group of nonlinear soil elements, an effective strain factor of 0.60 and 2 soil material curves.

The order number of the nonlinear soil group is 2, the number of soil materials in the group is 5, and total number of elements in the group is 180.

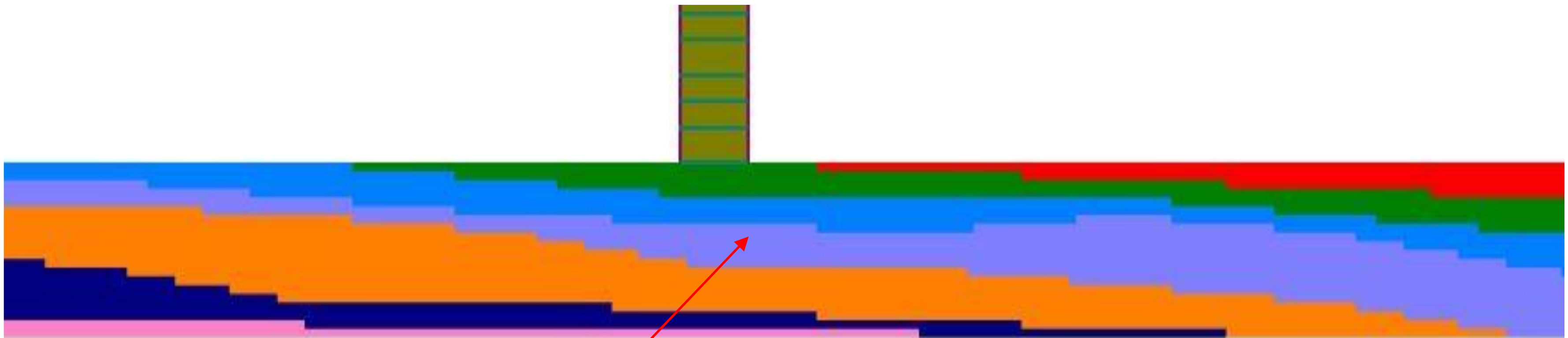
For each the 5 soil material lines, we input 1.0 for the scale factor of G, 1.0 for the scale factor of D, and 1 for material curve (curve number are defined in SOIL).





2D Soil Nonlinear Soil for Site or SSI Response

Use of nonlinear 2D PLANE elements similar to the use of nonlinear 3D SOLID elements for 3D SSI analysis

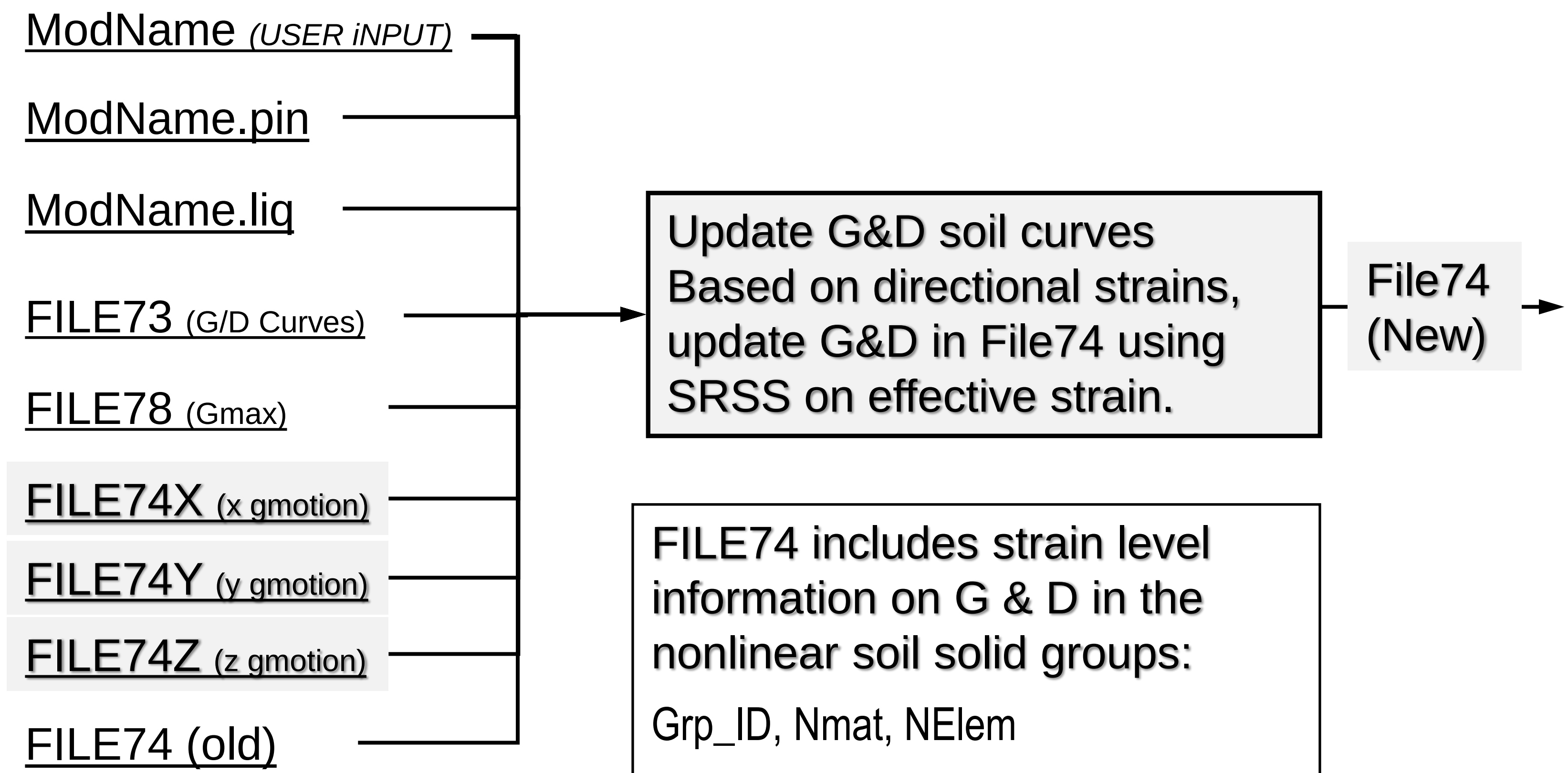


COMB_XYZ_STRAIN Module for 2D SSI Models

Constitutive Model Criteria based on:

1) Maximum Component Shear Strain (X)
2) Maximum Shear Plane Criterion

Auxiliary Program COMB_XYZ_STRAIN for Combining SSI Soil Shear Strains in 3D Space



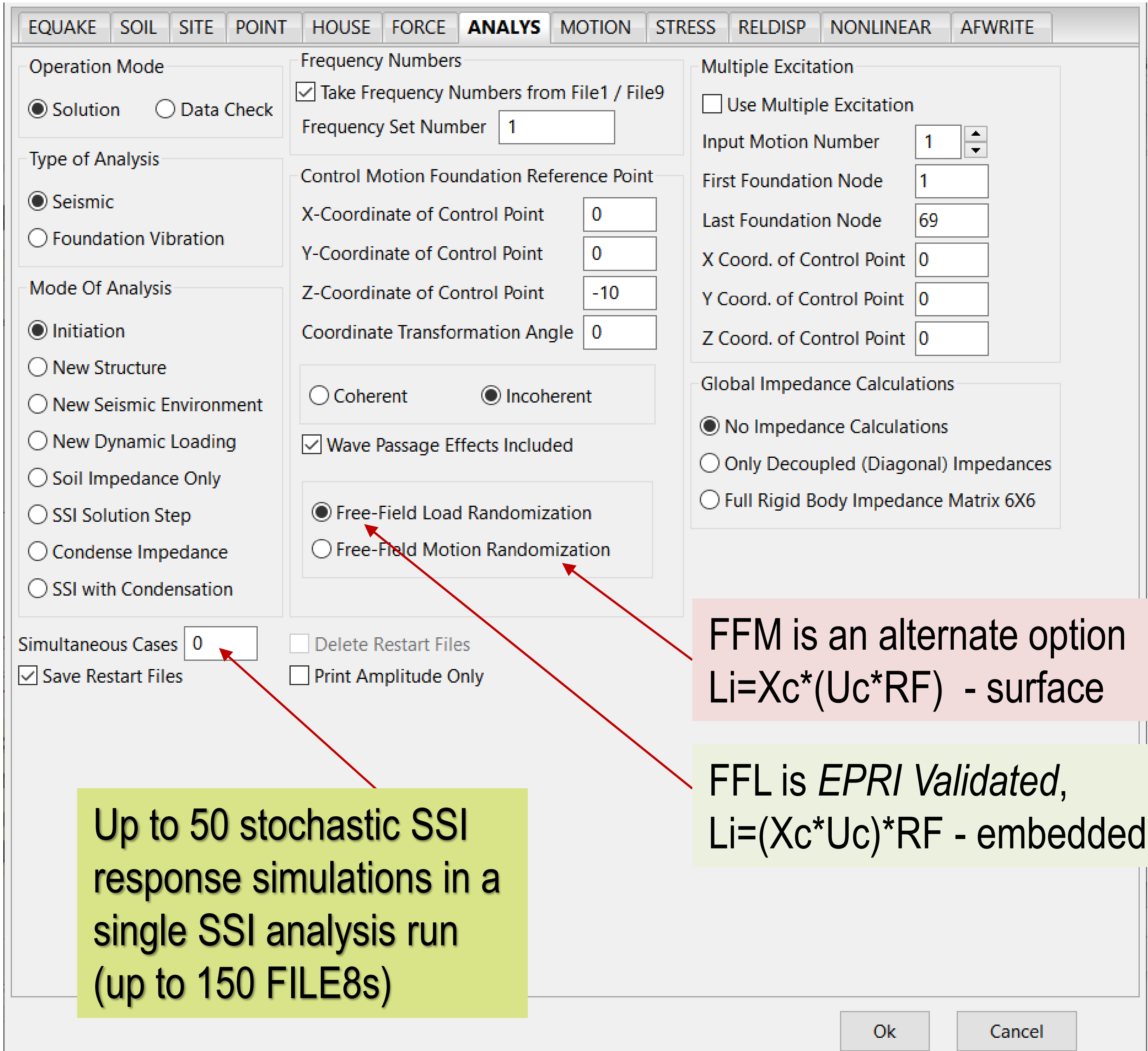
FILE74 includes strain level information on G & D in the nonlinear soil solid groups:

Grp_ID, Nmat, NElem

E_ID, g_{eff}, G_0, D_0, G_1, D_1, G_1/G_0, D_1/D_0

• • •   • • •

ANALYS Module Coherent & Incoherent SSI



FFM is an alternate option
Li=Xc*(Uc*RF) - surface

FFL is *EPRI Validated*,
Li=(Xc*Uc)*RF - embedded

Up to 50 stochastic SSI response simulations in a single SSI analysis run (up to 150 FILE8s)

Simultaneous Cases Option

To save significant SSI runtime for batch runs, the ANALYSFS can solve simultaneously all three X, Y and Z input directions for seismic SSI analysis (coherent, seismic option), or to up to 500 load cases of external forcing function (vibration option).

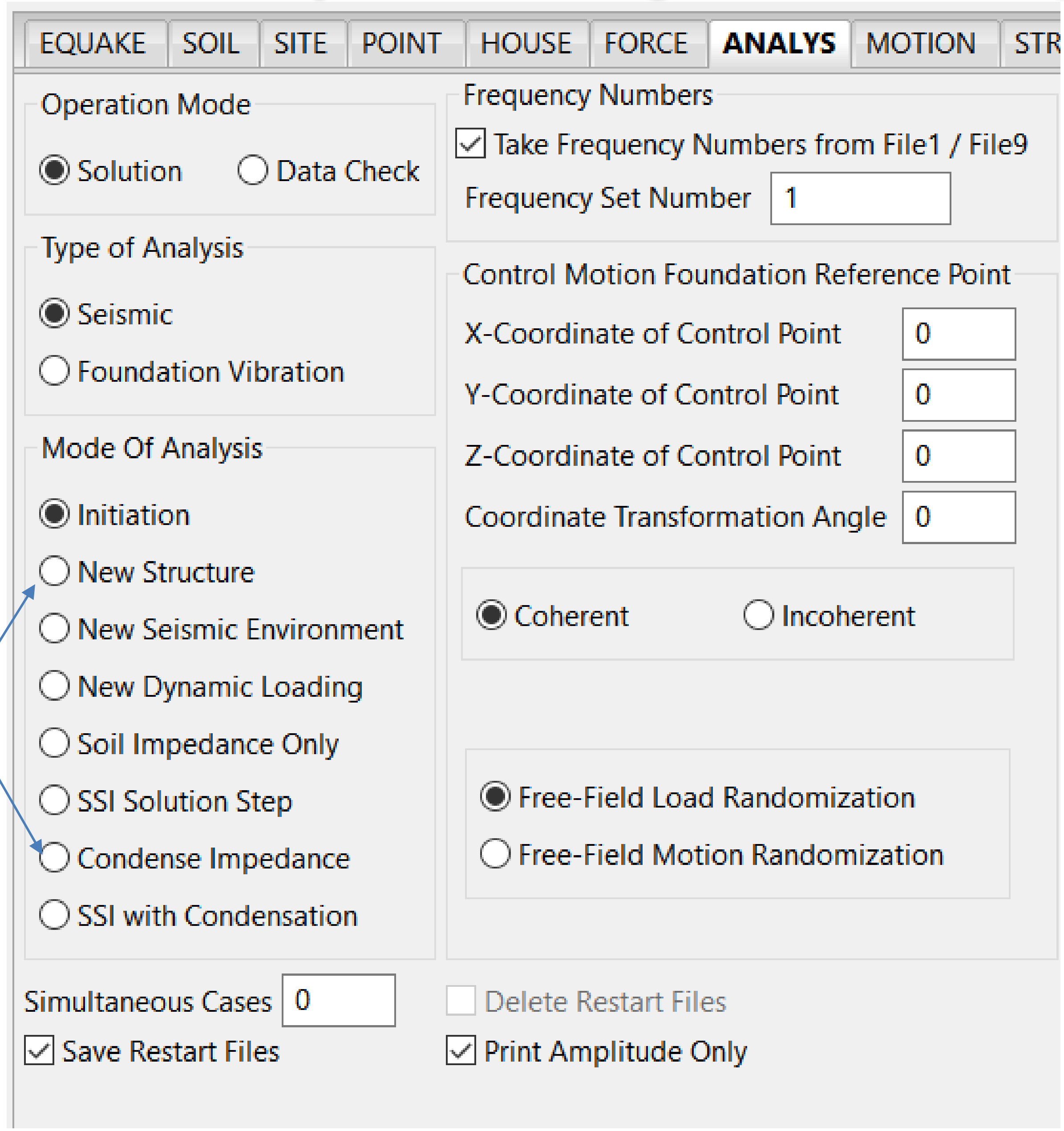
For seismic analysis the user should type 1 to get all three X,Y and Z direction results in a single run. The ANALYS run results will be saved in three FILE8 files computed for X,Y and Z inputs named FILE8X, FILE8Y and FILE8Z. To use this option for seismic analysis, the SITE module should be run before ANALYS run for X, Y and Z direction inputs and generate the FILE1X, FILE1Y and FILE1Z files.

The user should define for seismic input the SV waves for the X-direction by selecting x' direction and 0 angle in the .sit SITE input file, the SH waves for the Y-direction by selecting y' direction and 0 degree angle and the P waves in the Z-direction by selecting z direction and 0 angle.The coordinate transformation angle in the .anl ANALYS input file should be 0.

It should be noted that if the “***Simultaneous Cases***” is selected for *incoherent SSI analysis*, then, up 50 simulations can be solved in a single run. The FILE77 produced by HOUSE should be also replicated in the FILE77001, FILE77002, up to FILE77050 before the ANALYS module is run.

For *external force/vibration analysis* the user should type a three-digit number to input multiple load cases, up to 500 load external force cases. To use multiple external force cases up to 500 load cases in a single ANALYS run, the FORCE module should be run before ANALYS for all considered load cases and generate the FILE9001, FILE9002, FILE9003...up to FILE500. For the 50 load cases, ANALYSFS produces the 150 FILE8 files named FILE8001, FILE8002, FILE8003,...up to FILE8150 depending on the number of load cases.

Restart SSI Analyses Using ANALYS Options



Eight running modes are available in ANALYS

ANALYS Restart Options

The restart analyses imply that large files were saved. The following changes of problem parameters need different levels for the restart analyses:

1. **New Control Motion (No ANALYS restart)**

   Suppose results are required for a different time history (or response spectrum) of the control motion. Then, as long as the nature of seismic environment, i.e., the type of wave field, is not changed, only the module MOTION has to be re-executed

.

2. **New Structure or Near-Field Soil (Mode 2)**

   If changes are made in the superstructure or near field soil, the HOUSE, ANALYS, STRESS or MOTION and RELDISP modules have to be re-executed.

   The restart files for the fast-solver are the COOXxxx and COOTKxxx, DOFSMAP, FILE90 and FILE91 files.'

3. New Seismic Environment (Mode 3)

Suppose that structure was originally analyzed for the effects of vertically propagated body waves and that results are required for the case of incident Rayleigh waves causing the same motion at the control point as in the free field. In this case only a part of the SITE module and ANALYS module have to be re-executed.

4. New Dynamic Loading (Mode 4)

Similar with Mode 3 for new external force cases.

5. Compute only Soil Impedance Matrix (Mode 5)

Computes the COOXxxx for the selected SSI frequencies.

6. Compute SSI Solution (Mode 6)

Computes SSI solution only. The SSI system matrices COOTKxxx and load vectors LOADXYZxxx should be available in working folder

7. Condense Excavation Impedance Matrix for Embedded Models (Mode 7)

Condense Impedance (Mode 7, Limited Analysis Run): This option is for computing the frequency-dependent condensed excavated soil impedance and condensed seismic load vector. Currently this ANALYS option should be used in conjunction with the Option AA-R only.

This ANALYS running option will create a set of *IMP_EXCV_Fxxx* and *IMP_EXCVxxx* files for each SSI frequency, as well as a *DOFSMAP_IMP_EXCV* file. These files will be used to create super-elements in ANSYS for performing the SSI harmonic analysis via Option AA-R.

Demos 15 and 16 provides the details on how to use the reduced soil impedance and reduced seismic load vector for seismic SSI solution in ANSYS via Option AA-R, and ACS SASSI, respectively.

8. SSI with Condensation (Mode 8)

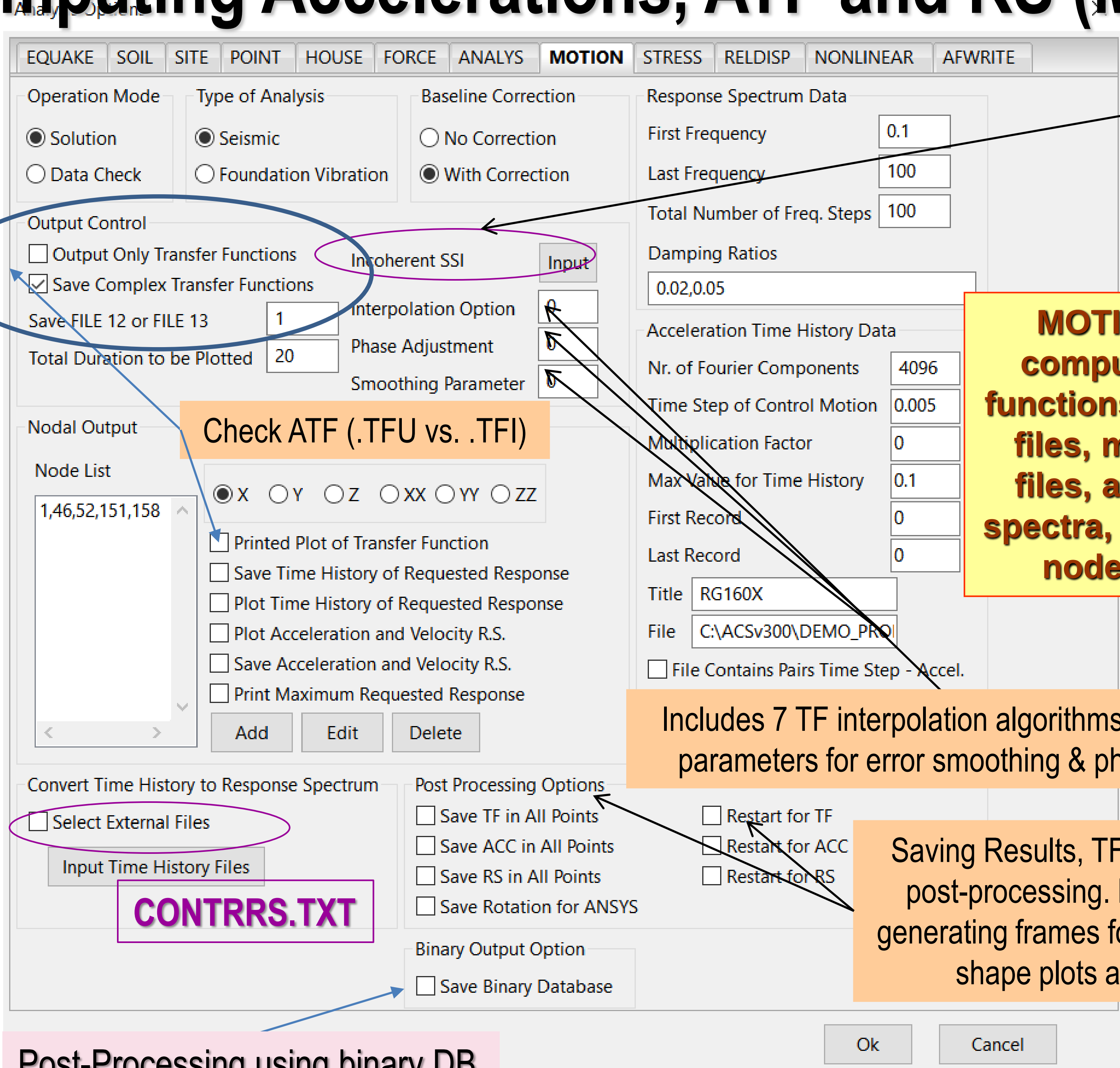
Restart for SSI Solution using condensed soil impedance matrix.

The generalized procedure for the **FVROM-INT** approach is as follows:

1) *Identify the key or condensation frequencies* using *SOIL* module
2) *Compute the condensed* the excavation impedance matrix and seismic load vectors for the *key frequencies* using *ANALYS* module
3) *Interpolate* these quantities *for all SSI frequencies*
4) *Compute SSI system solution* for *all SSI frequencies* for the *reduced excavation impedance matrix and load vectors using ANALYSI module*

Demo 16 shows in detail the use of the FVROM-INT approach

Computing Accelerations, ATF and RS (MOTION)



SRSSTF.TXT

MOTION Module computes transfer functions, TFU and TFI files, motions, ACC files, and response spectra, RS at selected nodes, RS files.

Check ATF (.TFU vs. .TFI)

Includes 7 TF interpolation algorithms and explicit input parameters for error smoothing & phase adjustment.

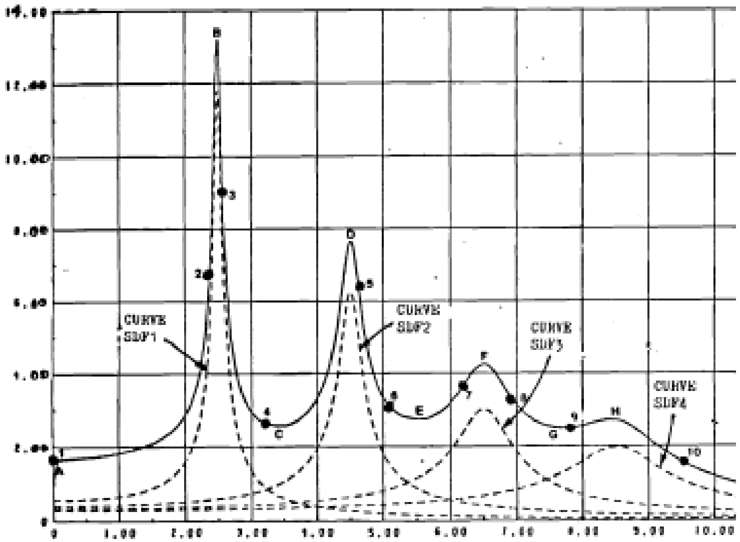
Saving Results, TFU, ACC and RS for post-processing. Restart is used for generating frames for contour, deformed shape plots and animations

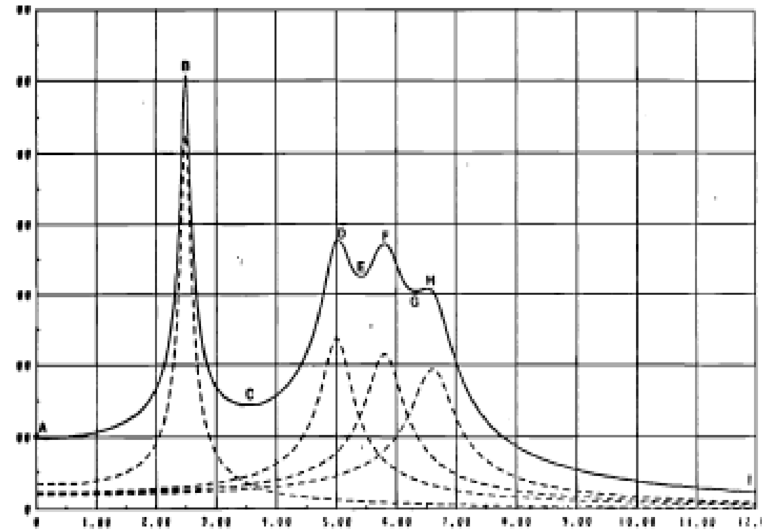
CONTRRS.TXT

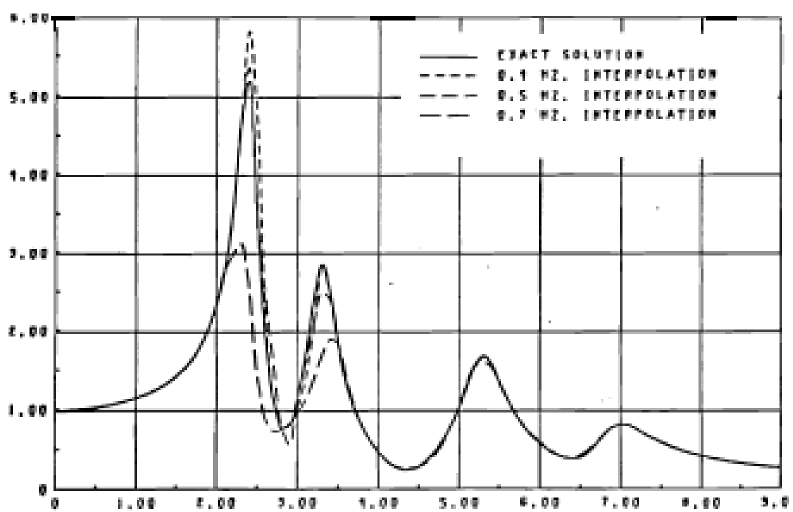
Post-Processing using binary DB

Notes

SSI Response Transfer Function Interpolation







(after Tajirian, 1983)

Transfer Function Interpolation Technique

- The frequency interpolation technique used to interpolate the response for frequencies in between the calculated and to obtain the response for all FFT frequencies is based on the frequency response function of a two-degree-of-freedom system.

- The total response of a two-degree-of-freedom system subjected to harmonic base excitation for each degree-of-freedom has the following general from

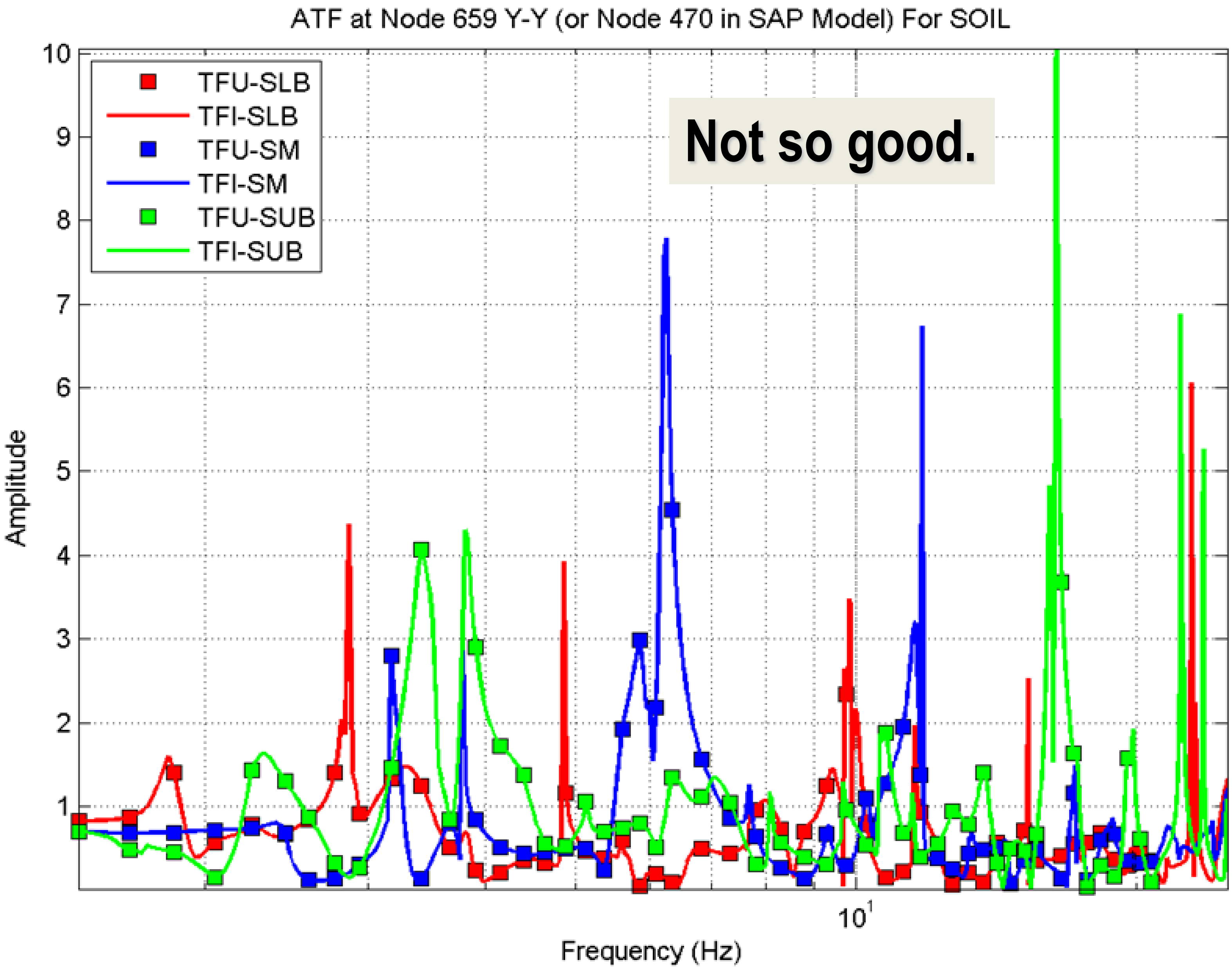
$$U^i(\omega) = \frac{C_1^i \omega^4 + C_2^i \omega^2 + C_3^i}{\omega^4 + C_4^i \omega^2 + C_5^i}$$

To compute the complex coefficients a five equation system needs to be solved

$$\begin{bmatrix} \omega_1^4 & \omega_1^2 & 1 & -\omega_1^2 U_1 & U_1 \\ \omega_2^4 & \omega_2^2 & 1 & -\omega_2^2 U_2 & U_2 \\ \omega_3^4 & \omega_3^2 & 1 & -\omega_3^2 U_3 & U_3 \\ \omega_4^4 & \omega_4^2 & 1 & -\omega_4^2 U_4 & U_4 \\ \omega_5^4 & \omega_5^2 & 1 & -\omega_5^2 U_5 & U_5 \end{bmatrix} \begin{bmatrix} C_1 \\ C_2 \\ C_3 \\ C_4 \\ C_5 \end{bmatrix} = \begin{bmatrix} \omega_1^4 U_1 \\ \omega_2^4 U_2 \\ \omega_3^4 U_3 \\ \omega_4^4 U_4 \\ \omega_5^4 U_5 \end{bmatrix}$$

Note:
Based on our experience that the two-degree-of-freedom-system interpolation technique may sometimes introduce some spurious spectral peaks and valleys. Thus, it is recommended when significant spectral peaks are identified between the frequency solution points to add new frequency points in that range.

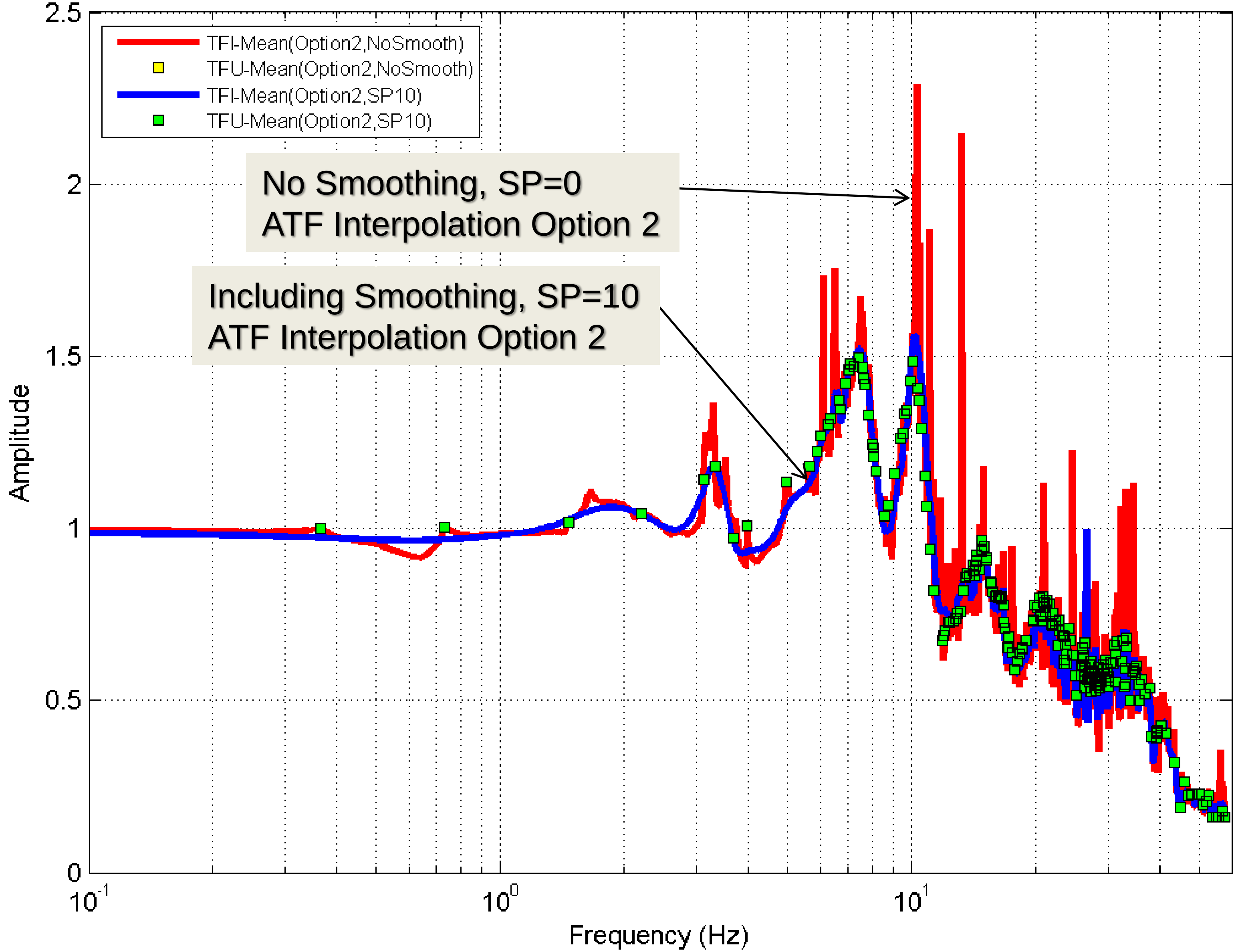
Comparative ATF Results w/ Less Frequencies



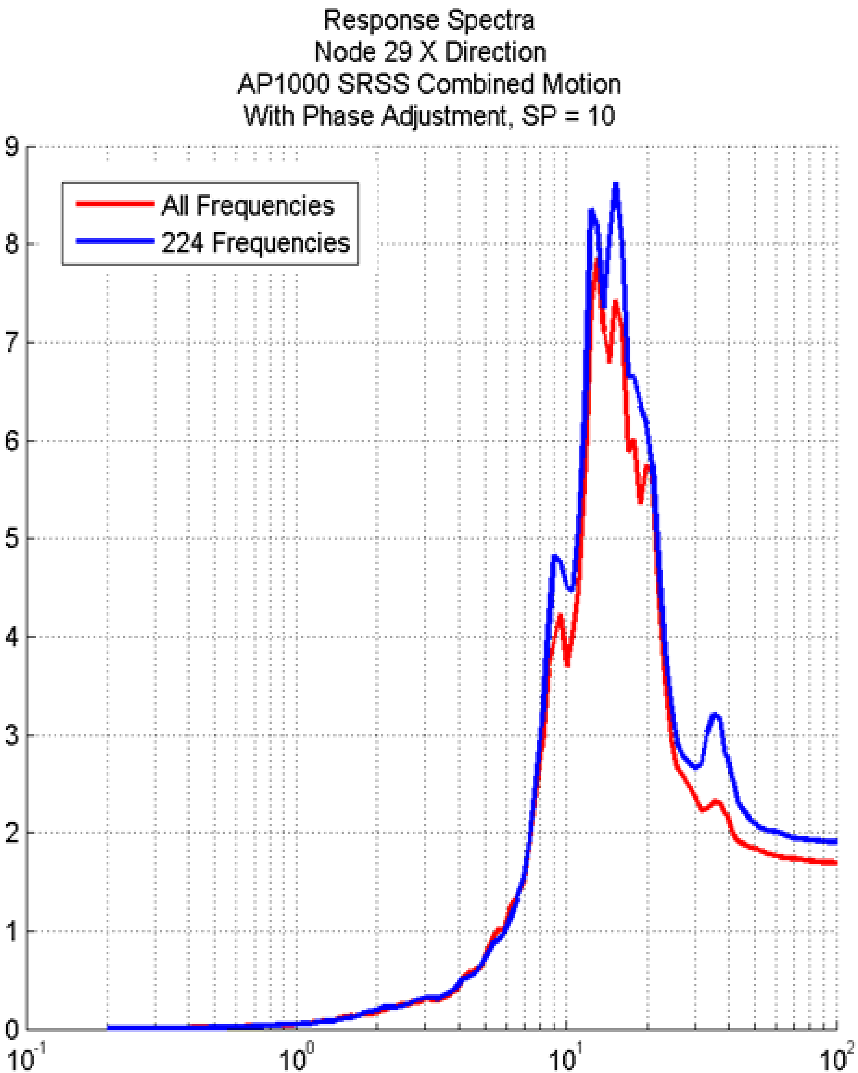
Criteria for Selecting Frequency Solution Points

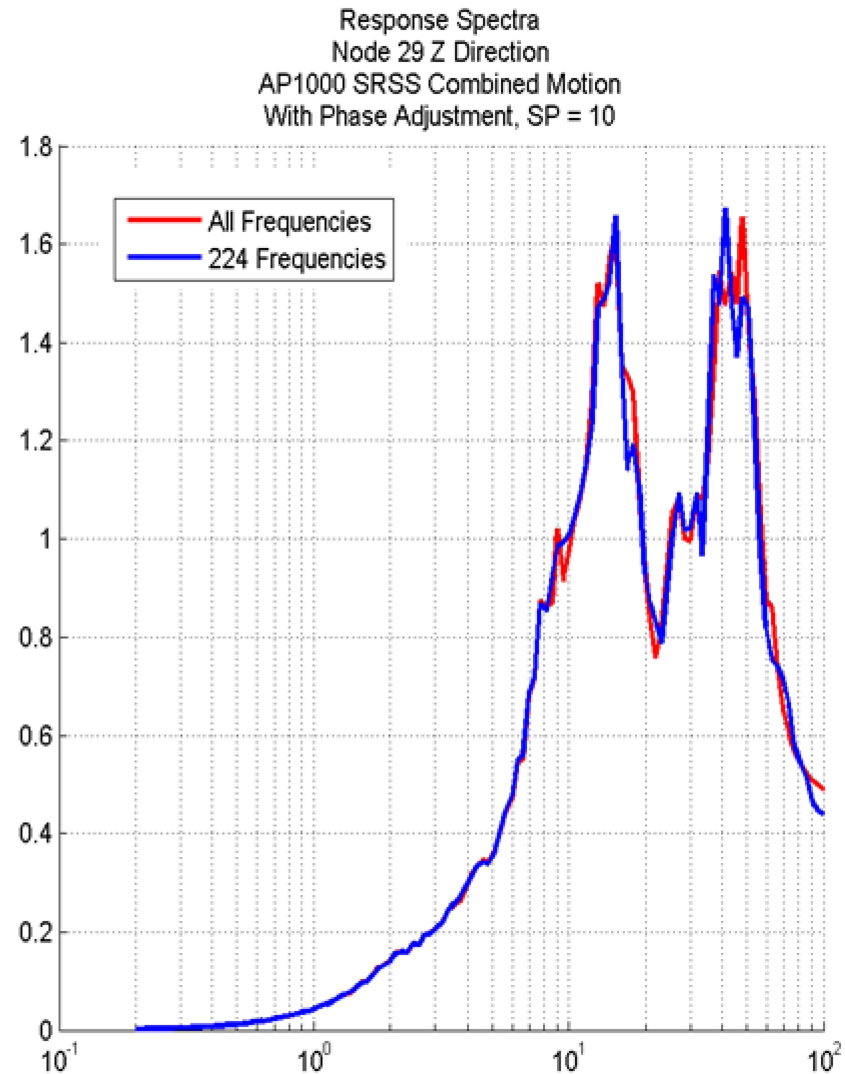
- Depend on the number of peaks in the transfer function at the specific response location and how close these peaks are located relative to each other.
- The frequencies of analysis can be selected by recognizing that the SSI effects usually shift the frequencies to the lower frequency range and tend to flatten the sharp peaks or sometimes even eliminate the fixed-base response peaks.
- Most of the practical problems are sufficient to solve SSI solution for a limited number of frequencies; about 40-50 frequencies for stick SSI models and about 150-250 frequencies for 3D SSI models. A larger number of frequencies needed for rock sites than soil sites.
- If no information on natural frequencies of the system are is available, it is necessary to selected adequate number of frequencies with an uniform increment throughout the frequency range of interest. Then, after revising the results, more frequencies are added to reconstruct the missing spectral peaks.

Including ATF Interpolation Error Smoothing. Results Using New Option 2. With Overlapping Moving Windows, A=(A1+A2+A3+A4+A5)/5)

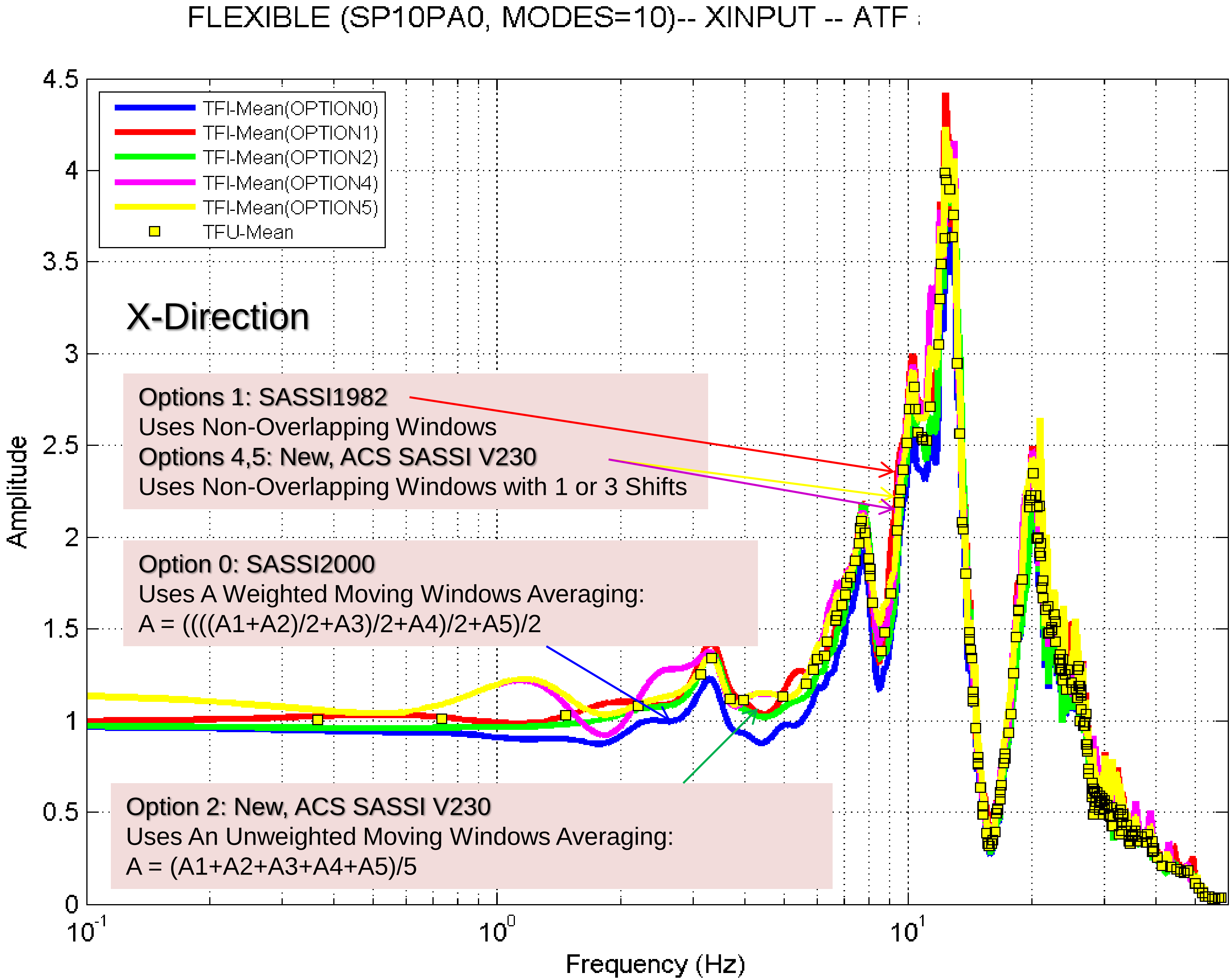


ATF Interpolation Error Smoothing Results for EPRI AP1000 Stick Model.

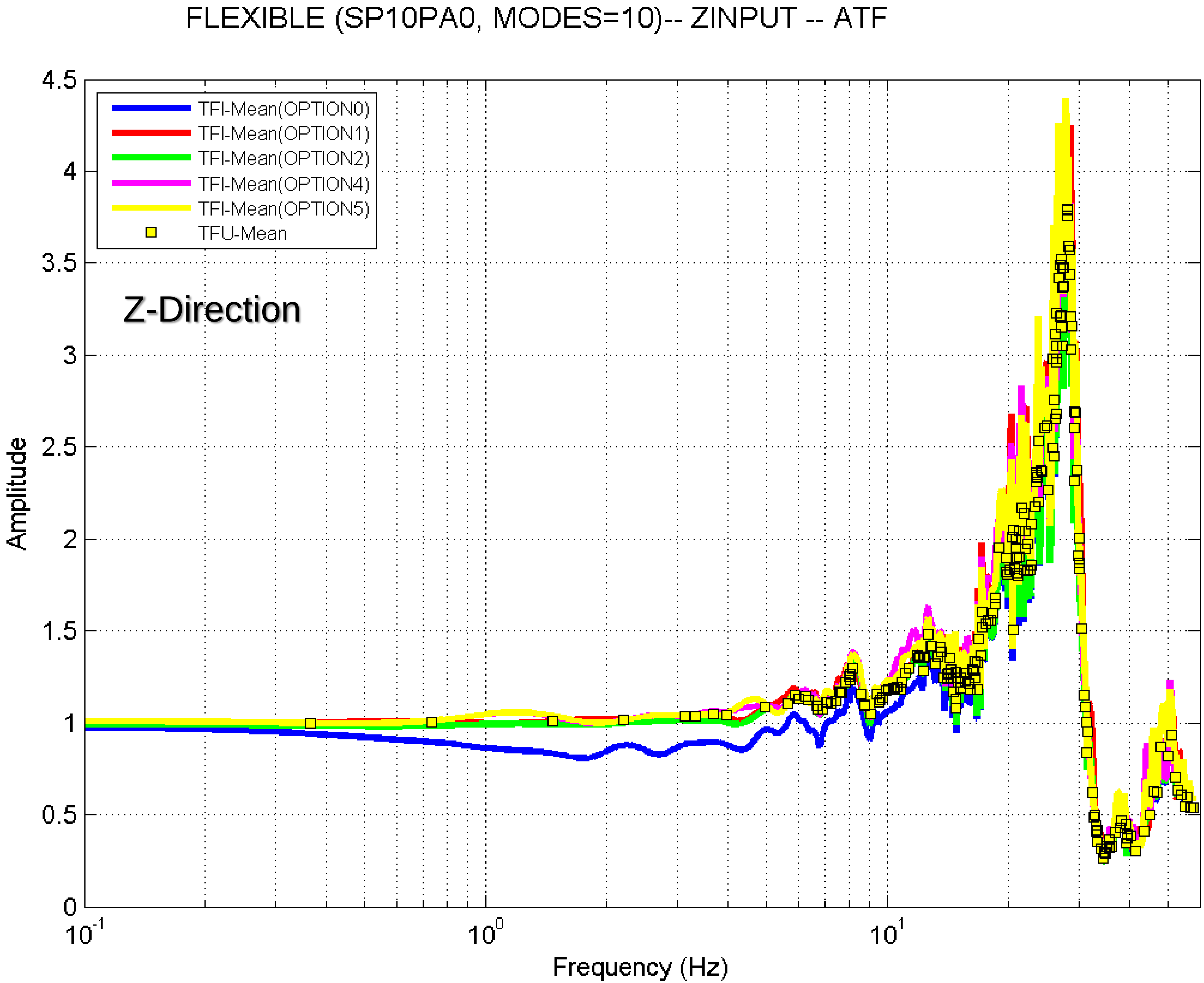




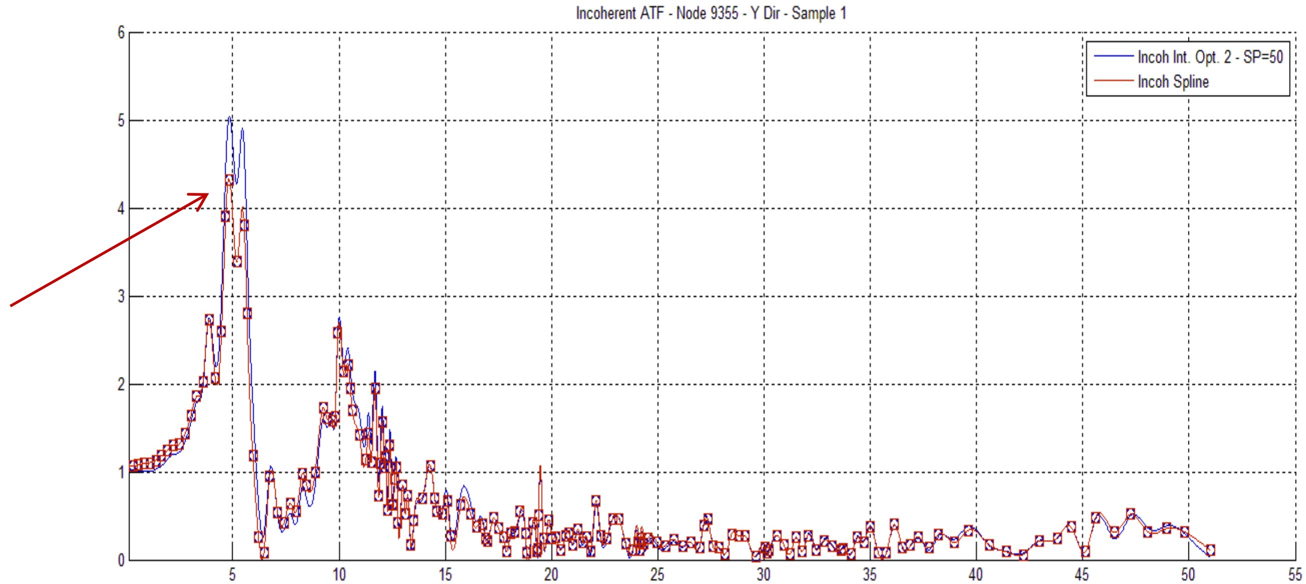
(Mean) Incoherent Interpolated ATF Using Different Interpolation Schemes

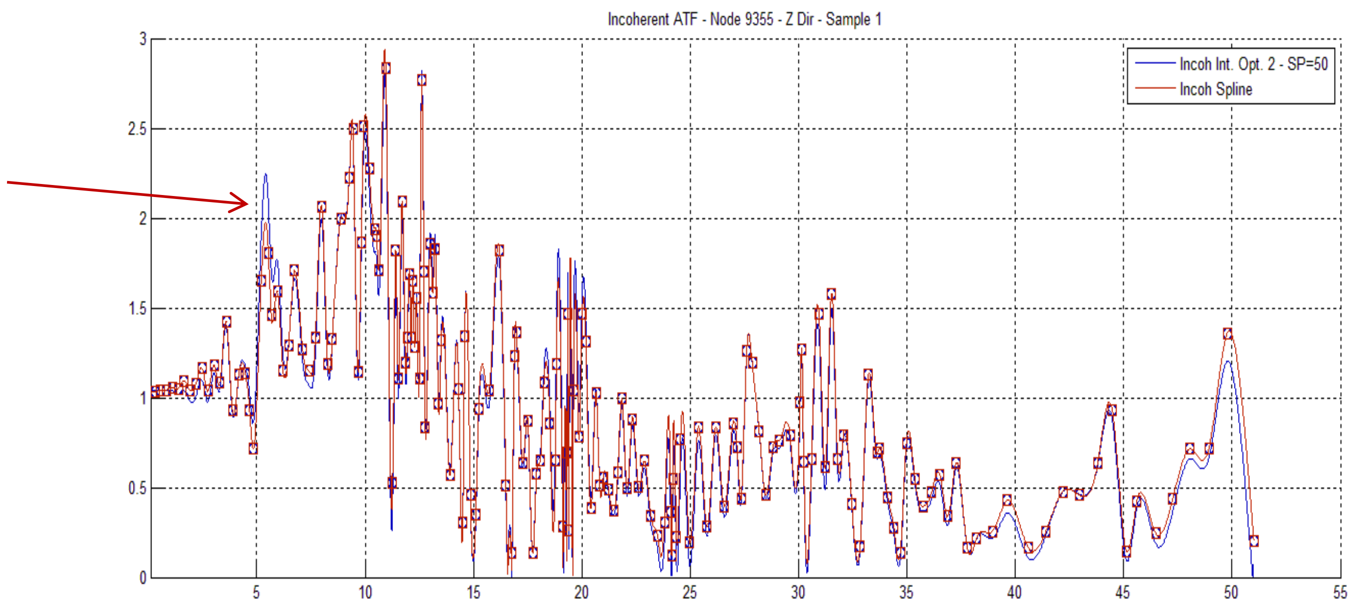


(Mean) Incoherent Interpolated ATF Using Different Interpolation Schemes

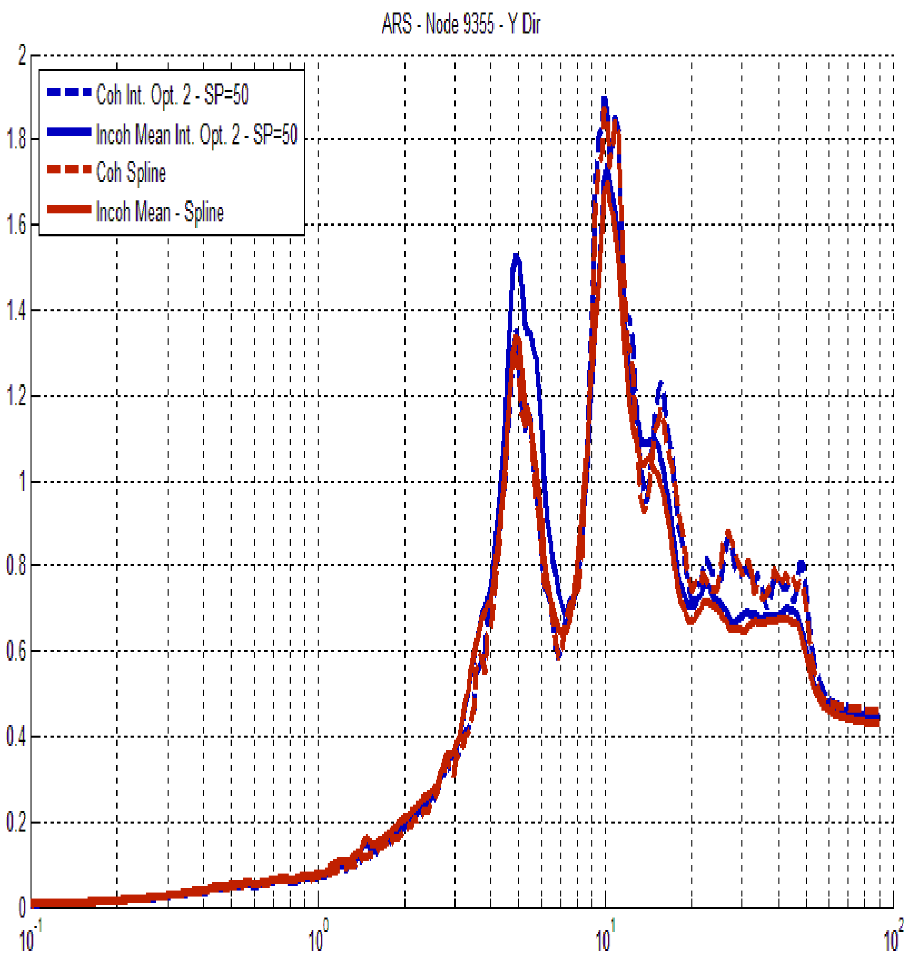


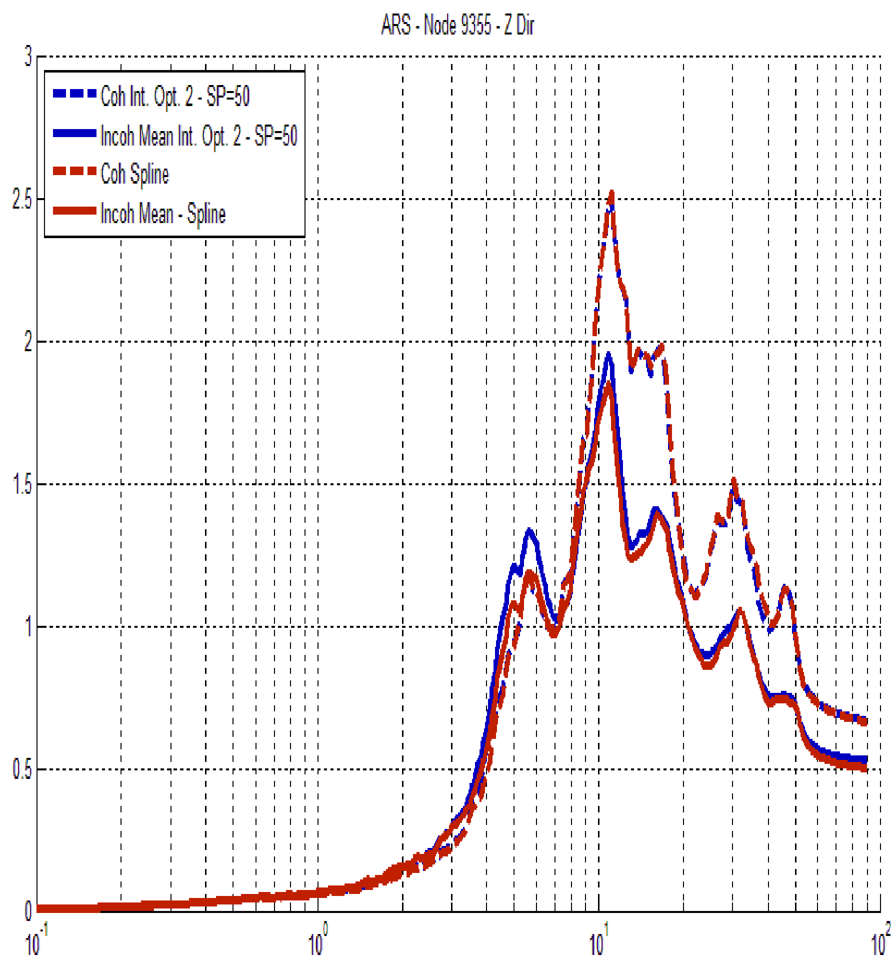
Spline Interpolation Applied to SSI Simulations





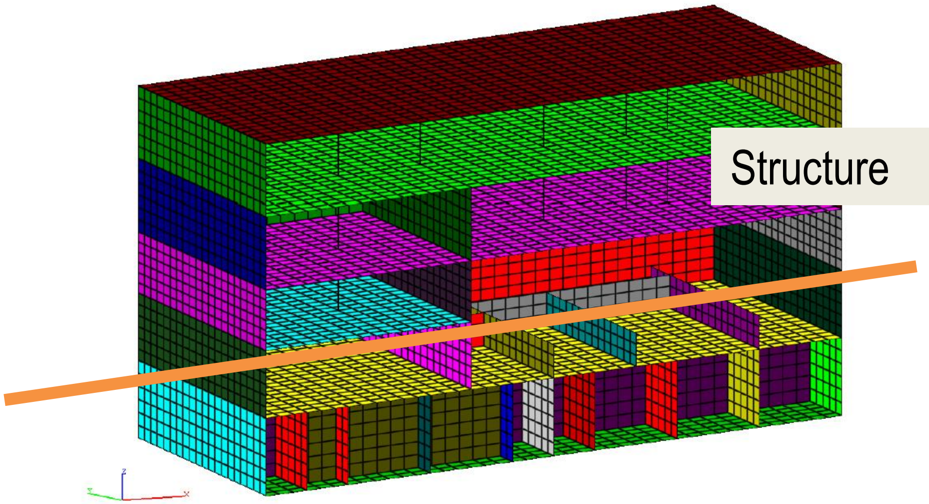
Spline Interpolation Applied to Incoherent SSI Simulation Approach (RECOMMENDED)

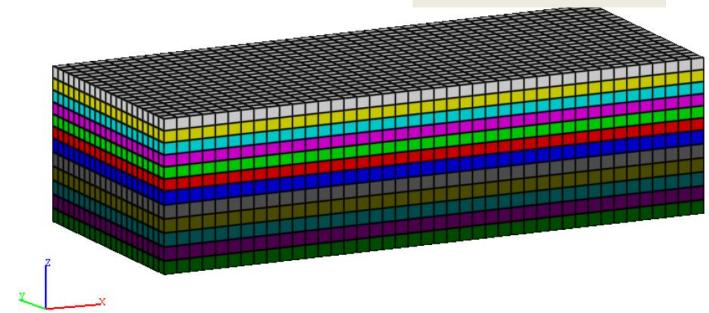


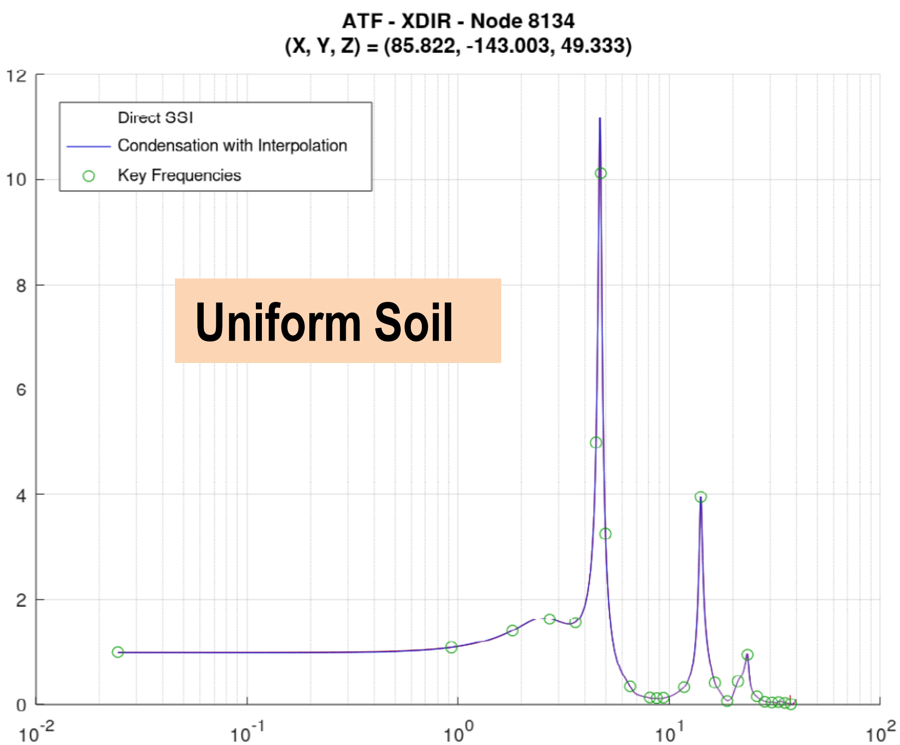


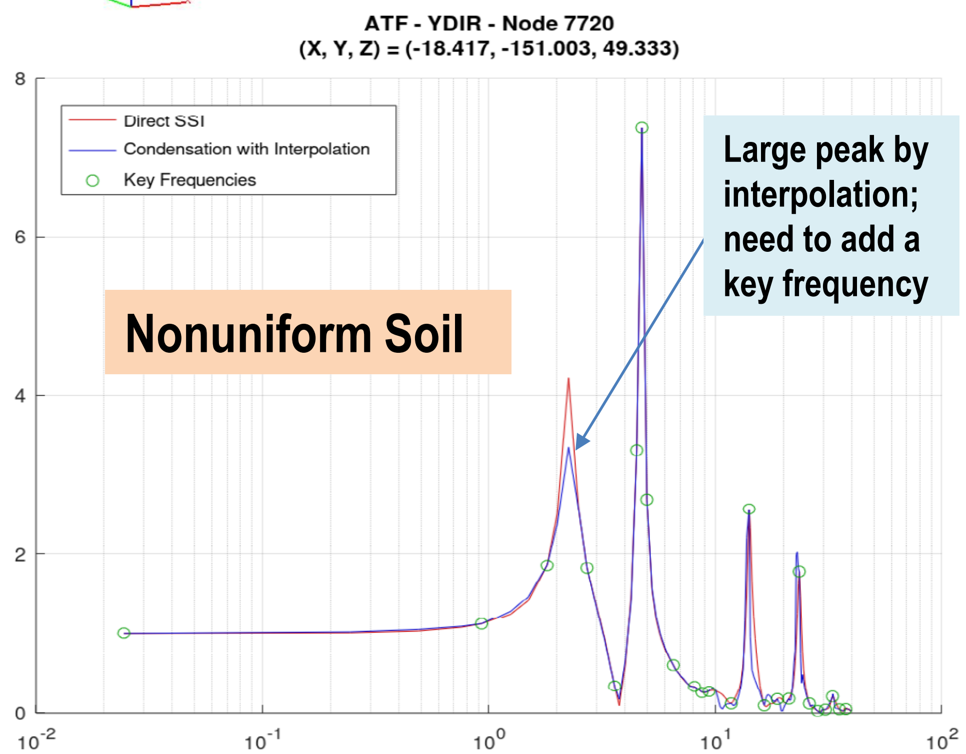
ATF Checking for FVROM Key SSI Frequencies

See Demo 16

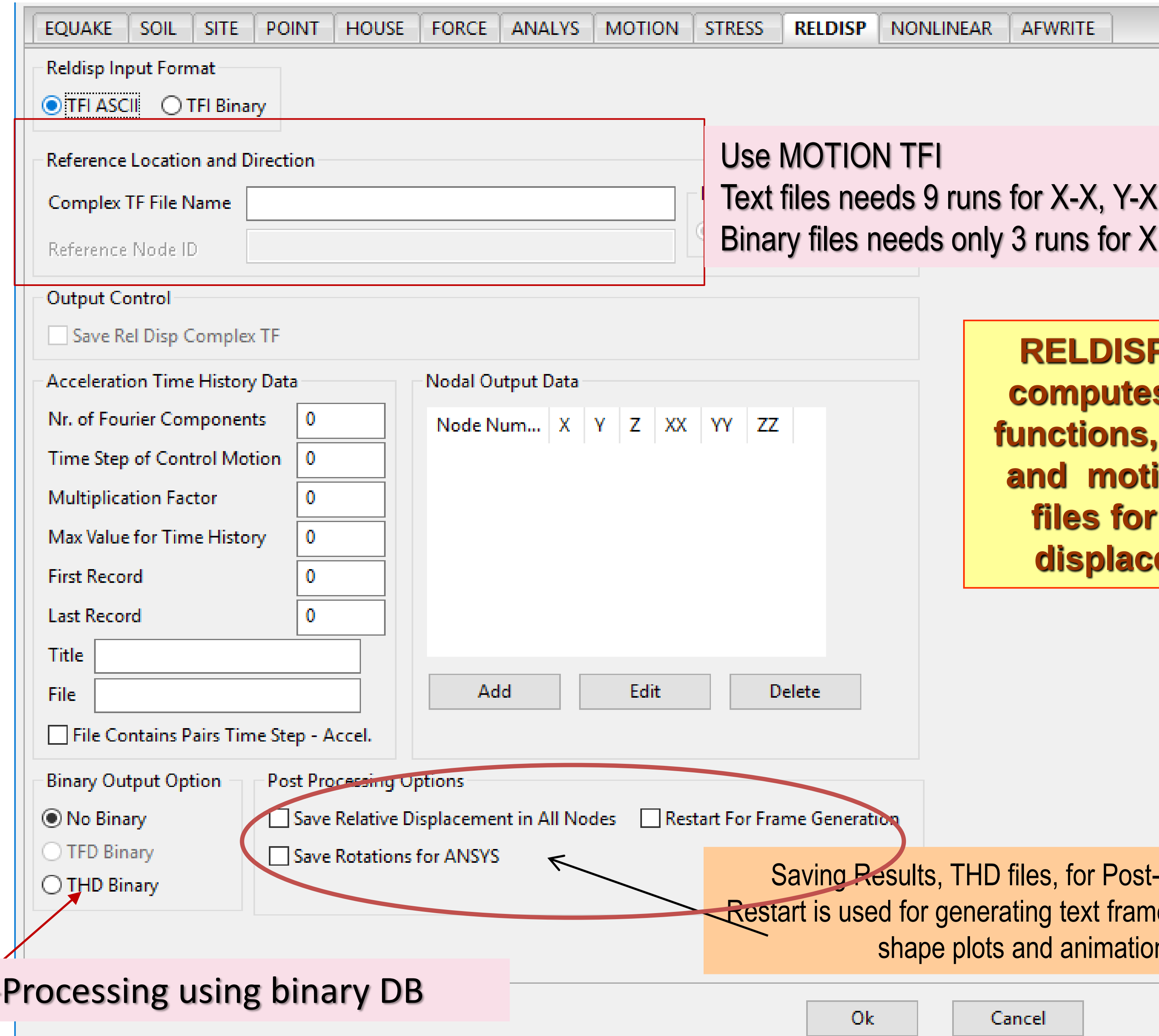








Computing Relative Displacements (RELDISP)



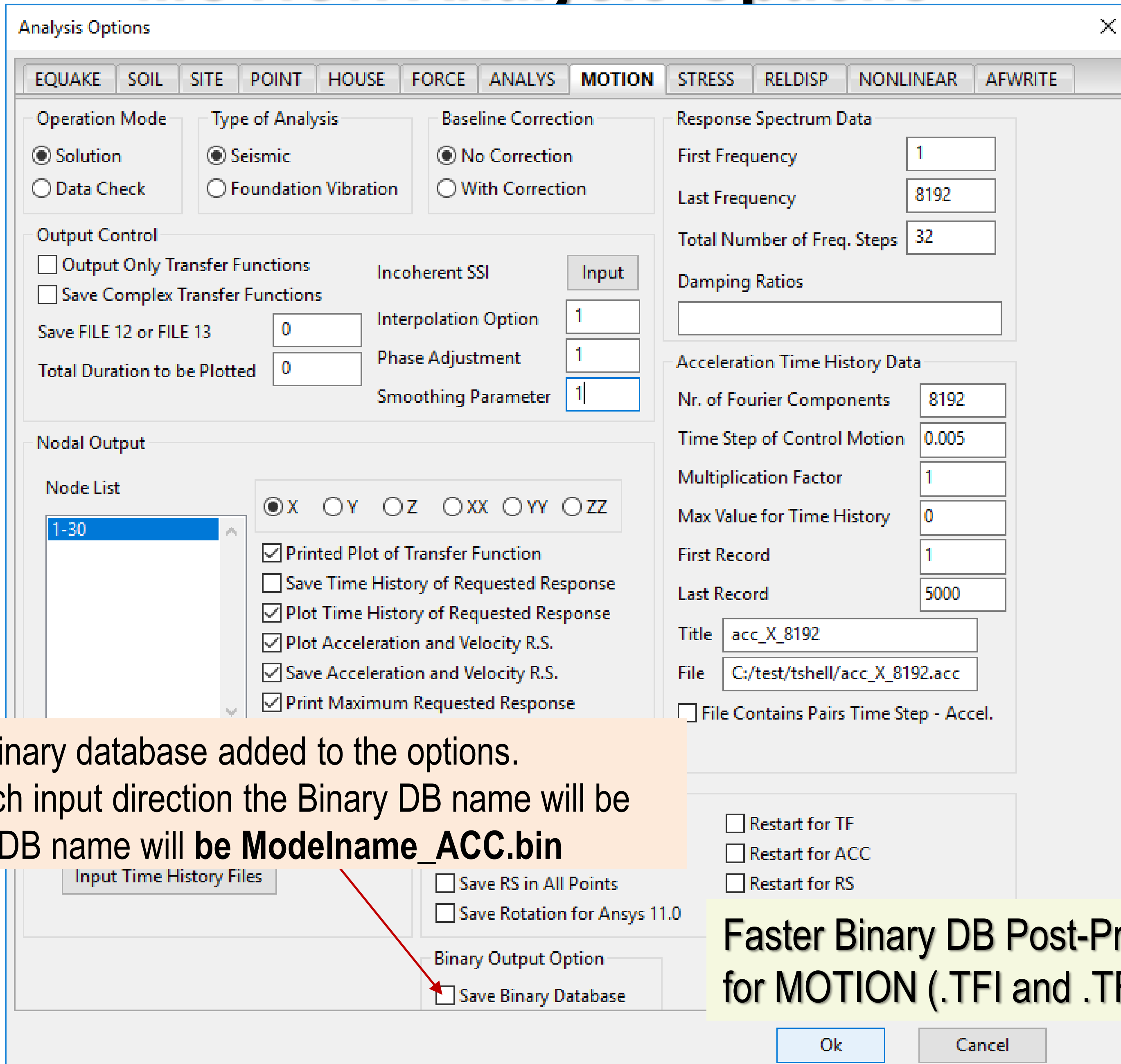
Use MOTION TFI
Text files needs 9 runs for X-X, Y-X, Z-X….
Binary files needs only 3 runs for X, Y and Z

RELDISP Module computes transfer functions, TFD files, and motions, THD files for relative displacements.

Saving Results, THD files, for Post-processing.
Restart is used for generating text frames for deformed shape plots and animations

Post-Processing using binary DB

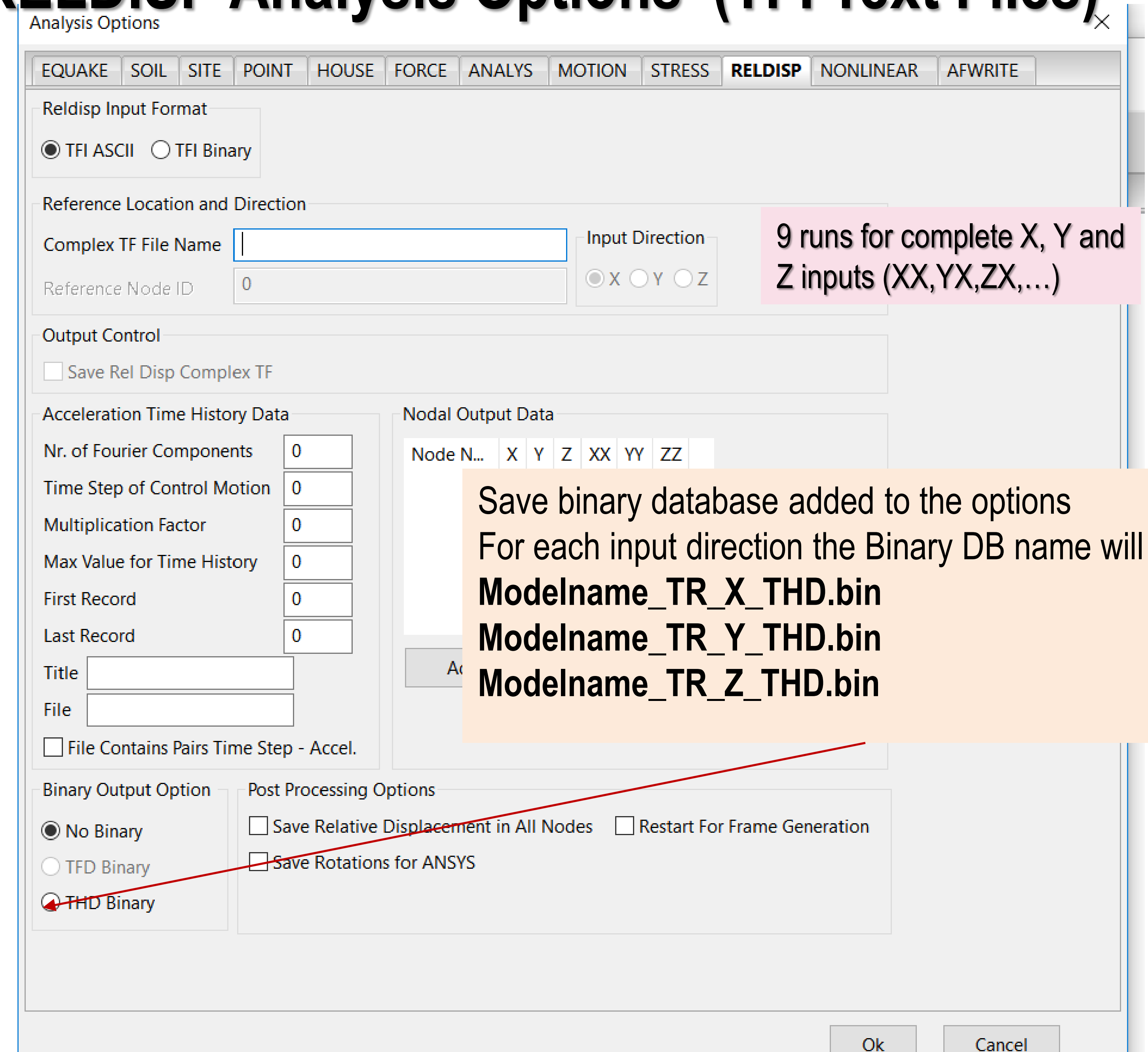
MOTION Analysis Options



Save binary database added to the options.
For each input direction the Binary DB name will be
Binary DB name will **be Modelname_ACC.bin**

Faster Binary DB Post-Processing
for MOTION (.TFI and .TFU)

RELDISP Analysis Options (TFI Text Files)



9 runs for complete X, Y and Z inputs (XX,YX,ZX,…)

Save binary database added to the options

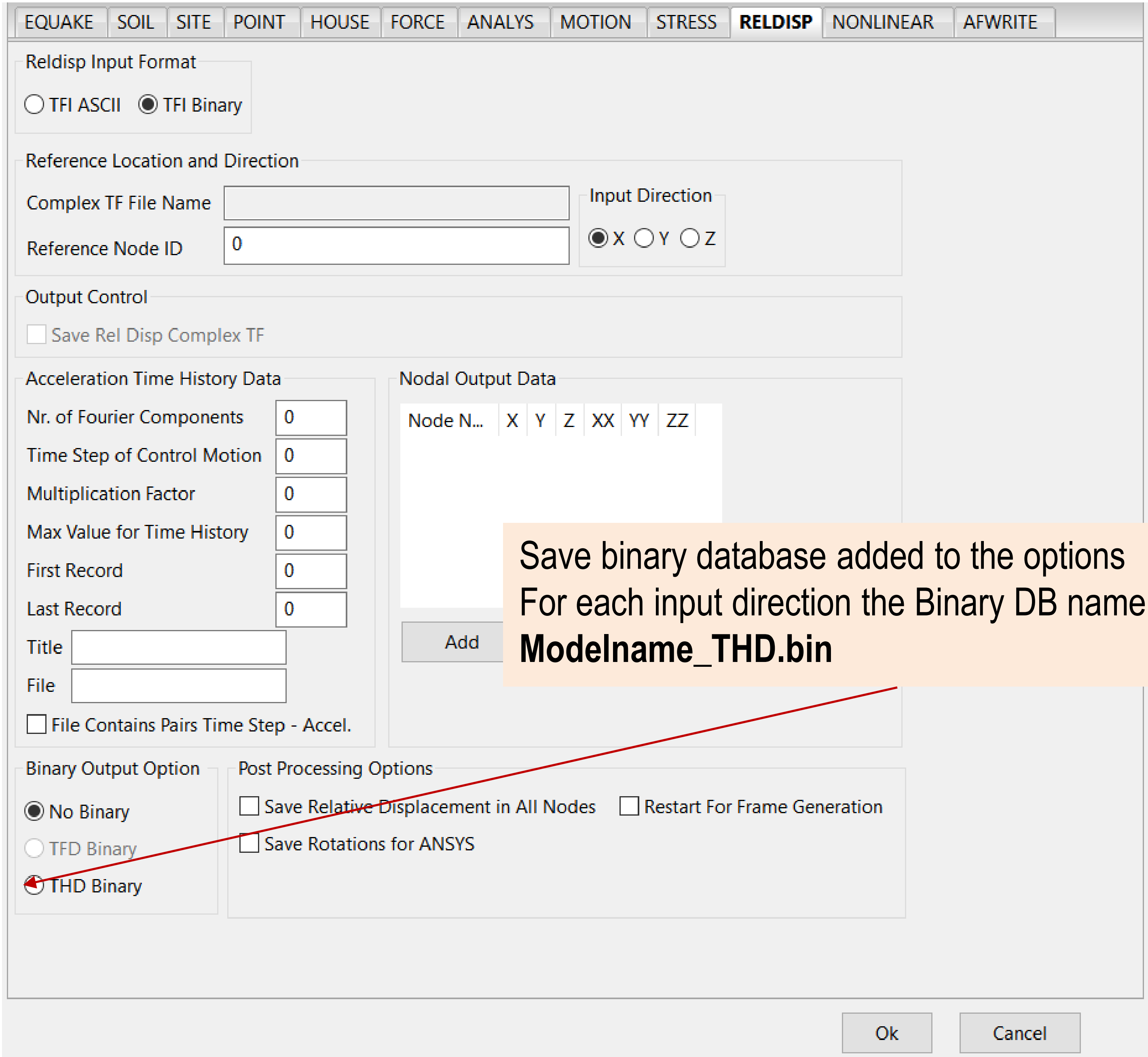
For each input direction the Binary DB name will be

Modelname_TR_X_THD.bin

Modelname_TR_Y_THD.bin

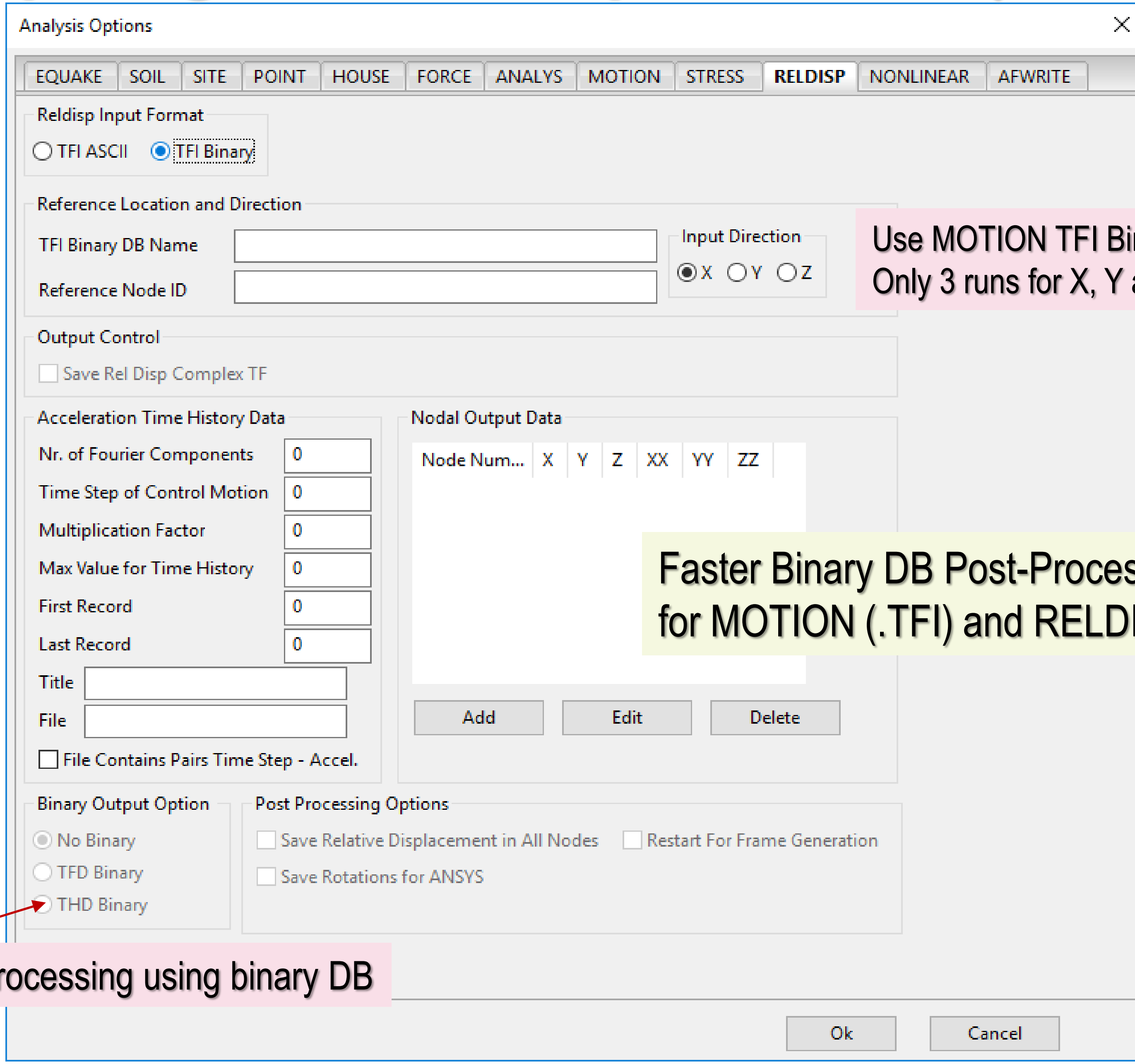
Modelname_TR_Z_THD.bin

RELDISP Analysis Options (TFI Binary Files)



Save binary database added to the options
For each input direction the Binary DB name will be
Modelname_THD.bin

Computing Relative Displacements (RELDISP)

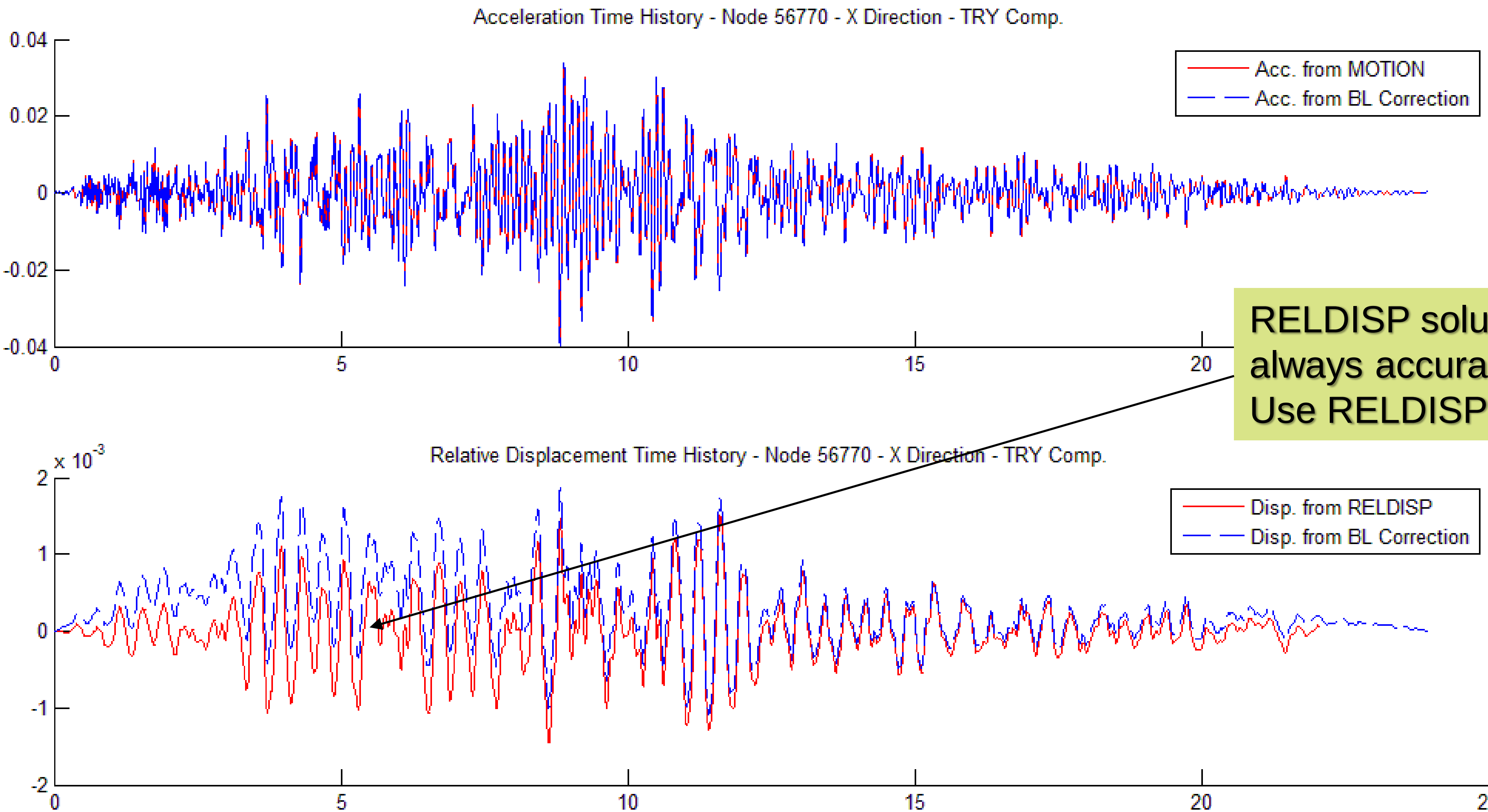


Use MOTION TFI Binary DB.
Only 3 runs for X, Y and Z inputs.

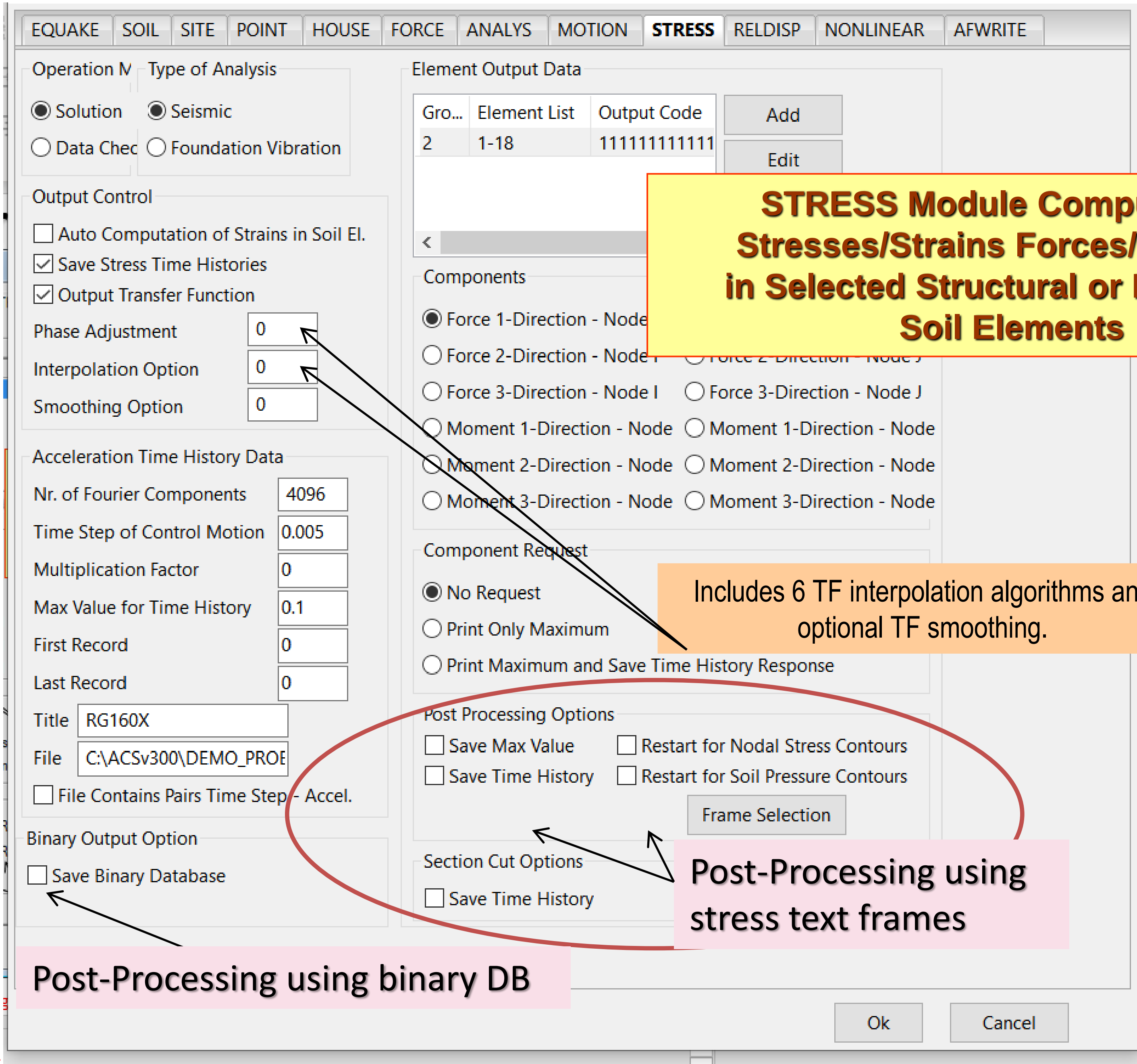
Faster Binary DB Post-Processing
for MOTION (.TFI) and RELDISP (.thd)

Post-Processing using binary DB

Relative Displacements Computed By Baseline Correction (“Approximate”) and RELDISP (“Exact”)



Computing Output Stresses (STRESS)



STRESS Module Computes Stresses/Strains Forces/Moments in Selected Structural or Near-Field Soil Elements

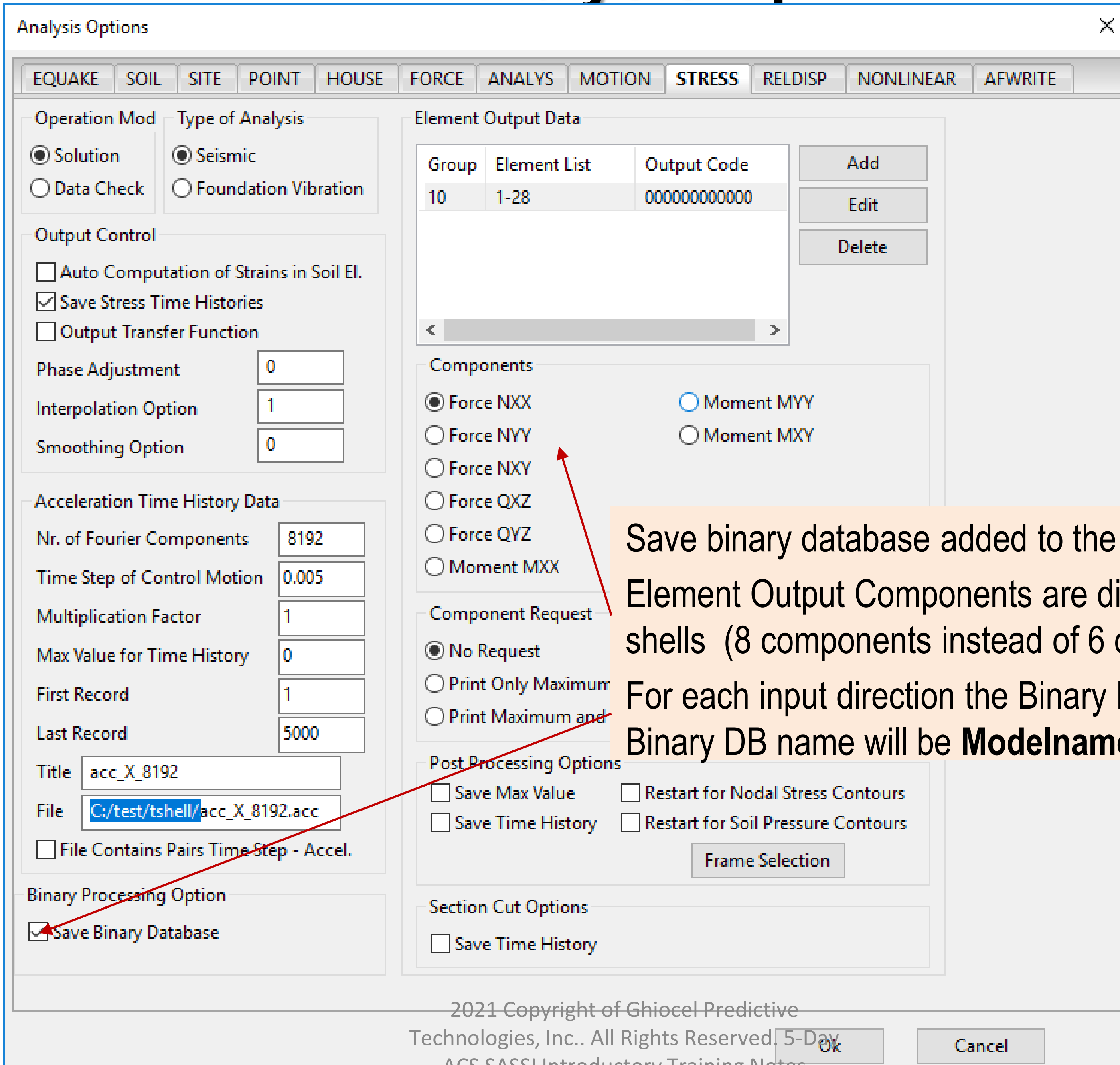
Includes 6 TF interpolation algorithms and optional TF smoothing.

Post-Processing using stress text frames

Post-Processing using binary DB

202

STRESS Analysis Options

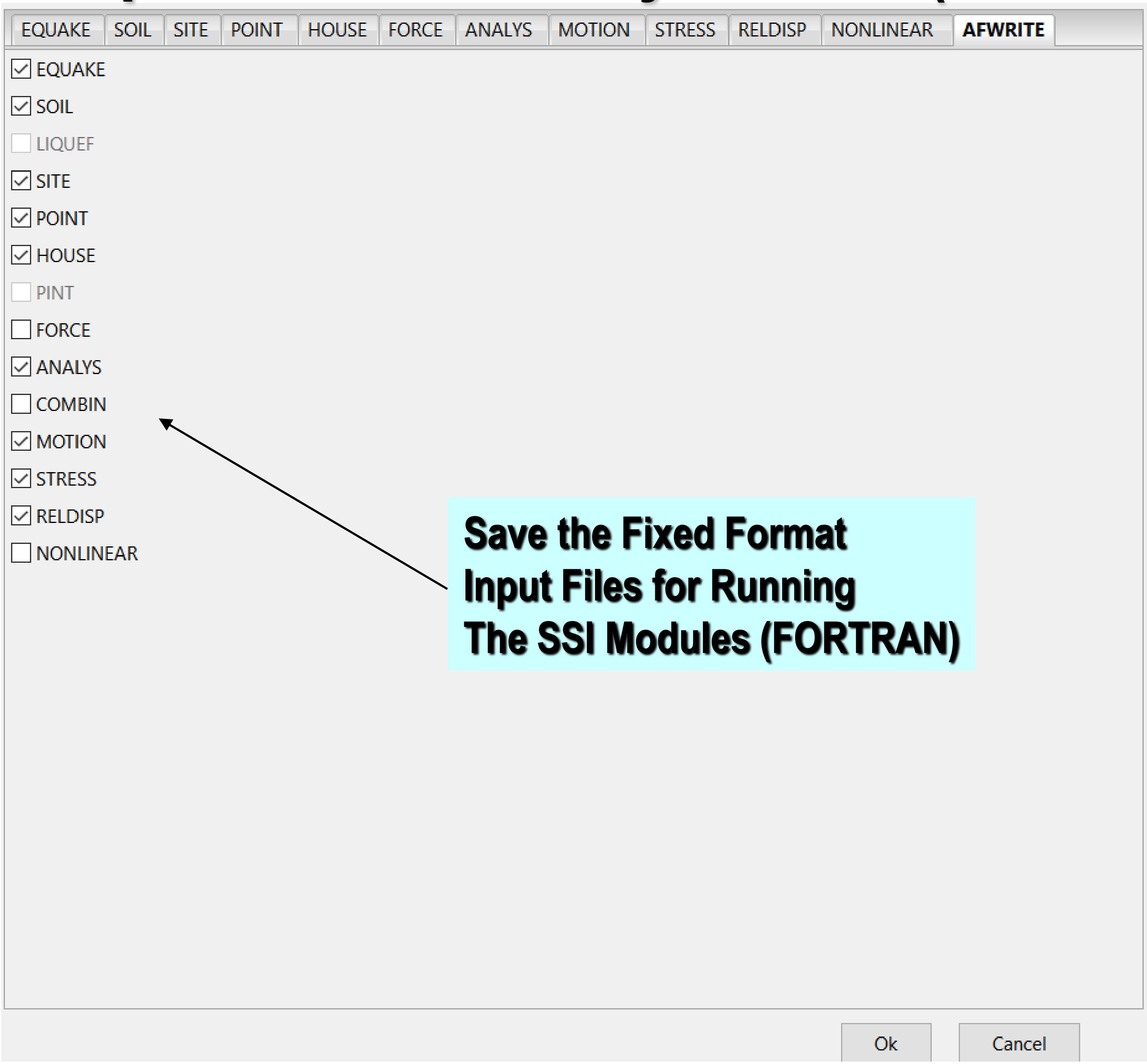


Save binary database added to the options

Element Output Components are different for thick shells (8 components instead of 6 components)

For each input direction the Binary DB name will be Binary DB name will be **Modelname_STRESS.bin**

Save Inputs for SSI Analysis Run (AFWRITE)



Save the Fixed Format Input Files for Running The SSI Modules (FORTRAN)

4. Description of SSI Response Text Files and Frames (for Animations)

Transfer Function (TF), Response Spectra (RS) and Time History (TH) Text Files for Post-Processing

RS	Response spectra data files generated by the motion module		
	Naming Scheme for TFU, TFI, TFD, ACC Files		
	Characters 1-5	Node Number	
	Characters 6-9	Translation (TR) or Rotational (R) degree of freedom	
	Characters 10-11	Damping ratio number	
TFU	Uninterpolated acceleration transfer functions written by the motion module and stress transfer functions		
TFI	Interpolated acceleration transfer functions written by the motion module and stress transfer functions written by the stress module		
TFD	Displacement transfer functions generated by the reldisp module		
THD	Displacement time history written by reldisp module		
ACC	Acceleration time history written by motion module		
	Naming Scheme for Acceleration TFU, Acceleration TFI, TFD, THD, and ACC Files		
	Characters 1-5	Node Number	
	Characters 6-9	Translation (TR) or Rotational (R) degree of freedom	
TH	Soil time history for layers		
	Naming Scheme		
	ACC***	Acceleration time history for soil layer ***	i.e. ACC001.TH is the acceleration time history for soil layer 1
	SN***	Strain time history for soil layer ***	i.e. SN001.TH is the strain time history for soil layer 2
	SS***	Stress time history for soil layer ***	i.e. SS001.TH is the stress time history for soil layer 3
THS	Stress time history written by stress module		
	Naming Scheme for THS, stress TFU, and Stress TFI		
	etype_gnum_enum_comp		e.g. BEAMS_012_00001_FXI.THS
	etype =	element type	
	gnum =	group number	
	enum =	element number	
	comp =	stress component	
Frames.txt	Post processing frames for stress and motion		
ELEMENT_CENTER_ABS_MAX_STRESSES.TXT	List of maximum stresses for each element		
STATIC_SOIL_PRESSURES.TXT	Defines additional soil pressure (geological pressure) to be included in soil pressure frames		
SRSSTF.txt	SRSS option in motion		

Frame Files for Post-Processing

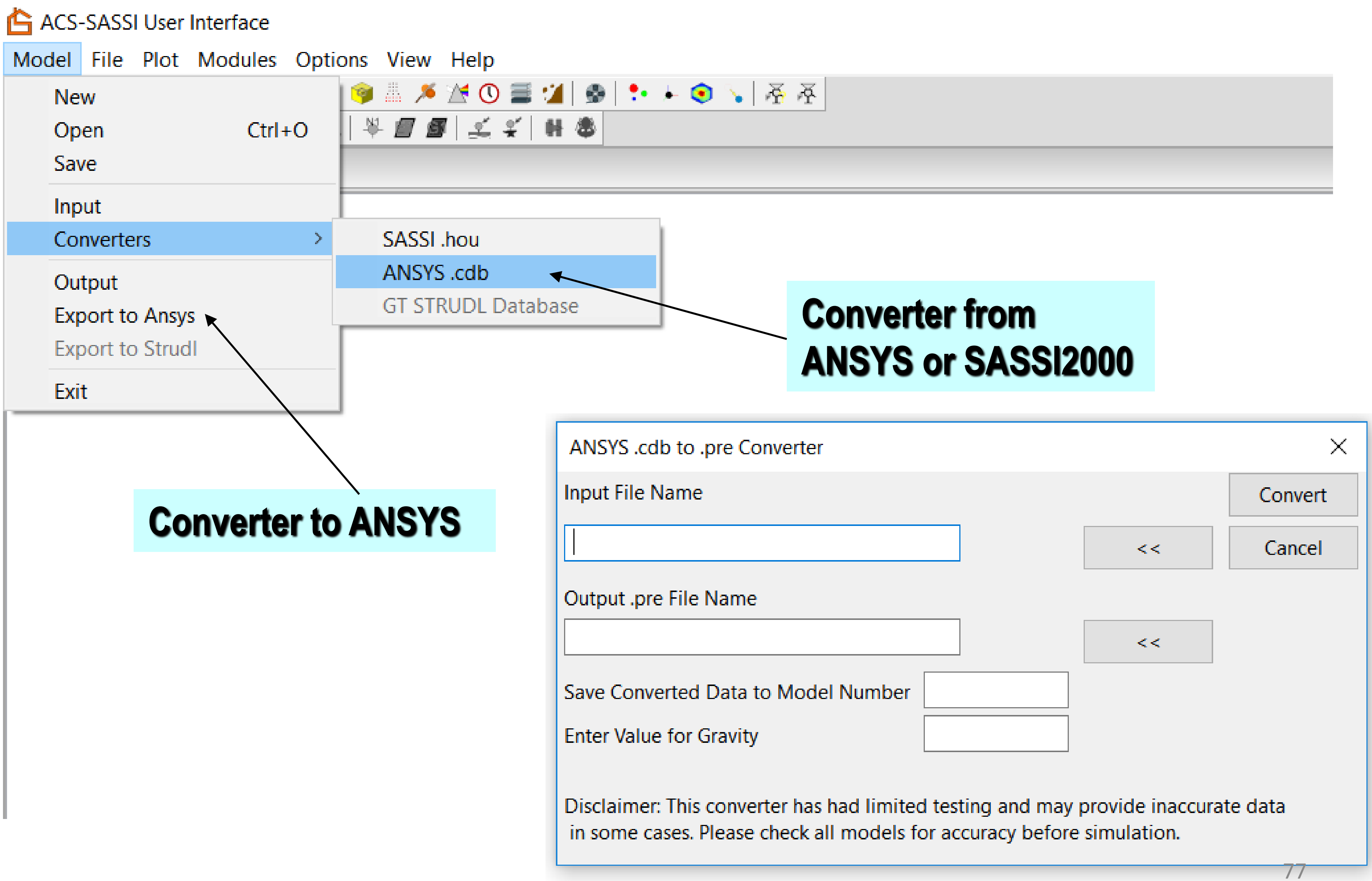
RS Frames Naming Scheme				
RS##_freq_filenum				e.g. \RS\RS01_000.10_00001
	## =	Damping number		
	freq =	frequency		
	fnum =	Frame number		
TFU Frames Naming Scheme				
TFU_freq_filenum				e.g. \TFU\TFU_000.02_00001
	freq =	frequency		
	fnum =	Frame number		
ACC Frames Naming Scheme				e.g. \ACC\ACC_00.000_00001
ACC_time_filenum				
	time =	time		
	fnum =	Frame number		
THD Frames Naming Scheme				e.g. \THD\THD_00.000_00001
THD_time_filenum				
	time =	time		
	fnum =	Frame number		
Stress Frame Naming Scheme				
stress_time_fnum_comp				e.g. \NTRESS\stress_00.000_00001_sig
	time =	time		
	fnum =	Frame number		
	comp =	Stress Component		
		sig	Solids	Normal Stress
			Shells	Membrane Stress
		tau	Solids	Shear Stress
			Shells	Membrane Shear
		bdsig	Bending Stress (shell elements only)	
		bdtau	Bending Shear (shell elements only)	
Soil Pressure Frame Naming Scheme				
press_time_fnum_type				e.g. \SOILPRES\pres_00.000_00001_nod
	time =	time		
	fnum =	Frame number		
	type =	Element Values or Nodal Values		
		ele	Element Values	
		nod	Nodal Values	

Frame Files for Post-Processing (cont')

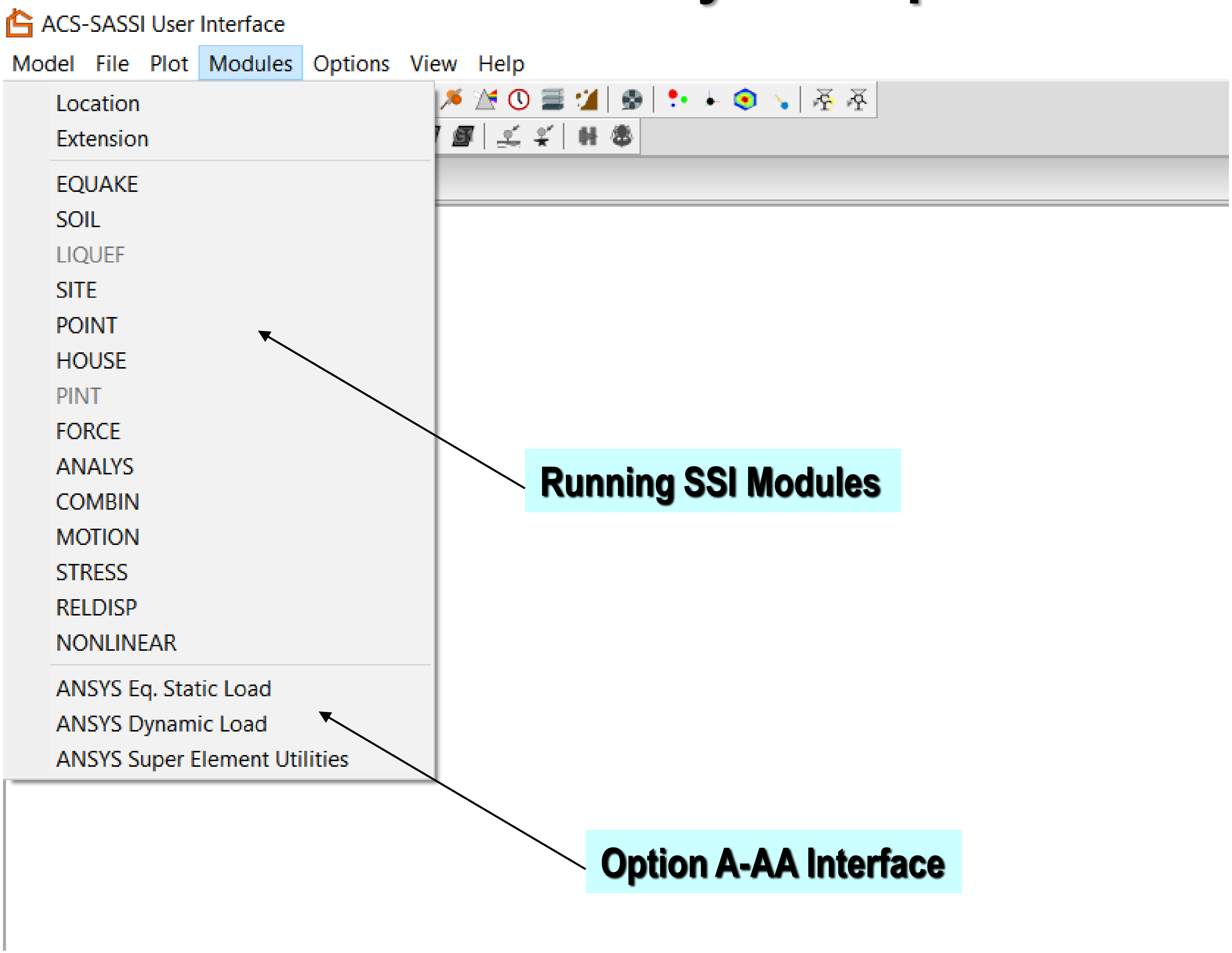
Maximum Value Frames				
Stress				
stress_ABS_MAX_comp				e.g. \NSTRESS\stress_ABS_MAX_sig
	comp =	Stress Component		
		sig	Solids	Normal Stress
			Shells	Membrane Stress
		tau	Solids	Shear Stress
			Shells	Membrane Shear
		bdsig	Bending Stress (shell elements only)	
		bdtau	Bending Shear (shell elements only)	
Soil Pressure				
press_ABS_MAX_type				e.g. \SOILPRES\pres_ABS_MAX_nod
	type =	Element Values or Nodal Values		
		ele	Element Values	
		nod	Nodal Values	

5. Main UI Menus for Model Inputs, Performing SSI Analysis and Plotting Results

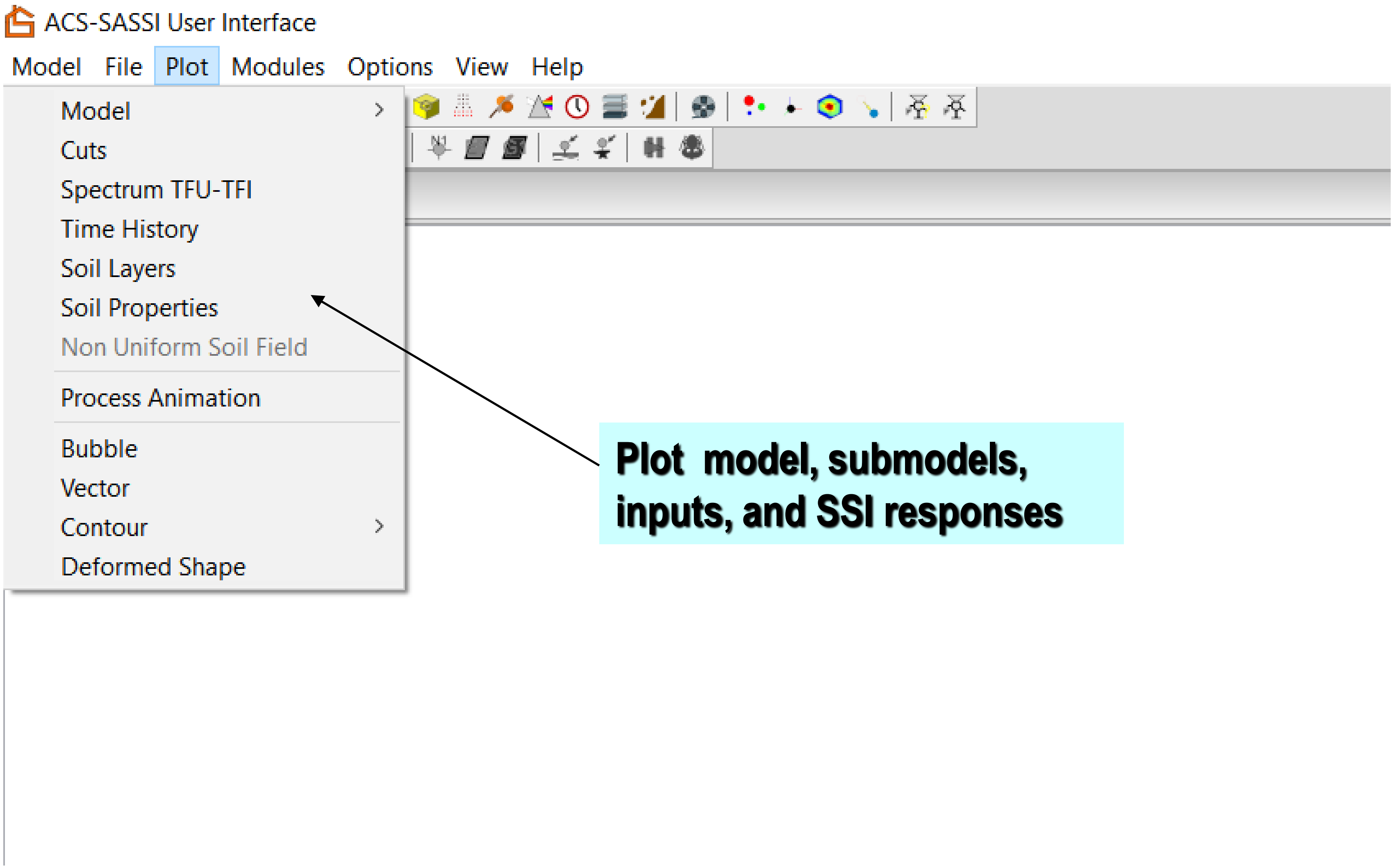
ACS SASSI Model Input File Capabilities



ACS SASSI SSI Analysis Capabilities



ACS SASSI Graphical Processing Capabilities



Plot model, submodels, inputs, and SSI responses

SSI Response Post-Processing Options

User Interface:

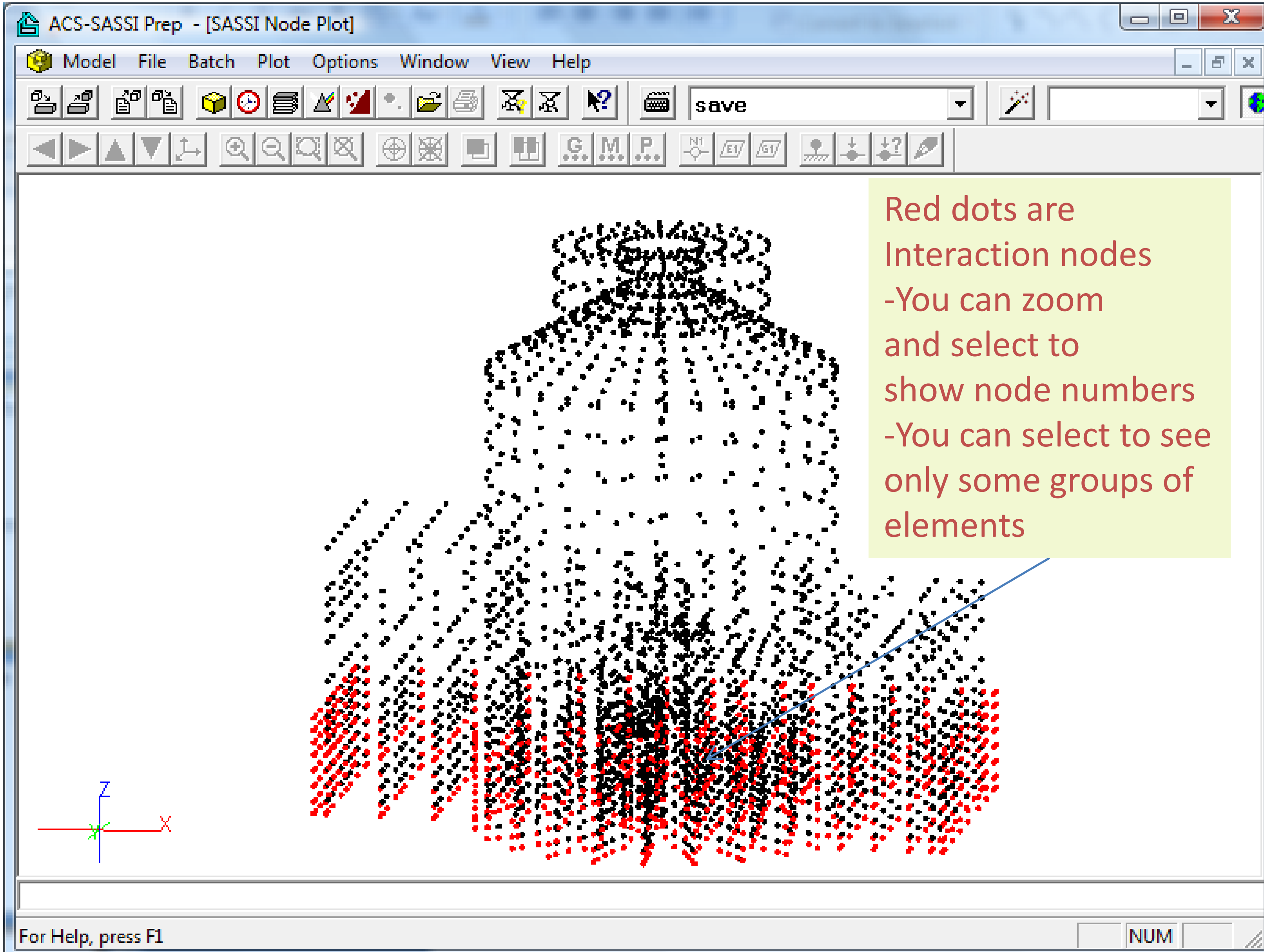
Binary Databases:

- Generate BDBs for each input direction; select flag for BDB
- Combine BDBs for three input direction using UI commands
- Use the XYZ combined BDB to extract frames at selected time steps or maximum values (text frame tables)
- UI command is designed for extracting selected time histories from BDB (Demos 3 and 13)

Text Files (.acc or .thd files):

- Combine .acc or .thd node/dof history
- *Use UI macros* (Demo 3)

Checking SSI Interaction Nodes



6. UI General Commands

General Commands

Older General Commands

Command	Action	Description
ACCIN	Sets the acceleration time history input file for ACS SASSI EQUAKE module.	Section 9.2.1
ACCOUT	Sets the acceleration time history output file for ACS SASSI EQUAKE module.	Section 9.2.2
AFWRITE	Writes the analysis files.	Section 9.2.3
AMP	Defines spectral amplification ratios for ACS SASSI HOUSE module.	Section 9.2.4
ANALYS	Defines analysis options for ACS SASSI ANALYS module.	Section 9.2.5
AOPT	Sets options for AFWRITE and CHECK.	Section 9.2.6
CHECK	Checks data.	Section 9.2.7
CORR	Sets spectra correlation values for ACS SASSI EQUAKE module.	Section 9.2.8
DAMP	Adds / resets damping ratios for RS analysis.	Section 9.2.9
DYNP	Defines data for dynamic soil properties.	Section 9.2.10
EOUT	Adds element output request for ACS SASSI STRESS module.	Section 9.2.11
EQTIT	Defines spectra title for ACS SASSI EQUAKE module.	Section 9.2.12
EQUAKE	Defines analysis options for ACS SASSI EQUAKE module.	Section 9.2.13
FORCE	Defines analysis options for ACS SASSI FORCE module.	Section 9.2.14
FREQ	Adds / deletes frequency numbers to / from frequency set.	Section 9.2.15
HOUSE	Defines analysis options for ACS SASSI HOUSE module.	Section 9.2.16
INCOH	Defines incoherence analysis options for ACS SASSI HOUSE module.	Section 9.2.17
INP	Switches input to file.	Section 9.2.18
LFREQ	Lists frequency sets.	Section 9.2.19

General Commands (cont.)

ME	Defines input motion data for ACS SASSI HOUSE module.	Section 9.2.20
MOPT	Changes the model options.	Section 9.2.21
MOTION	Defines analysis options for ACS SASSI MOTION module.	Section 9.2.22
NOUT	Adds nodal output request for ACS SASSI MOTION module.	Section 9.2.23
POINT	Defines analysis options for ACS SASSI POINT module.	Section 9.2.24
RESUME	Re-loads the active model.	Section 9.2.25
RSIN	Sets the response spectrum input file for ACS SASSI EQUAKE module.	Section 9.2.26
RSOUT	Sets the response spectrum output file for ACS SASSI EQUAKE module.	Section 9.2.27
SACC	Sets the acceleration output options for ACS SASSI SOIL module.	Section 9.2.28
SAVE	Saves active model.	Section 9.2.29
SFOU	Sets the Fourier spectrum output options for ACS SASSI SOIL module.	Section 9.2.30
SITE	Defines analysis options for ACS SASSI SITE module.	Section 9.2.31
SOIL	Defines analysis options for ACS SASSI SOIL module.	Section 9.2.32
SPRO	Defines soil profile data for ACS SASSI SOIL module.	Section 9.2.33
SRS	Sets the response spectrum output options for ACS SASSI SOIL module.	Section 9.2.34
SSAF	Sets the spectral amplification factor output options for ACS SASSI SOIL module.	Section 9.2.35
SSTR	Sets the stresses and strains output options for ACS SASSI SOIL module.	Section 9.2.36

General Commands (cont.)

STATUS	Lists general information.	Section 9.2.37
STRESS	Defines analysis options for ACS SASSI STRESS module.	Section 9.2.38
SYMM	Sets information for symmetry / anti-symmetry plane / line	Section 9.2.39
THFILE	Sets acceleration time history file.	Section 9.2.40
THTIT	Sets title for acceleration time history.	Section 9.2.41
TPSD	Use the Target PSD functionality of ACS SASSI EQUAKE	Section 9.2.42
TIT	Sets the model title.	Section 9.2.43
TOPL	Adds / deletes top layers for ACS SASSI SITE module.	Section 9.2.44
WAVE	Defines wave information for ACS SASSI SITE module.	Section 9.2.45
WPASS	Defines wave passage data for ACS SASSI HOUSE module.	Section 9.2.46
WRITE	Writes model data to an input file.	Section 9.2.47
RELD	Writes Relative Displacement Options	Section 9.2.48
RELFILE	Writes the Relative Displacement Reference File.	Section 9.2.49
RDND	Write a node to the Relative Displacement Output Node List	Section 9.2.50

General Commands (cont.)

Newer General Commands

Command	Action	Description
ACTM	Change the Active model	Section 9.6.1
AFWRBAT	Split SSI analysis set into multiple sets	Section 9.6.2
CPMODEL	copy the active model to another model	Section 9.6.3
CRITFREQ	frequencies where the interpolation and actual simulated results differ	Section 9.6.4
DMODEL	Delete a model from ACS SASSI User Interface memory	Section 9.6.5
ETYPEGEN	Modify the element types for the models	Section 9.6.6
FIXSLDROT	Fix the solid rotations in a model	Section 9.6.7
FIXSHLROT	Fix the shell rotations in a model	Section 9.6.8
FIXSPRROT	Fix the spring rotations in a model	Section 9.6.9
FIXROT	Fix the rotations in a model	Section 9.6.10
FRAMECOMBIN	Combine frame files	Section 9.6.11
FRAMESEL	Identify local max/min of time history	Section 9.6.12
GCOM	Compress group numbers	Section 9.6.13
GETENV	Display solver environment variables	Section 9.6.14
GRAVITY	Sets Gravity value for the model	Section 9.6.15
GROUNDELEV	Sets Ground Elevation for the model	Section 9.6.16
GROUPMAT	Sets a material for each group	Section 9.6.17
HVDBAT	Writes a batch run file for models with HVD elements	Section 9.6.18
INTGEN	generate interaction node if embedment is explicitly defined	Section 9.6.19
MDL	Change the Model name and path for uses with save, resume and AFWRITE	Section 9.6.20
MDLNAME	Change the Model Name without changing the path or model title	Section 9.6.21
NCOM	Compress Node Numbers in a Model	Section 9.6.22
MODELLIST	Show the models that have been defined and their names	Section 9.6.23
MODFRAMES	Modify legacy frames files for the new frame format	Section 9.6.24
RADIUS	Write a radius for non uniform soil	Section 9.6.25
RMVUNUSED	Remove Unused nodes from the model	Section 9.6.26
SETENV	Set solver environmental variables	Section 9.6.27

WARNING: *Most of the commands shown below can be alternatively defined more conveniently using the UI menu selection for the Option/Analysis. Then, the entire input file (the modelname.pre extension text file) including all the user inputs in the UI windows dialogs can be save using the WRITE command. The created .pre input file will contain all the user inputs organized in a logical sequence convenient for review and checking.*

WRITE Command

Write Model data to a Prep Input file.

WRITE,[<file>],[<path>]

creates the file named <file> in which all existent data is stored as instruction lines, so that the active model can be reloaded using the INP instruction (see section , page). If the parameter is missing, the file will have the model's name with the ".pre" extension. The default path for <file> is the model's path.

Once you have created and saved a model, the data is stored in a sequence of files in the model's directory. If you want to keep the model only for further reference (the model will not be used often) you can produce an input file using the WRITE instruction. The file <file> is an ASCII file in which data is stored in PREP instructions. When you need to reload the model, type INP,<file> and the model's files will be rewritten.

AFWRITE Command

AFWRITE

Afwrite will write the module input data files that the user requests. These files are written to the modeldirectory using the modelname with a different extension for each input file requested. Setting up an analysis can be difficult and required multiple commands or the use of the user interface to do properly.

Running Module Commands

Module Commands

Command	Action	Description
RUNANALYS	Run the ANALYS module in the ACS SASSI User Interface	Section 9.11.1
RUNCOMBIN	Run the COMBIN module in the ACS SASSI User Interface SASSI User Interface	Section 9.11.2
RUNFORCE	Run the FORCE module in the ACS SASSI User Interface	Section 9.11.4
RUNHOUSE	Run the HOUSE module in the ACS SASSI User Interface	Section 9.11.5
RUNMOTION	Run the MOTION module in the ACS SASSI User Interface	Section 9.11.6
RUNPOINT	Run the POINT module in the ACS SASSI User Interface	Section 9.11.7
RUNRELDISP	Run the RELDISP module in the ACS SASSI User Interface	Section 9.11.8
RUNSOIL	Run the SOIL module in the ACS SASSI User Interface	Section 9.11.9
RUNSTRESS	Run the STRESS module in the ACS SASSI User Interface	Section 9.11.10

SETENV Command

SETENV,<mem>

<mem> - The Memory size limit that the fast solver is allowed to use in megabytes. It is suggested that user set the size limit at 90 - 95% of physical RAM on the system. If the user attempts to allocate more than 100% of physical memory the fast solver has shown sometime to return incorrect results.

This command shall be used after software installation when the UI is launched for the first time. Sets the environment variables for the fast solver modules. The command sets three environment variables in the users registry. The environment variables are local to each user account and are persistent once they have been set. This command should be run by each user account.

GETENV Command

GETENV

This command shows the environment variables for the fast solver and the values of those environment variables.

AFWRBAT Command

AFWRBAT,<splits>
<splits> - number of sections the frequency set will be split into.

The AFWRBAT command allows the user split a simulation across multiple systems by frequency set into multiple models each with a separate frequency subset in separate folders. The folders can then be transferred to different systems and batch files created by this command run the necessary modules, then the data from each model can be combined by another batch file created by this command.

The command uses model data defined by MDL command to determine the name and location of the new folders that are created.

CRITFREQ Command

CRITFREQ,<tol>,<minfilter>,<TF>,<Var>

<tol> - percentage difference between the TFU and TFI that will cause the frequency to be added to the result.

<minfilter> - percentage below the global maximum where differences between the TFU and TFI should be ignored.

<TF> - name of the transfer function file for which the .TFI peak and .TFU values are compared.

<Var> - Variable name that includes the results of the critical frequency command application.

This command allows the user to identify automatically the frequencies where the interpolated ATF peaks are significantly different from the computed ATF values in the vicinity of the frequency of the interpolated ATF peak. These are identified frequencies should be added to the SSI input analysis. The user controls the results of this command with the <tol> and the <minfitler> argument. The full path name of the .TFU or .TFI files without these extensions should be included in the TF argument. The identified frequencies will be stored in the variable named in the last argument.

FIXROT to automatically add the needed soft rotational springs to improve numerical conditioning for detailed flat SHELL models (for the Kirckhoff plate element the drilling degree of freedom has no stiffness associated with it, and therefore could produce poorly conditioned or unstable numerical models). Not required for the TSHELL elements.

FIXROT,<Stiff>.

Example code for fixing free shell drilling rotations in a FEA model.

Actm,1
Inp, Example_Model.pre
** Add soft springs with overall stiffness 10 at the oblique SHELL nodes;*
FixRot,10

8. Batch Run Mode Commands for SSI Modules and Auxiliary Programs

Batch SSI Module Runs

If the SSI runs are done in the batch mode under a DOS window, then, a batch file needs to be created. To run a SSI module in batch mode, the following DOS command is required:

SSI_module_name.exe < SSI_module_name.inp

where SSI_module_name could be SITE, or POINT or ANALYS. The SSI module executables are installed by default in the ACS_C directory on the hard drive, and are also provided on the ACS SASSI installation DVD in the Batch. Each input file with the SSI_module_name and the extension .inp contains only three input lines:

modelname
modelname.ext_input
modelname_SSI_module_name.out

where ext_input is the extension provided by the ACS SASSI UI AFWRITE command.

For complex batch run file scripts, see Demo 16, 17 or 18.

Batch File for Nonlinear Soil SSI Analysis for X-Dir

```
REM Embedbment Example Batch Mode
REM This batch file is used to run the SSI modules for the embedded example.
@echo off

REM  FIRST ITERATION
C:\acsv300\EXEB\Soilb.exe < Soil.in            REM  Runs the soil module with inputs defined in the Soil.in file
C:\acsv300\EXEB\Siteb.exe < Site.in            REM  Runs the site module with inputs defined in the Site.in file
C:\acsv300\EXEB\Point3b.exe < Point.in         REM  Runs the point module with inputs defined in the Point.in file
C:\acsv300\EXEB\Houseb.exe < House.in          REM  Runs the house module with inputs defined in the House.in file
C:\acsv300\EXEB\Analysb.exe < Analys.in        REM  Runs the analys module with inputs defined in the Analys.in file
C:\acsv300\EXEB\Stressb.exe < Stress.in        REM  Runs the stress module with inputs defined in the Stress.in file

REM SECOND ITERATION
C:\acsv300\EXEB\Houseb.exe < House.in          REM  Runs the house module with inputs defined in the House.in file
C:\acsv300\EXEB\Analysb.exe < Analys.in        REM  Runs the analys module with inputs defined in the Analys.in file
C:\acsv300\EXEB\Stressb.exe < Stress.in        REM  Runs the stress module with inputs defined in the Stress.in file

REM THIRD ITERATION
C:\acsv300\EXEB\Houseb.exe < House.in          REM  Runs the house module with inputs defined in the House.in file
C:\acsv300\EXEB\Analysb.exe < Analys.in        REM  Runs the analys module with inputs defined in the Analys.in file
C:\acsv300\EXEB\Stressb.exe < Stress.in        REM  Runs the stress module with inputs defined in the Stress.in file

REM POST PROCESSING
C:\acsv300\EXEB\Motionb.exe < Motion.in                     REM Runs the motion module with inputs defined in the Motion.in file
C:\acsv300\EXEB\Stressb.exe < Stress_Post.in   REM  Runs the stress module for post processing with inputs defined in the Stress_P
C:\acsv300\EXEB\Reldispb.exe < ReldispX.in     REM  Runs the reldisp module for x direction with inputs defined in the ReldispX.in
C:\acsv300\EXEB\Reldispb.exe < ReldispY.in     REM  Runs the reldisp module for y direction with inputs defined in the ReldispY.in
C:\acsv300\EXEB\Reldispb.exe < ReldispZ.in     REM  Runs the reldisp module for z direction with inputs defined in the ReldispZ.in
```

Explain .in files

```
Demo4
Demo4.soi
Demo4_Soil.out

Demo4
Demo4.sit
Demo4_Site.out

Demo4
Demo4.poi
Demo4_Point.out

Demo4
Demo4.hou
Demo4_House.out

Demo4
Demo4.anl
Demo4_Analys.out

Demo4
Demo4.mot
Demo4_Motion.out

Demo4
Demo4.str
Demo4_Stress.out
```

ACS SASSI V4.3 Includes 6 Auxiliary Programs

BuildFile77_Embedded_Incoherency – Incoherent SSI for DES

COMB_XYZ_ACC_for_RS – Combine X, Y and Z acceleration histories

COMB_XYZ_THD_Option_NON – Combine X, Y, Z nodal displacements

COMB_XYZ_STRAIN_Soil_Nonlinear – Combine X, Y, Z soil shear strains

File8_add_cp_resp_Option_AA – Adjust FILE8 after ANSYS runs w/ CP

Remove_Frequencies_from_FILE8 – Remove unstable solutions

Auxiliary Programs Used Only for Specific Tasks

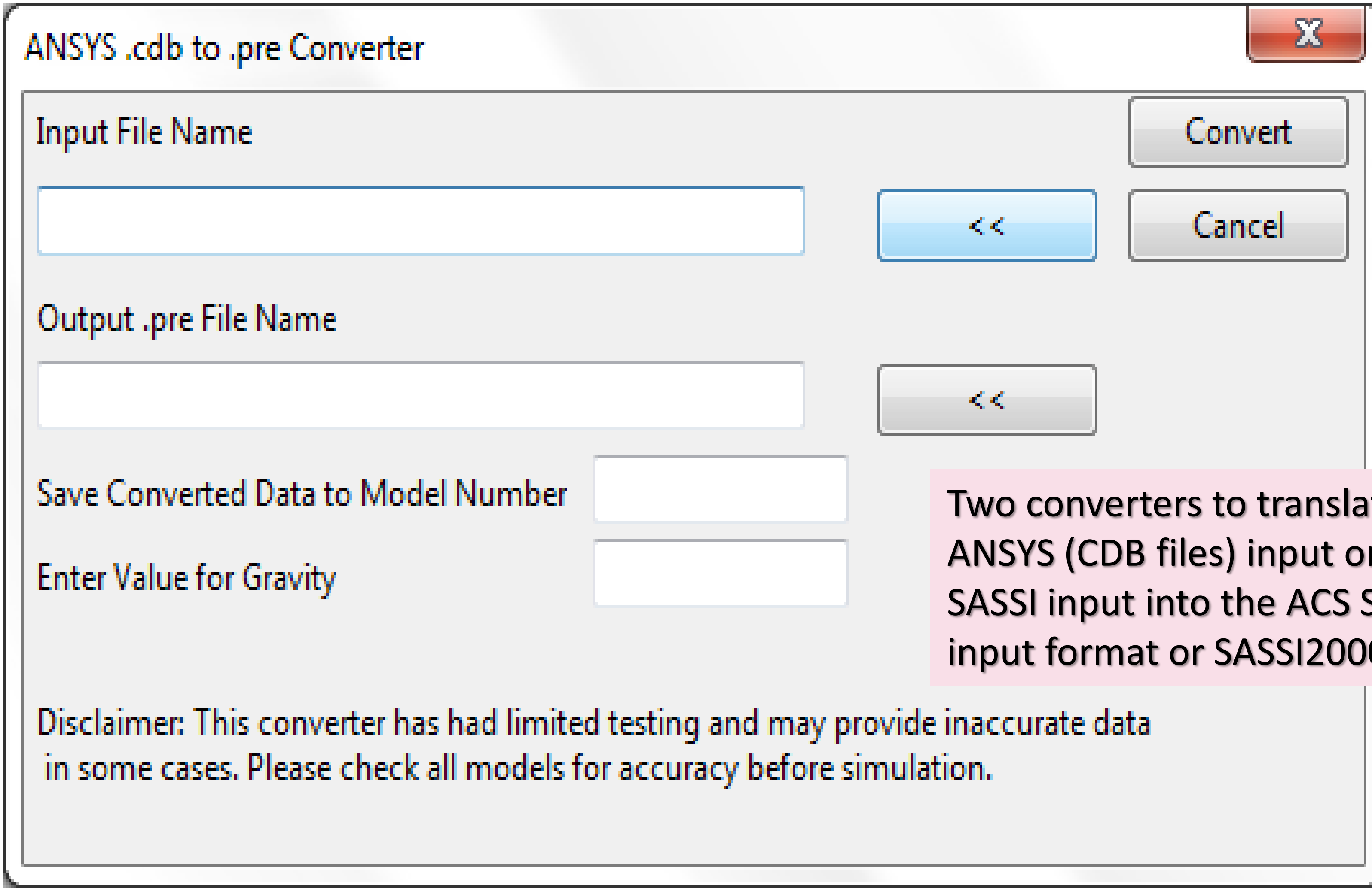
COMB_XYZ_STRAIN applicable to 3D nonlinear soil site response or SSI analysis using iterative equivalent linear procedure. Combines three directional shear-strains for X, Y and Z inputs in the nonlinear soil solid elements at each SSI iteration.

COMB_XYZ_THD applicable to nonlinear structure SSI analysis using iterative equivalent linear procedure (*for Option NON*). Combines directional node displacements for nonlinear shell walls and spring elements at each SSI iteration.

BuildFile77 applicable to incoherent SSI analysis of deeply embedded structures, such as SMRs. Permits performing incoherent SSI analysis for basements with different level node coordinates & configurations or when incoherent input has different coherence functions at different depths.
NOTE: Still needs extra checks on incoherent mode signs

7. UI Commands for Generating and Checking FE Models

Building ACS SASSI Model Using UI Converters



Two converters to translate ANSYS (CDB files) input or SASSI input into the ACS SASSI input format or SASSI2000.

ANSYS Model Converters To and From ACS SASSI

File Conversion Commands

Command	Action	Description
ANSYS	Write model to ANSYS® input format	Section 9.10.1
ANSYSMODELTYPE	Change the model type for the Advanced ANSYS® option	Section 9.10.2
ANSYSREFORMAT	Reformat the beam groups to work better for ANSYS® write	Section 9.10.3
CONVERT	Use one of the file converters to translate a file	Section 9.10.4

Model Generation & Combination Commands

Model Generation & Combination Commands

Command	Action	Description
CONDMAP	Create node mapping file for impedance condensation	Section 9.8.1
EXCAV	Create an Excavation volume for a model	Section 9.8.2
MERGE	Merge two models	Section 9.8.3
MERGEGROUP	Merge two groups together	Section 9.8.4
MERGEPANEL	Merge a Panel Model to the Solids and beams of the original model	Section 9.8.5
MERGESOIL	Merge a Structural Model With a Matching Soil	Section 9.8.6
ROTATE	Rotate the model around a point	Section 9.8.7
SOILCONTACT	Generate contact elements for Option A	Section 9.8.8
SOILMESH	Create a soil mesh for the active model	Section 9.8.9
TRANSLATE	Translate all nodes a specified distance	Section 9.8.10
WELD	Combine nodes that share the same location	Section 9.8.11

EXCAV command creates an excavation model for a structural model that doesn't have an excavation

Example code to create an excavation model for a structural model (.pre).

```
Actm,1
INP, Example_model.pre
EXCAV,2
ACTM,2
* Write .pre file for the excavation model 2
Write, Example_Excavation.pre
```

MERGESOIL,

<Struct>,<Soil>,[Mode],[StiffStiff],[StiffSoft],[SepLevel],[Mapping]

This command is used to merge the structural and the excavation volume models together in a new
active SSI FEA model.

<Struct> - Model Number of the Structure

<Soil> - Model Number of the Excavation volume

[Mode] - Merging nodes on the structure excavation interface

= 0 Unbonded lateral foundation-soil interface with side solid

= 1 Bonded lateral foundation-soil interface (default)

= 2 Bonded foundation-soil interface using duplicate nodes connected by stiff springs

= 3 Unbonded foundation soil-interface using duplicate nodes connected by soft springs

[StiffStiff] - Stiff spring stiffness for Modes 2 and 4. (Default = 10^7)

[StiffSoft] - Soft spring stiffness for Modes 3. (Default = 10)

[SepLevel] - Global z-coordinate level for depth where soil separation occurs

[Mapping] - This is mapping filename for the duplicate node merging

INTGEN to generate automatically interaction nodes for different substructuring approaches FV, FI-FSIN (SM), FI-EVBN (MSM) and Fast FV.

INTGEN,<type>,<skip> to generate the interaction nodes based on the selected SSI substructuringapproach. The excavation volume must be explicitly defined by the ETYPE command for options 1-3.
If the ETYPE is left to default values, this command will not work.

<type> :Type of iteration node generation
= 1 for Embedded Foundation - Flexible Volume (FV)
= 2 for Embedded Foundation - Flexible Interface with Excavation Volume Boundary Nodes, denoted FI-EVBN or Modified Subtraction Method (MSM)
= 3 for Embedded Foundation - Flexible Interface with Foundation-Soil Interface Nodes, denoted FI-FSIN or Subtraction Method (SM)
= 4 for Surface Foundation (interaction nodes are only at ground surface)
= 5 for FFV with multiple internal interaction node layers using <skip level>
= 6 add Z-level interaction nodes using <skip level>

CONDMAP Command (for FVROM approach)

CONDMAP,<excav>

This command builds a mapping file between interaction nodes and coincidental structural nodes. The command finds all of the interaction nodes in the excavation model. Then the command will find the first coincidental structural node found in the structural model.

The structural model is considered to be the current active model. The excavation model must have the interaction nodes defined before using this command.

The file written will have the file name "INT_NODES_IF" and will be found in the active model directory. Users should have defined the active model directory by using the MDL command. The first line of the file is number of interaction nodes in the model. Then each line subsequent line will have the interaction node and structural node separated by a space.

- excav - excavation model number.

Building SSI Model by Merging Structure and Excavation Models (as used in Option AA)

It is assumed that the ground surface is at Z=0. and the FV method will be used

**Convert ANSYS Structure.cdb in Model 1*

Actm,1

Convert,ansys,struct.cdb,32.2

Etypegen,1

Actm,2

Convert, ansys,Soil.cdb,32.2

** Define excavation elements of type 2*

Etypegen,2

** Create SSI model by combining Models 1 and 2 in Model 3*

Actm,3

MergeSoil,1,2,1,,,,mappingfile.txt

Groundelev, 0

Intgen, 1

SOILCONTACT Command

SOILCONTACT

This command will add two groups of planes to the active model. The first group of plane elements will share nodes with models interface with the excavation volume. The second group of plane elements will be built with coincident nodes to the first group of quadrilateral elements and all the nodes of the second group are created by this command. All of the new nodes will be added to the end of the node list of the active model.

A map of the coincident node of the 2 Planes will be written to an ASCII format file modelname_Contactmap.txt.

This command is intended to be used to create contact/target surface elements when creating an ANSYS® model with contact surfaces between foundation and surrounding soil for Option A.

SSI Model Checking UI Commands

Model Checking Commands

Command	Action	Description
EXCSTRCHK	Checks excavation nodes for potential errors	Section 9.7.1
FIXEDINT	Find Fixed Interaction Nodes	Section 9.7.2
FREESPRING	Find Free Spring Nodes	Section 9.7.3
HINGED	Check model for possible hinged connections	Section 9.7.4
INTCOUNT	Display a count of interaction nodes	Section 9.7.5
KINT	Find K node that are interaction nodes	Section 9.7.6
USED	Check and fix Unused nodes	Section 9.7.7

EXCSTRCHK Commands

EXCSTRCHK

This command checks if the Excavation interior nodes are common with the Structure basement nodes. This will be incorrect from a SASSI modeling point of view. A list detailing the shared nodes will be printed to the command history. The number of entries in the list is controlled by the Check Options break message number.

This command does not change the active model in any way.

HINGED checks model to find all hinged connections between solids and shell and beams and beams and shells. Write warnings for hinged nodes.

These hinged connections could be potentially indicate incorrect FE modelling, since the node rotations from beams and shells are not transmitted to solids at the common nodes, and the node rotations from beams are not transmitted the in-plane shell rotations at the common nodes (the drilling dof equations have no stiffness terms by default)

FIXEDINT *checks if there are interaction nodes that are fixed by mistake*

Nonlinear Modeling Commands (for Option NON and MS)

Nonlinear & Panel Commands

Command	Action	Description
B	Nonlinear Beam Definition	Section 9.16.1
BBC	Add a backbone curve from a file	Section 9.16.2
BBCGEN	Generate BBC curves for defined panels	Section 9.16.3
BBCI	Sets BBC information	Section 9.16.4
BBCP	Defines single point on BBC	Section 9.16.5
BBCX	Backbone curve definition command	Section 9.16.6
BBCY	Backbone curve definition command	Section 9.16.7
BEAMPILE	Separates piles from model, and reconnects them with a spring at pile interface	Section 9.16.8
DCOUPLEBEAM	Add Springs at beam intersections	Section 9.16.9
DELBBC	Delete Backbone Curves	Section 9.16.10
DELBM	Delete Nonlinear Beam	Section 9.16.11
DELNLS	Delete Nonlinear Soil Layers	Section 9.16.12
DELSPR	Delete Nonlinear Spring	Section 9.16.13
DGRDFLR	Modify Young's modulus of floor panel	Section 9.16.14
EDGE	Split panel based on edges	Section 9.16.15
EDGEMODEL	Applies the EDGE command to all of the wall groups	Section 9.16.16
EQL	Nonlinear analysis options	Section 9.16.17
MERGEPANEL	Merge a Panel Model to the Solids and beams of the original model	Section 9.16.18
NLSLAYER	Add a nonlinear soil layer definition	Section 9.16.19
NLSOIL	Set Parameters for the nonlinear soil option	Section 9.16.20
NONLINBAT	Create a Generic Nonlinear Batch Run	Section 9.16.21
NONLINMOTDISP	Add panel corner nodes to output request list	Section 9.16.22

Nonlinear Modeling Commands (for Option NON and MS)

P	Add a panel to the active model	Section 9.16.23
PANELIZE	Separate Shells in the model into Panels	Section 9.16.24
PDEL	Delete panel(s) from the active model	Section 9.16.25
PLIST	List or check panel(s) in the active model	Section 9.16.26
PNLGEN	Create panel definitions for shell groups	Section 9.16.27
S	Nonlinear Spring definition command	Section 9.16.28
SOILREDEF	Redefine excavation volume layers based on a 2D soil model	Section 9.16.29
SOLIDPILE	Separates piles from model, and reconnects them with a spring at pile interface	Section 9.16.30
UNIPNL	Create unique group for each element of a group	Section 9.16.31
WALLFLR	Separate shells walls and floors into separate groups	Section 9.16.32

9. Post-Processing of SSI Responses Using UI Plotting Commands and Binary Databases

Post-Processing Plotting Commands

Plot Commands

Command	Action	Description
ADDITION	Add line together and store it	Section 9.13.1
AVERAGE	Calculate the average of a group of line and store in a line	Section 9.13.2
AXES	Change the axes in a 2d model plot	Section 9.13.3
BROADEN	Broaden Selected line and stave results to selected line	Section 9.13.4
BUBBLEPLOT	Plot Bubble (ZPA) for the model	Section 9.13.5
CAPTUREPLOT	Screen capture of the current plot	Section 9.13.6
CLOSEPLOT	Close the current plot	Section 9.13.7
CNGCENTER	Change the center of rotation for the 3d model plot	Section 9.13.8
CNGVIEW	Change the view of the current 3d model plot	Section 9.13.9
COLOR	Change the color in the color list	Section 9.13.10
CONTOURPLOT	Plot a 3d Contour plot	Section 9.13.11
CUTPLOT	Plot the 3d model with what elements are in the cut	Section 9.13.12
DEBUG	Display 3d debug text to the screen	Section 9.13.13
DEFORMPLOT	plot a 3d deformed shape plot	Section 9.13.14
ELECOLOR	Change element property coloration	Section 9.13.15
ELENUM	Show/Hide Element numbers on active plot	Section 9.13.16
GROUPNUM	Show/Hide Group numbers on active plot	Section 9.13.17

Post-Processing Commands

LBINCORS	Calculate the Lower Bound Incoherent Response Spectra	Section 9.13.18
LAYERPLOT	Plot a 2d cross section of the models soil layers	Section 9.13.19
LINECOMBIN	Linearly combine a set of lines with scalars	Section 9.13.20
LINENAME	Change the name of a line in memory	Section 9.13.21
MARKERS	Add markers to a line	Section 9.13.22
MODELPLOT	Plot a 3d model plot	Section 9.13.23
NODENUM	Show/Hide Node Number on active plot	Section 9.13.24
NODEPLOT	Plot a 3d node plot	Section 9.13.25
NODESEL	Select a node on a 3d plot	Section 9.13.26
PAUSE	Start or stop and animation	Section 9.13.27
PLOTRANGE	Change the range of the axes on a 2d plot	Section 9.13.28
PLOTTITLE	Change the title of a plot	Section 9.13.29
PROCFRAME	Process the frame animation files	Section 9.13.30
READSPEC	Read a spectrum file	Section 9.13.31
READTH	Read a File in the Time history format	Section 9.13.32
RSTCENTER	Reset the Center to Default	Section 9.13.33
RSTVIEW	Reset the View to Default	Section 9.13.34
SHADEROPTIONS	Change the shader options for the current plot	Section 9.13.35
SHOWDOF	Show the fixed degrees of freedom in some 3D plots	Section 9.13.36

Post-Processing Plotting Commands

SHOWMASS	Show the lumped masses in some 3D plots	Section 9.13.37
SHRINK	Shrink the elements in the Element Plot	Section 9.13.38
SOILPROPPLOT	Create a 2d soil property plot	Section 9.13.39
SPECPLOT	Plot a spectrum plot	Section 9.13.40
SRSS	Combine lines using srss	Section 9.13.41
STIPPLE	Stipple line in a plot for Black and white printing	Section 9.13.42
SUBTRACTION	Subtract lines and store into another line	Section 9.13.43
THPLOT	Create a 2d time history plot	Section 9.13.44
VECTORPLOT	Plot a 3d vector plot with the current model	Section 9.13.45
WINDOWSETTINGS	Change the Windows Setting for the current plot	Section 9.13.46
WIREFRAME	Show the wireframe of the Element Plot	Section 9.13.47
WRITESPEC	Write a line in the spectrum file format	Section 9.13.48
WRITETH	Write a line in the time history file format	Section 9.13.49
XTITLE	Change the X axis title for a 2D graph	Section 9.13.50
YTITLE	Change the left side Y axis title for a 2D graph	Section 9.13.51
YTITLE2	Change the right side Y axis title for a 2D graph	Section 9.13.52

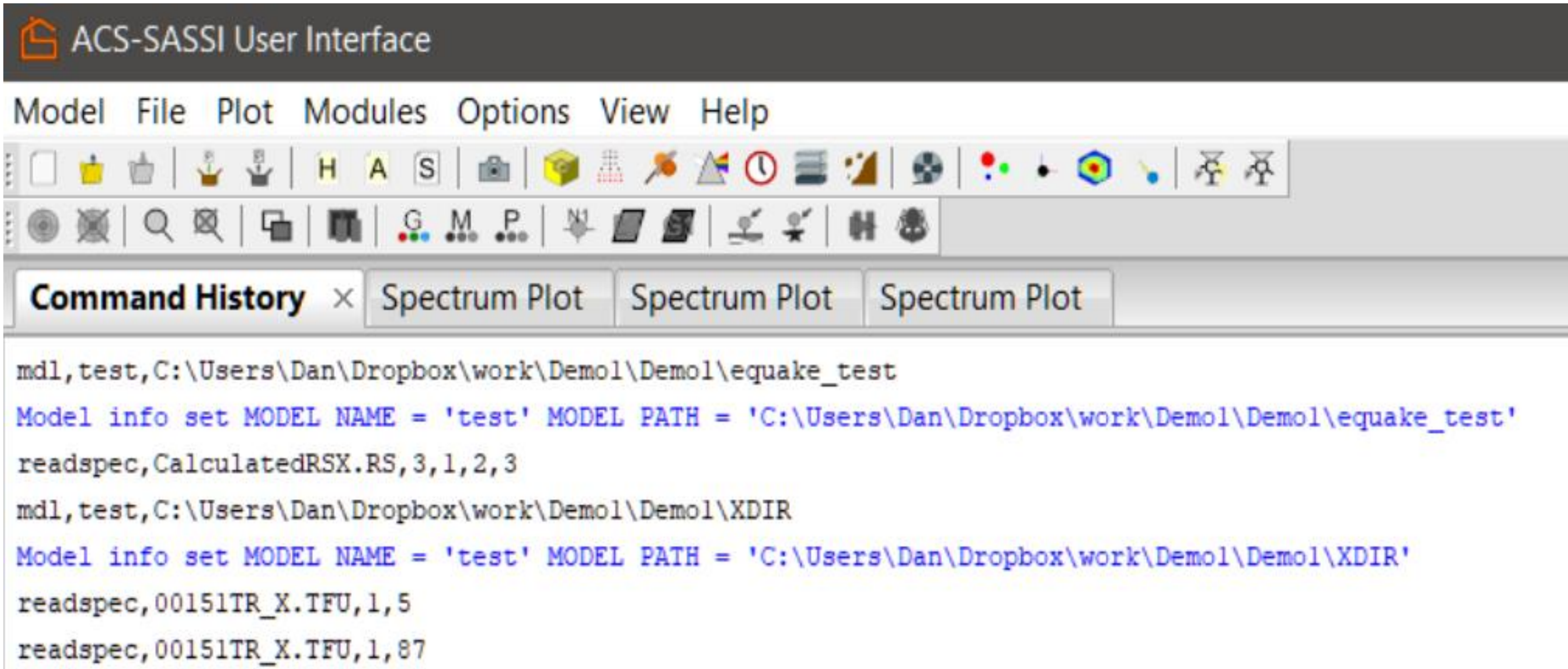
READSPEC Command

READSPEC,<SpecFile>,<numLines>,<Line1> … <LineN>

Reads a spectrum file into a line object for plotting.
- SpecFile – Full path of the spectrum file to be read
- numLines – number of data columns to be read from the file
- Line - Line Reference Number for each line object to be read

NOTE: Line objects in ACS SASSI are read from the columns in a data file. For TFI, TFI, TFD, and RS files, the frequency column should not be counted in the <numLines> argument.
For example, an RS file has a column for frequency and a column for acceleration. In this case, <numLines> is 1 because there is only one column of data to be read into a line object for plotting.

UI Commands for Post-Processing and Plotting of SSI Result Text Files – Example for RS and ATF



Post-Processing Commands Using Binary Databases

Binary Database Commands

Command	Action	Description
ACCDBANI	Creates an animation from the acceleration binary database in memory	Section 9.17.1
ACCDBTHFILE	Write ASCII time history files from the binary ACC database	Section 9.17.2
BINFRAMEOUT	Write frame files from a database	Section 9.17.3
BINOUT	output flags for binary files	Section 9.17.4
BINSTRTBL	Output selected stresses from a group in an ACSII CSV format	Section 9.17.5
COMBACCDB	Combine three databases acc binary format	Section 9.17.6
COMBDISPDB	Combine three databases int the disp/acc binary format	Section 9.17.7
COMBDISPDIR	Combine three displacement component binary files into a complete displacement database	Section 9.17.8
COMBTHSDB	combine stress binary database from file	Section 9.17.9
DELDB	Delete a database from User Interface Memory	Section 9.17.10
DISPDBANI	Creates a animation from the displacement binary database in memory	Section 9.17.11
DISPDBTHFILE	Write ASCII time history files from the binary Displacement database	Section 9.17.12
LOADACCDB	Load a acceleration binary database from file	Section 9.17.13
LOADDISPDB	Load a displacement binary database from file	Section 9.17.14
LOADTHSDB	Load a stress binary database from file	Section 9.17.15
MAXDBFRAME	create a frame with the global maximum for all components	Section 9.17.16
RELBIN	Sets options for using binary TFI files with RELDSIP	Section 9.17.17
THSDBANI	Creates a animation from the stress binary database in memory	Section 9.17.18
THSDBTHFILE	Write ASCII time history files from the binary stress database	Section 9.17.19

BINOUT Command

BINOUT,[mot],[str],[reldisp]

BINOUT command sets the binary options for the inputs of the SSI modules such as MOTION, RELDISP and STRESS. If an argument is left blank the value of the associated flag remains unchanged

- MOTION nodal acceleration history binary database
  - 0 - Do not write database
  - 1 - Write binary database
- STRESS the element stress/force/moment history binary database
  - 0 - Do not write database
  - 1 - Write binary database
- RELDISP nodal displacement history binary database
  - 0 - Do not write database
  - 1 - Not used in this version
  - 2 - Write binary database

COMBACCDB Command

COMBACCDB,<Xfile>,<Yfile>,<Zfile>,<Comb>

Combine *three acceleration history binary databases* for X, Y and Z seismic input into a single database.

•Xfile - full path name of the x direction binary database
•Yfile - full path name of the y direction binary database
•Zfile - full path name of the z direction binary database
•Comb - full path name of combined direction binary database

COMBDISPDB Command

three displacement history binary databases

COMBTHSDB Command

Three element stress history binary databases

LOADACCDB Command

LOADACCDB,<file>

Load the MOTION created acceleration history binary database into the UI memory. Only a single acceleration binary database can be loaded into the UI memory at a time

•file - full path name of binary acceleration database

LOADDISPDB Command

Load the RELDISP created displacement history binary database

LOADTHSDB Command

Load the STRESS created element stress history binary database

Post-Processing SSI Responses Using Binary Databases Described in Detail in Demo 13

*** Define Variable for Working Directory**
VAR,PATH,C:\ACSV300\Demo_Problems\Demo13

*** Combining SSI Responses Computed for X, Y and Z Inputs Using Binary Databases for MOTION, RELDISP and STRESS**
COMBACCDB,@PATH[1]\XDIR\ABShear_ACC.bin,@PATH[1]\YDIR\ABShear_ACC.bin,@PATH[1]\ZDIR\ABShear_ACC.bin,@PATH[1]\Combined\ABShear_ACC.bin,0
COMBDISPDB,@PATH[1]\XDIR\ABShear_thd.bin,@PATH[1]\YDIR\ABShear_thd.bin,@PATH[1]\ZDIR\ABShear_thd.bin,@PATH[1]\Combined\ABShear_thd.bin,0
COMBTHSDB,@PATH[1]\XDIR\ABShear_STRESS.bin,@PATH[1]\YDIR\ABShear_STRESS.bin,@PATH[1]\ZDIR\ABShear_STRESS.bin,@PATH[1]\Combined\ABShear_STRESS.bin,0

*** Loading Binary Databases into the ACS SASSI UI for Fast SSI Response History Post-Processing**
LOADACCDB,@PATH[1]\Combined\ABShear_ACC.bin
LOADDISPDB,@PATH[1]\Combined\ABShear_THD.bin
LOADTHSDB,@PATH[1]\Combined\ABShear_STRESS.bin

ACCDBANI Command

ACCDBANI,<dir>,[label]

Create the SSI model animation from the nodal acceleration binary database that is loaded in the UI memory

•dir – work directory for the animation files.

•label – description label of the animation data. This label is stored in the animation database under Process Animation, and is used to identify the selected animation file when the animation data is loaded/reloaded

DISPDBANI Command

Create the SSI model animation from the nodal displacement binary database

THSDBANI Command

Create the SSI model animation from the element stress binary database

MAXDBFRAME Command

MAXDBFRAME,<Type>,[dir]

This command will find the maximum component values of for a binary database loaded into UI memory. The data will be written in a binary database file located in either the working directory or an user defined directory.

- Type - Database used to make the frame
    - THS - Stress
    - DISP - Displacement
    - ACC - Acceleration
- dir - directory where the frame file will be written. (default: current working directory)

BINSTRTBL Command

BINSTRTBL,<group>,<EVar>,<file>,[step]

Create a text table format for selected stresses.

Gr.	Elem.	SXX	SYY	TXY	MXX	MYY	MXY
34	1	0.0619	1.3769	1.7051	0.2896	0.2343	0.3228
34	2	0.0092	1.6048	2.6930	0.2101	0.8928	0.1681
34	3	0.1953	1.0696	2.6461	0.5233	0.8652	0.1771
34	4	0.1861	0.7625	2.1855	0.2927	1.1031	0.1119
34	5	0.0589	0.8005	4.2785	0.5619	1.8940	0.0472
34	6	0.0849	0.7012	4.9323	0.8285	1.8190	0.1373
34	7	0.0262	0.6604	3.4275	0.6116	0.2215	0.4655
34	8	0.0329	0.4823	1.7195	0.2640	1.7506	0.5377
34	9	0.0498	0.7906	1.7139	0.7083	1.3001	0.2326
34	10	0.0864	0.5886	2.4700	0.5667	0.5384	0.1734

The 1st line of the file will be column labels for each column. Each subsequent line will be the Group and element number then each component stress for that element.

If the step argument is left blank or set to -1 the Maximum values of each component will be written to the table. This Maximum is the absolute maximum for the component.

BINFRAMEOUT Command

BINFRAMEOUT db>,<frame>,<TS>,[Split],<dir>

Create ASCII frame files from binary databases in the UI frame format. This command will write a frame(s) for a single database. The user can request the frame by either integer order in the database or by time value.

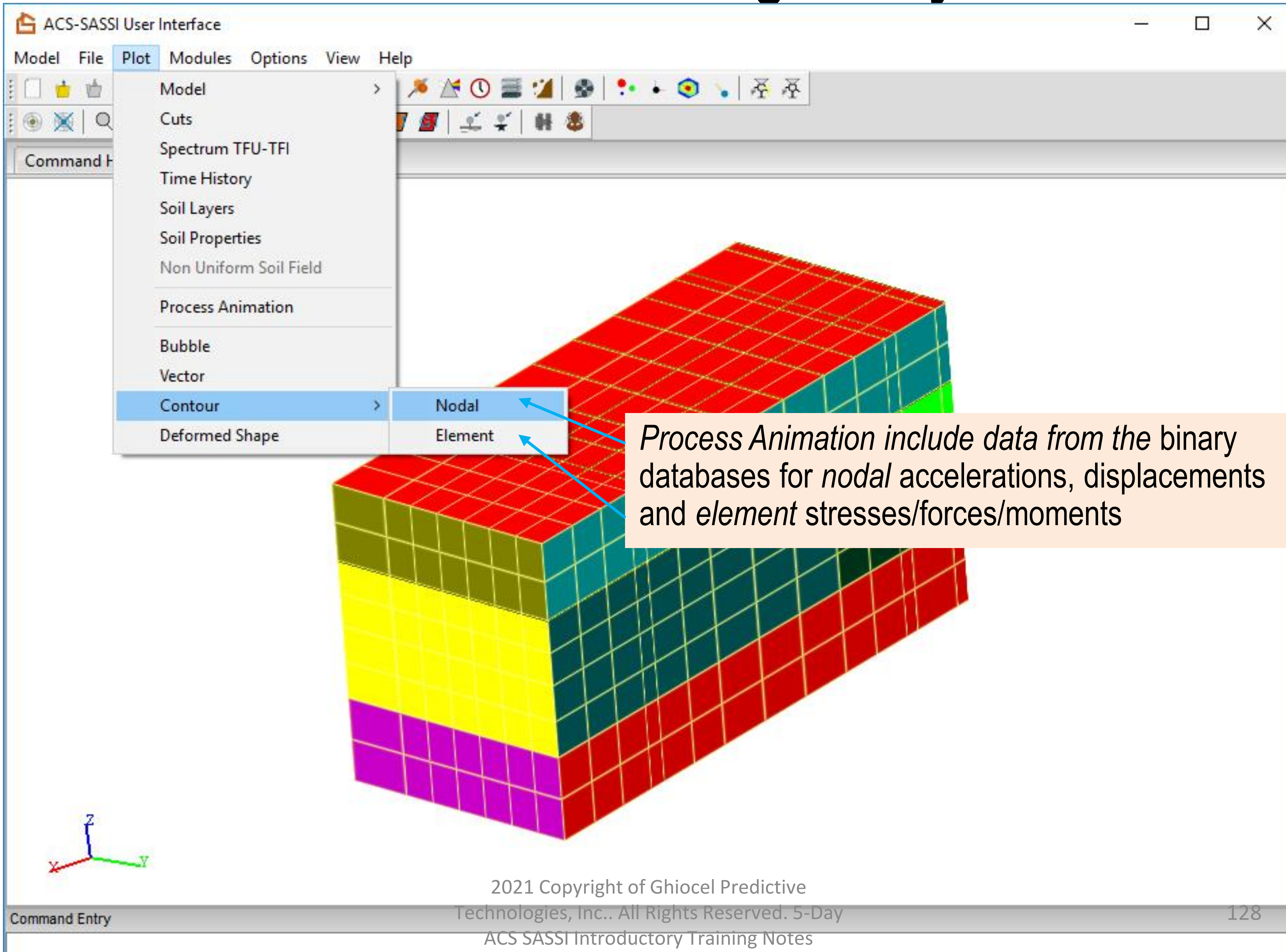
- •db - database used fro frame creation
  - •THS - Stress Database
  - •DISP - Displacement Database
  - •ACC - Acceleration Database
- •frame - frame number or frame time. If using the command to create a frame of the maximum components use -1 for frame num and TS < = 0
- •TS - time step for the database if using frame time
  - •if TS < = 0 use integer frame number
  - •if TS > 0 use frame time
- •Split - split the frame file into to separate files
  - •0 - no frame split
  - •1 - frame split – separate translation and rotations
- •dir - directory where the frame file will be written

UI Extracting STRESS Maximum Element Outputs

```
LOADTHSDB,C:\ACSV300\DEMO_PROBLEMS\DEMO9\XYZ_Analysis\STRESS\Combined\THS\AB_SHEAR_NL_STRESS.BIN
Database read took 3.906000 Seconds
LOADVAR,elist,C:\ACSV300\Demo_Problems\Demo9\XYZ_Analysis\element_output_list.txt
Variable Loaded Sucessfully from file
BINSTRTBL,34,elist,1,C:\ACSV300\Demo_Problems\Demo9\XYZ_Analysis\STRESS_MAX.txt
Table File : C:\ACSV300\Demo_Problems\Demo9\XYZ_Analysis\STRESS_MAX.txt  written
```

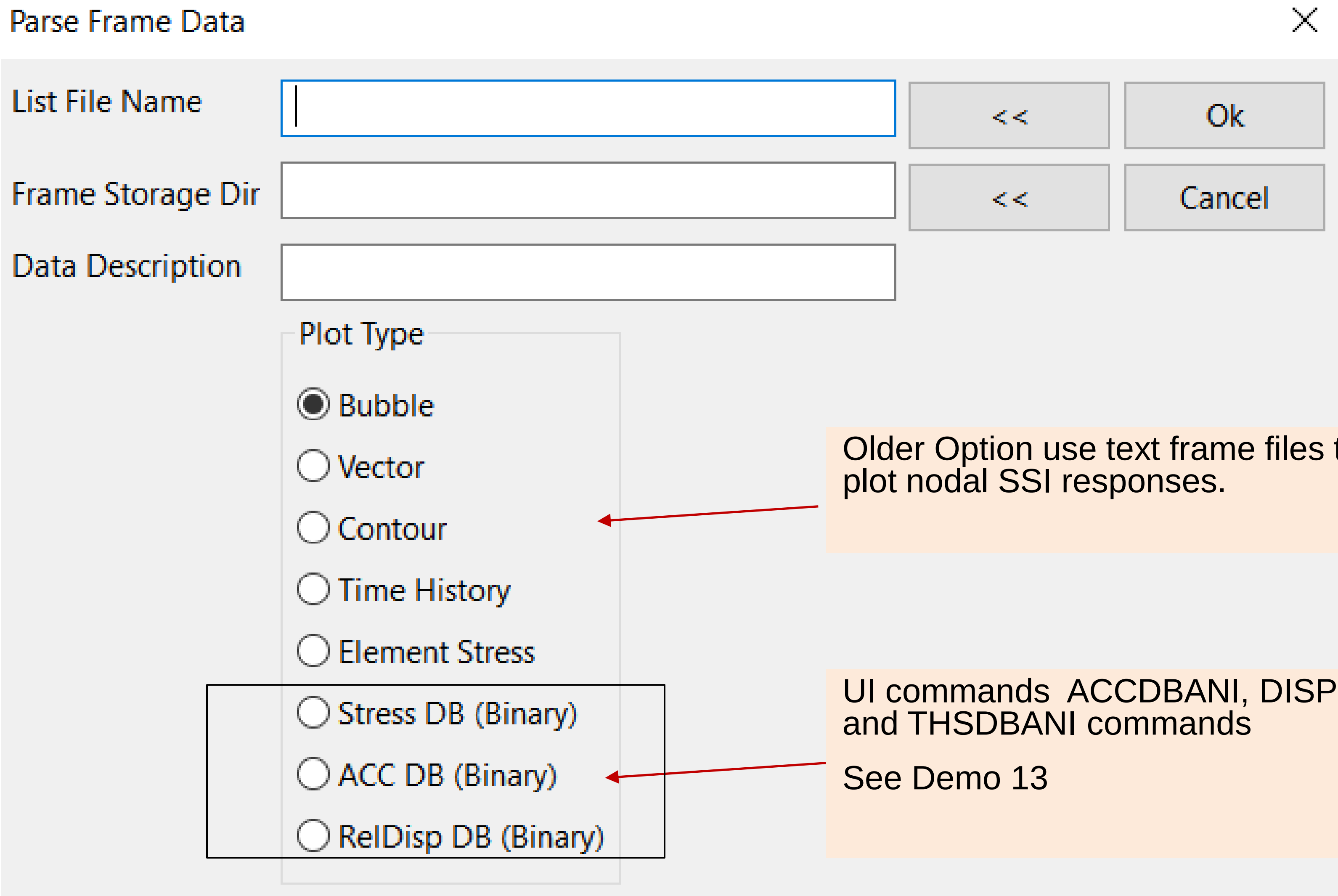
Gr.	Elem.	SXX	SYY	TXY	MXX	MYY	MXY
34	1	0.0619	1.3769	1.7051	0.2896	0.2343	0.3228
34	2	0.0092	1.6048	2.6930	0.2101	0.8928	0.1681
34	3	0.1953	1.0696	2.6461	0.5233	0.8652	0.1771
34	4	0.1861	0.7625	2.1855	0.2927	1.1031	0.1119
34	5	0.0589	0.8005	4.2785	0.5619	1.8940	0.0472
34	6	0.0849	0.7012	4.9323	0.8285	1.8190	0.1373
34	7	0.0262	0.6604	3.4275	0.6116	0.2215	0.4655
34	8	0.0329	0.4823	1.7195	0.2640	1.7506	0.5377
34	9	0.0498	0.7906	1.7139	0.7083	1.3001	0.2326
34	10	0.0864	0.5886	2.4700	0.5667	0.5384	0.1734

UI Plot Nodal Contours Using Binary Databases



Process Animation include data from the binary databases for *nodal* accelerations, displacements and *element* stresses/forces/moments

UI Processing for Structural Animations Using Text Frames or Binary Databases



Older Option use text frame files to plot nodal SSI responses.

UI commands ACCDBANI, DISPDBANI and THSDBANI commands

See Demo 13

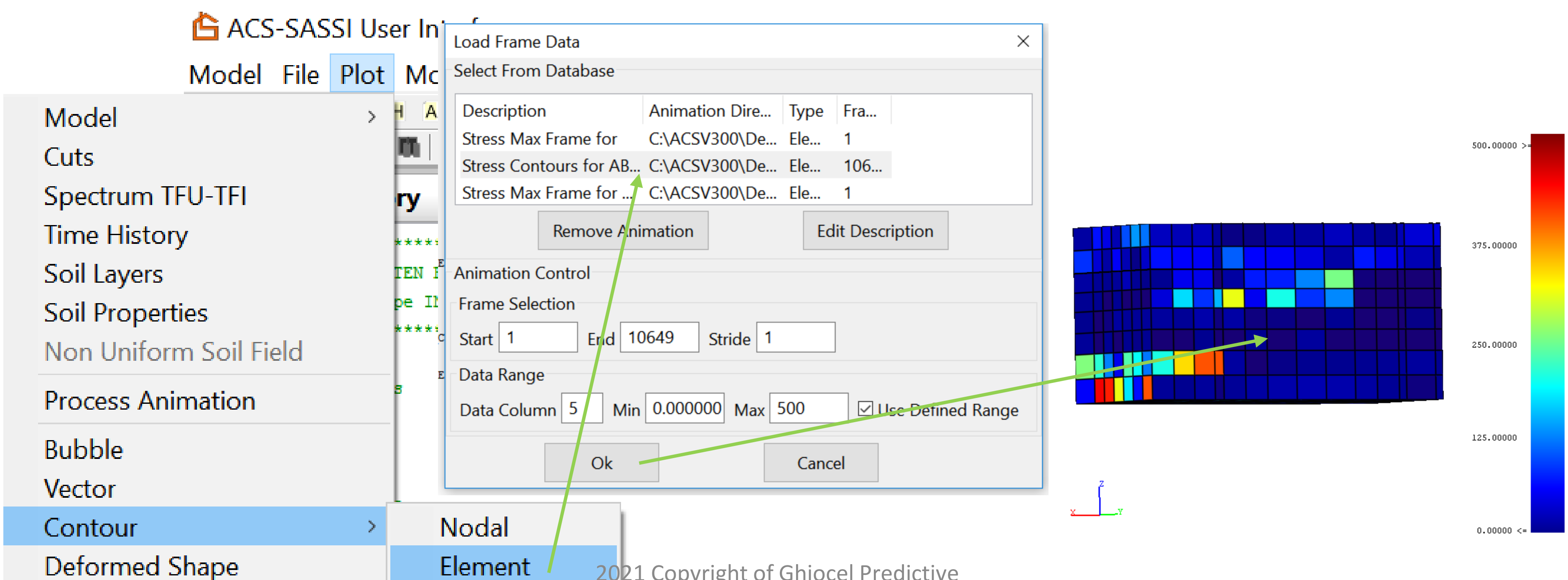
Plotting STRESS History of Max Element Outputs

```
INPUT FILE REACHED EOF, INPUT SWITCHED TO KEYBOARD
LOADTHSDB,C:\ACSV300\DEMO_PROBLEMS\DEMO9\XYZ_Analysis\STRESS\Combined\THS\AB_SHEAR_NL_STRESS.BIN
Database read took 7.110000 Seconds
LOADVAR,elist,C:\ACSV300\Demo_Problems\Demo9\XYZ_Analysis\element_output_list.txt
Variable Loaded Sucessfully from file
MAXDBFRAME,THS,C:\ACSV300\Demo_Problems\Demo9\XYZ_Analysis\STRESS\Combined\THS\Max_Frame_Contour
Max Frame added user interface animation database
```

1

1-10649

THSDBANI,C:\ACSV300\Demo_Problems\Demo9\XYZ_Analysis\STRESS\Combined\THS\All_Frame_Contour



Post-Processing SSI Responses Using Binary Databases Described in Detail in Demo 13

*** Plotting and Saving Results as Text Files from Binary Databases**

```
* Nodal Accelerations (MOTION Module)
ACCDBANI,@PATH[1]\Combined\ACC,Demo 13 Acceleration Time History
MAXDBFRAME,ACC,@PATH[1]\Combined\ACC_Max

* Nodal Relative Displacements (RELDISP Module)
DISPDBANI,@PATH[1]\Combined\THD,Demo 13 Relative Displacement
MAXDBFRAME,DISP,@PATH[1]\Combined\THD_Max

* Element Stresses/Forces (STRESS Module)
THSDBANI,@PATH[1]\Combined\STRESS,Demo 13 Stress Contour
MAXDBFRAME,THS,@PATH[1]\Combined\Stress_Max

* Saving Binary Database Results to Text Files
VAR,OUTNODES,63,137,205,219,253,271
VAR,OUTELEM,1,2,3,4,5,6,7,8,9,10,11,12,13,14

* Accelerations (MOTION Module)
ACCDBTHFILE,OUTNODES,1,@PATH[1]\COMBINED
BINFRAMEOUT,ACC,-1,0,0,@PATH[1]\Combined
BINFRAMEOUT,ACC,1000,0,0,@PATH[1]\Combined
BINFRAMEOUT,ACC,5.5,0.005,0,@PATH[1]\Combined

* Displacement (RELDISP Module)
DISPDBTHFILE,OUTNODES,1,@PATH[1]\COMBINED

* Stress (STRESS Module)
THSDBTHFILE,19,OUTELEM,1,@PATH[1]\Combined
BINSTRTBL,19,OUTELEM,-1,@PATH[1]\Combined\Group19_Max_Stress.txt
BINSTRTBL,19,OUTELEM,1000,@PATH[1]\Combined\Group19_tstep_1000_Stress.txt
```

A list of nodes or elements must be assigned to an ACS SASSI UI variable when saving specific nodes or elements to text files. These are not required when saving a frame containing all nodes or elements

The output direction is controlled by this argument, so 1 = X, 2 = Y, and 3 = Z

Use **ACC**, **DISP**, or **THS** for this argument in the BINFRAMEOUT command to save frames for acceleration, displacement, or stress, respectively

The output group number and stress component is controlled by this arguments, respectively

Coordinate Transformation for Forces and Moments in Shell Elements – Details in Demo 14

To apply coordinate transformations to element forces and moments, the requested elements must first be added to a (section cut) submodel in the ACS SASSI UI.

This submodeling operation conveniently allows for multiple walls or floors to be stored independently without having to redefine any grouping currently present in the model, as well as plotted separately to visualize the elements selected to apply coordinate transformation to.

Coordinate Transformation Commands

For Planar Geometry - CTRVEC,1,0,1,0,@PATH[1],MAX
For Cylindrical Geometry - CTRCCV,2,@PATH[1],MAX

Defines a vector for the X-axis of the coordinate system to transform to. CTRCCV does not require this argument, as it is calculated internally for the cylindrical surface

Some Useful Commands for Selecting Elements t

CutAdd,<cut num>,<group num>,<elem 1>, … <elem N>
CutAdd,<cut num>,<group num>,RANGE,<elem start>,[elem end], [stride]
CutVol,<cutnum>,[Xmin],[Xmax],[Ymin],[Ymax],[Zmin],[Zmax]
SLICE,<cutnum>,<pointx>,<pointy>,<pointz>,<normalx>,<normaly>,<normalz>

10. Automatic Post-Processing Using Multilevel UI Command Macros

Post-Processing Programming Commands

Programming Commands

Command	Action	Description
ADDRND	Append random numbers to the end of a variable	Section 9.14.1
CD	Change Current working directory	Section 9.14.2
FOREACH	Loop on a variable	Section 9.14.3
LOADMACRO	Load a Macro pre file	Section 9.14.4
LOADVAR	Load a Variable from a file	Section 9.14.5
MACRO	Use a Macro	Section 9.14.6
MACROLIST	Show all macro names currently in ACS SASSI User Interface memory	Section 9.14.7
MKDIR	Make a new directory	Section 9.14.8
REDUCESET	Sort variable contents and remove duplicate entries	Section 9.14.9
RND	Fills a variable with a random number	Section 9.14.10
RNDSEED	Seeds or reseeds random number generator	Section 9.14.11
SETVAR	Set Variables	Section 9.14.12
SHOWVAR	Show contents of a variable	Section 9.14.13
VAR	Create a variable	Section 9.14.14
VARLIST	Show all variables in ACS SASSI UI memory	Section 9.14.15

UI Macros for Post-Processing SSI Results

- Perform fast repeated UI operations easily
- Post-Process SSI results (SRSS, averaging, time superposition by algebraic summation, etc.)
- Identify missing frequencies based on ATF or STF results, and build list of new frequencies to be added
- Automate the running of SSI Analysis and post-processing results for coherent and incoherent seismic inputs

See also Demo 3

Macro Basic Functions

- Macros are loaded into the UI with the LOADMACRO command
- Macros are called with the MACRO command
- Input arguments for a macro are written in the form n, where n is the number of the argument, i.e. 1 is the first input argument, 2 is the second, etc.
  - In the example below, the macro call would look like **macro,srss,.\XDIR\00001TR_X01.rs,.\YDIR\00001TR_X01.rs,.\ZDIR\00001TR_X01.rs,.\SRSS\00001TR_X01_SRSS.rs**
  - From this example, the first argument is the X-Direction .rs file, the second is the Y-Direction .rs file, the third is the Z-Direction .rs file, and the last argument is the file to save the computed SRSS to.

```
* Compute SRSS
READSPEC,$1$,1,1
READSPEC,$2$,1,2
READSPEC,$3$,1,3
SRSS,4,1,2,3
WRITESPEC,$4$,4
```

Defining Variables for Efficient Post-Processing

- Defined in the ACS SASSI UI using the VAR and LOADVAR commands
- Can be viewed with VARLIST and SHOWVAR commands
- Reference in the ACS SASSI UI by the defined name preceded by @, i.e. *@variablename[1]*
  - *The number within the brackets identifies the index number to use to access the values stored in a variable i.e. @name[1] for the first value, @name[2] for the second, etc.*

FOREACH Command for Building Variable Loops

- The FOREACH command is extremely useful for post-processing SSI results for many nodes or elements
- ***FOREACH, variable,<command to be executed>***
- The looped variable should appear in the command to be executed in the form of *variable[#]*
  - The [#] indicates that the specified variable is to be looped on
  - Only one variable can be looped on in a single FOREACH command
- When used with the MACRO command, the execution of operations such as computing SRSS for spectra or summation of time histories can be looped through for a list of nodes or elements

Nested Macros for Efficient Post-Processing

Macros can be nested to easily perform repeated operations. This example calls the nesting macro to pass file name information to the macro performing the SRSS operation. The nesting macro is passed file name information from the top level .pre file, looping through nodes to build file names to pass to the nesting macro with the FOREACH command.

Top Level .pre File Calling Nesting Macro

```
* Define Variables
mdl,temp,<work folder>\Coherent\Post-Processing
var,path,.
loadvar,nodes,@Path[1]\Nodes.txt

* Load Macros
loadmacro,srss,SRSS-macro.pre
loadmacro,nestSRSS,Nested-SRSS.pre

* Combine Results
foreach,nodes,macro,nestSRSS,@nodes[#],01,X,@path[1]
```

Macro Calling SRSS Macro (Nested-SRSS.pre)

```
* NEST SRSS
macro,srss,$4$\XDIR\$1$TR_$3$$2$.rs,$4$\YDIR\$1$TR_$3$$2$.rs,
$4$\ZDIR\$1$TR_$3$$2$.rs,$4$\Combined\ISRS\$1$TR_$3$$2$.RS
```

Macro to Perform SRSS Calculation (SRSS-macro.pre)

```
* SRSS MACRO
READSPEC,$1$,1,1
READSPEC,$2$,1,2
READSPEC,$3$,1,3
SRSS,4,1,2,3
WRITESPEC,$4$,4
```

Example of UI Macros for Adding Acceleration Histories

Macros can be nested to easily perform repeated operations. This example calls the nesting macro to pass file name information to the macro performing the ADDITION operation.
The nesting macro is passed file name information from the top level .pre file, looping through nodes to build file names to pass to the nesting macro with the FOREACH command.

Top Level .pre File Calling Nesting Macro

```
* Define Variables
mdl,temp,<work folder>\Coherent\Post-Processing
var,path,.
loadvar,nodes,@Path[1]\Nodes.txt

* Load Macros
loadmacro,ADD,ADD-Macro.pre
loadmacro,NESTADD,Nested-ADD.pre

* Combine Results
foreach,nodes,macro,nestADD,@nodes[#],X,@path[1]
```

NESTADD Macro Calling ADD Macro (Nested-ADD.pre)

```
* NESTADD Macro
macro,ADD,$3$\XDIR\$1$TR_$2$.acc,$3$\YDIR\$1$TR_$2$.rs,$3$\ZDI
R\$1$TR_$2$.acc,$4$\Combined\ISRS\$1$TR_$2$.acc
```

ADD Macro for ADDITION Calculations (ADD-Macro.pre)

```
* ADD Macro
READTH,$1$,0,1
READTH,$2$,0,2
READTH,$3$,0,3
ADDITION,4,1,2,3
WRITETH,$4$,4
```

Nodes.txt

```
09201
09202
09203
09204
09205
09206
09207
09208
09209
09210
09211
09212
09213
09214
09215
09233
09960
```

Combination of ISRS, ACC, and THD Files for Incoherent SSI Analysis (Combine_Results.pre)

```
* Combine_Results.pre
* Macro to run SRSS and Average for ISRS, and time history
combination for * THD, ACC, and THS files

* Define Variables
mdl,temp,<work directory>\Incoherent\Post-Processing
var,path,.
loadvar,xnodes,@Path[1]\XDIR_Nodes.txt
loadvar,ynodes,@Path[1]\YDIR_Nodes.txt
loadvar,znodes,@Path[1]\ZDIR_Nodes.txt
loadvar,samples,@Path[1]\Samples.txt

* Load Macros
loadmacro,srss,SRSS-macro.pre
loadmacro,add,Addition-macro.pre
loadmacro,mean,Average-macro.pre

loadmacro,xnestsrssfor,Nested-SRSS-foreach-X.pre
loadmacro,ynestsrssfor,Nested-SRSS-foreach-Y.pre
loadmacro,znestsrssfor,Nested-SRSS-foreach-Z.pre

loadmacro,xnestaddaccfor,Nested-Add-ACC-foreach-X.pre
loadmacro,ynestaddaccfor,Nested-Add-ACC-foreach-Y.pre
loadmacro,znestaddaccfor,Nested-Add-ACC-foreach-Z.pre
loadmacro,xnestaddthdfor,Nested-Add-THD-foreach-X.pre
loadmacro,ynestaddthdfor,Nested-Add-THD-foreach-Y.pre
loadmacro,znestaddthdfor,Nested-Add-THD-foreach-Z.pre

loadmacro,nestSRSS,Nested-SRSS.pre
loadmacro,nestAddACC,Nested-Add-ACC.pre
loadmacro,nestAddTHD,Nested-Add-THD.pre
```

See Demo 3

```
* Macro continued
* Create Combined Folders for Each Sample
foreach,samples,mkdir,@Path[1]\Sample_@samples[#]\Combined
foreach,samples,mkdir,@Path[1]\Sample_@samples[#]\Combined\ISRS
foreach,samples,mkdir,@Path[1]\Sample_@samples[#]\Combined\THD
foreach,samples,mkdir,@Path[1]\Sample_@samples[#]\Combined\ACC

* Calculate SRSS for ISRS for Each Sample
foreach,samples,macro,xnestsrssfor,@samples[#]
foreach,samples,macro,ynestsrssfor,@samples[#]
foreach,samples,macro,znestsrssfor,@samples[#]

* Calculate Sum for ACC for Each Sample
foreach,samples,macro,xnestaddaccfor,@samples[#]
foreach,samples,macro,ynestaddaccfor,@samples[#]
foreach,samples,macro,znestaddaccfor,@samples[#]
```

Combination of ISRS, ACC, and THD Files for Incoherent SSI Analysis (Combine_Results.pre)

```
* Calculate Sum for THD for Each Sample
foreach,samples,macro,xnestaddthdfor,@samples[#]
foreach,samples,macro,ynestaddthdfor,@samples[#]
foreach,samples,macro,znestaddthdfor,@samples[#]

* Calculate Mean for Each Node
mkdir,.\Mean
foreach,xnodes,macro,mean,@path[1],@xnodes[#],X,01
foreach,ynodes,macro,mean,@path[1],@ynodes[#],Y,01
foreach,znodes,macro,mean,@path[1],@znodes[#],Z,01
```

Macros Are Defined as .pre Input Files

```
* Addition-Macro.pre for Add
Time History for 3 Directions
READTH,$1$,0,1
READTH,$2$,0,2
READTH,$3$,0,3
ADDITION,4,1,2,3
WRITETH,$4$,4
```

```
* SRSS-Macro.pre for SRSS
READSPEC,$1$,1,1
READSPEC,$2$,1,2
READSPEC,$3$,1,3
SRSS,4,1,2,3
WRITESPEC,$4$,4
```

```
* Nested-SRSS.pre
macro,srss,$4$\XDIR\$1$TR_$3$$2$.rs,$4$\YDIR\$1$TR_$3$$2$.rs,$4$\
ZDIR\$1$TR_$3$$2$.rs,$4$\Combined\ISRS\$1$TR_$3$$2$.RS
```

```
* Nested-SRSS-foreach-X.pre
foreach,xnodes,macro,nestSRSS,@xnodes[#],01,X,@path[1]\Sample_$1$
```

```
* Nested-Add-ACC-foreach-X.pre
foreach,xnodes,macro,nestAddACC,@xnodes[#],X,@path[1]\Sample_$1$
```

```
* Average-Macro.pre for Average for 20 Samples
READSPEC,$1$\Sample_1\Combined\ISRS\$2$TR_$3$$4$.RS,1,1
READSPEC,$1$\Sample_2\Combined\ISRS\$2$TR_$3$$4$.RS,1,2
READSPEC,$1$\Sample_3\Combined\ISRS\$2$TR_$3$$4$.RS,1,3
READSPEC,$1$\Sample_4\Combined\ISRS\$2$TR_$3$$4$.RS,1,4
READSPEC,$1$\Sample_5\Combined\ISRS\$2$TR_$3$$4$.RS,1,5
READSPEC,$1$\Sample_6\Combined\ISRS\$2$TR_$3$$4$.RS,1,6
READSPEC,$1$\Sample_7\Combined\ISRS\$2$TR_$3$$4$.RS,1,7
READSPEC,$1$\Sample_8\Combined\ISRS\$2$TR_$3$$4$.RS,1,8
READSPEC,$1$\Sample_9\Combined\ISRS\$2$TR_$3$$4$.RS,1,9
READSPEC,$1$\Sample_10\Combined\ISRS\$2$TR_$3$$4$.RS,1,10
READSPEC,$1$\Sample_11\Combined\ISRS\$2$TR_$3$$4$.RS,1,11
READSPEC,$1$\Sample_12\Combined\ISRS\$2$TR_$3$$4$.RS,1,12
READSPEC,$1$\Sample_13\Combined\ISRS\$2$TR_$3$$4$.RS,1,13
READSPEC,$1$\Sample_14\Combined\ISRS\$2$TR_$3$$4$.RS,1,14
READSPEC,$1$\Sample_15\Combined\ISRS\$2$TR_$3$$4$.RS,1,15
READSPEC,$1$\Sample_16\Combined\ISRS\$2$TR_$3$$4$.RS,1,16
READSPEC,$1$\Sample_17\Combined\ISRS\$2$TR_$3$$4$.RS,1,17
READSPEC,$1$\Sample_18\Combined\ISRS\$2$TR_$3$$4$.RS,1,18
READSPEC,$1$\Sample_19\Combined\ISRS\$2$TR_$3$$4$.RS,1,19
READSPEC,$1$\Sample_20\Combined\ISRS\$2$TR_$3$$4$.RS,1,20
AVERAGE,21,1,2,3,4,5,6,7,8,9,10,11,12,13,14,15,16,17,18,19,20
WRITESPEC,$1$\Mean\$2$TR_$3$$4$.RS,21
```

The THS, THD, and ACC addition nested macros follow a similar structure to the Nested-SRSS.pre and Nested-SRSS-foreach-X.pre

Combination of THS Files for Incoherent Analysis

```
* Define Variables
mdl,temp,<work directory>\Post-Processing
var,path,.
loadvar,thslist,@path[1]\ths_list.txt
loadvar,samples,@Path[1]\Samples.txt

* Load Macros
loadmacro,add,Addition-macro.pre
loadmacro,nestaddthsfor,Nested-Add-THS-foreach.pre
loadmacro,nestAddTHS,Nested-Add-THS.pre

* Create Combined Folders for Each Sample
foreach,samples,mkdir,@Path[1]\Sample_@samples[#]\Combined
foreach,samples,mkdir,@Path[1]\Sample_@samples[#]\Combined\THS

* Calculate Sum for THS for Each Sample
foreach,samples,macro,nestaddthsfor,@samples[#]
```

```
* Addition-Macro.pre for Add Time History for 3 Directions
READTH,$1$,0,1
READTH,$2$,0,2
READTH,$3$,0,3
ADDITION,4,1,2,3
WRITETH,$4$,4
```

```
* Nested-Add-THS-foreach.pre
foreach,thslist,macro,nestAddTHS,@path[1]\Sample_$1$,@thslist[#]
```

```
* Nested-Add-THS.pre
macro,add,$1$\XDIR\$2$,$1$\YDIR\$2$,$1$\ZDIR\$2$,$1$\Combined\THS\$2$
```


```
* Nested-Add-THS.pre
macro,add,$1$\XDIR\$2$,$1$\YDIR\$2$,$1$\ZDIR\$2$,$1$\Combined\THS\$2$
```


THS_list.txt

```
BEAMS_002_00001_FXI.THS
BEAMS_002_00001_FXJ.THS
BEAMS_002_00001_FYI.THS
BEAMS_002_00001_FYJ.THS
BEAMS_002_00001_FZI.THS
BEAMS_002_00001_FZJ.THS
BEAMS_002_00001_MXI.THS
BEAMS_002_00001_MXJ.THS
BEAMS_002_00001_MYI.THS
BEAMS_002_00001_MYJ.THS
BEAMS_002_00001_MZI.THS
BEAMS_002_00001_MZJ.THS
.
.
.
BEAMS_011_00095_FXI.THS
BEAMS_011_00095_FXJ.THS
BEAMS_011_00095_FYI.THS
BEAMS_011_00095_FYJ.THS
BEAMS_011_00095_FZI.THS
BEAMS_011_00095_FZJ.THS
BEAMS_011_00095_MXI.THS
BEAMS_011_00095_MXJ.THS
BEAMS_011_00095_MYI.THS
BEAMS_011_00095_MYJ.THS
BEAMS_011_00095_MZI.THS
BEAMS_011_00095_MZJ.THS
```

Variables Loaded in Combine_Results.pre

XDIR_Nodes.txt

```
09201
09202
09203
09204
09205
09206
09207
09208
09209
09210
09211
09212
09213
09214
09215
09233
09960
```

YDIR_Nodes.txt

```
09201
09202
09203
09204
09205
09206
09207
09208
09209
09210
09211
09212
09213
09214
09215
09233
09960
```

ZDIR_Nodes.txt

```
09201
09202
09203
09204
09205
09206
09207
09208
09209
09210
09211
09212
09213
09214
09215
09233
09301
09302
09303
09304
09305
09306
09307
09308
09309
09310
09311
09312
09313
09314
09315
```

Samples.txt

```
1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
```

Macro for Finding Additional Frequencies for Improving ATF and STF Interpolation Errors

```
* Modify path var to location of the demo directory
var,path,C:ACSV300\Demo_Problems\demo3\
var,dirs,@path[1]\TFU-TFI
mdl,,@path[1]
loadvar,TF,tflist.txt
mdl,,@dirs[1]
var,FREQ
foreach,TF,CRITFREQ,90,50,@TF[#],FREQ
reduceset,FREQ,FLOAT
showvar,FREQ
```

11. Section-Cuts Capabilities for Shell and Solid Elements

Section-Cut Related Commands

Cut & Submodeling Commands

Command	Action	Description
CSECT	create a cross sectional model	Section 9.9.1
CUT2SUB	Transfer Elements in a cut to another model	Section 9.9.2
CUTADD	Add elements from a group to a cut	Section 9.9.3
CUTCLR	Clear a cut in memory	Section 9.9.4
CUTRMV	Remove elements from a cut	Section 9.9.5
CUTVOL	Add elements in a volume to a cut	Section 9.9.6
EXTRACTEXCAV	Make a submodel of the excavation volume	Section 9.9.7
SLICE	Add elements to a cut that lie along a plane	Section 9.9.8
SPLITGROUP	Split a group into 2 different groups	Section 9.9.9
TRANELEM	Transfer a list of element in the current model to another model	Section 9.9.10
TRANVOL	Transfer all elements in a volume to another model	Section 9.9.11

Calculation Section-Cut UI Commands

Calculation Commands

Command	Action	Description
CALCC	Calculate the center of area for the model	Section 9.12.1
CALCSECTHIST	Calculate the stress history of a cross section based on multiple files	Section 9.12.2
CALCSECTHISTDB	Calculate the stress history of a cross section using the data from the binary stress database	Section 9.12.3
CALCM	Calculate the mass of the object	Section 9.12.4
CALCMOI	Calculate the moment of inertia	Section 9.12.5
CALCPAR	Calculate all six response quantities for a cross-section	Section 9.12.6
READSTR	Read the .ess stress file and apply to the model	Section 9.12.7
SECDATAOPT	Set the output request in stress for section cut data	Section 9.12.8
SHEAR	Calculate ultimate shear for a nonlinear panel	Section 9.12.9

Section-Cuts Using ESTRESS Text Frames

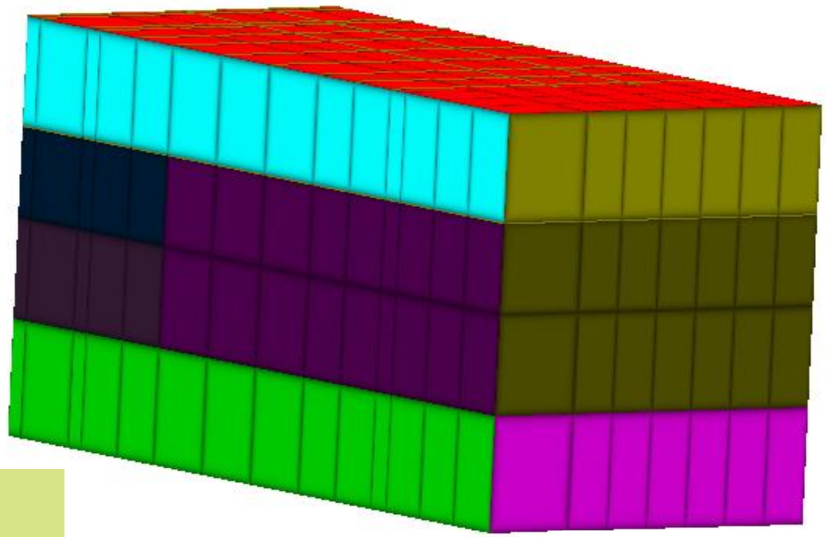
The UI Section-Cut capability has two options:

1) Uses *a single frame of stress data* (single .ess frame file) to compute the section-cut forces and moments on a cross-section at a specific time step.

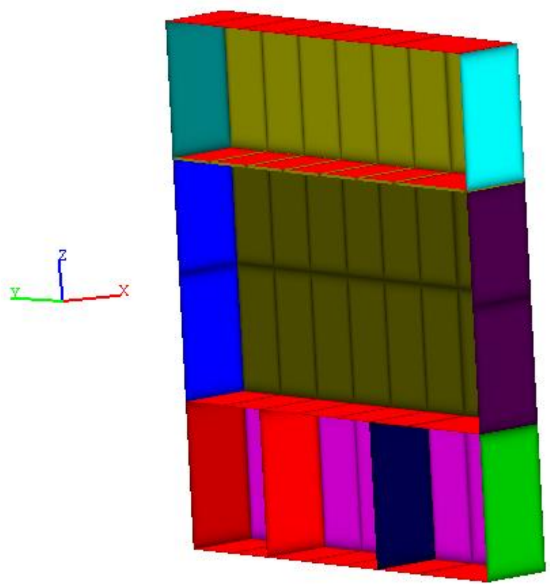
2) Uses a *multiple frames of stress data* (all .ess frame files) to compute the full time-history of the section-cut forces and moments.

Demo 8

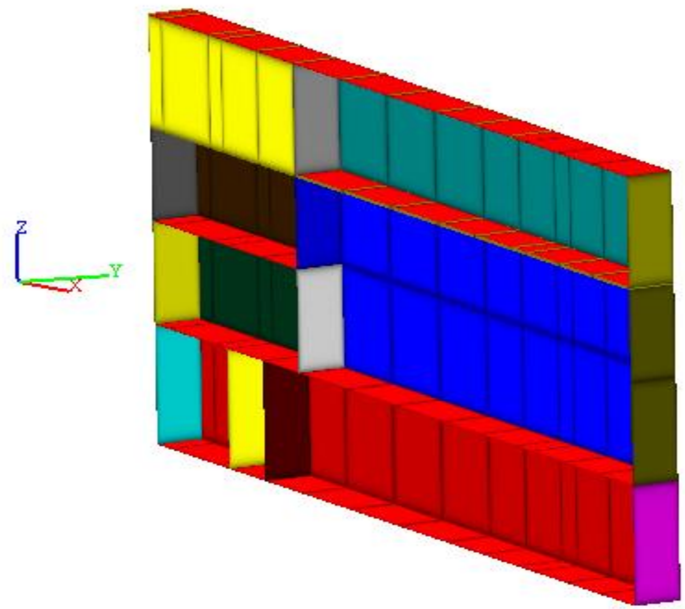
Section-Cut Submodel Models



Transverse Wall



Longitudinal Wall



Section-Cut for Single ESTRESS Stress Frame

* Read element center stress frame

READSTR, estress_02617.ess, C:\DEMOS\DEMO8\ESS_STRESS

*For the 1st section-cut in the SUBMODELER command line, type

CUTVOL,1,132.4

*The blank arguments to this command are interpreted as the respective
*minimum or maximum extent of the building model geometry. This cut volume
*is saved to cut #1.

CSECT,1,1,0,0,15.3,0,0,1

*This creates a cross-section model from cut #1 through point (0.0, 0.0, 15.3),
*with a cross-section plane normal unit vector of (0.0, 0.0, 1.0). The cut cross-
*section is saved to model #1

CALCPAR, 0.0, 0.0, 1.0, 1.0, 0.0, 0.0" in the command SUBMODELER window to calculate the cross-section parameters, seismic forces and moments

Section-Cut Results

```
Model Parameters
Centroid X =145.443 Y =-149.003 Z = 15.8 Area = 342
Ixx = 305990 Iyy = 5183.71 Izz = 311174
Fx = -28.0657 Fy = 11456.9 Fz = 109.184
Mx = -323054 My = 124.862 Mz = 97618.6
```

NOTE: If the element stress frame data is not read properly or not input, the force and moment parameters will be set to "0".

CALCPAR Command Script (.pre) Example

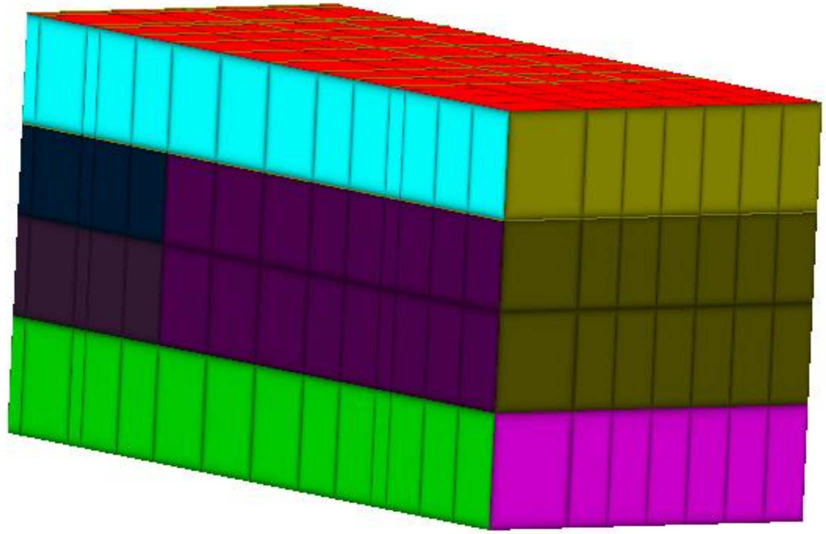
```
actm,0
* Load Model and stress user must change path
inp,Demo8.pre,C:\DEMO_PROBLEMS\DEMO8\
readstr,estress_02617.ess,C:\DEMO_PROBLEMS\DEMO8\ESS_STRESS
*define structural components to be cut
cutvol,1,132.4
cutvol,2,,,-120
* create cross sectional models of selected components along a plane
csect,1,1,0,0,15.3,0,0,1
csect,2,2,0,0,-1,0,0,1
* calculate parameters for each of the cross sections
actm,1
calcpar,0,0,1,1,0,0,1
actm,2
calcpar,0,0,1,1,0,0,1
* output cross sections for visualization with PREP(optional)
actm,0
cut2sub,1,3
cut2sub,2,4
actm,3
write,XSub.pre,C:\DEMO_PROBLEMS\DEMO8\
actm,4
* write,YSub.pre,C:\DEMO_PROBLEMS\DEMO8\
```

CALCSECTHIST Command Batch Input

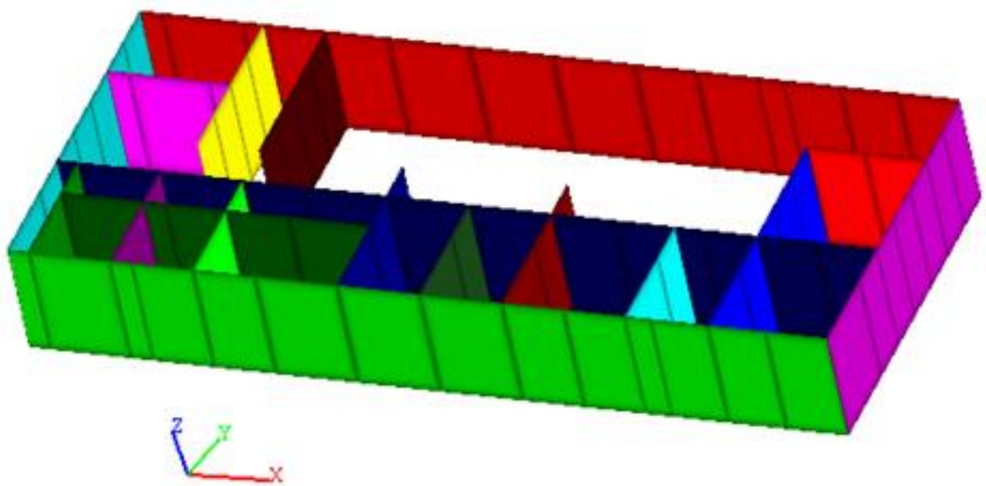
```
1  401   1
C:\ DEMO_PROBLEMS\DEMO8\ESS_FRAMES\
estress_02401.ess
estress_02402.ess
estress_02403.ess
estress_02404.ess
estress_02405.ess
estress_02406.ess
estress_02407.ess
estress_02408.ess
estress_02409.ess
estress_02410.ess
 ......    ......   ......
 estress_02795.ess
estress_02796.ess
estress_02797.ess
estress_02798.ess
estress_02799.ess
estress_02800.ess
estress_02801.ess
```

Batch input file has a similar configuration with the animation files, .thani or .rsani.

Section-Cut for Time History ESTRESS Frames



Section-Cut Model



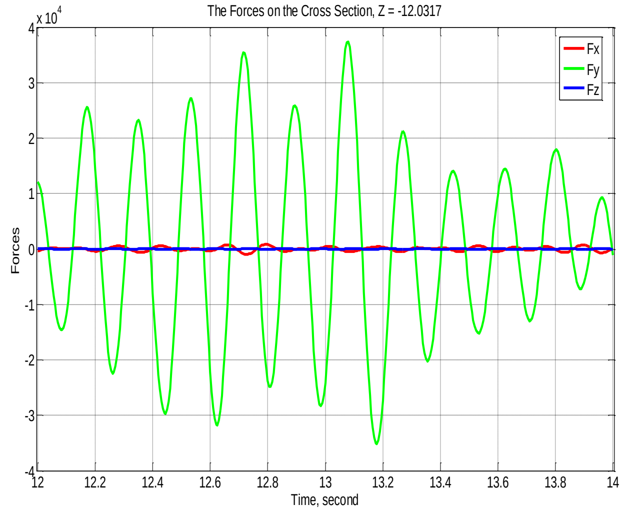
CALCSECTHIST Command Script (.pre) Example

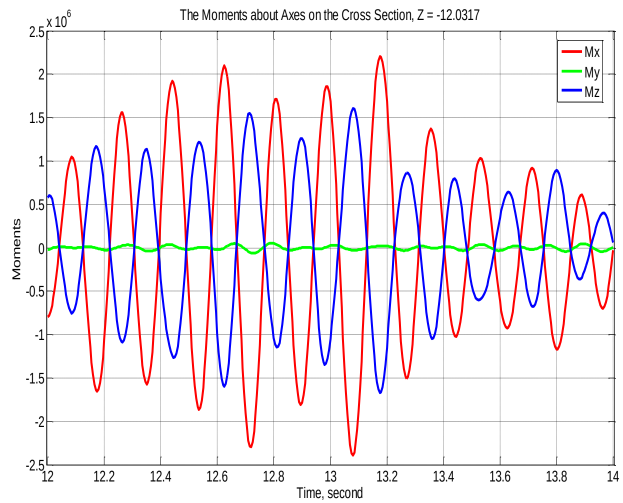
```
*
*Batch .pre input file of section cut for multiple frame data
*
actm,0
*Replace Directory Path
inp,demo8.pre,C:\DEMO_PROBLEMS\DEMO8\
*
* Define structure component to be cut
slice,1,0.0,0.0,-12.0317,0.0,0.0,1.0
*
* Cut the selected structure component using cutting plane
* Calculate the parameters on it, and output to given file
Calcsecthist,C:\DEMO_PROBLEMS\DEMO8\estr_frame_files.lst,1,0.0,0.0,-
12.0317,0.0,0.0,1.0,1.0,0.0,0.0,1,.005,C:\DEMO_PROBLEMS\DEMO8\frc_
mmt_on_cut02.txt
```

CALCSECTHIST Results

C:\DEMO_PROBLEMS\DEMO8\frc_mmt_on_cut02.txt file:

0.005 -343.777 12065.2 0.229168 -800746 -21965.1 581499
0.01 -268.056 11539.5 0.300395 -779248 -16700.6 600032
0.015 -189.851 10623.5 0.294853 -733816 -10990.5 593052
0.02 -114.229 9320.15 0.227868 -665089 -5282.46 558887
0.025 -44.3936 7645.78 0.144148 -573880 2.20757 497620
0.03 18.0423 5647.78 0.0908182 -461901 4514.26 411756
0.035 72.3351 3391.01 0.0894927 -331126 8011.35 305474
0.04 117.752 967.345 0.124164 -184604 10378.6 184558
... ...
1.97 -676.309 9226 0.149393 -698463 -42784.3 401711
1.975 -631.161 8753.78 0.143451 -677218 -40340.6 396997
1.98 -553.34 7862.11 0.158777 -627501 -35783.4 376102
1.985 -450.432 6584.62 0.16545 -550419 -29410.2 339107
1.99 -330.715 4981.56 0.133199 -448530 -21628.8 287164
1.995 -202.2 3116.14 0.0512961 -325035 -12940.6 221694
2 -72.0058 1058.89 -0.0630017 -184306 -3907.81 144578
2.005 53.7767 -1132.73 -0.172686 -30951.4 4886.85 57394.5





CALCSECTHISTDB Command Using STRESS Binary Database

CALCSECTHISTDB,<cutnum>,<px>,<py>,<pz>,<nx>,<ny>,<nz>,<rx>,<ry>,<rz>
,<sys>,<ts>,[start],[end],[Stride],<outfile>

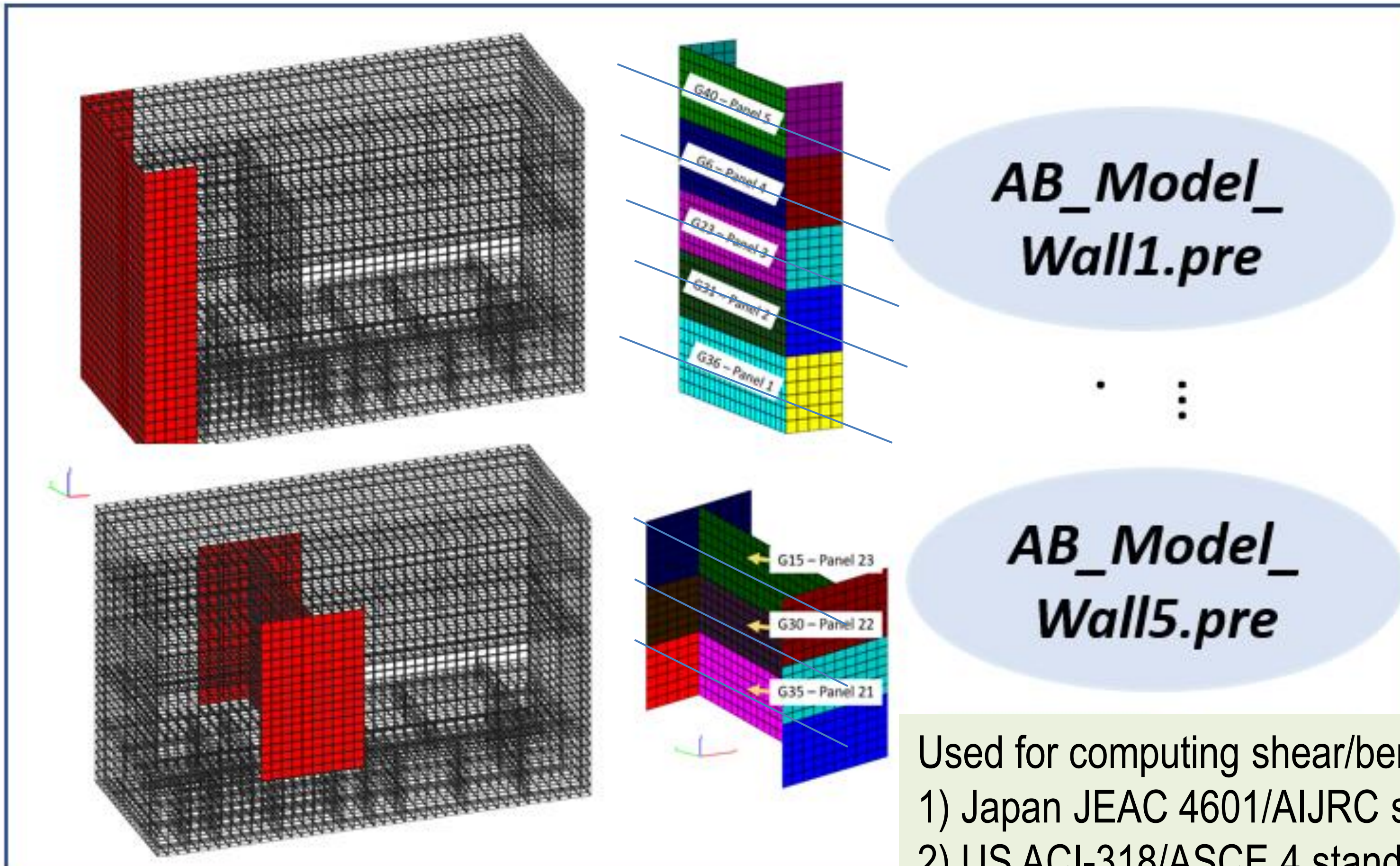
Calculate the Stress history from a cross section using the database currently loaded into User Interface memory. This command has the functionality as CALCSECTHIST does for ASCII generate stress data.

The output of this command is a 7-column text table where the 1st column is either the simulation time or step number if it is set to 0. The rest of the six columns are the stress components of the cross section. The final line of the file will have the word MAX in the1st column. This line will contain the absolute maximum for each component in the table, but the sign of the stress will be maintained on this line.

Before using this command, the stress database for the calculation must be loaded into the UI. The associated model must be loaded into memory and set as the active model. The cut input by the user *cutnum* must be defined before this command.

Section-Cuts w/ Section_Cuts_for_BBC (Opt. NON)

Use UI Section-cut commands to split the 3DFEM model in wall submodels (including several shell groups).



Used for computing shear/bending wall BBCs:
1) Japan JEAC 4601/AIJRC standards, or
2) US ACI-318/ASCE 4 standards

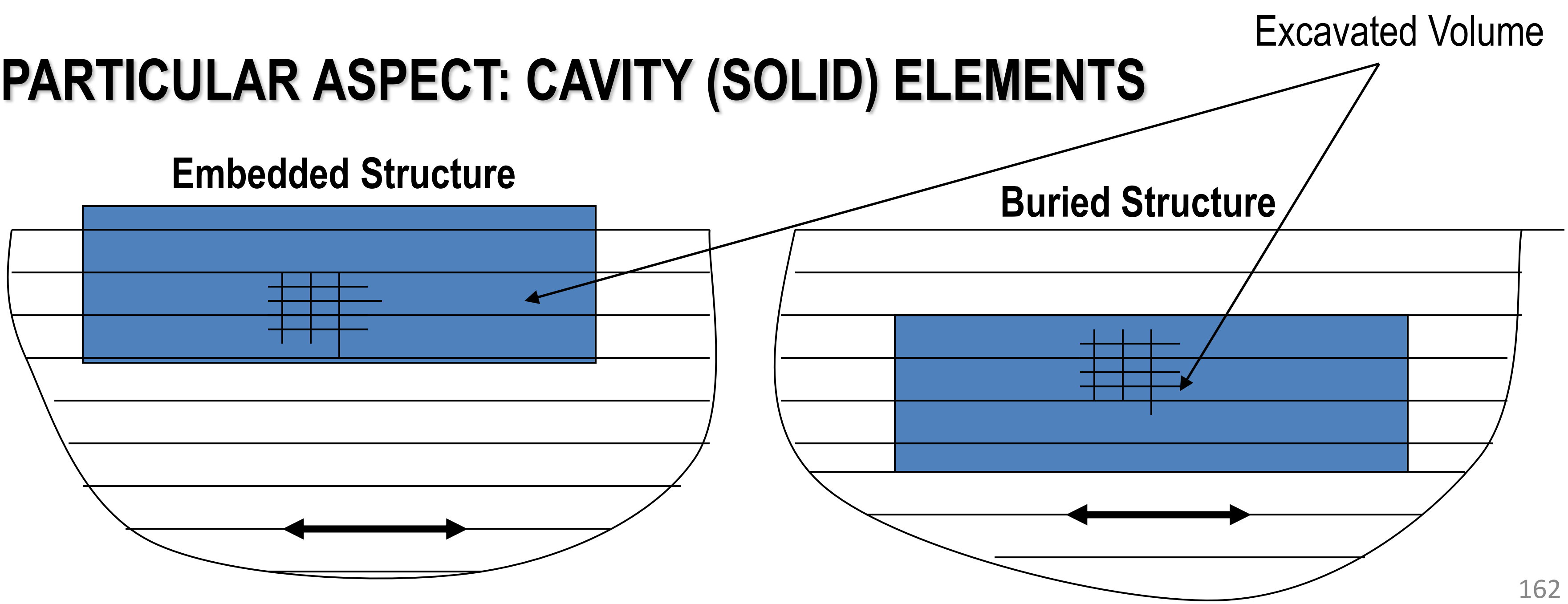
The *Section_Cuts_for_BBC* module is used for computing automatic section-cuts for seismic and gravity loads for each submodel for all floor levels

12. UI Parametric Language Commands for Building SSI FE Models

Building A SSI Analysis Models

Step 1: Define Dynamic Inputs (Seismic Motion or Forces)
Step 2: Define Soil Layering
Step 3: Define Structure and Near Field Zone Using FE Modeling
Step 4: Define Seismic Motion Spatial Incoherency
Step 5: Select SSI Analysis Options (Assumptions, Methods, Parameters)
Step 6: Manage SSI Analysis Runs
Step 7: Post Processing for Extracting Results

PARTICULAR ASPECT: CAVITY (SOLID) ELEMENTS



SSI Model Building Recommendations

The user manuals contain a large number of comments on various SSI modeling aspects.

Top-level recommendations of node and element numbering:

- Soil layering to be numbered from ground surface to baserock
- Excavation volume nodes to be numbered from baserock to ground surface
- Interaction nodes defined in ascending number order
- Excavation volume layers to be numbered from ground surface to baserock
- Excavation volume elements to be numbered from ground surface to baserock; each layer of elements to be a separate group for post-processing.

We also recommend always check the consistency of your soil layer or material element assignments for the soil excavation volume and the structural embedment part by revising the HOUSE output (modelname_HOUSE.out).

For technical support please contact us acs.sassi@ghiocel-tech.com.

Node Commands

Node Commands

Command	Action	Description
D	Sets boundary conditions.	Section 9.3.1
FILL	Generates a node line.	Section 9.3.2
INT	Sets interaction, interface, intermediate, or internal nodes.	Section 9.3.3
INTLIST	Lists interaction, interface, intermediate, or internal nodes.	Section 9.3.4
LMOVE	Generates a node list by translation.	Section 9.3.5
N	Defines a node.	Section 9.3.6
NDEL	Deletes nodes.	Section 9.3.7
NGEN	Generates nodes by copying a node pattern.	Section 9.3.8
NLIST	Lists nodes.	Section 9.3.9
NMED	Defines a node with average coordinates.	Section 9.3.10
NMOVE	Generates a node list by scaling.	Section 9.3.11
NSCALE	Scales nodal coordinates.	Section 9.3.12

Element Commands

Element Commands

Command	Action	Description
DELL	Deletes soil layers.	Section 9.4.1
DELM	Deletes materials.	Section 9.4.2
DELR	Deletes real properties.	Section 9.4.3
DELSC	Deletes spring properties.	Section 9.4.4
E	Defines an element.	Section 9.4.5
ECOMPR	Compresses elements.	Section 9.4.6
EDEL	Deletes elements.	Section 9.4.7
EGEN	Generates elements by translation	Section 9.4.8
EINT	Sets integration order for SOLID elements.	Section 9.4.9
ELIST	Lists elements.	Section 9.4.10
ETYPE	Sets type for SOLID, PLANE, SHELL/TSHELL elements.	Section 9.4.11
GDEL	Deletes groups.	Section 9.4.12
GLIST	Lists groups.	Section 9.4.13
GROUP	Creates or activates a group.	Section 9.4.14
GTIT	Sets group title	Section 9.4.15
HVDCEDEL	Deletes HVD constant set	Section 9.4.16
HVDCELST	Lists the HVD constants	Section 9.4.17
HVDCEQ	Define HVD material constants	Section 9.4.18
KI	Defines end release code in node I of BEAMS elements.	Section 9.4.19
KJ	Defines end release code in node J of BEAMS elements.	Section 9.4.20

Element Commands (cont.)

L	Defines a soil layer.	Section 9.4.21
LLIST	Lists soil layers.	Section 9.4.22
M	Defines a material.	Section 9.4.23
MACT	Sets active material / soil layer index.	Section 9.4.24
MLIST	Lists materials.	Section 9.4.25
MSET	Sets element material / soil layer index.	Section 9.4.26
MXDEL	Delete matrix properties.	Section 9.4.27
MXI	Set terms for matrix property - imaginary part of stiffness matrix.	Section 9.4.28
MXLIST	List matrix property.	Section 9.4.29
MXM	Set terms for matrix property - mass / weight matrix.	Section 9.4.30
MXR	Set terms for matrix property - real part of stiffness matrix	Section 9.4.31
RACT	Sets active real / spring / matrix property index.	Section 9.4.33
RLIST	Lists real properties.	Section 9.4.34
RSET	Sets element real / spring / matrix property index	Section 9.4.35
SC	Defines a spring property.	Section 9.4.36
SCLIST	Lists spring properties.	Section 9.4.37
THICK	Sets thickness for SHELL elements.	Section 9.4.38

NGEN Command for Generating New Nodes

- **NGEN**, *ITIME*, *INC*, *NODE1*, *NODE2*, *NINC*, *DX*, *DY*, *DZ*

Generates additional nodes from a pattern of nodes.

ITIME, *INC*

Do this generation operation a total of *ITIME* times, incrementing all nodes in the given pattern by *INC* each time after the first. *ITIME* must be > 1 for generation to occur.

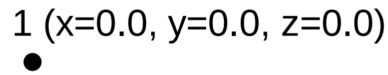
NODE1, *NODE2*, *NINC*

Generate nodes from the pattern of nodes beginning with *NODE1* to *NODE2* in steps of *NINC*

DX, *DY*, *DZ*

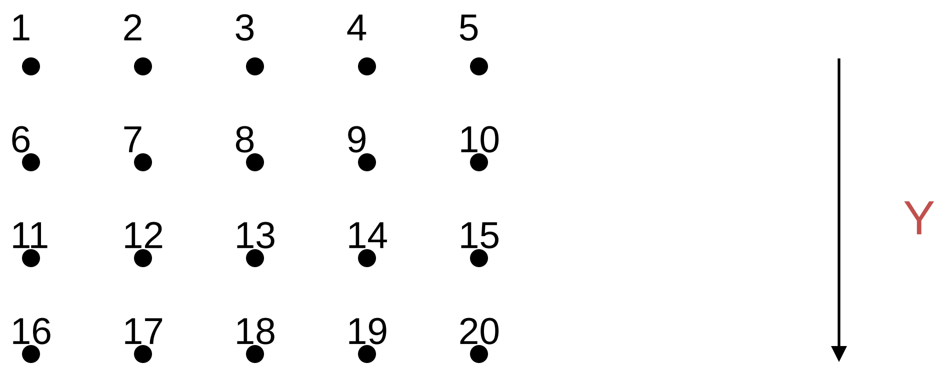
Node location increments

N, 1, 0.0, 0.0, 0.0



NGEN, 4, 1, 1,1,1, 1,0,0

NGEN, 3, 5,1, 5, 1,0,1,0



NGEN, 1, 20,1,20,1,0,0,1

EGEN Command for Generating New Elements

- **EGEN**, *ITIME*, *NINC*, *IEL1*, *IEL2*, *IEINC*

Generates elements from an existing pattern.

ITIME, NINC

Do this generation operation a total of *ITIME*s, incrementing all nodes in the given pattern by *NINC* each time after the first.

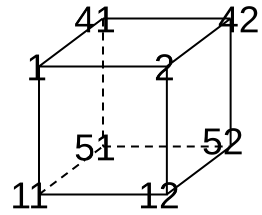
IEL1, IEL2, IEINC

Generate elements from selected pattern beginning with *IEL1* to *IEL2* in steps of *IEINC*

MINC

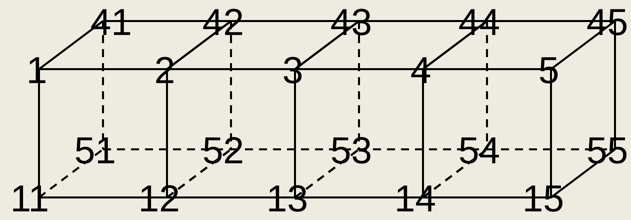
Increment material number of all elements in the given pattern by *MINC* each time after the first.

E,1,1,2,42,41,11,12,52,51



Element #1

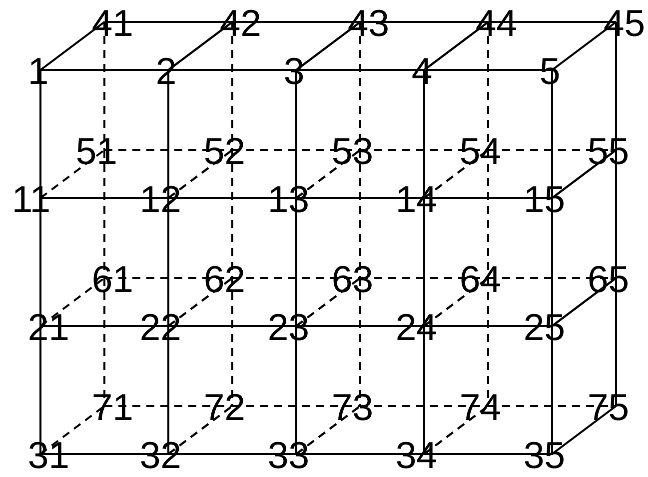
EGEN,3,1,1,1,1



E #1-4

(old 1) (new 2-4)

EGEN,2,10,1,4,1



E #1-12

(old 1-4, new 5-12)

Group Type Commands

Group Type	Element Data	Instruction
SOLID	material / soil layer index	MSET
	element type	ETYPE
BEAMS	material index	MSET
	real property index	RSET
	I node release code	KI
	J node release code	KJ
SHELL/ TSHELL	material index	MSET
	thickness	THICK
	element type	ETYPE
PLANE	material / soil layer index	MSET
	element type	ETYPE
SPRING	spring property index	RSET
GENERAL	matrix property index	RSET
HVD	HVD property constants defined by HVDCEQ	MSET

WARNING: *All the elements in the system must be grouped separately according to their type. Gaps in element numbering are not allowed. Use the ECOMPR command to compress groups with element gaps.*

Nodal Load Commands

Load Commands

Command	Action	Description
F	Defines a force.	Section 9.5.1
FDEL	Deletes forces.	Section 9.5.2
FLIST	Lists forces.	Section 9.5.3
FSCALE	Scales forces.	Section 9.5.4
MM	Defines a moment.	Section 9.5.5
MMDEL	Deletes moments.	Section 9.5.6
MMLIST	Lists moments.	Section 9.5.7
MR	Defines a rotational mass.	Section 9.5.8
MRGEN	Generates rotational masses by translation.	Section 9.5.9
MRDEL	Deletes rotational masses.	Section 9.5.10
MRSCALE	Scales rotational masses.	Section 9.5.11
MSCALE	Scales moments.	Section 9.5.12
MT	Defines a translational mass.	Section 9.5.13
MTDEL	Deletes translational masses.	Section 9.5.14
MTGEN	Generate translational masses by translation.	Section 9.5.15
MTLIST	Lists translational and rotational masses.	Section 9.5.16
MTSCALE	Scales translational masses.	Section 9.5.17
MUNITS	Sets units for translational and rotational masses.	Section 9.5.18

Example of ACS SASSSI .Pre Input File for Simple Embedded Cylinder Model (Solid Elements Only)

.PRE File Structure

```
******************************************************************************
* THIS FILE WAS WRITTEN BY THE ACS SASSI PREPROCESSOR
* To reload model type INP,<this file> in PREP
******************************************************************************
```

Comment lines starts by *

```
TIT,    EMBEDDED CYLINDER MODEL
```

Program title is defined by “TIT” command

```
* Nodes
N,1,0,0,-70
N,2,17.5,0,-70
N,3,12.374,12.374,-70
N,4,0,17.5,-70
N,5,-12.374,12.374,-70
N,6,-17.5,0,-70
N,7,-12.374,-12.374,-70
```

Input Node Coordinates by “N” command:

.PRE File Model Commands

Input constrained displacement by “D” command:

* Boundary Conditions
D,1,414,1,1,ROTX,ROTY,ROTZ

* Interaction Nodes
INT,1,414,1,1,0

Input interaction nodes by “INT” command.
Use INTGEN!

* Material Table
M,1,1e+012,0.2,0,0,0,1,

Input material properties by “M” command:

.PRE File Model Commands

Input soil layer table by “D” command:

```
* Soil Layer Table
L,1,14,0.12,1400,700,0.05,0.05
L,2,10,0.12,1400,700,0.05,0.05
```

Input element group information by “GROUP” command:

```
* Groups and Elements
GROUP,1,SOLID
```

.PRE File Model Commands

Input solid element by “E” command:

E,1,278,279,277,277,347,348,346,346
E,2,279,280,277,277,348,349,346,346
E,3,280,281,277,277,349,350,346,346
E,4,281,282,277,277,350,351,346,346

EINT command for solid element

EINT,1,440,1,1

MSET command for solid element

MSET,1,88,1,1
MSET,89,176,1,2
MSET,177,264,1,3

.PRE File Model Commands

Input solid element (Group #2) by "E" command:

GROUP,2,SOLID
E,1,278,279,277,277,347,348,346,346
E,2,279,280,277,277,348,349,346,346
E,3,280,281,277,277,349,350,346,346
E,4,281,282,277,277,350,351,346,346

ETYPE,1,440,1,1

"ETYPE" command for element group 2.
1 is for STRUCTURE

EINT,1,440,1,2

"EINT" command for element group 2

MSET,1,440,1,2

MSET command for beam element

.PRE File Model Commands

INPUT USING ANALYSIS/OPTIONS WINDOW DIALOGS

* Model Options
MOPT,,0,0,0

"MOPT" command

* Analysis Options
AOPT,0,0,0,1,1,1,0,0,1,0,0,0

Analysis options: "AOPT" command

EQUAKE,0,0,0,0,0,1

"EQUAKE" command

Other .pre Input Files Including Beam and Shell Elements

.PRE File Input for ABShear Model with Shell Groups

ABShear - Notepad

File Edit Format View Help

```
*****************************************************
* THIS FILE WAS WRITTEN BY THE ACS SASSI UI
* To reload model type INP,<this file> in UI
*****************************************************
TIT,
* Nodes
N,1,-46.5833,-149.003,26.9167
N,2,-46.5833,-149.003,49.3333
N,3,-21.5833,-149.003,26.9167
N,4,-21.5833,-149.003,49.3333
N,5,-46.5833,-149.003,75.9167
N,6,-21.5833,-149.003,75.9167
N,7,8.41667,-149.003,49.3333
N,8,8.41667,-149.003,75.9167
N,9,68.4167,-149.003,49.3333
N,10,68.4167,-149.003,75.9167
N,11,98.4167,-149.003,49.3333
N,12,98.4167,-149.003,75.9167
N,13,104.417,-149.003,75.9167
N,14,104.417,-149.003,49.3333
N,15,-51.5833,-128.003,75.9167
N,16,-46.5833,-128.003,75.9167
N,17,-26.5833,-128.003,75.9167
N,18,-21.5833,-128.003,75.9167
N,19,-6.58333,-128.003,75.9167
N,20,8.41667,-128.003,75.9167
N,21,28.4167,-128.003,75.9167
```

```
N,1034,147.417,-117.503,100
N,1035,147.417,-107.003,100
* Boundary Conditions
D,553,556,1,1,UX,UY,UZ,ROTX,ROTY,ROTZ
D,671,671,1,1,UX,UY,UZ,ROTX,ROTY,ROTZ
D,673,673,1,1,UX,UY,UZ,ROTX,ROTY,ROTZ
D,676,676,1,1,UX,UY,UZ,ROTX,ROTY,ROTZ
D,687,687,1,1,UX,UY,UZ,ROTX,ROTY,ROTZ
D,810,1035,1,1,UX,UY,UZ,ROTX,ROTY,ROTZ
* Interaction Nodes
INT,363,482,1,1,0
* Material Table
M,1,519100,0.17,0,0.07,0.07,1
M,2,519100,0.17,0,0.07,0.07,1
M,3,298257,0.17,0,0.07,0.07,1
M,4,362689,0.17,0,0.07,0.07,1
M,5,519100,0.17,0,0.07,0.07,1
M,6,260610,0.17,0,0.07,0.07,1
M,7,519100,0.17,0,0.07,0.07,1
M,8,455845,0.17,0,0.07,0.07,1
M,9,519100,0.17,0,0.07,0.07,1
M,10,360878,0.17,0,0.07,0.07,1
M,11,210410,0.17,0,0.07,0.07,1
M,12,107181,0.17,0,0.07,0.07,1
```

```
M,42,519100,0.17,0,0.07,0.07,1
M,43,519100,0.17,0,0.07,0.07,1
M,44,4.176e+006,0.3,0,0.07,0.07,1
M,45,1e+007,0.17,0,0.07,0.07,1
* Soil Layer Table
L,1,2.75,0.136,1821.14,817.967,0.0072,0.0072
L,2,2.4375,0.136,1888.03,845.657,0.00788,0.00788
L,3,2.4375,0.136,1997.61,912.117,0.0161,0.0161
L,4,2.4375,0.136,2021.25,889.741,0.01627,0.01627
L,5,2.4375,0.136,2080.06,880,0.0226,0.0226
L,6,2.5,0.136,2120,880,0.0226,0.0226
L,98,5,0.136,6900,1880,0.0318,0.0318
L,99,5,0.136,6900,1880,0.0318,0.0318
L,100,5,0.136,6883.94,1895.8,0.03172,0.03172
* Real Property Table
R,1,11.111,0,0,17.387,10.288,10.288
R,2,13,0,0,22.316,9.75,20.343
R,3,2.849,0,0,28.958,0.333,28.292
* Groups and Elements
GROUP,1,TSHELL
E,1,63,64,70,69
E,2,69,70,76,75
E,3,75,76,82,81
E,4,81,82,88,87
E,5,87,88,94,93
E,6,93,94,214,213
```

```
* Frequencies
FREQ,1,1,5,20,40,60,80,100,120,140,160
FREQ,1,180,200,220,240,260,280,300,320,340,360
FREQ,1,380,400,420,440,460,480,500,520,540,560
FREQ,1,580,600,620,640,660,680,700,720,740,760
FREQ,1,780,800,820,840,860,880,900,920,940,960
```

```
E,327,320,335,336,321
E,328,334,349,350,335
E,329,335,350,351,336
THICK,1,329,1,3
MSET,1,329,1,1
GROUP,2,TSHELL
E,1,308,309,577,576
E,2,309,310,574,577
THICK,1,2,1,3
MSET,1,2,1,2
GROUP,3,TSHELL
E,1,106,105,766,767
E,2,105,104,759,766
E,3,104,103,765,759
E,96,423,438,439,424
E,97,438,453,454,439
E,98,453,468,469,454
THICK,1,98,1,3
MSET,1,98,1,45
* Masses
MT,1,4,4,4
MT,2,4,4,4
MT,3,4,4,4
MT,4,4,4,4
MT,527,4,4,4
MT,528,4,4,4
* Model Options
MOPT,1,0,1,1
* Analysis Options
AOPT,0,0,0,0,0,0,0,0,0,0,0,0,0,0
```

Suggestion: After FE model is defined, user should use the UI Analysis/Option Window Input

.PRE File Input for Beam Groups

```
* Real Property Table
R,1,1400,700,700,1e+009,2.8e+006,2.8e+006
R,2,990,500,500,1e+009,1.9e+006,1.9e+006
R,3,990,500,500,1e+009,1.5e+006,1.5e+006
R,4,990,500,500,1e+009,800000,800000
R,5,990,500,500,1e+009,200000,200000
R,6,2000,1320,1320,1e+009,1.1e+006,1.1e+006
R,7,2560,1560,1560,1e+009,1.2e+006,1.2e+006
R,8,2210,1460,1460,1e+009,1.2e+006,1.2e+006
R,9,1960,730,730,1e+009,1.3e+006,1.3e+006
R,10,1740,600,600,1e+009,900000,900000
R,11,780,360,360,1e+009,200000,200000
R,12,190,70,70,1e+009,4000,4000
R,13,100000,500000,500000,1e+012,2e+009,2e+009
```

```
GROUP,2,BEAMS
E,1,139,141,46
E,2,141,142,46
E,3,142,143,46
E,4,143,144,46
E,5,144,145,46
E,6,145,146,46
E,7,146,147,46
E,8,147,148,46
E,9,148,149,46
E,10,149,150,46
E,11,150,151,46
E,12,140,152,46
E,13,152,153,46
E,14,153,154,46
E,15,154,155,46
E,16,155,156,46
E,17,156,157,46
E,18,157,158,46
MSET,1,18,1,2
RSET,1,7,1,1
RSET,8,8,1,2
RSET,9,9,1,3
RSET,10,10,1,4
RSET,11,11,1,5
RSET,12,12,1,6
RSET,13,13,1,7
RSET,14,14,1,8
RSET,15,15,1,9
```

13. Brief Review of Demo Contents

Demo Problem	Software Features		Description
Demo 1	X	Base Software	Introductory demo for ACS SASSI. This demo covers basic ACS SASSI functionality, such as loading model files, running modules, and basic post-Processing and result visualization.
		Option A	
		Option AA	
		Option AA-R	
		Option NON	
		Option UPLIFT	
Demo 2	X	Base Software	This demo includes a procedure for stress post Processing, including creating stress contour plot animations for critical time steps using the ACS SASSI UI.
		Option A	
		Option AA	
		Option AA-R	
		Option NON	
		Option UPLIFT	
Demo 3	X	Base Software	This demo introduces macros for combination of post-processing results, as well as using the ACS SASSI UI to determine critical frequencies.
		Option A	
		Option AA	
		Option AA-R	
		Option NON	
		Option UPLIFT	
Demo 4	X	Base Software	This demo includes a procedure for performing an SSI analysis with nonlinear soil.
		Option A	
		Option AA	
		Option AA-R	
		Option NON	
		Option UPLIFT	
Demo 5	X	Base Software	This demo includes a procedure for for transferring SSI loads to an ANSYS model for equivalent static or dynamic analysis. The conversion of ANSYS models to the ACS SASSI format is demonstrated as well.
	X	Option A	
		Option AA	
		Option AA-R	
		Option NON	
		Option UPLIFT	

Demo 6	X	Base Software	This demo includes a procedure for creating soil finite element models for equivalent static linear soil pressure analysis, and nonlinear soil pressure analysis including foundation separation.
	X	Option A	
		Option AA	
		Option AA-R	
		Option NON	
		Option UPLIFT	
Demo 7	X	Base Software	This demo includes a procedure for performing SSI analysis for surface and embedded structures using mass, stiffness, and damping matrices from ANSYS.
		Option A	
	X	Option AA	
		Option AA-R	
		Option NON	
		Option UPLIFT	
Demo 8	X	Base Software	This demo includes a procedure for computing section-cut forces and moments in a shearwall structure using the ACS SASSI UI commands.
		Option A	
		Option AA	
		Option AA-R	
		Option NON	
		Option UPLIFT	
Demo 9	X	Base Software	This demo includes a procedure for performing nonlinear structure SSI analysis with Option NON for a shearwall structure. It demonstrates how to split a model into nonlinear panels and define backbone curves and nonlinear analysis options.
		Option A	
		Option AA	
		Option AA-R	
	X	Option NON	
		Option UPLIFT	
Demo 10	X	Base Software	This demo includes a procedure for performing nonlinear SSI analysis for a base-isolated shearwall structure with nonlinear hysteretic isolators.
		Option A	
		Option AA	
		Option AA-R	
	X	Option NON	
		Option UPLIFT	

Demo 11	X	Base Software	This demo includes a procedure for using ANSYS MATRIX50 elements in an SSI analysis in ACS SASSI.
		Option A	
	X	Option AA	
		Option AA-R	
		Option NON	
		Option UPLIFT	
Demo 12	X	Base Software	This demo performs a pushover analysis of a reactor building reinforced concrete containment shell
		Option A	
		Option AA	
		Option AA-R	
	X	Option NON	
Demo 13	X	Base Software	The demo includes a procedure for post-processing SSI analysis results using binary databases. It includes combination of binary database, extracting text files from databases, and creating animations from binary databases.
		Option A	
		Option AA	
		Option AA-R	
		Option NON	
		Option UPLIFT	
Demo 14	X	Base Software	This demo includes a procedure for performing coordinate transformation and strain calculations for shell elements in the ACS SASSI UI.
		Option A	
		Option AA	
		Option AA-R	
		Option NON	
		Option UPLIFT	
Demo 15	X	Base Software	This demo includes a procedure for performing SSI analysis in ANSYS via harmonic analysis using condensed impedances generated by ACS SASSI using Option AA-R.
		Option UPLIFT	
		Option AA	
	X	Option AA-R	
		Option NON	
		Option UPLIFT	

Demo 16	X	Base Software	This demo describes the procedure for performing fast SSI analysis using the FVROM-INT approach for a deeply embedded shearwall structure. Also applicable to AA-R option.
		Option A	
		Option AA	
	X	Option AA-R	
		Option NON	
		Option UPLIFT	
Demo 17	X	Base Software	This demo describes the uplift SSI analysis for a surface and an embedded RB model using the UPLIFT option.
		Option A	
		Option AA	
		Option AA-R	
		Option NON	
	X	Option UPLIFT	
Demo 18	X	Base Software	This demo applies Option NON Advanced to typical RC shearwall building using the US standard and Japan standard recommendations.
		Option A	
		Option AA	
		Option AA-R	
	X	Option NON	
Demo 19	X	Base Software	This demo applies Option NON Advanced to typical deeply embedded SMR structure using the US standard and Japan standard recommendations
		Option A	
		Option AA	
		Option AA-R	
	X	Option NON	
		Option UPLIFT	

Option PRO and RVT-SIM have separate illustrative example problems provided in their user manuals

14. Verification and Validation

ACS SASSI NQA V4.3.2 Verification & Validation

Verification Manual includes 65 Selected SSI Problems (with more than 100 subproblems, about 500 figures, with more than 5,000 files) to cover ACS SASSI functionalities (except PRO and RVT-SIM):

- Verify Results Against Other Codes: SHAKE91, ANSYS, etc.
- Verify Against Analytical Solutions
- Verify Against Experiments
- Verify by Engineering Body of Knowledge/Judgment
- Verify by a) Result Accuracy and b) Expected Behavior

NQA Maintenance Service: Bugs and Error Reports, Periodic and Focused Memos with comments, Technical Investigation Reports

End of Part 3 Presentation
Thank You!