ACS SASSI Application to Linear and Nonlinear Seismic SSI Analysis of Nuclear Structures Subjected to Coherent and Incoherent Inputs

1-Day Training Notes

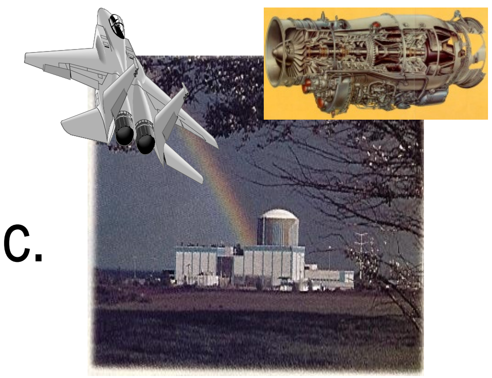
Ghiocel Predictive Technologies Inc.

Dr. Dan M. Ghiocel

Email: dan.ghiocel@ghiocel-tech.com

Ghiocel Predictive Technologies Inc.

http://www.ghiocel-tech.com



"Dan Ghiocel" International Research Center

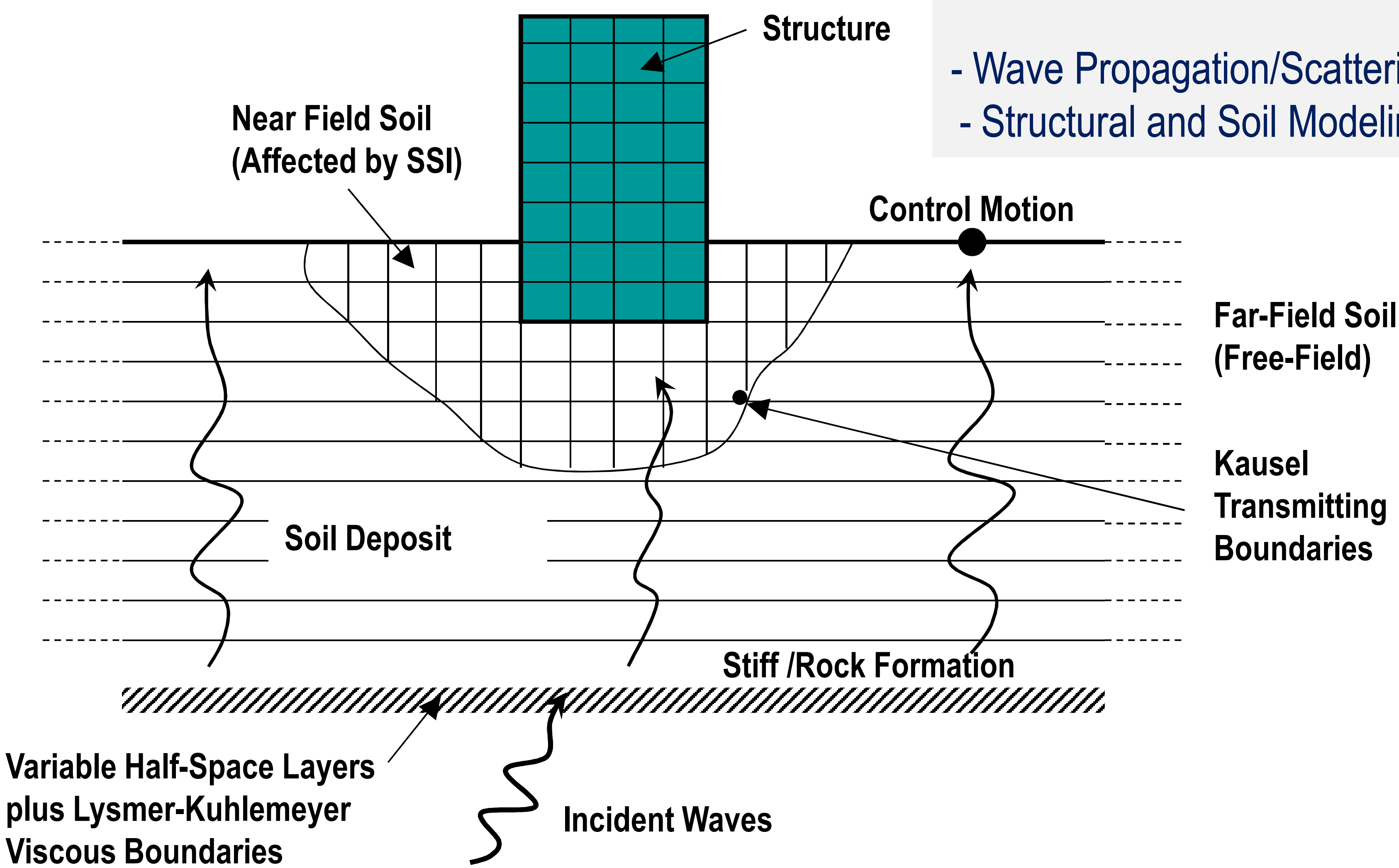
UTCB, Bucharest, Romania

February 27, 2013

Introduction to ACS SASSI Software Capabilities

SSI Modeling and Algorithms
Modular Configuration
Description of GUI and SSI Software Modules
SSI Analysis Runs, Restart and Post-Processing
Special Reporting on SSI Substructuring Methods
ACS SASSI-ANSYS Integration (New, Option A)

Seismic SSI Analysis Problem



Seismic SSI Analysis Aspects:

- Wave Propagation/Scattering
- Structural and Soil Modeling

SSI Analysis Complex Frequency Approach

The equation of motion of the SSI system is:

$$[M]\{\ddot{u}\} + [C]\{\dot{u}\} + [K]\{u\} = -\{m\}\ddot{y}$$

$$[M]\{\ddot{u}\} + [K^*]\{u\} = -\{m\}\ddot{y}$$

Assume: $\ddot{y} = \ddot{Y}e^{i\omega t}$

Then: $\{u\} = \{U\}e^{i\omega t}$

$$\left([K^*] - \omega^2 [M]\right)\{U\} = -\{m\}\ddot{Y}$$

Solve for complex transfer functions for each frequency:

$$\left([K^*] - \omega_s^2 [M]\right)\{A_s\} = -\{m\}$$

Then the solution in frequncy domain:

$$\{U_s\} = \{A_s\}\ddot{Y}$$

Use Fourier Transform for transient time histories, and the compute solution in time domain:

$$u_j(t) = \mathrm{Re}\sum_{s=0}^{N/2} U_{j,s}e^{i\omega_s t}$$

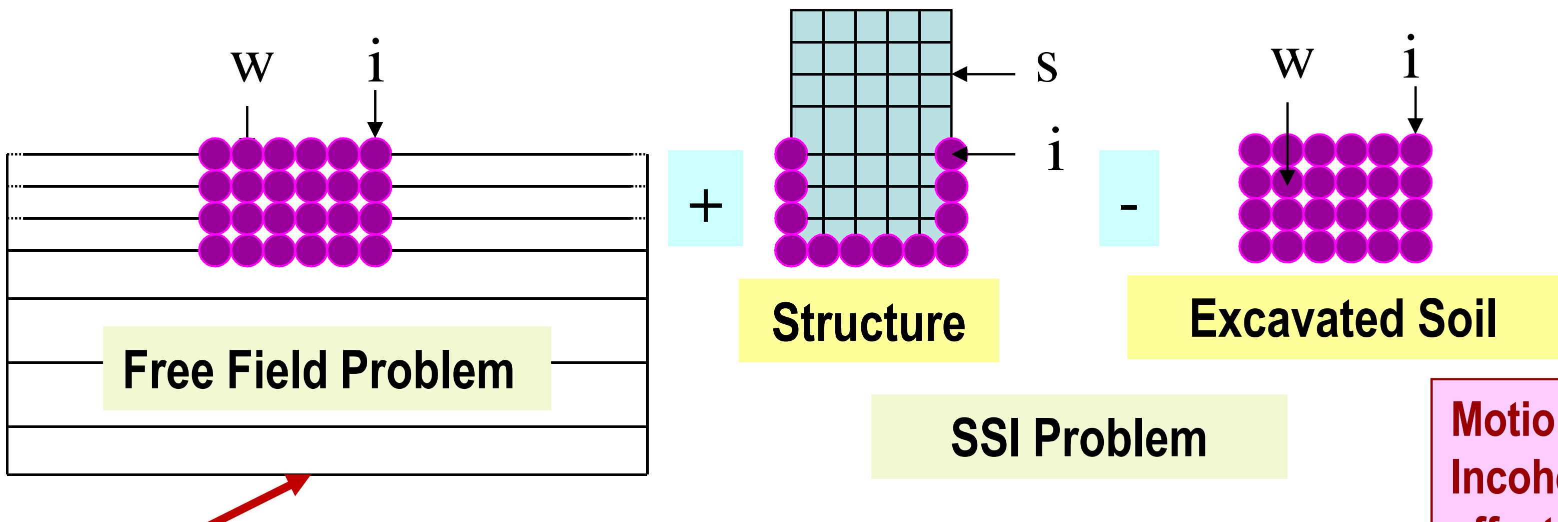
Linear Soil-Structure Interaction Analysis

Computational Steps for Linear SSI Solution:

Soil structure interaction problem subjected to the seismic excitation is solved in the frequency domain following steps:

1. Solve the site response problem
2. Solve the impedance problem
3. Form the load vector
4. Form the complex stiffness matrix
5. Solve the system of linear equations of motion

Incoherent SSI Analysis in ACS SASSI



Motion Incoherency affects free-field motion at interaction nodes

Flexible Volume Method (using all excavated volume nodes)

$$\begin{bmatrix} \mathbf{C}^s_{ii}-\mathbf{C}^e_{ii}+\mathbf{X}_{ii} & -\mathbf{C}^e_{iw}-\mathbf{X}_{iw} & \mathbf{C}^s_{is}\\ -\mathbf{C}^e_{wi}+\mathbf{X}_{wi} & -\mathbf{C}^e_{ww}+\mathbf{X}_{ww} & \mathbf{0} \\ \mathbf{C}^s_{si} & \mathbf{0} & \mathbf{C}^s_{ss}\end{bmatrix}\begin{Bmatrix}\mathbf{U}_i\\ \mathbf{U}_w\\ \mathbf{U}_s\end{Bmatrix} = \begin{Bmatrix} \mathbf{X}_{ii}\mathbf{U}'_i + \mathbf{X}_{iw}\mathbf{U}'_w \\ \mathbf{X}_{wi}\mathbf{U}'_i+\mathbf{X}_{ww}\mathbf{U}'_w \\ \mathbf{0}\end{Bmatrix}$$

Flexible Interface Methods (using excavation volume outer surface nodes)

$$\begin{bmatrix} \mathbf{C}^s_{ii}-\mathbf{C}^e_{ii}+\mathbf{X}_{ii} & -\mathbf{C}^e_{iw} & \mathbf{C}^s_{is}\\ -\mathbf{C}^e_{wi} & -\mathbf{C}^e_{ww} & \mathbf{0} \\ \mathbf{C}^s_{si} & \mathbf{0} & \mathbf{C}^s_{ss}\end{bmatrix}\begin{Bmatrix}\mathbf{U}_i\\ \mathbf{U}_w\\ \mathbf{U}_s\end{Bmatrix} = \begin{Bmatrix} \mathbf{X}_{ii}\mathbf{U}'_i \\ \mathbf{0} \\ \mathbf{0}\end{Bmatrix}$$

$$\mathbf{C}(\omega)\mathbf{U}(\omega) = \mathbf{Q}(\omega)$$

where $\mathbf{C}(\omega) = \mathbf{K} - \omega^2\mathbf{M}$

Seismic Site Response

- The original site is assumed to consist of horizontal soil layers overlying a halfspace.
- All material properties are assumed to be visco-elastic materials.
- Solutions for inclined body waves and surface waves
- Only the free-field displacements of the layer interfaces where the structure is connected are of interest. For the vertically propagating wave types, displacement amplitudes are:

$$\mathbf{u}_f^{'}(x) = \mathbf{U}_f^{'} \exp[i(\omega t - kx)]$$

- Effective discrete methods are used for determining appropriate mode shapes and wave numbers corresponding to control motions at any layer interface for inclined P-, SV-, and SH- waves, Rayleigh waves and Love waves.
- Soil hysteretic behavior is idealized using the Seed-Idriss Equivalent Linear Model

Soil Layer Sizes (It impacts on SSI model)

- For such elements the accuracy of the solution is function of the method used to compute the mass matrix and an accuracy better than 10 percent on wave amplitude is obtained if the element size **h** follows the relations shown below:

$$h \le \begin{cases} 1/8\ \lambda_s & \text{for lumped mass matrix} \\ 1/5\ \lambda_s & \text{for consistent mass matrix} \\ \boxed{1/5\ \lambda_s \text{ for mixed mass matrix}} & \longleftarrow \end{cases}$$

- The wave length is obtained from

$$\lambda_s = \frac{V_s}{f_{max}}$$

Site Response Model for Body Waves

Incident Plane SV and P Waves

The equation of motion to the soil system subjected to inclined P- and/or SV- waves:

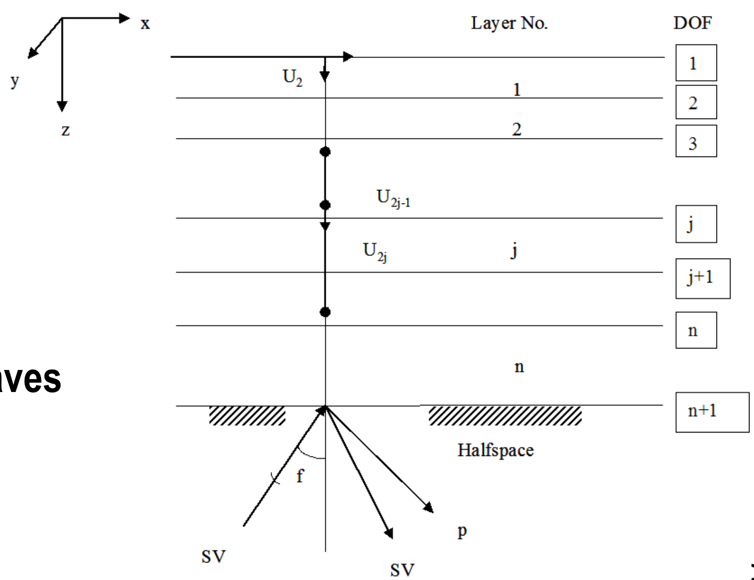
$$([A]k^2 + i[B]k + [G] - \omega^2[M])\{U\} = \begin{vmatrix} 0 \\ P_b \end{vmatrix}$$

Solution to the above equation yields the displacement vector { U }.

For vertically propagating waves it reduces to much simpler equation (Chen, 1980). The free-field motion at any distance x can be obtained from the solution using the relation

$$\{U(x)\} = \{U\}\exp(-ikx)\delta$$

Incident Plane SV Waves



Modeling of Semi-Infinite Halfspace Baserock

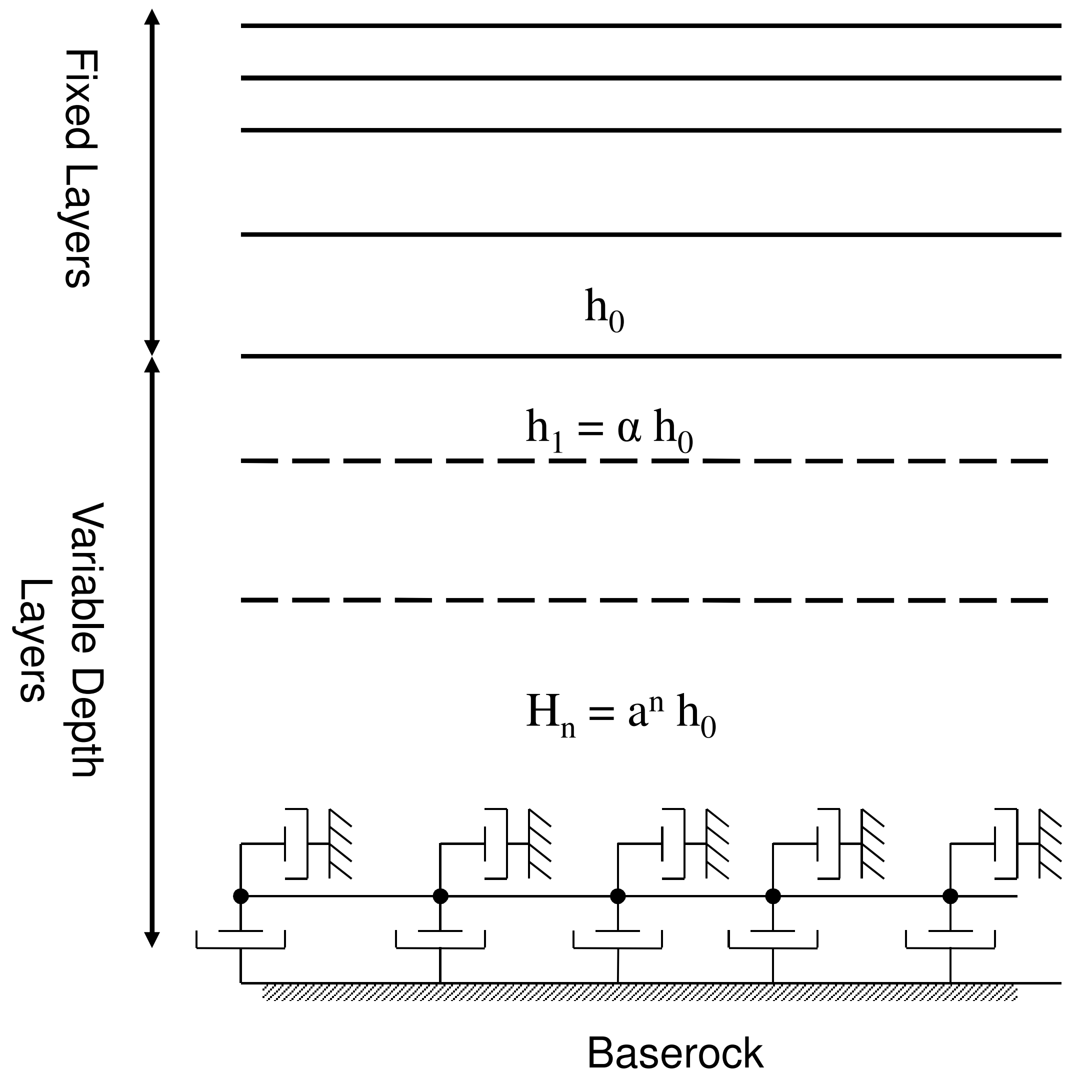
The Variable Depth Method

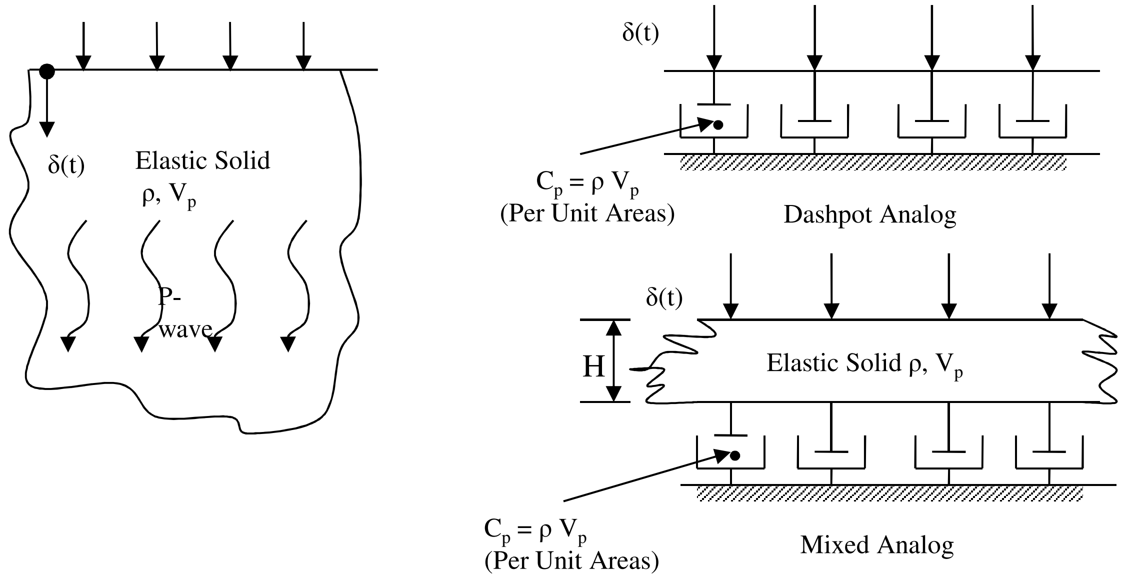
The total depth H of the added layers varies with frequency and is set to:

$$H = 1.5\frac{V_s}{f}$$

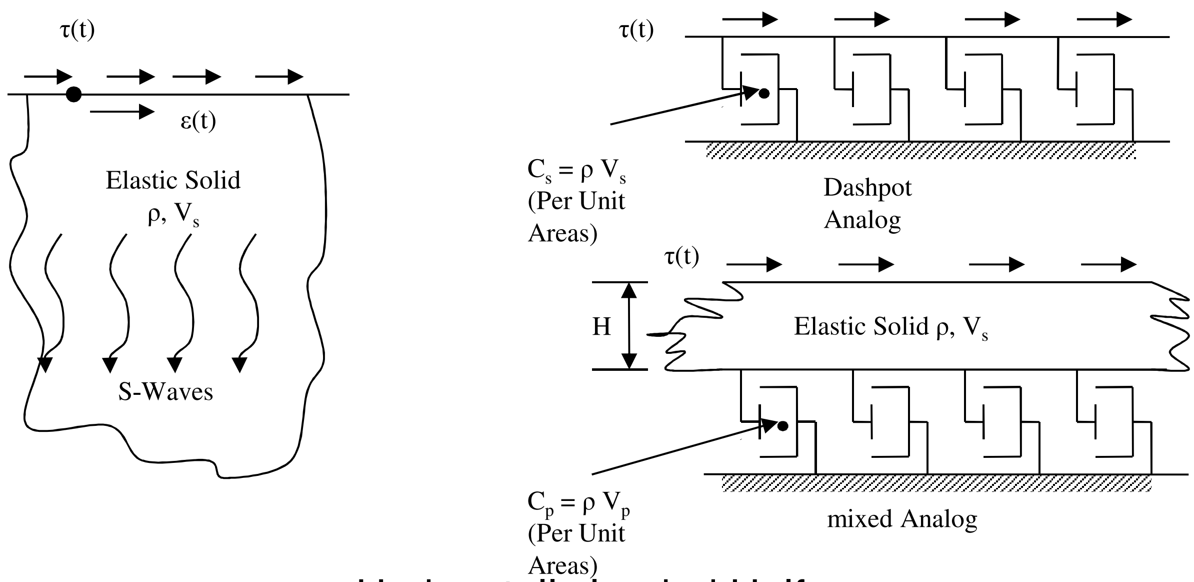
The total thickness of the n layers is:

$$H = h_0 + \alpha^2 h_0 + --- + \alpha n h_0 = \frac{(\alpha^n - 1)h_0}{\alpha - 1}$$





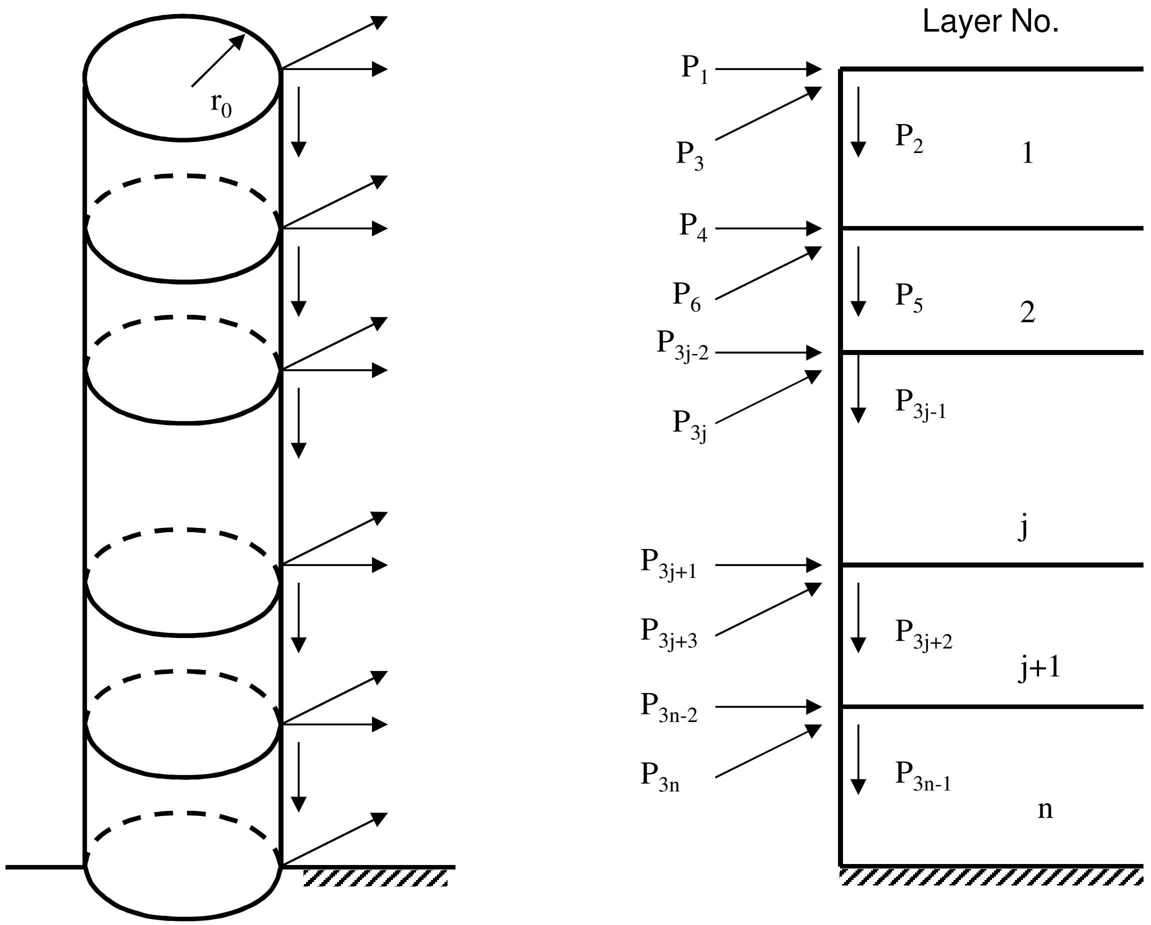
Vertically Loaded Halfspace



Horizontally loaded Halfspace

3D Transmitting Boundary Matrices

The 3D Transmitting Boundary Matrix Uses An Axisymmetric Model (Kausel, 1976):



Degrees of Freedom on Axisymmetric Transmitting Boundary

3D Transmitting Boundary Matrices

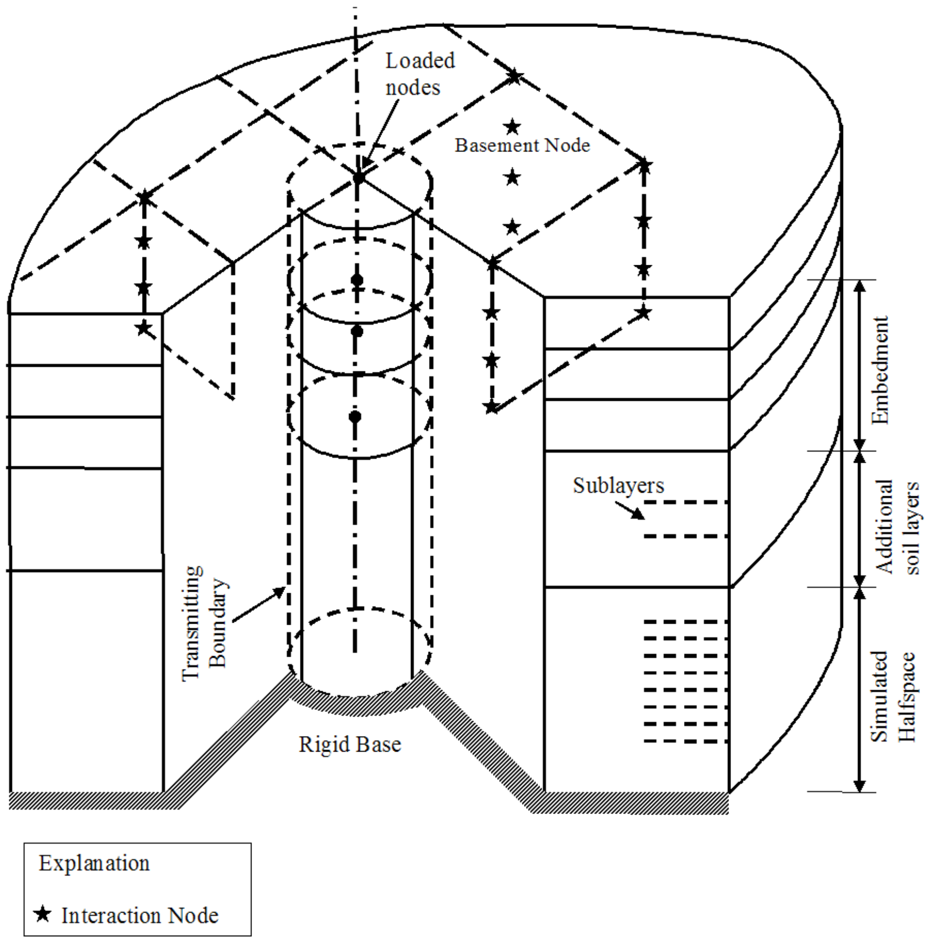
•Generalized Rayleigh and Love waves eigen-solutions and Fourier expansion are used to compute the force-displacement relationship for site response:

$$\{P\}_m = [R]_m \{U\}_m$$

$$[R]_m = r_0 \{[A][\psi]_m [K^2] + ([D] - [E] + m[N][\phi]_m [K] - m(\frac{m+1}{2}[L] + [Q])[\psi]_m [W(r_0)]^{-1}\}$$

Compute Flexibility Matrix

For each node dof the flexibility is computed using an axisymmetric model that includes a central zone with radius of cylindrical elements enclosed by an axisymmetric transmitting boundary.



Impedance Analysis

Computational Steps:

1. Compute Flexibility Matrix

2. Compute Impedance Matrix using
   - Flexible Volume Method (uses all the interaction nodes)
   - Skin Method (more approximate, not V&V)
   - Flexible Interface Method (used just the excavated interface nodes)

3. Equivalent Global Impedances (Optional)

Flexible Volume/Interface Method

In this method, the flexibility matrix need be computed for all the interacting nodes using the methods described above.

The impedance matrix is obtained by inverting the flexibility matrix, i.e.,

$$\mathbf{X}_{ff} = \mathbf{F}_{ff}^{-1}$$

- The inversion of the matrix is computationally intensive and needs to be performed for every frequency of analysis.
- An efficient in-place inversion routine is used to invert the flexibility matrix which is a full matrix in the direct method of analysis.
- For total number of i interacting nodes, the resultant impedance matrix of the order of 3i x 3i for three-dimensional problems.

Global Impedances

- The ACS SASSI code computes also the equivalent global impedance functions for the surface foundations.

- The global impedance functions are determined through a rigid body transformation.

$$K_{\theta,Y}(\omega) = \sum_i (X_i - X_C) \sum_j (X_j - X_C) k_{i,j}(\omega)$$

- Frequency dependent damping ratios corresponding to the equivalent global impedances are computed by analogy with a viscously damped SDOF system.

For a degree of freedom m, m = x, y, z, xx, yy, zz, the damping ratio is:

$$\xi_m(\omega) = \frac{\mathrm{Imag}[K_m(\omega)]}{2\,\mathrm{Real}[K_m(\omega = 0)]}$$

Global Impedances for A Circular Rigid Disk

Soil Layering

S O I L   L A Y E R   D A T A

N	H	W	VS	VP	DS	DP	G
1	0.1000E+02	0.1300E+00	0.9800E+03	0.2500E+04	0.1400E-01	0.1400E-01	0.3877E+04
2	0.1000E+02	0.1300E+00	0.9267E+03	0.2500E+04	0.2700E-01	0.2700E-01	0.3467E+04
3	0.1000E+02	0.1300E+00	0.8699E+03	0.2500E+04	0.3800E-01	0.3800E-01	0.3055E+04
4	0.1000E+02	0.1300E+00	0.8222E+03	0.2500E+04	0.4700E-01	0.4700E-01	0.2729E+04
H		0.1300E+00	0.1000E+04	0.2500E+04	0.5000E-01	0.5000E-01	0.4037E+04

IMPEDANCE ANALYSIS RESULTS:

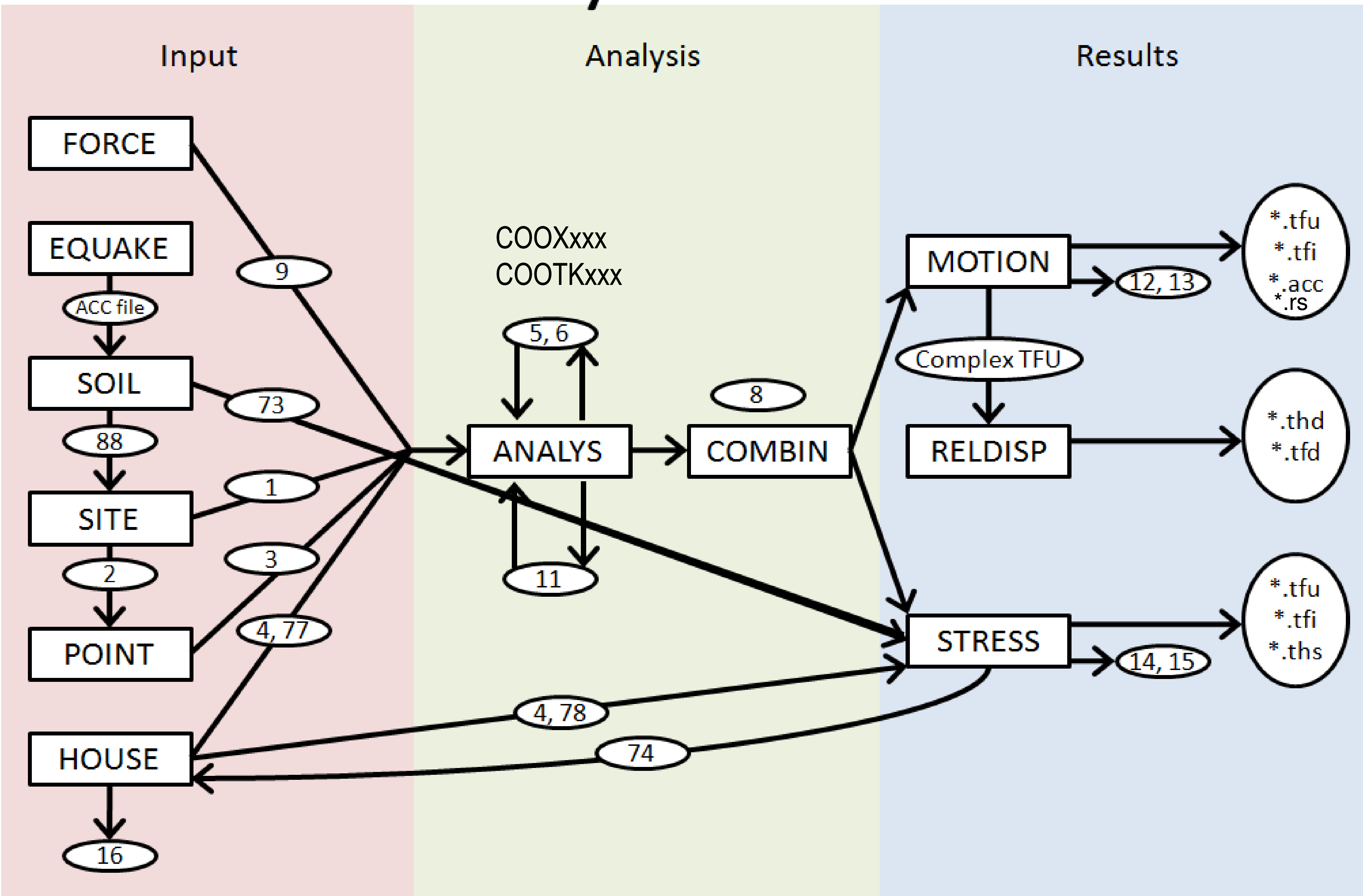
Global Impedances

	DYN. STIFF.	VISC. DAMP.	DAMP. RATIO
FREQUENCY = .05			
X	0.12333E+07	0.29910E+06	.04
Y	0.12333E+07	0.29910E+06	.04
Z	0.17374E+07	0.57989E+06	.05
XX	0.47310E+10	0.10835E+10	.04
YY	0.47310E+10	0.10835E+10	.04
ZZ	0.64420E+10	0.97004E+09	.02
FREQUENCY = .98			
X	0.12229E+07	58515.	.15
Y	0.12229E+07	58515.	.15
Z	0.16677E+07	0.10748E+06	.19
XX	0.44906E+10	0.63635E+08	.04
YY	0.44906E+10	0.63635E+08	.04
ZZ	0.62360E+10	0.58461E+08	.03
FREQUENCY = 1.95			
X	0.11923E+07	51694.	.26
Y	0.11923E+07	51694.	.26
Z	0.15560E+07	96036.	.34
XX	0.40767E+10	0.55045E+08	.07
YY	0.40767E+10	0.55045E+08	.07
ZZ	0.57881E+10	0.54443E+08	.05
FREQUENCY = 2.93			
X	0.11419E+07	50326.	.38
Y	0.11419E+07	50326.	.38
Z	0.13731E+07	93976.	.50
XX	0.36258E+10	0.63139E+08	.12
YY	0.36258E+10	0.63139E+08	.12
ZZ	0.52989E+10	0.67180E+08	.10
FREQUENCY = 4.35			
X	0.10439E+07	50586.	.56
Y	0.10439E+07	50586.	.56
Z	0.98731E+06	96869.	.76
XX	0.29810E+10	0.75676E+08	.22
YY	0.29810E+10	0.75676E+08	.22
ZZ	0.46914E+10	0.85966E+08	.18
FREQUENCY = 4.88			
X	0.99937E+06	51050.	.63
Y	0.99937E+06	51050.	.63
Z	0.80895E+06	99279.	.88
XX	0.27330E+10	0.79879E+08	.26
YY	0.27330E+10	0.79879E+08	.26
ZZ	0.45010E+10	0.92041E+08	.22

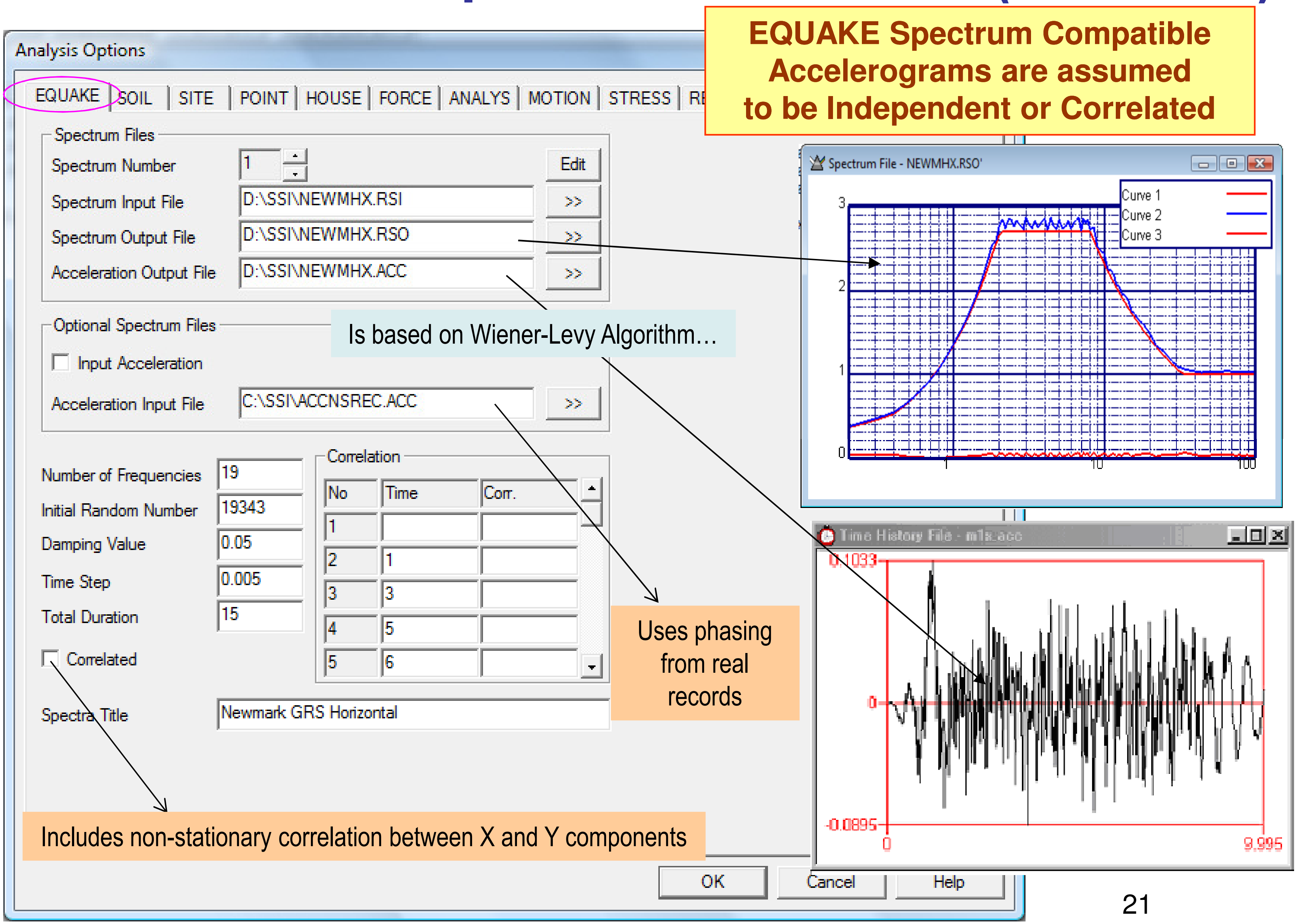
ACS SASSI SSI Modules:

1. **EQUAKE** – Generates 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. **HOUSE** – Defines the Structure and Near-Field Soil and Incoherence
6. **ANALYS** – Compute Impedances & Solves SSI Problem (ATF solution)
7. **MOTION** – Computes Accelerations, RS in Structure/Near-Soil
8. **RELDISP** - Computes Relative Displacements
9. **STRESS** – Computes Stresses/Strains in Structure and Near-Soil
10. **COMBIN** – Combine ANALYS Solutions with Different Frequencies

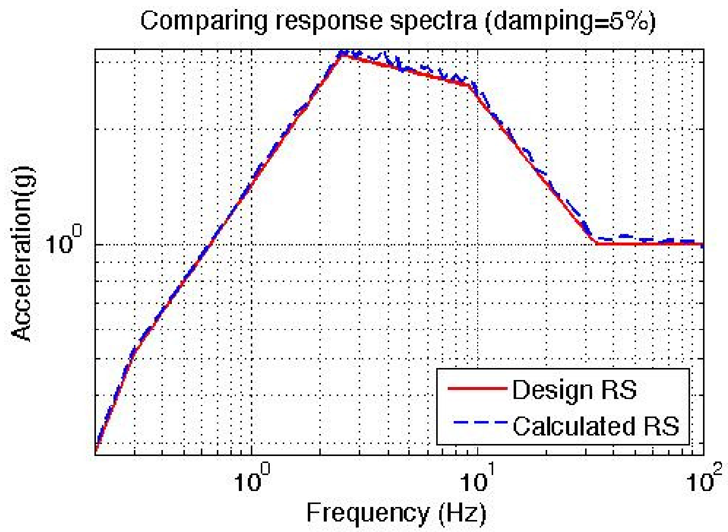
SSI Analysis Flowchart

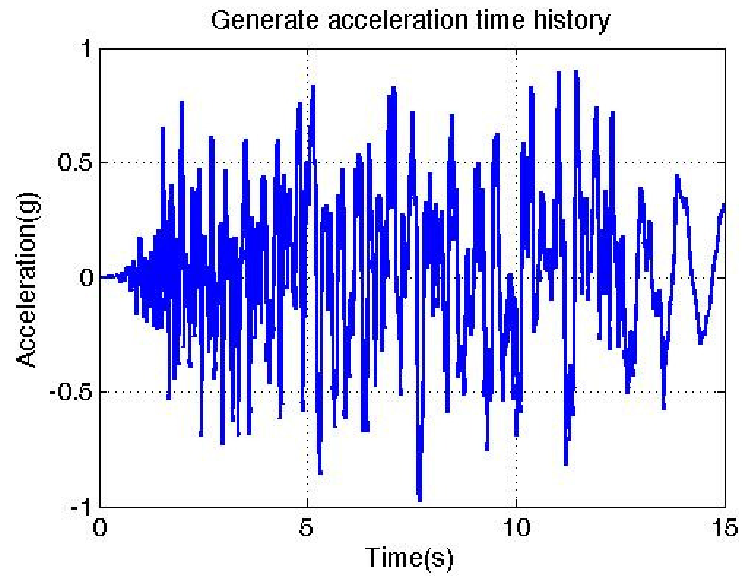


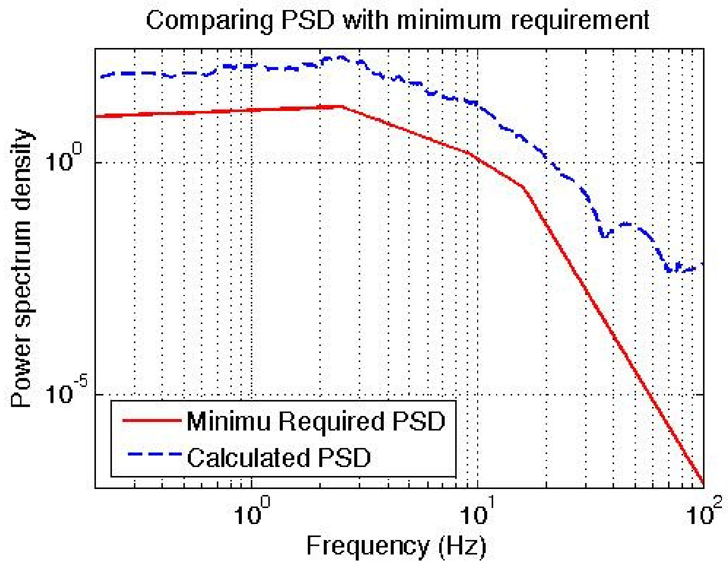
1. Simulation of Input Control Motion (EQUAKE)



RG 1.60 Spectrum Compatible Accelerograms Using Random Phasing

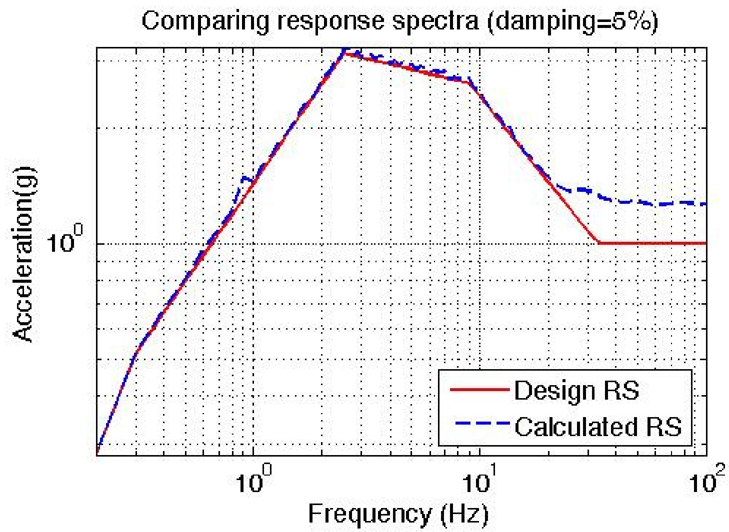


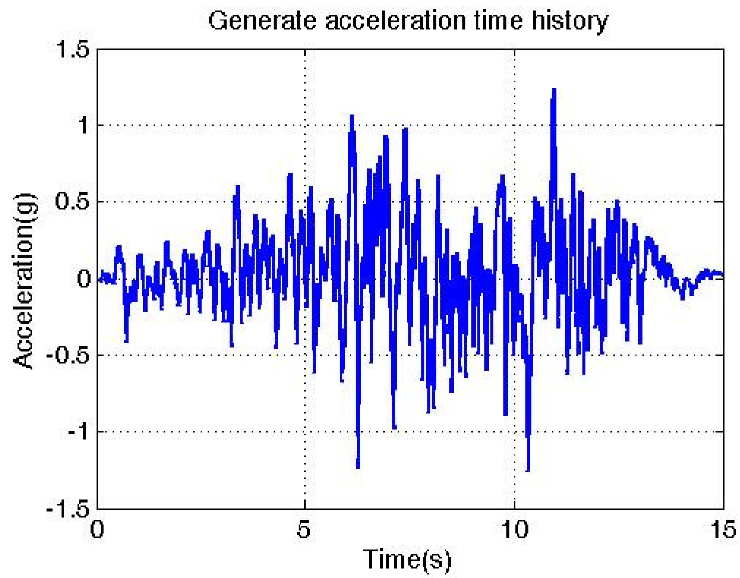




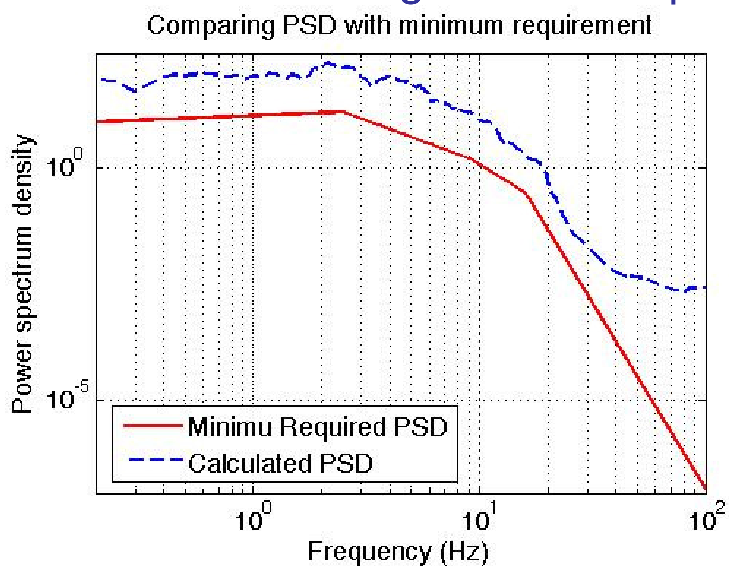
US NRC SRP 3.7.1 Target PSD Requirement

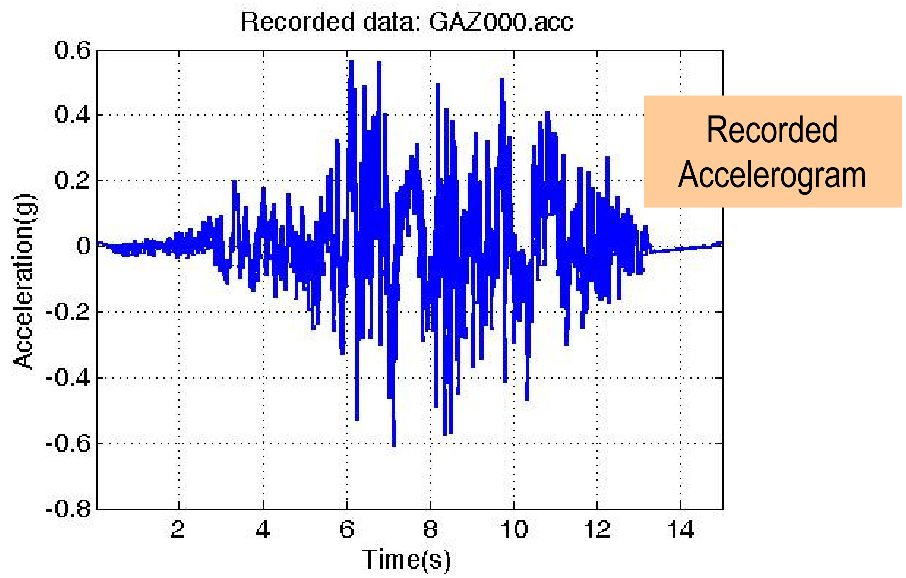
RG 1.60 Spectrum Compatible Accelerograms Using Recorded Phasing



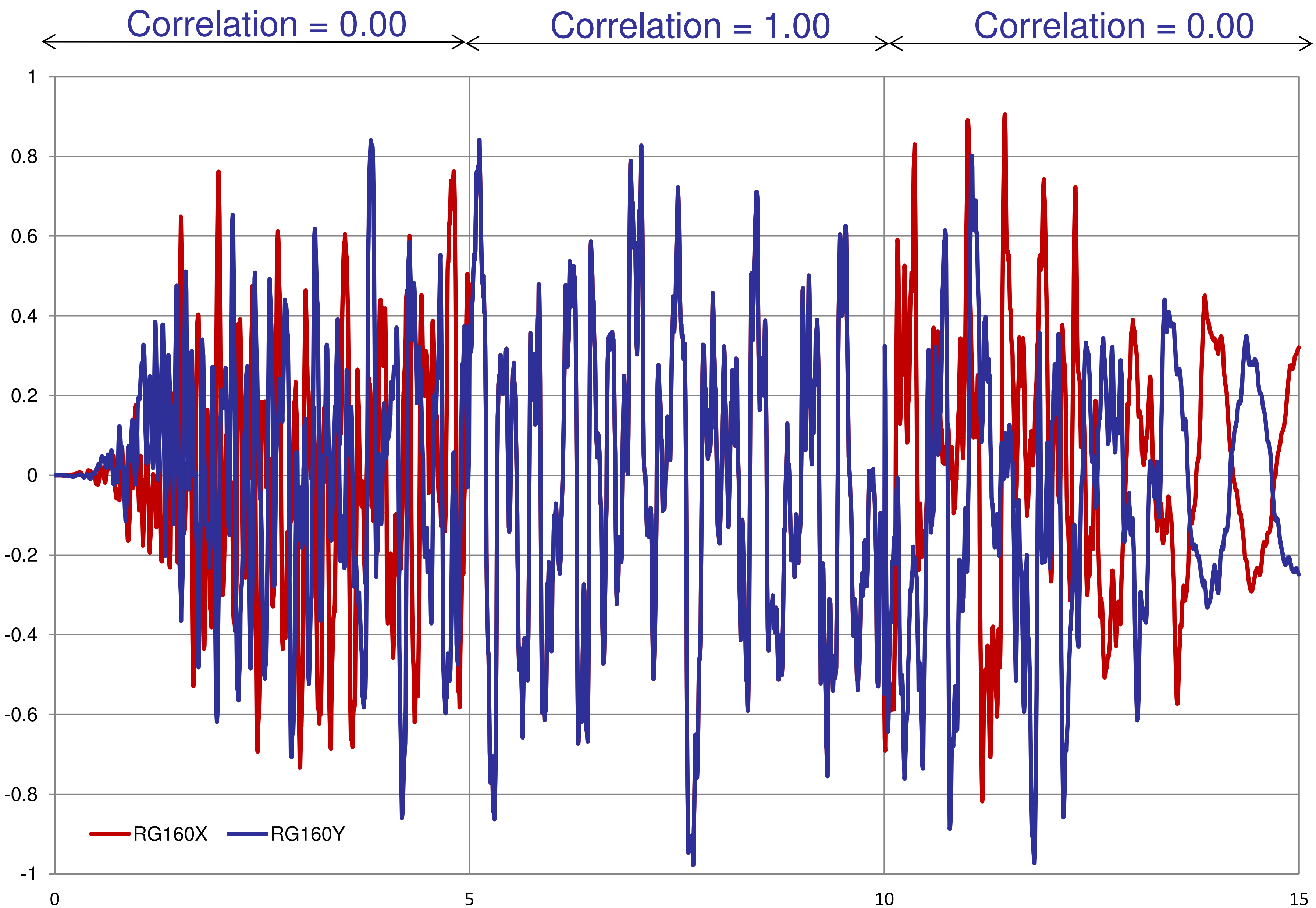


US NRC SRP 3.7.1 Target PSD Requirement

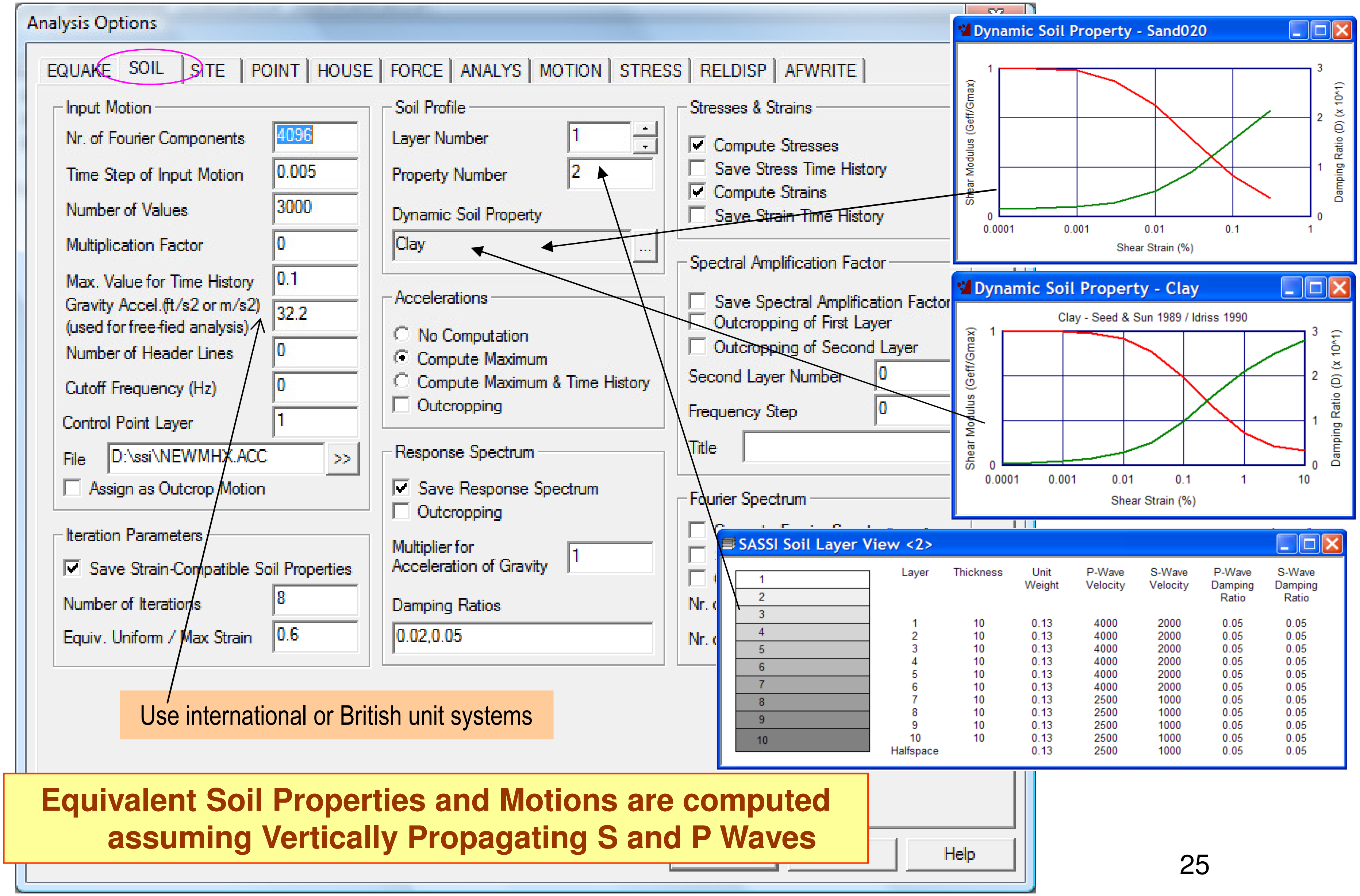


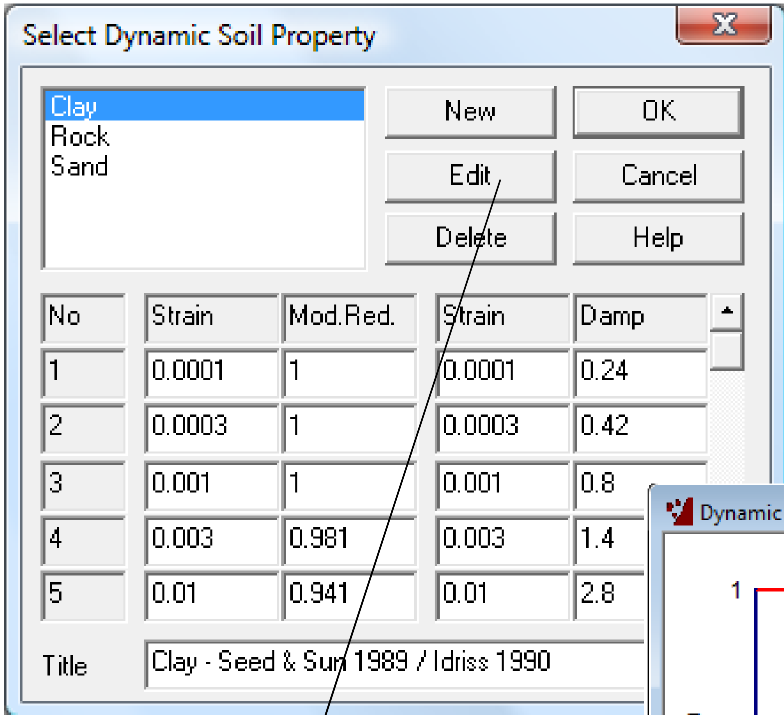


Spectrum Compatible Accelerograms with Nonstationary Correlation

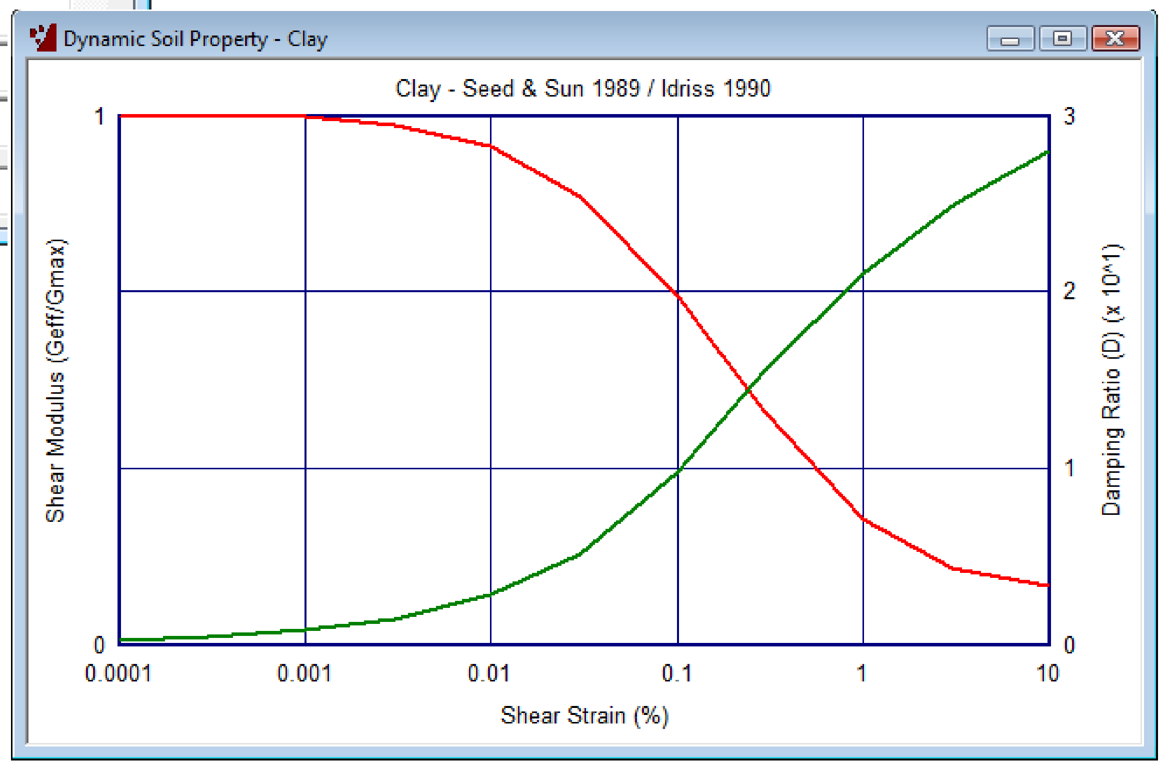


2. Nonlinear Site Response Analysis (SOIL)





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.
```

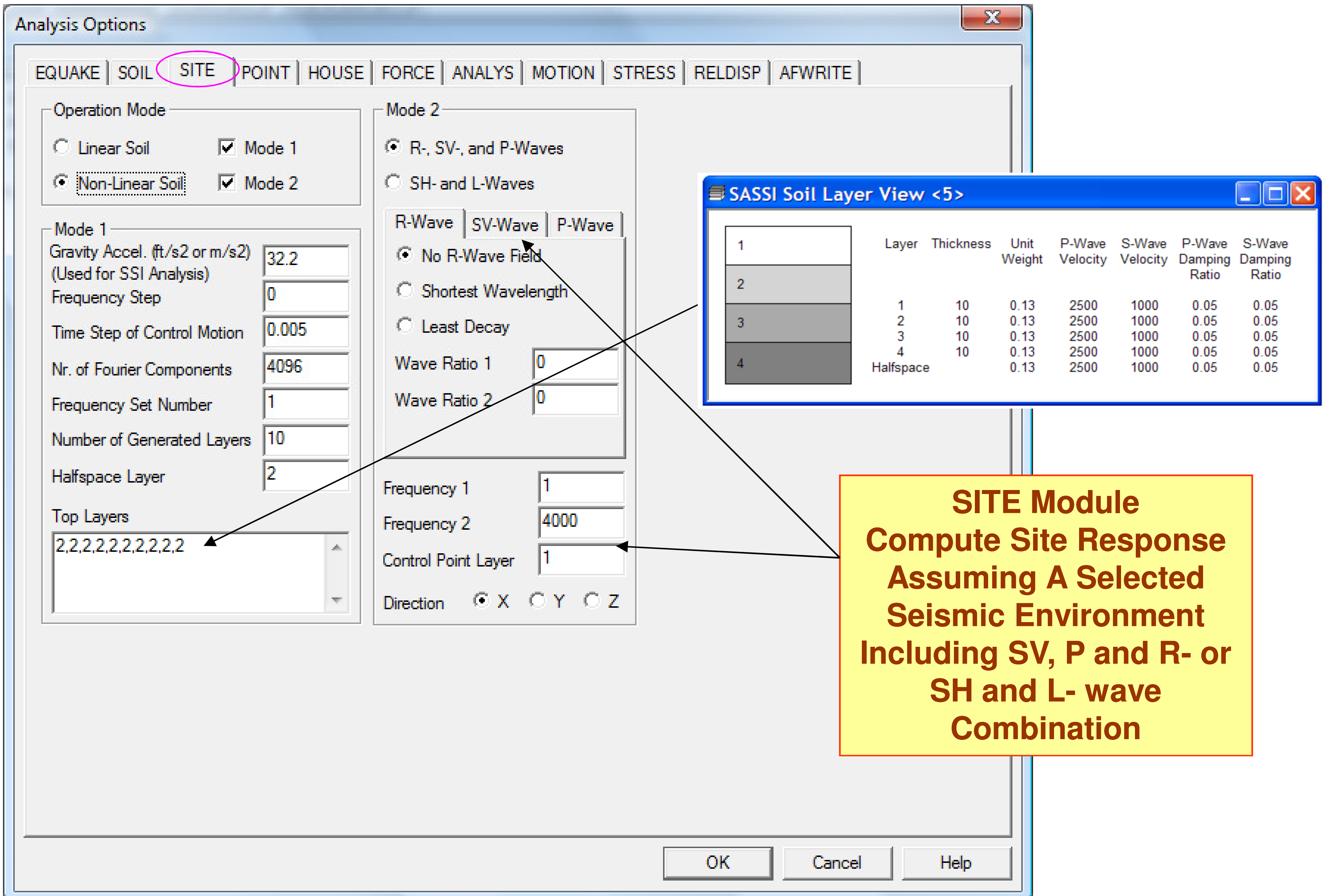
Final Soil Layering Properties

```
ITERATION NUMBER  8

VALUES IN TIME DOMAIN
```

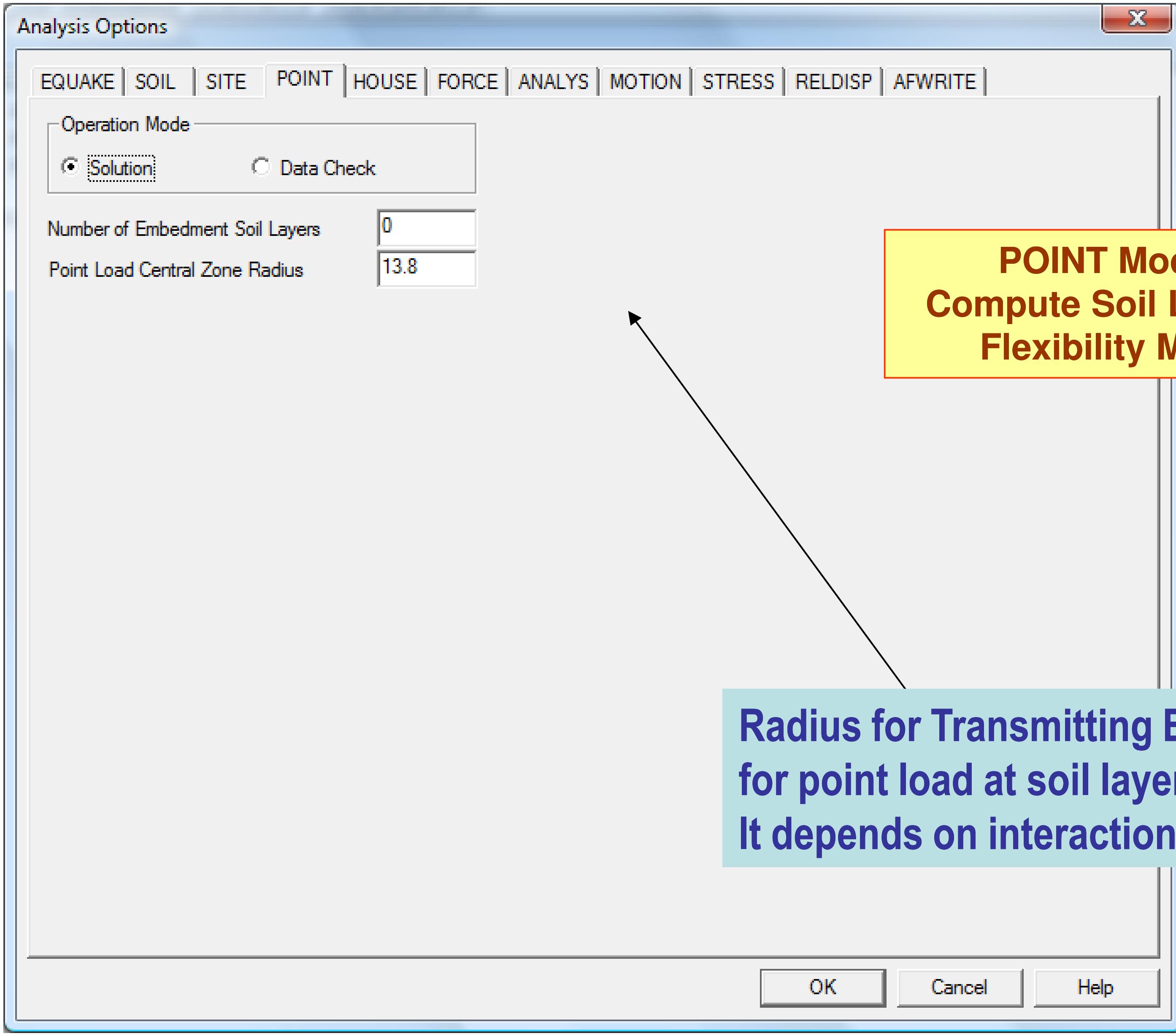
NO	TYPE	DEPTH (FT)	UNIFRM. STRAIN	<---- DAMPING NEW	USED	----> ERROR	<---- SHEAR MODULUS NEW	USED	-----> ERROR	G/Go RATIO
1	1	5.0	.00296	.014	.014	.0	3877.2	3877.2	.0	.960
2	1	15.0	.00909	.027	.027	0.0	3466.9	3466.9	.0	.859
3	1	25.0	.01629	.038	.038	0.0	3055.3	3055.3	0.0	.757
4	1	35.0	.02485	.047	.047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

```
PERIOD =   .18 FOR AVERAGE SHEAR VELOCITY =    900.
```

Selection of Seismic Wave Environment (SITE)



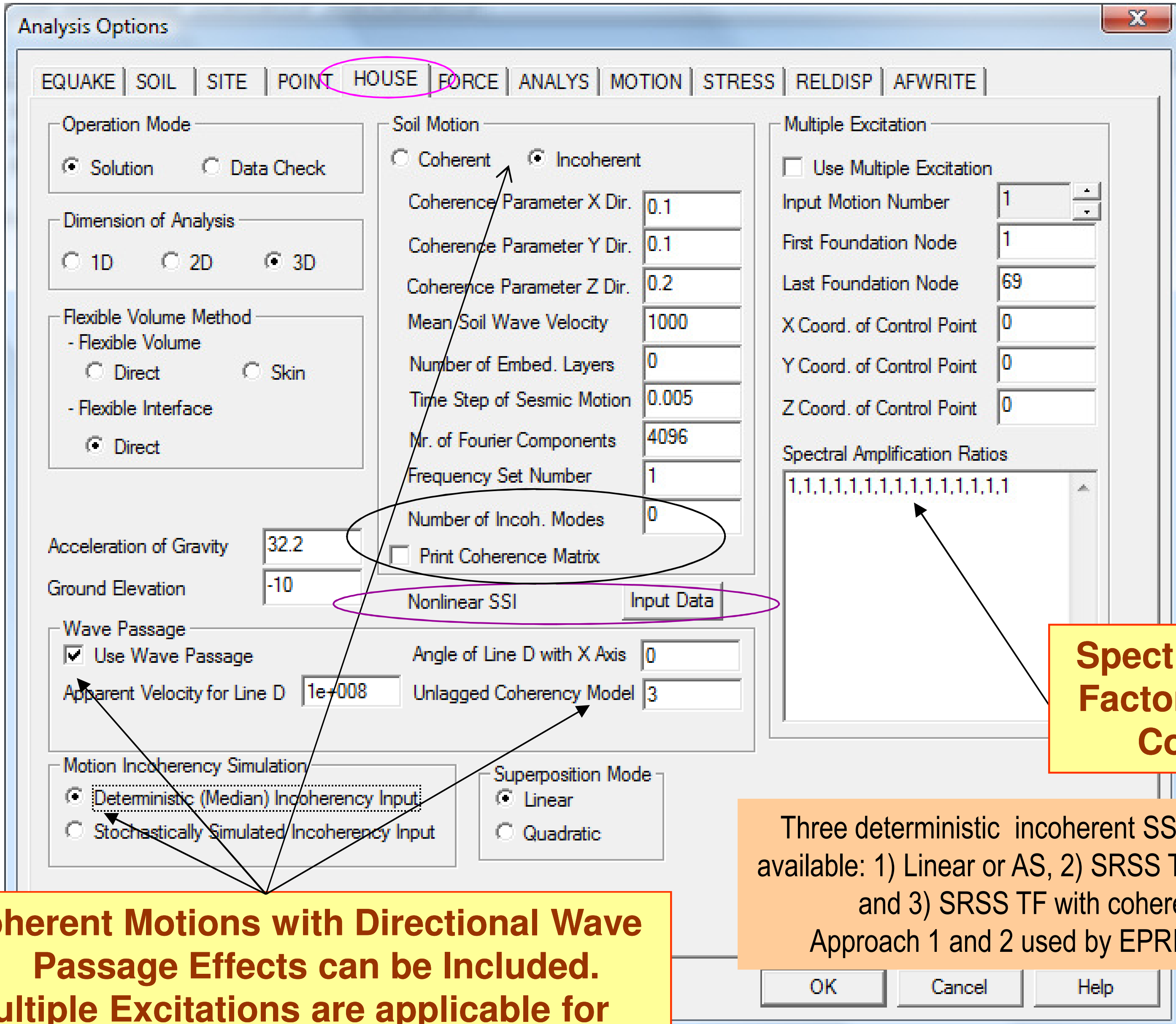
Input for Computing Soil Flexibility Matrix (POINT)



POINT Module
Compute Soil Layering Flexibility Matrix

Radius for Transmitting Boundary for point load at soil layer interface. It depends on interaction node mesh.

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.

ACS SASSI Plane-Wave Incoherency Models

6 plane-wave incoherency models incorporated in the code:

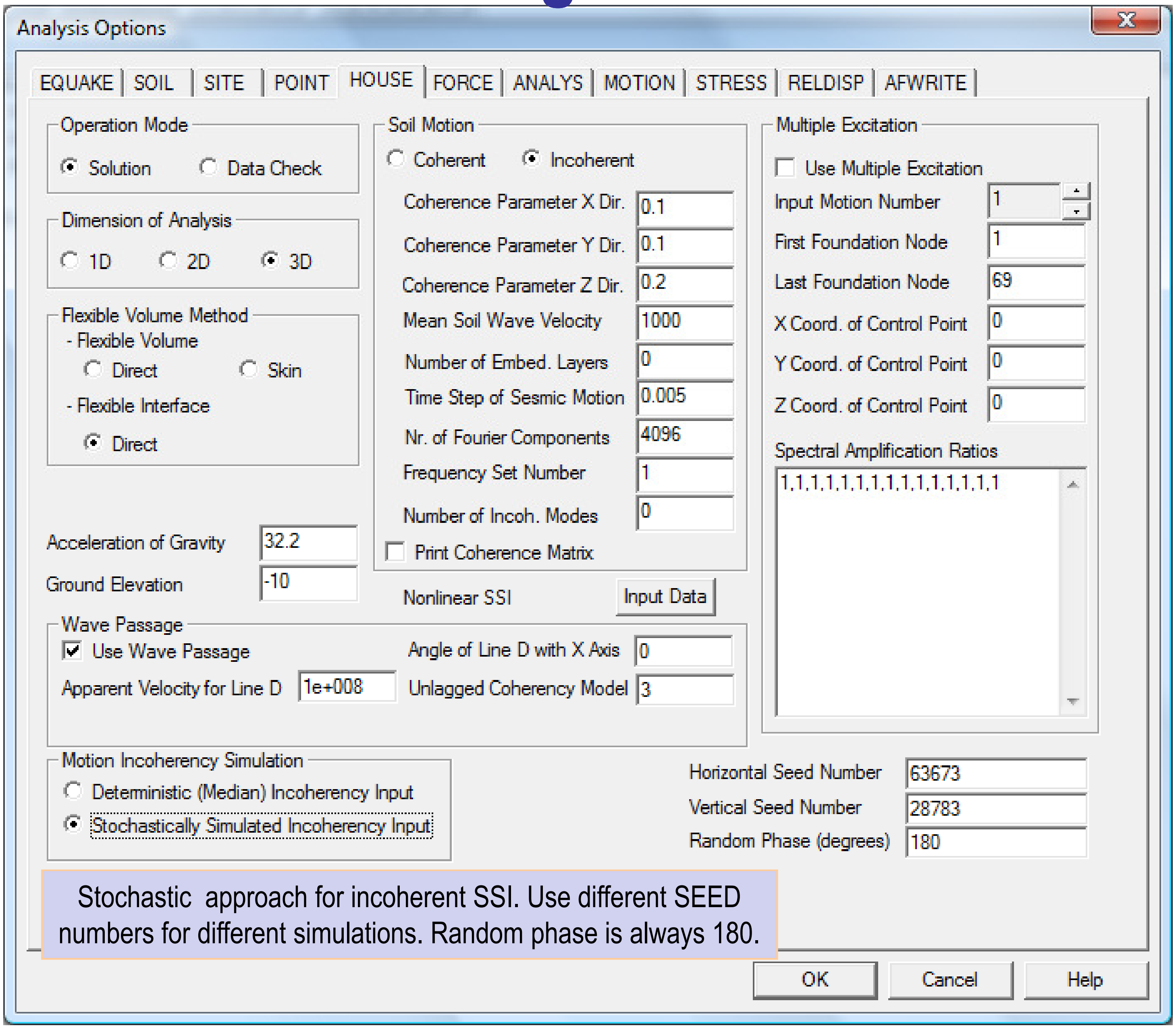
HOUSE Input:

1) For Luco-Wong model, 1986 (theoretical, but unvalidated model)

2) For 1993 Abrahamson model for all sites and surface foundations
3) For 2005 Abrahamson model for all sites and surface foundations
4) For 2006 Abrahamson model for all sites and embedded foundations
5) For 2007 Abrahamson model for hard-rock sites and all foundations (NRC)
6) For 2007 Abrahamson model for soil sites and surface foundations

NOTE:
It should be noted that at this time only the 2007 Abrahamson for hard-rock site conditions is permitted by US NRC.

Incoherent SSI Using Stochastic Simulation



Stochastic approach for incoherent SSI. Use different SEED numbers for different simulations. Random phase is always 180.

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.

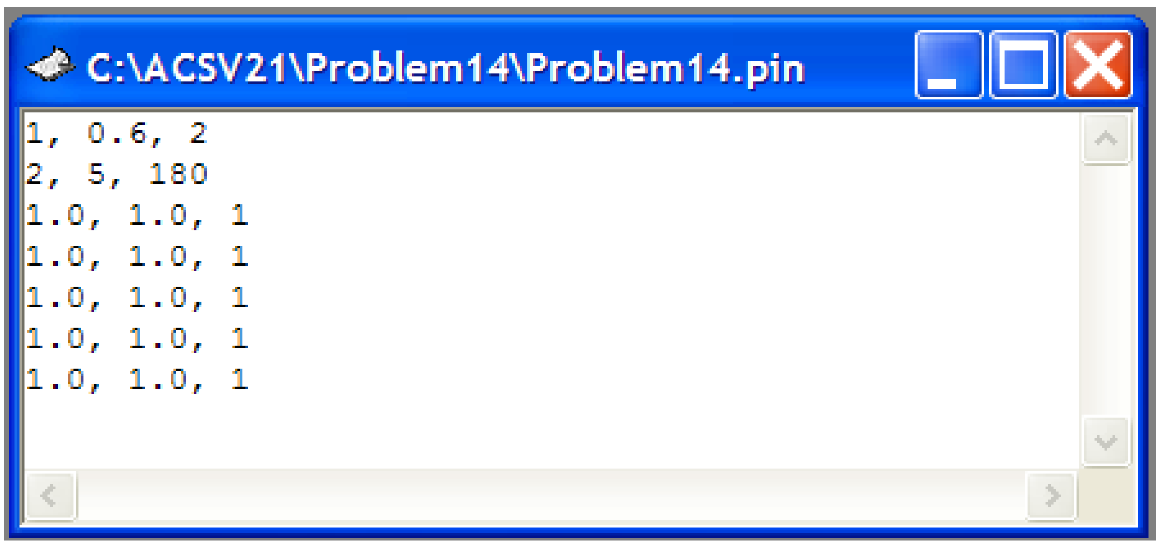
The block of lines after 1st line, needs to be input for all nonlinear soil element groups.

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.

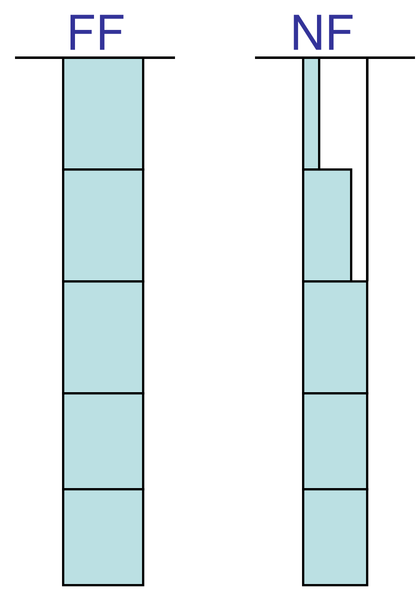
The order number of the nonlinear soil group is 2, the number of soil materials in the group is 5, and total number of elements in the group is 180.

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).

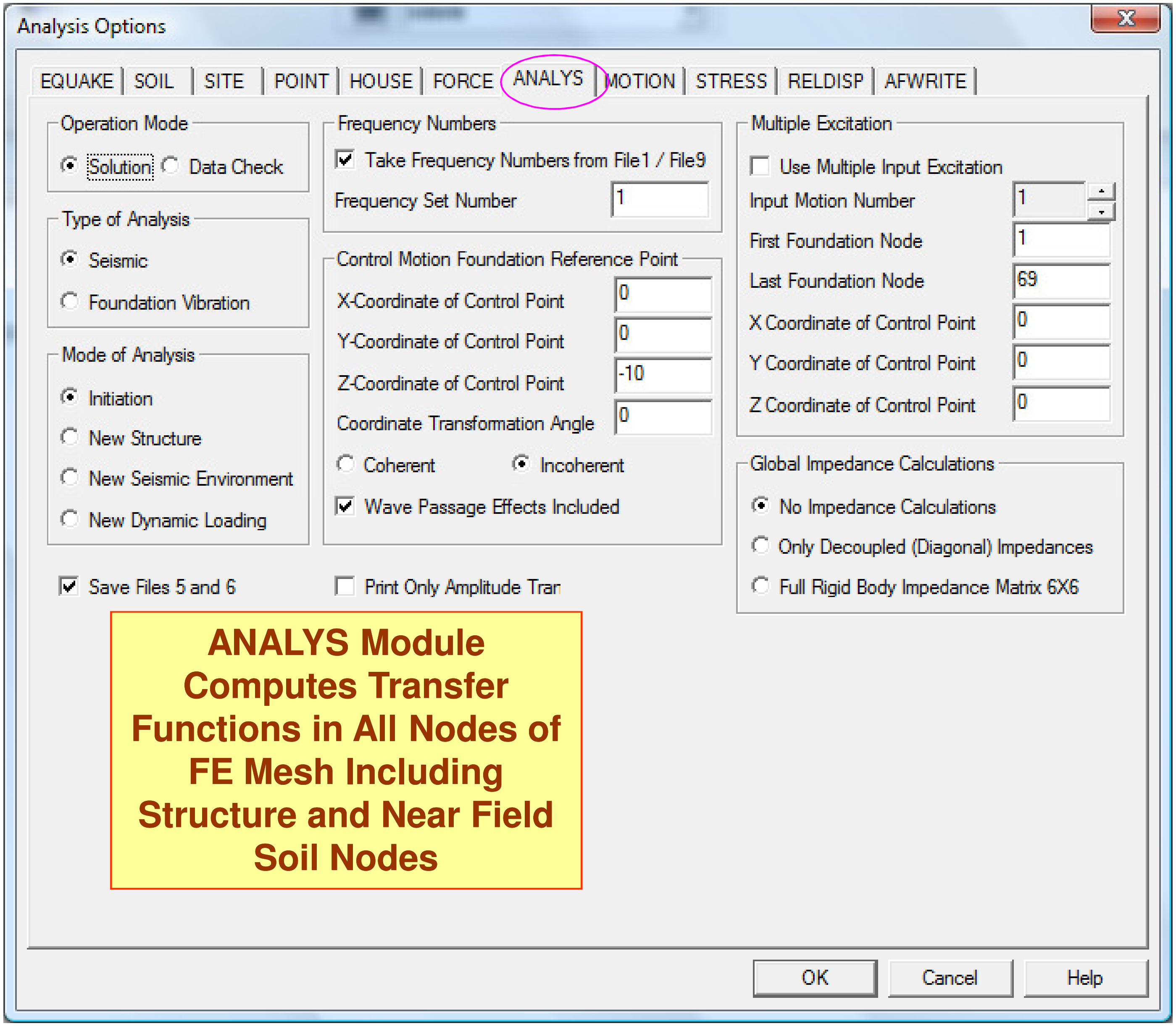


C:\ACSV21\Problem14\Problem14.pin

```
1, 0.6, 2
2, 5, 180
1.0, 1.0, 1
1.0, 1.0, 1
1.0, 1.0, 1
1.0, 1.0, 1
1.0, 1.0, 1
```

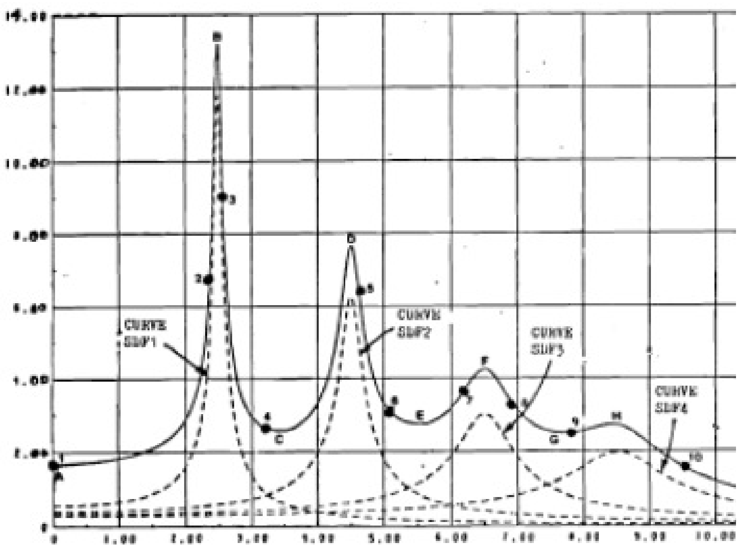


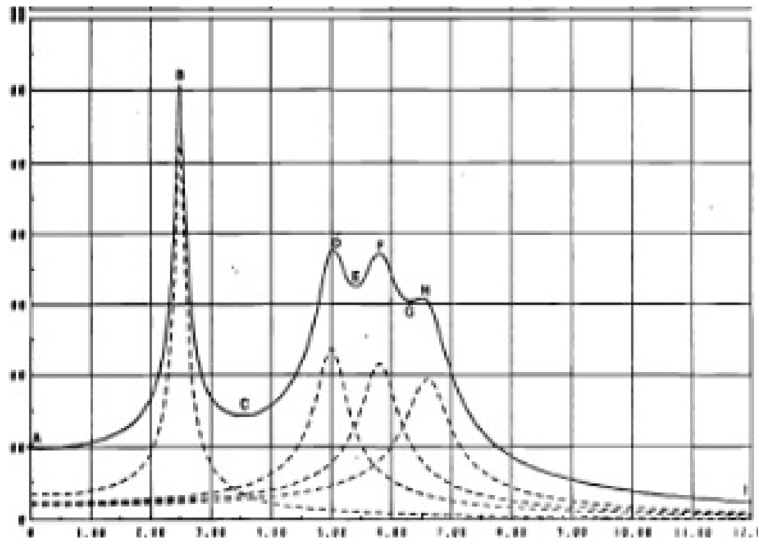
Performing Seismic SSI Analysis (ANALYS)

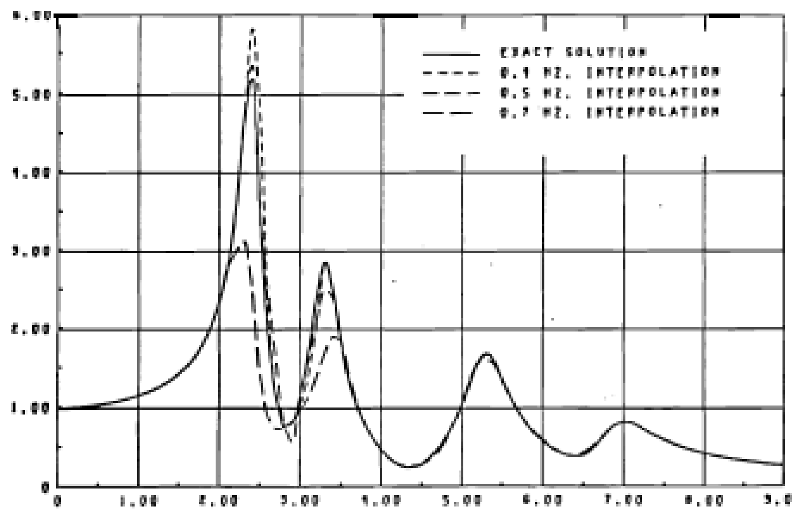


ANALYS Module Computes Transfer Functions in All Nodes of FE Mesh Including Structure and Near Field Soil Nodes

Transfer Function Interpolation Technique







(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

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

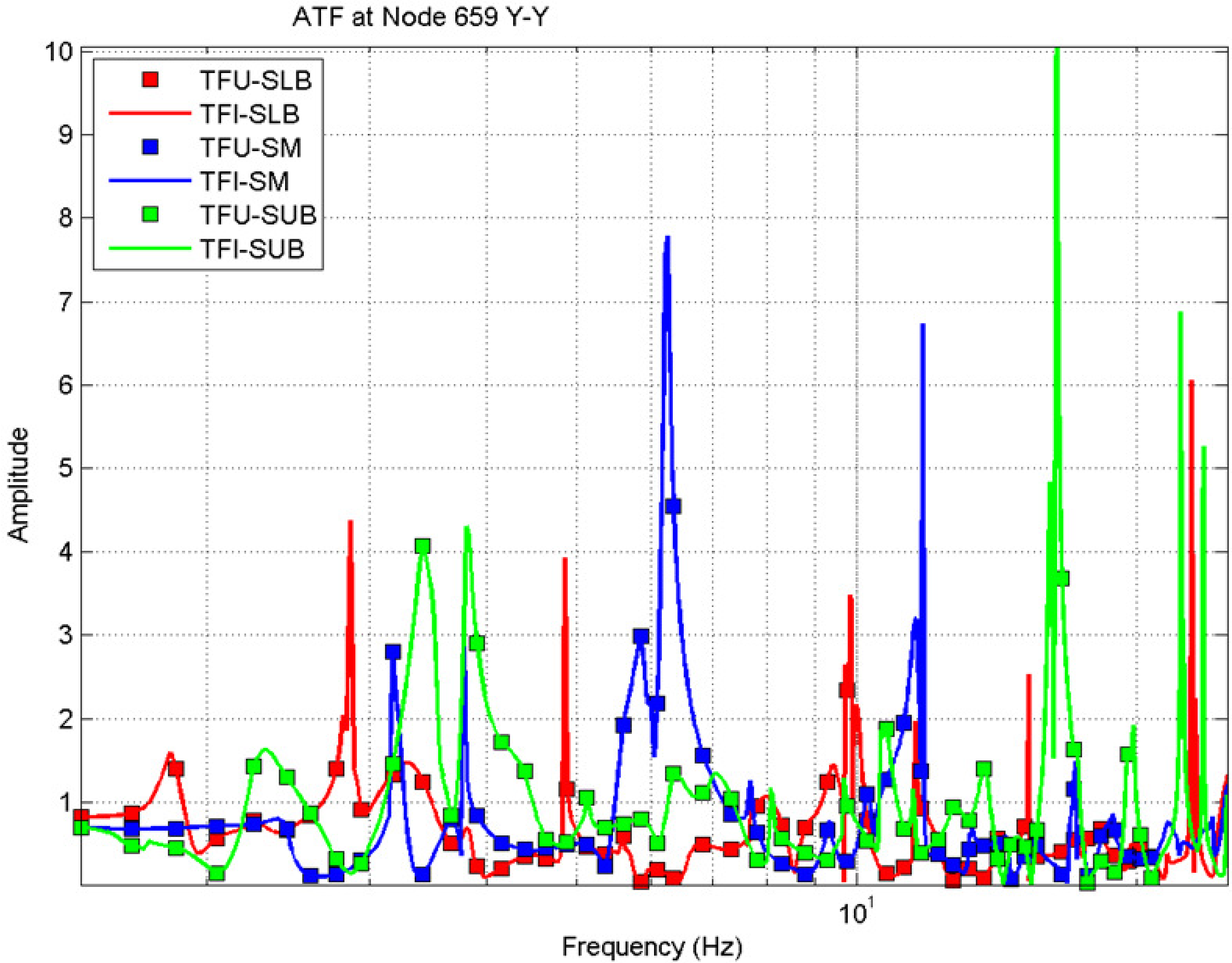
To compute the complex coefficients a five equation system needs to be solved

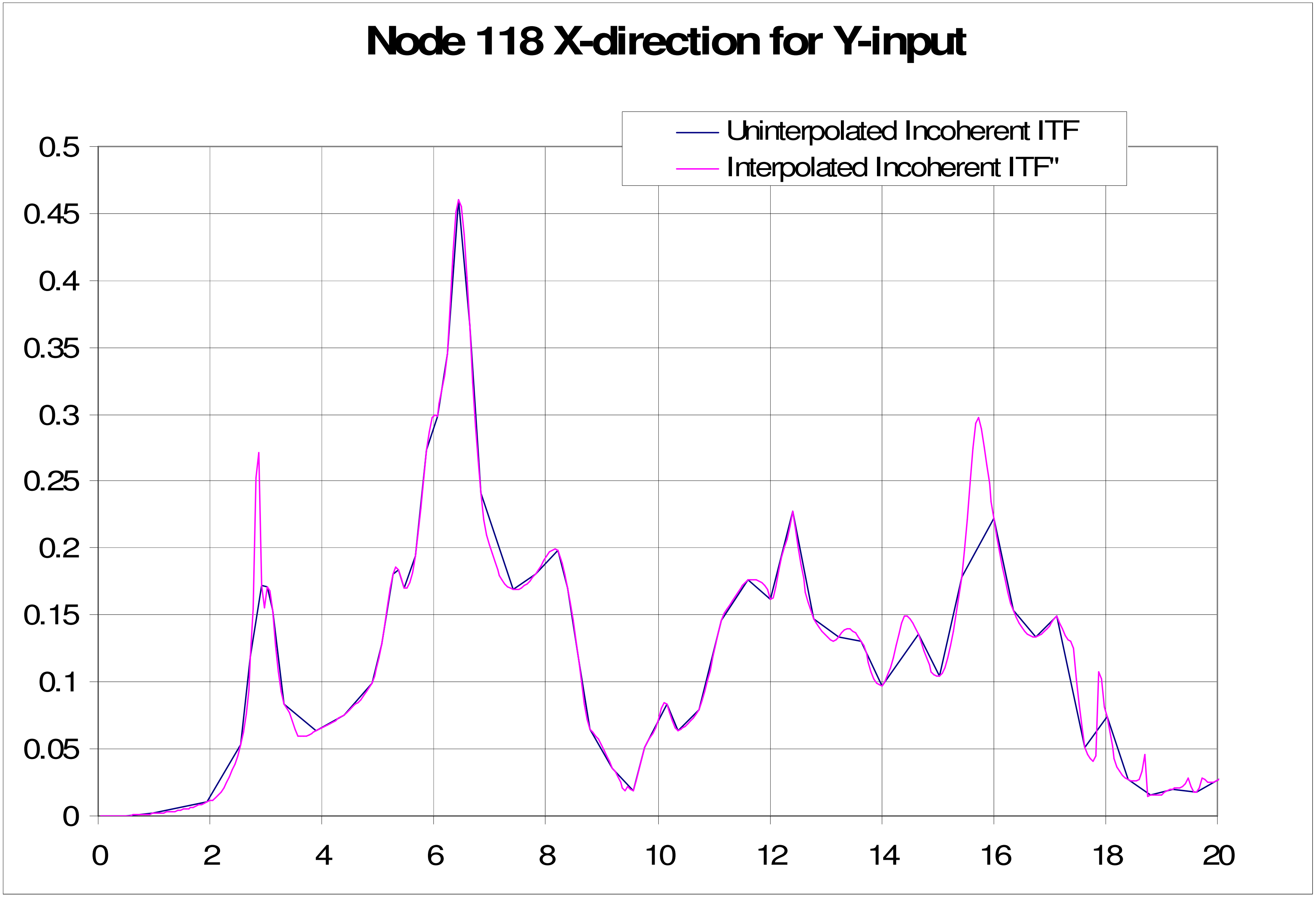
$$\begin{bmatrix} \omega_1^4 & \omega_1^2 & 1 & -\omega_1^2 U_1 & U_1 \\ \omega_2^4 & \omega_2^2 & 1 & -\omega_2^2 U_2 & U_2 \\ \omega_3^4 & \omega_3^2 & 1 & -\omega_3^2 U_3 & U_3 \\ \omega_4^4 & \omega_4^2 & 1 & -\omega_4^2 U_4 & U_4 \\ \omega_5^4 & \omega_5^2 & 1 & -\omega_5^2 U_5 & U_5 \end{bmatrix}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

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.

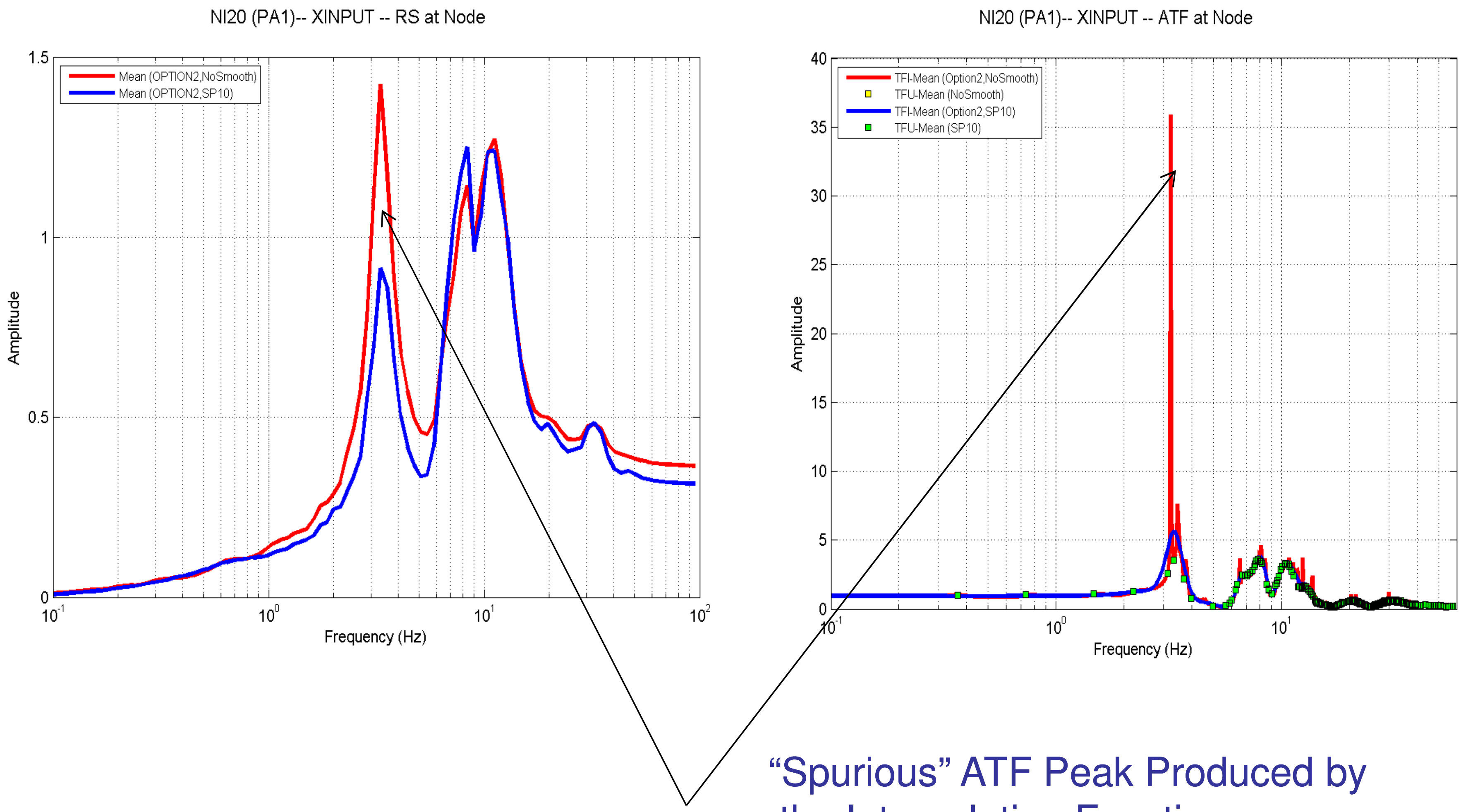
Criteria for Selecting Frequency Solution Points

- 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 50-200 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.


ATF at Node 659 Y-Y
TFU-SLB
TFI-SLB
TFU-SM
TFI-SM
TFU-SUB
TFI-SUB
Amplitude
Frequency (Hz)
10
9
8
7
6
5
4
3
2
1
10^1

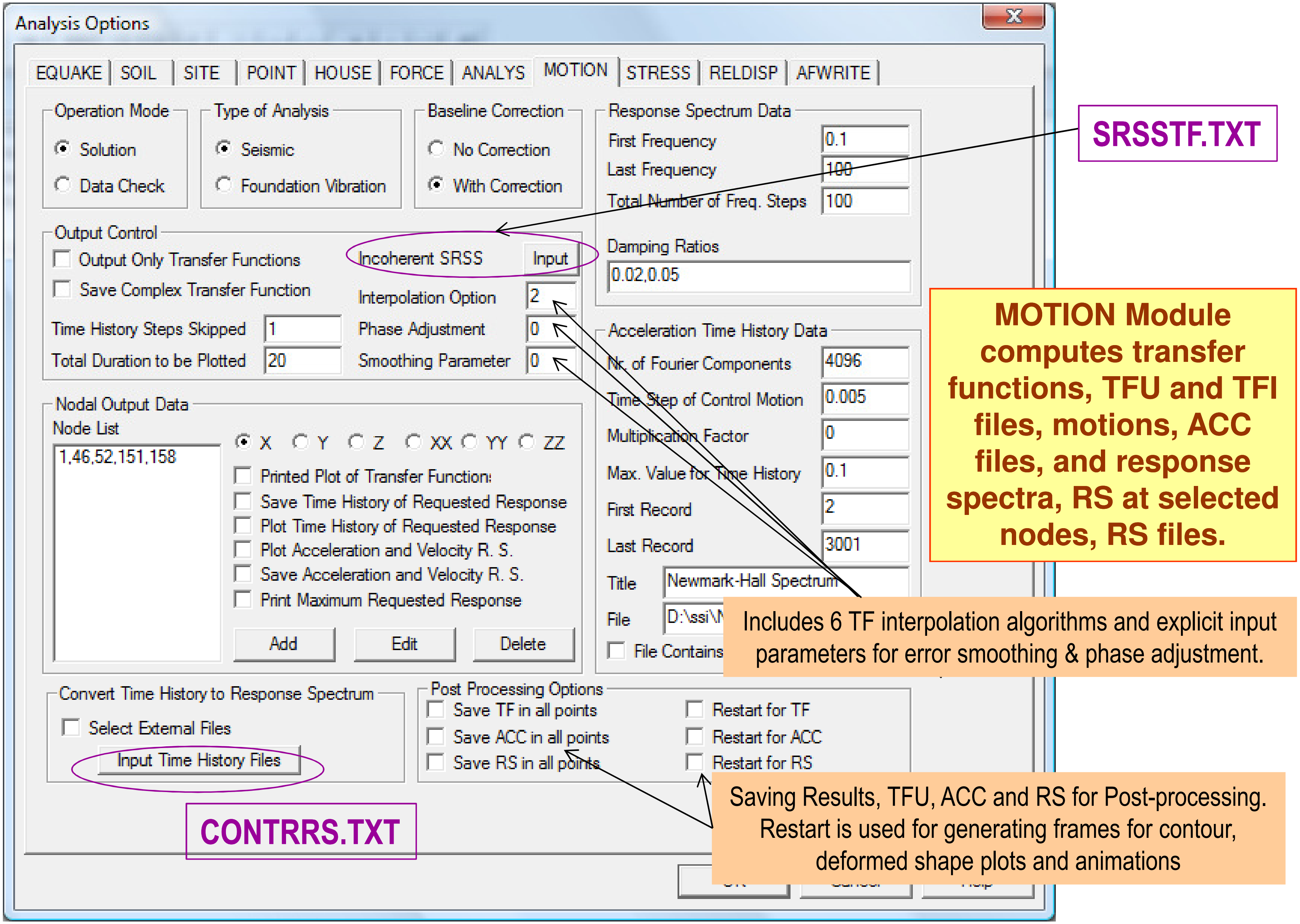

Node 118 X-direction for Y-input
Uninterpolated Incoherent ITF
Interpolated Incoherent ITF"
0.5
0.45
0.4
0.35
0.3
0.25
0.2
0.15
0.1
0.05
0
0
2
4
6
8
10
12
14
16
18
20

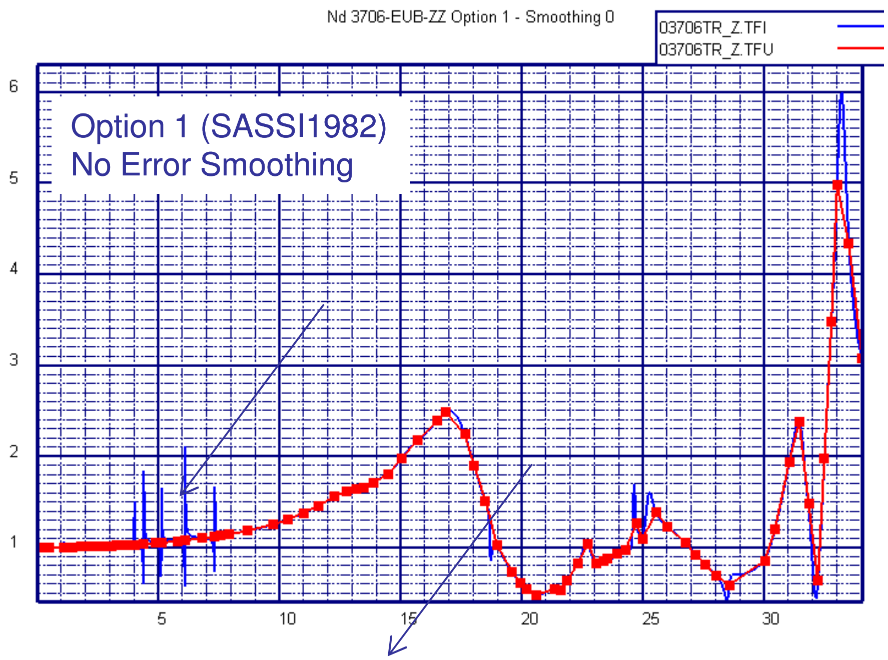
ATF Interpolation Error Smoothing Results; No Smoothing vs. Smoothing For Interpolated ATF. Need to Correlate RS and ATF Results

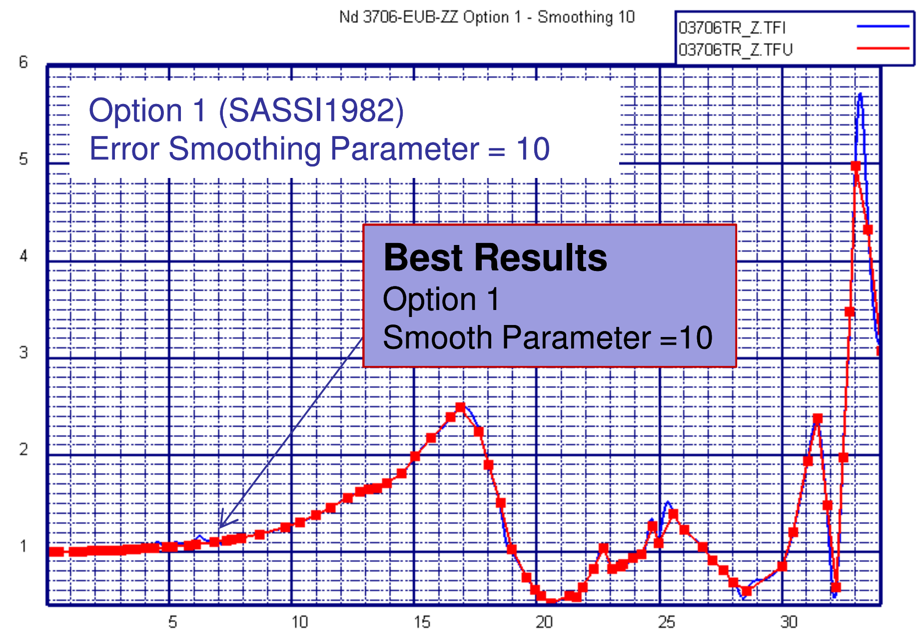


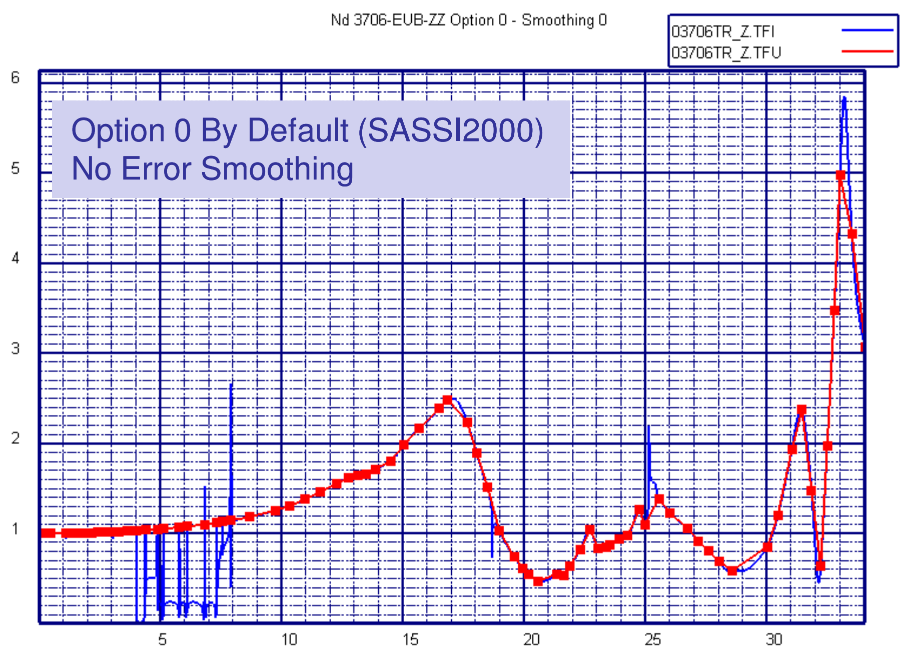
"Spurious" ATF Peak Produced by the Interpolation Function

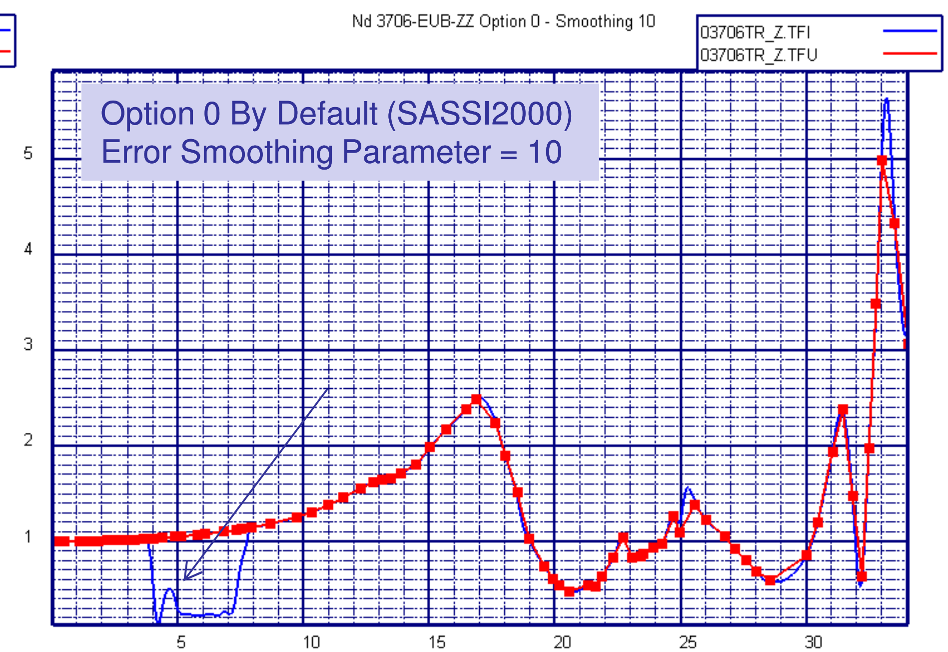
Computing Nodal Accelerations (MOTION)

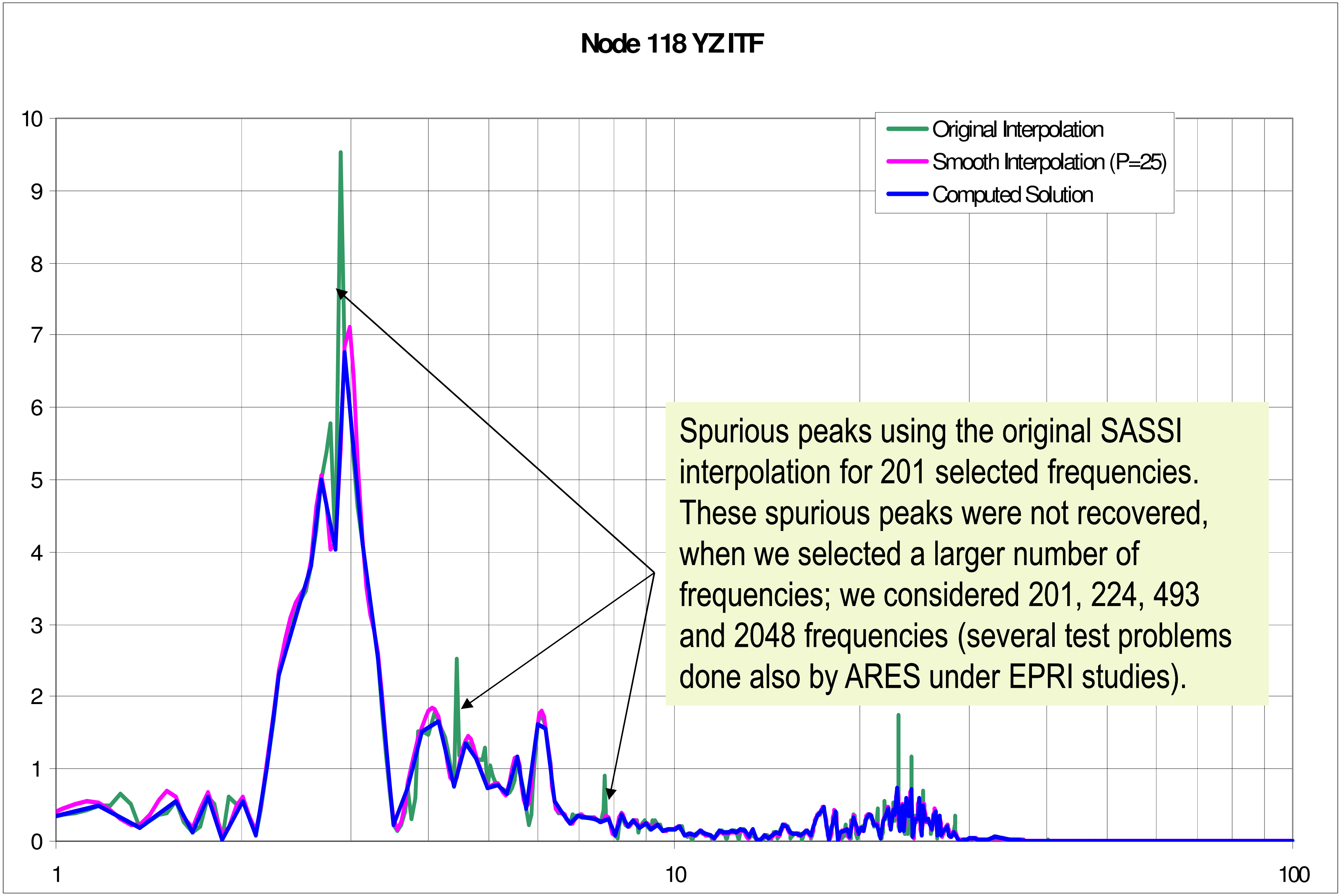



Nd 3706-EUB-ZZ Option 1 - Smoothing 0
03706TR_Z.TFI
03706TR_Z.TFU
Option 1 (SASSI1982)
No Error Smoothing

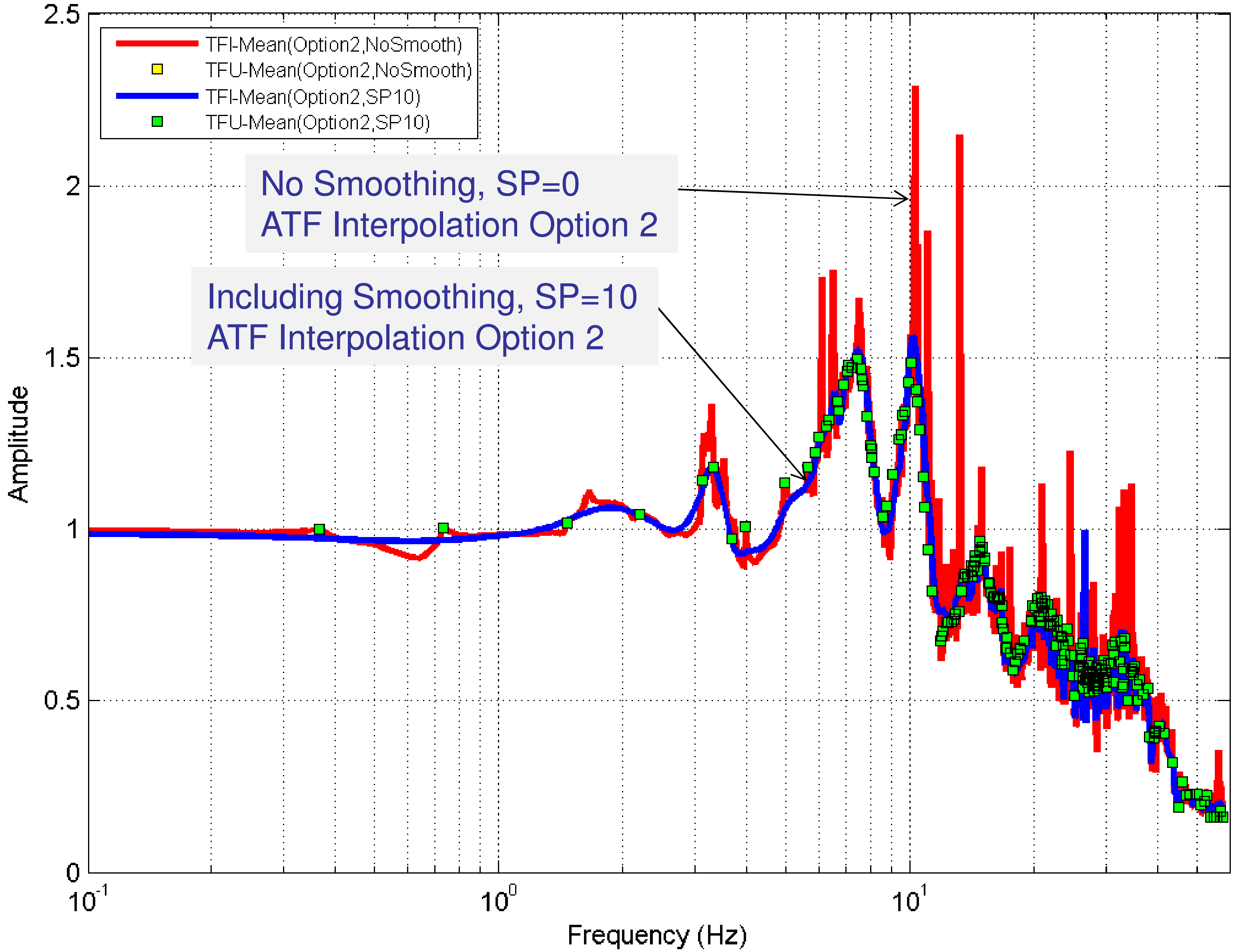

Nd 3706-EUB-ZZ Option 1 - Smoothing 10
03706TR_Z.TFI
03706TR_Z.TFU
Option 1 (SASSI1982)
Error Smoothing Parameter = 10
Best Results
Option 1
Smooth Parameter =10


Nd 3706-EUB-ZZ Option 0 - Smoothing 0
03706TR_Z.TFI
03706TR_Z.TFU
Option 0 By Default (SASSI2000)
No Error Smoothing

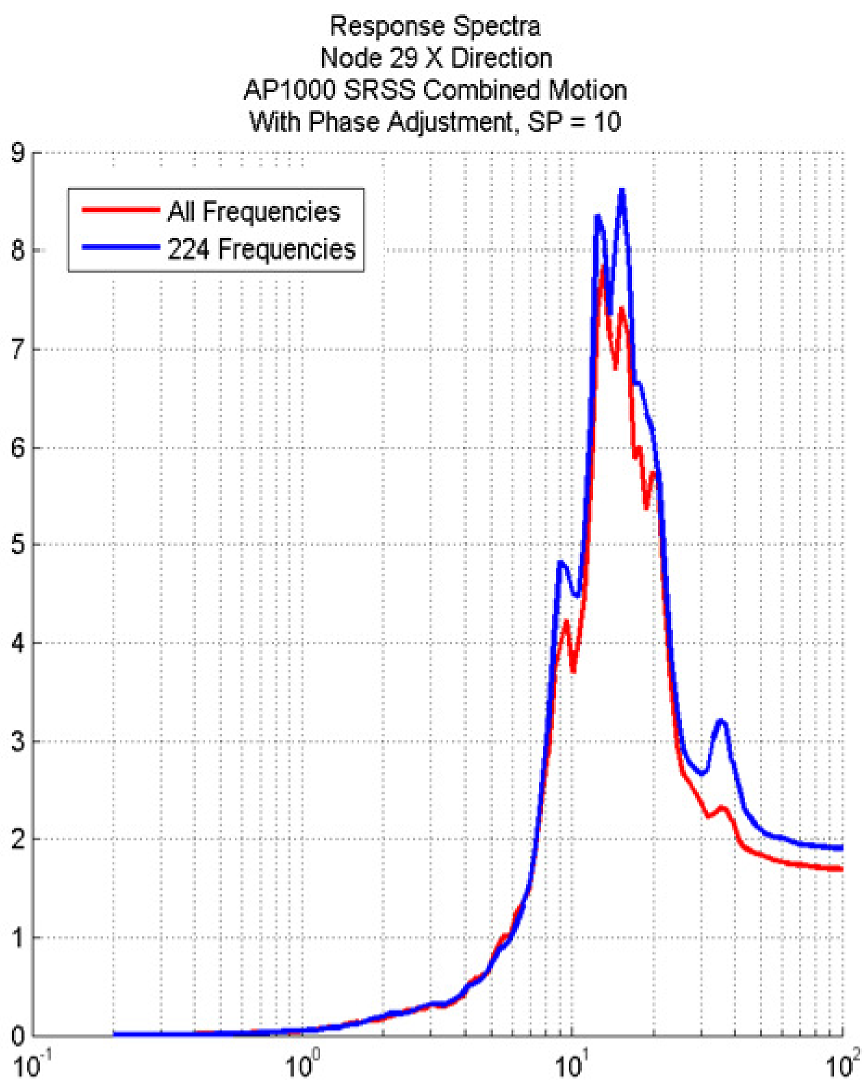

Nd 3706-EUB-ZZ Option 0 - Smoothing 10
03706TR_Z.TFI
03706TR_Z.TFU
Option 0 By Default (SASSI2000)
Error Smoothing Parameter = 10

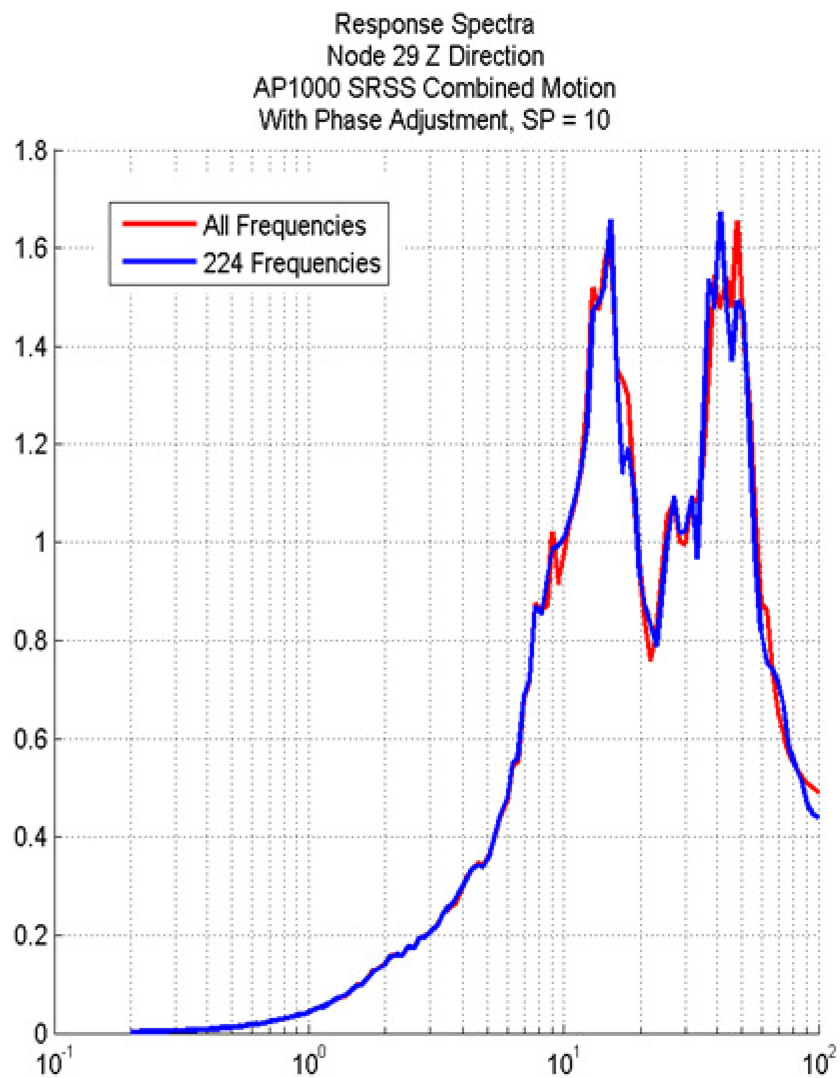

Node 118 YZ ITF
Original Interpolation
Smooth Interpolation (P=25)
Computed Solution
Spurious peaks using the original SASSI interpolation for 201 selected frequencies. These spurious peaks were not recovered, when we selected a larger number of frequencies; we considered 201, 224, 493 and 2048 frequencies (several test problems done also by ARES under EPRI studies).
10
9
8
7
6
5
4
3
2
1
0
1
10
100

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

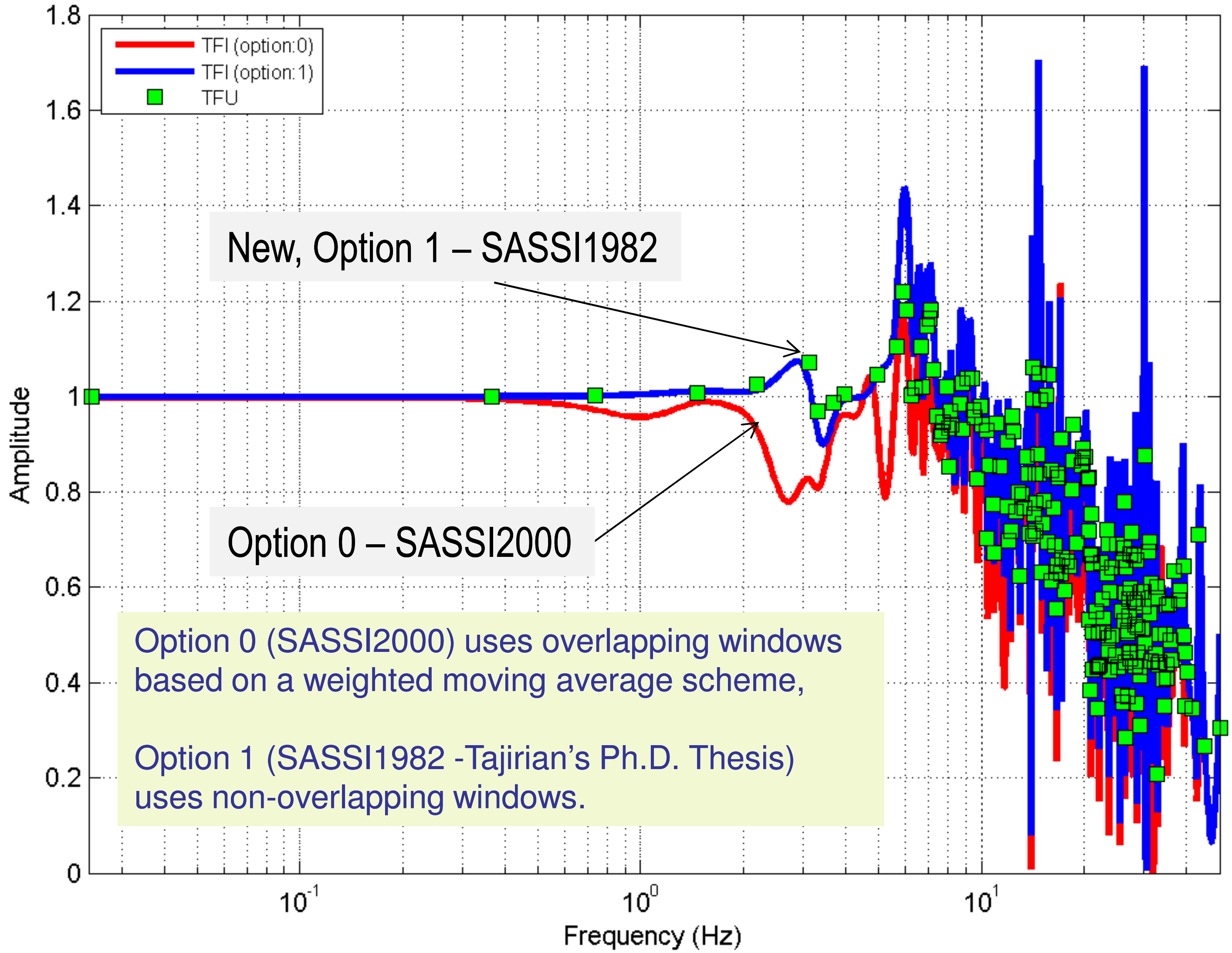


ATF Interpolation Error Smoothing Results for EPRI AP1000 Stick Model. Comparisons for 224 SSI Frequencies vs. 2048 Fourier Frequencies

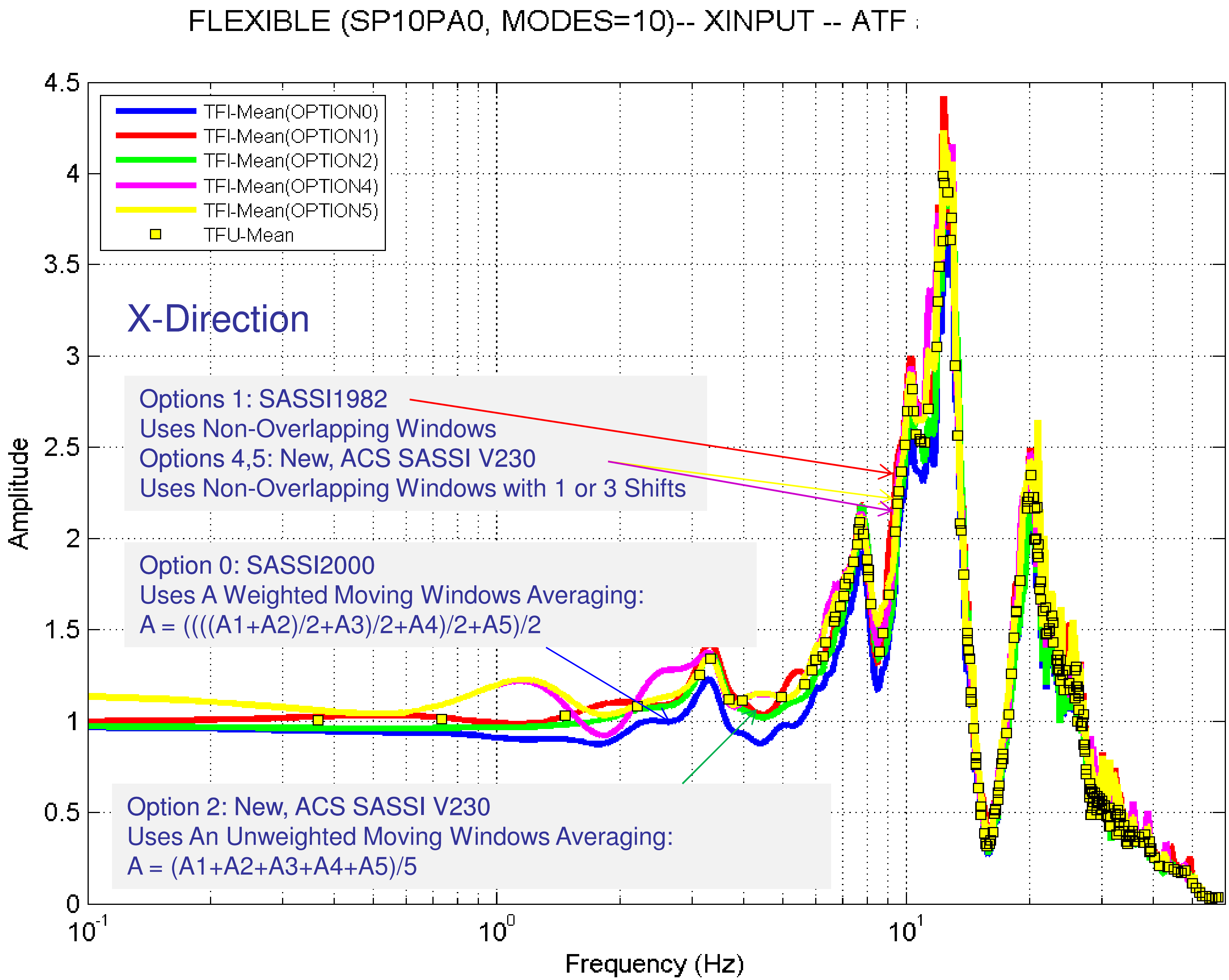




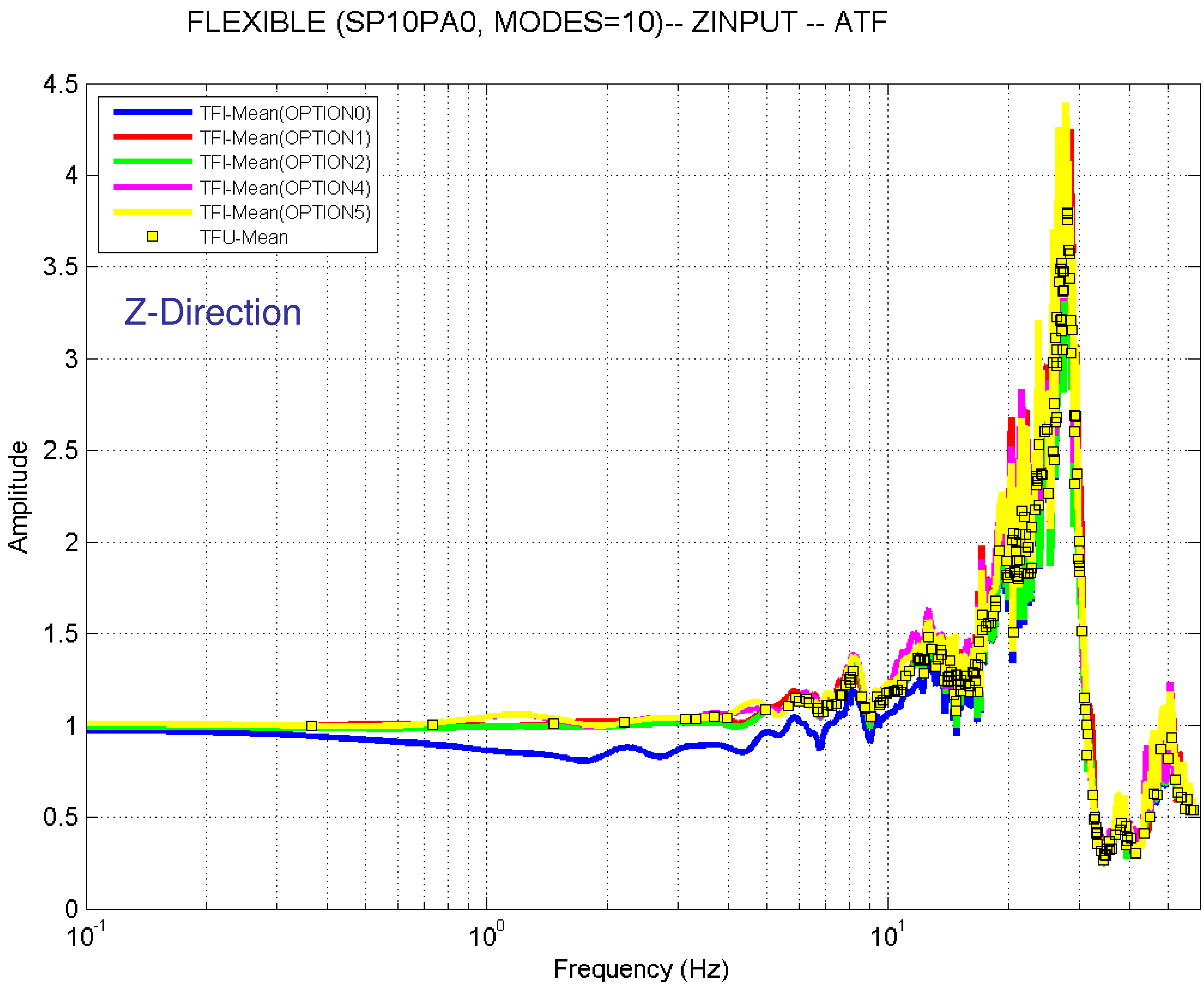
Simulated Incoherent Interpolated ATF Using SASSI2000 (Option 0) and SASSI1982 (Option 1) Interpolation Schemes



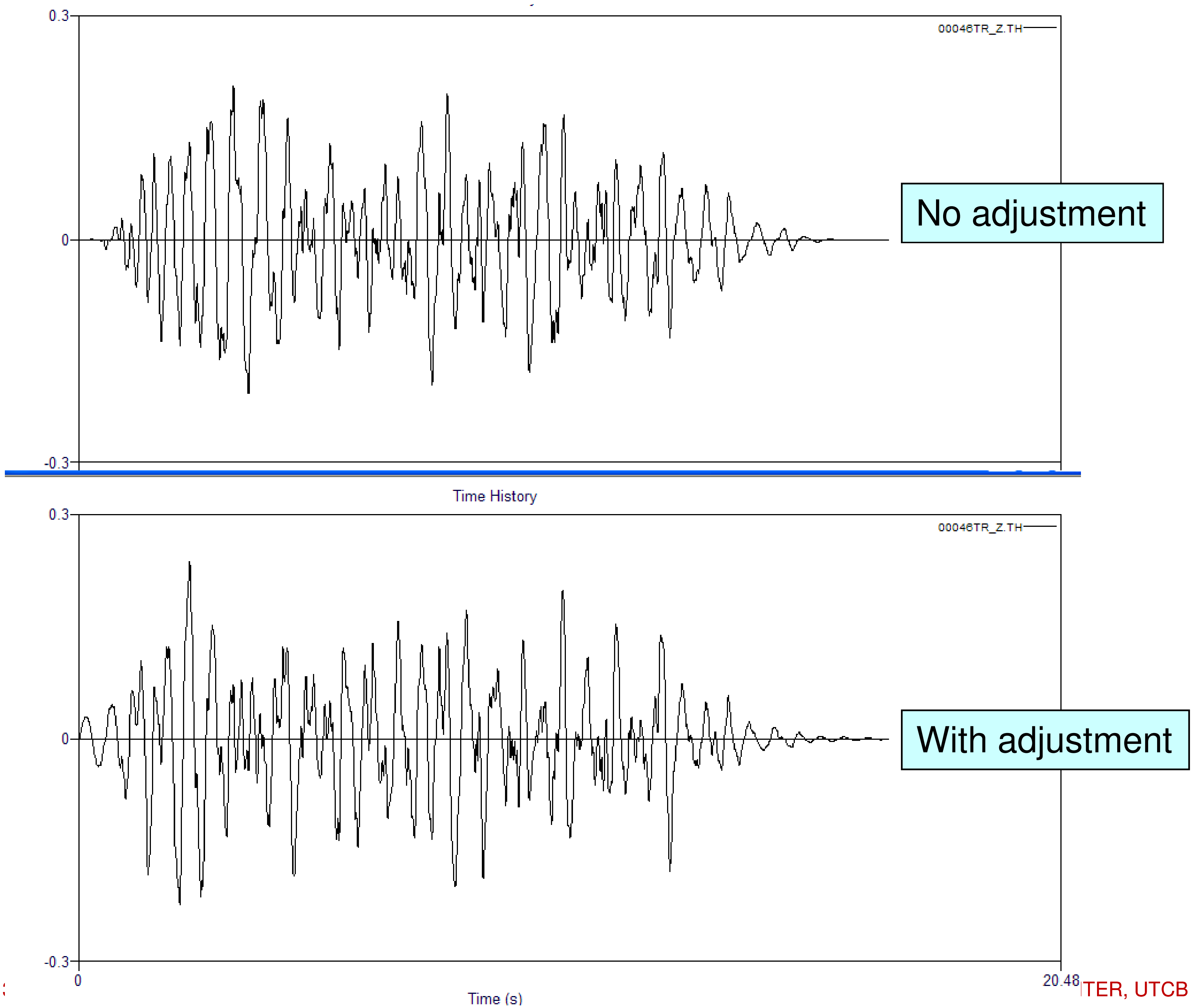
(Mean) Incoherent Interpolated ATF Using Different Interpolation Schemes



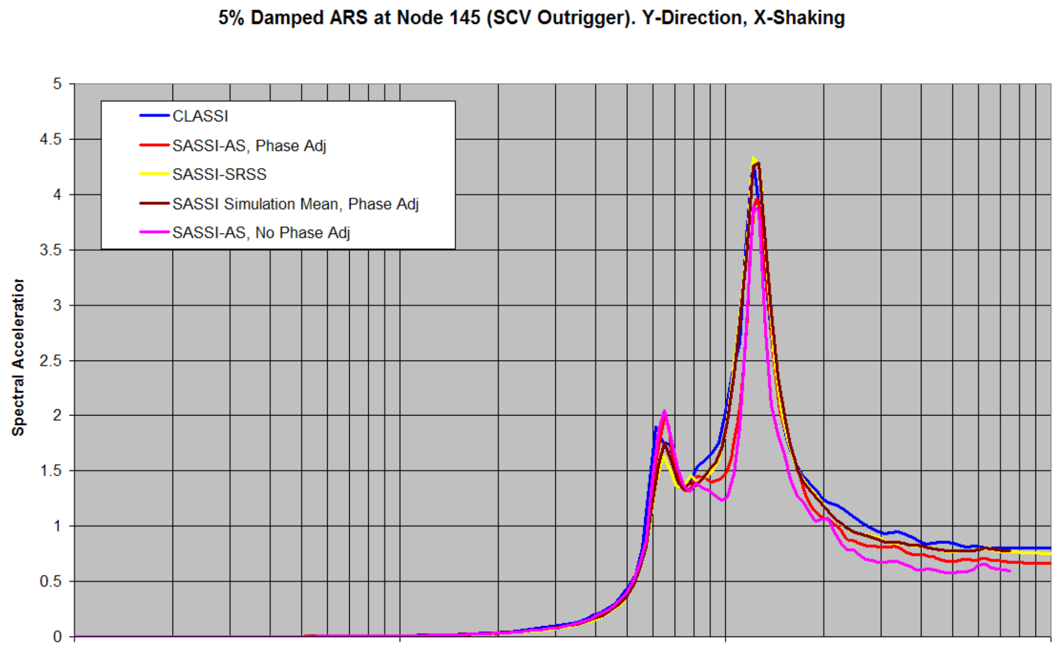
(Mean) Incoherent Interpolated ATF Using Different Interpolation Schemes



Effects of Phase Adjustment on Response Time History

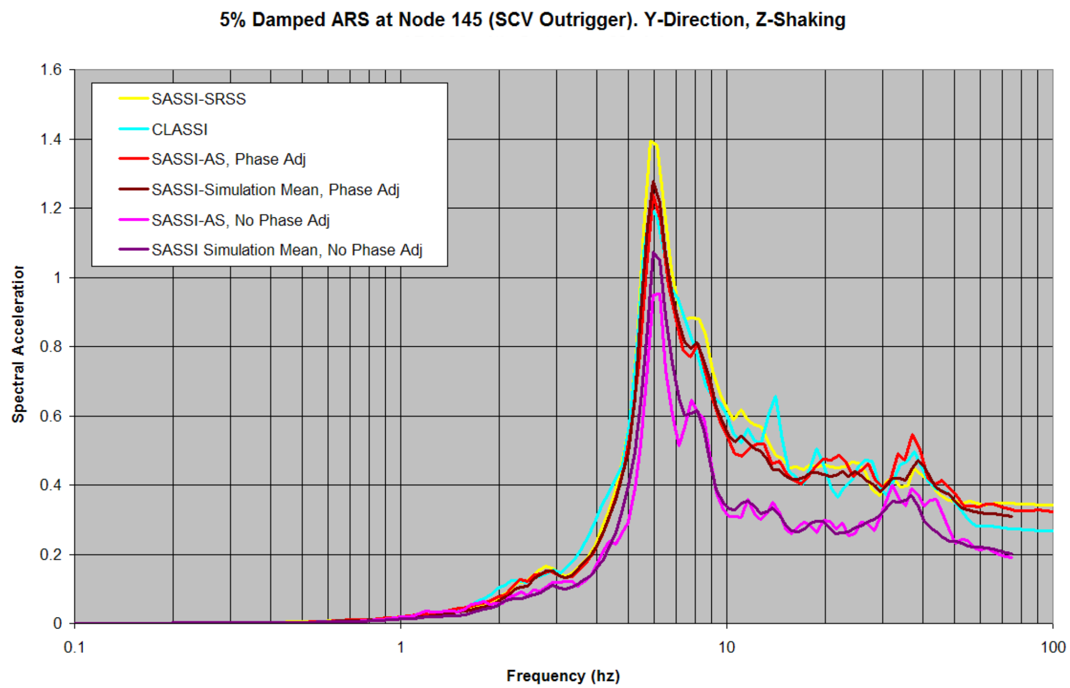


X-input



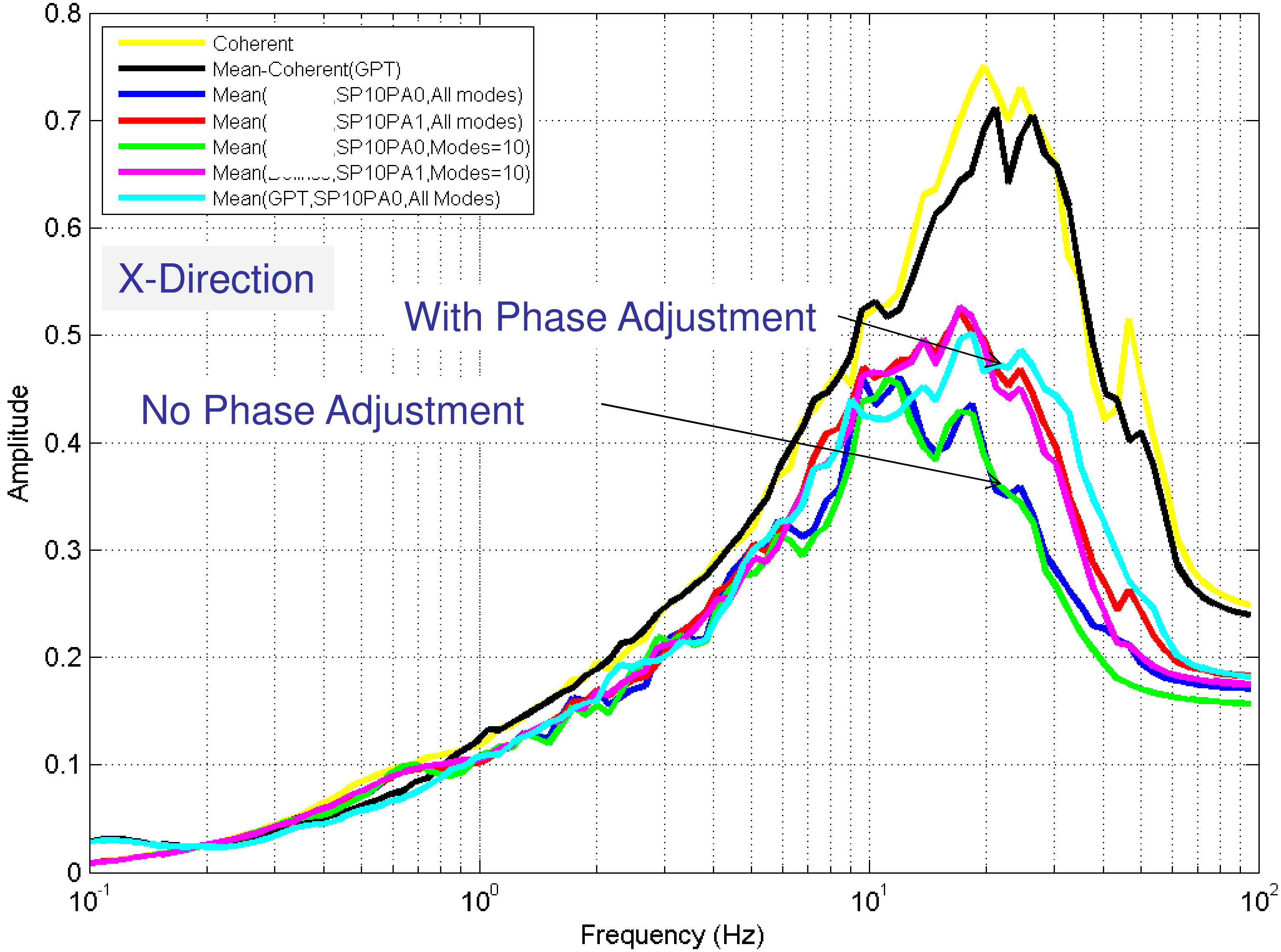
No phase adjustment has no visible effect… Provides close values with CLASSI Inco or SRSS SASSI

Z-input

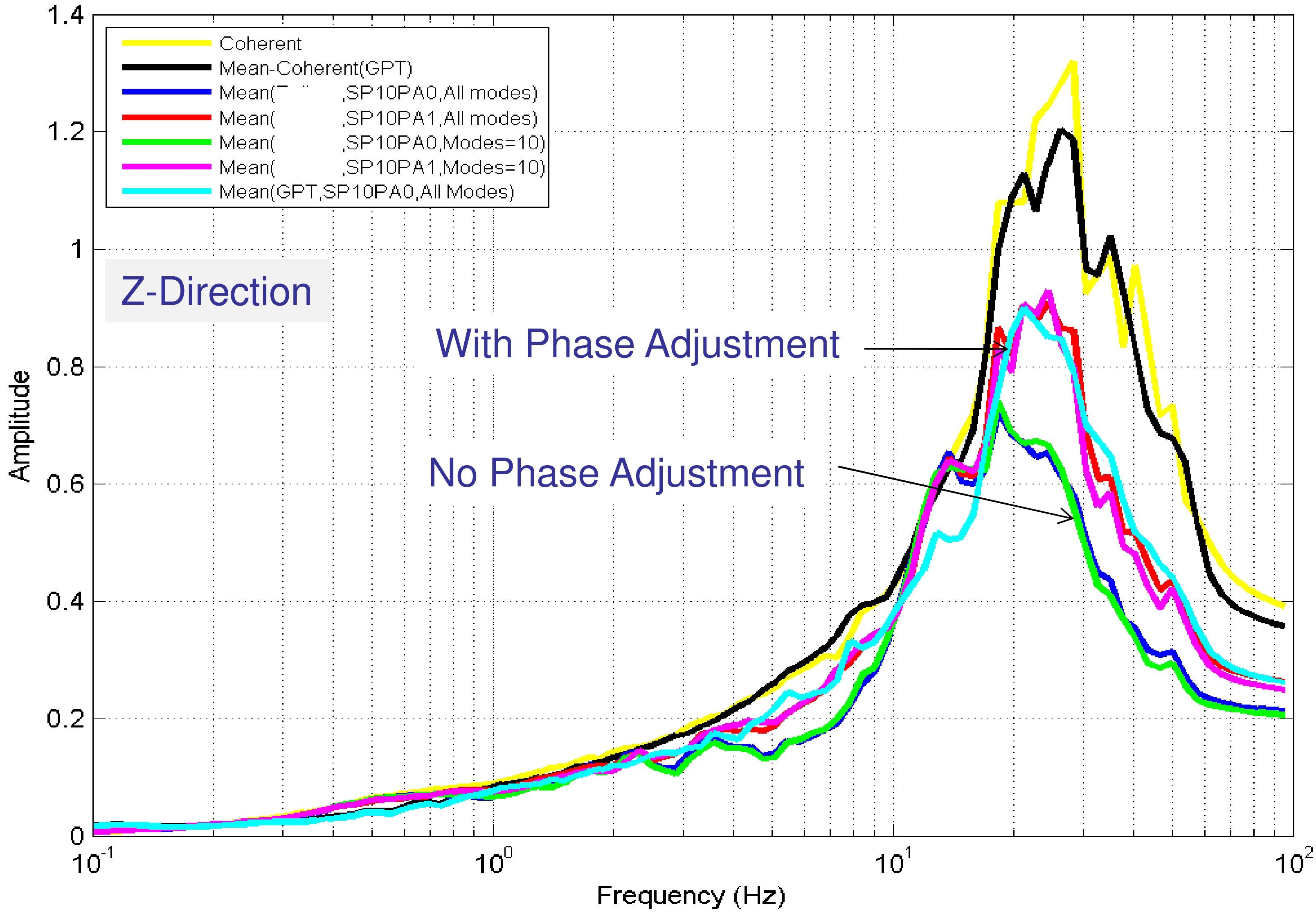


No phase adjustment provides lower response…

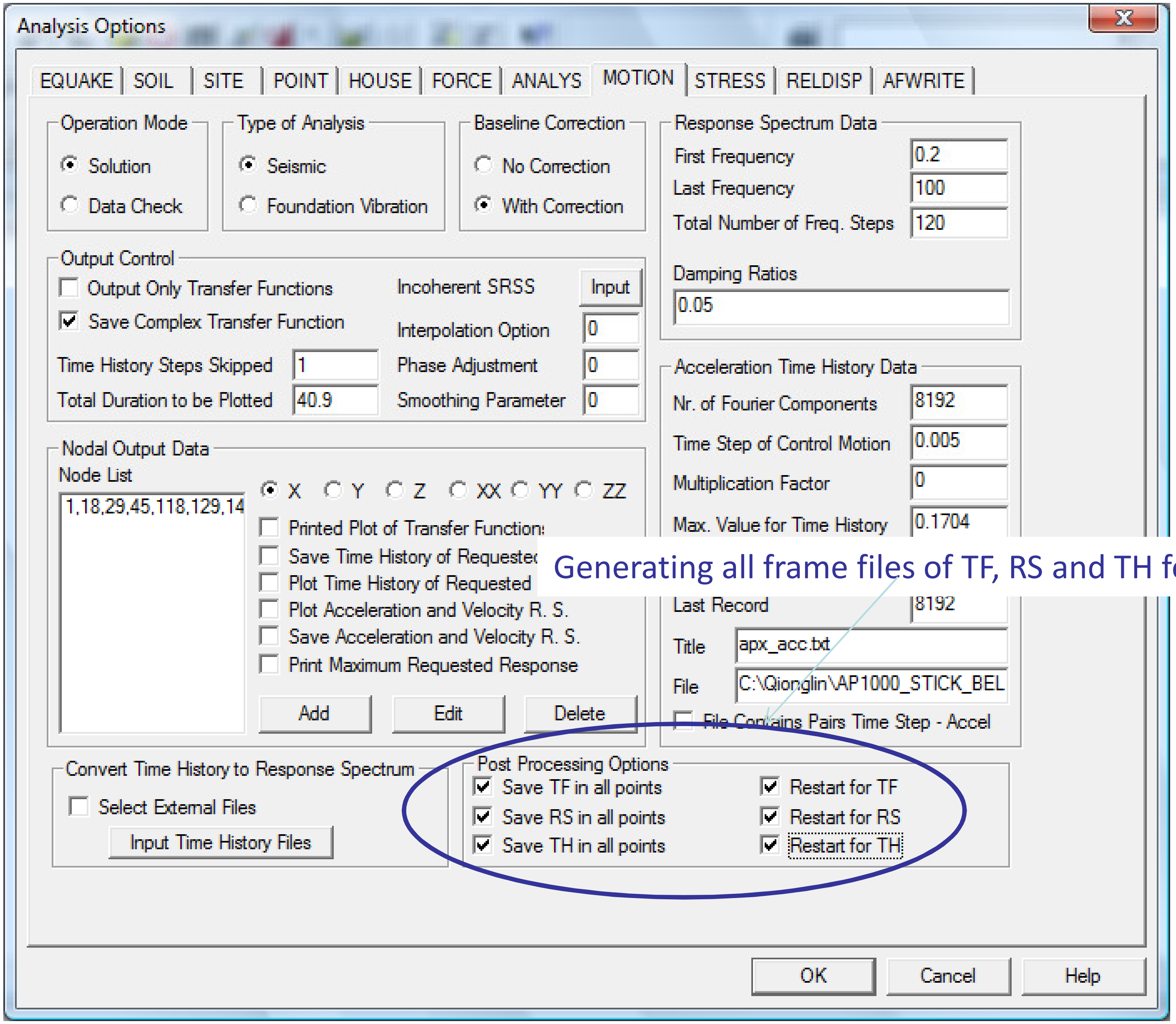
FRS Results With and Without ATF Phase Adjustment; With Single Accel Input and Multiple Accel Inputs



FRS Results With and Without ATF Phase Adjustment;
With Single Acc Input and Multiple Acc Inputs

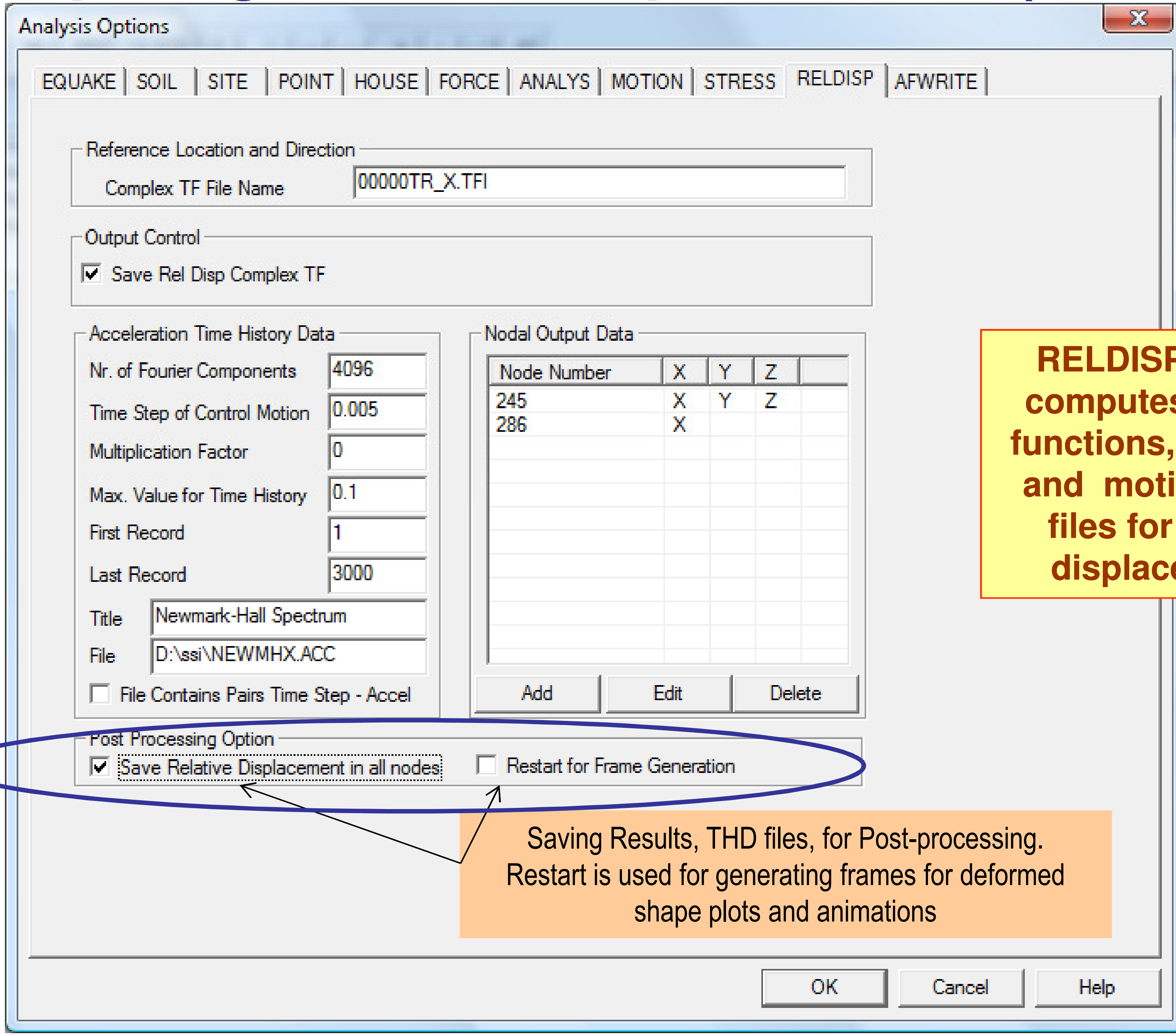


Generating Frame Files of TF, RS and TH Options



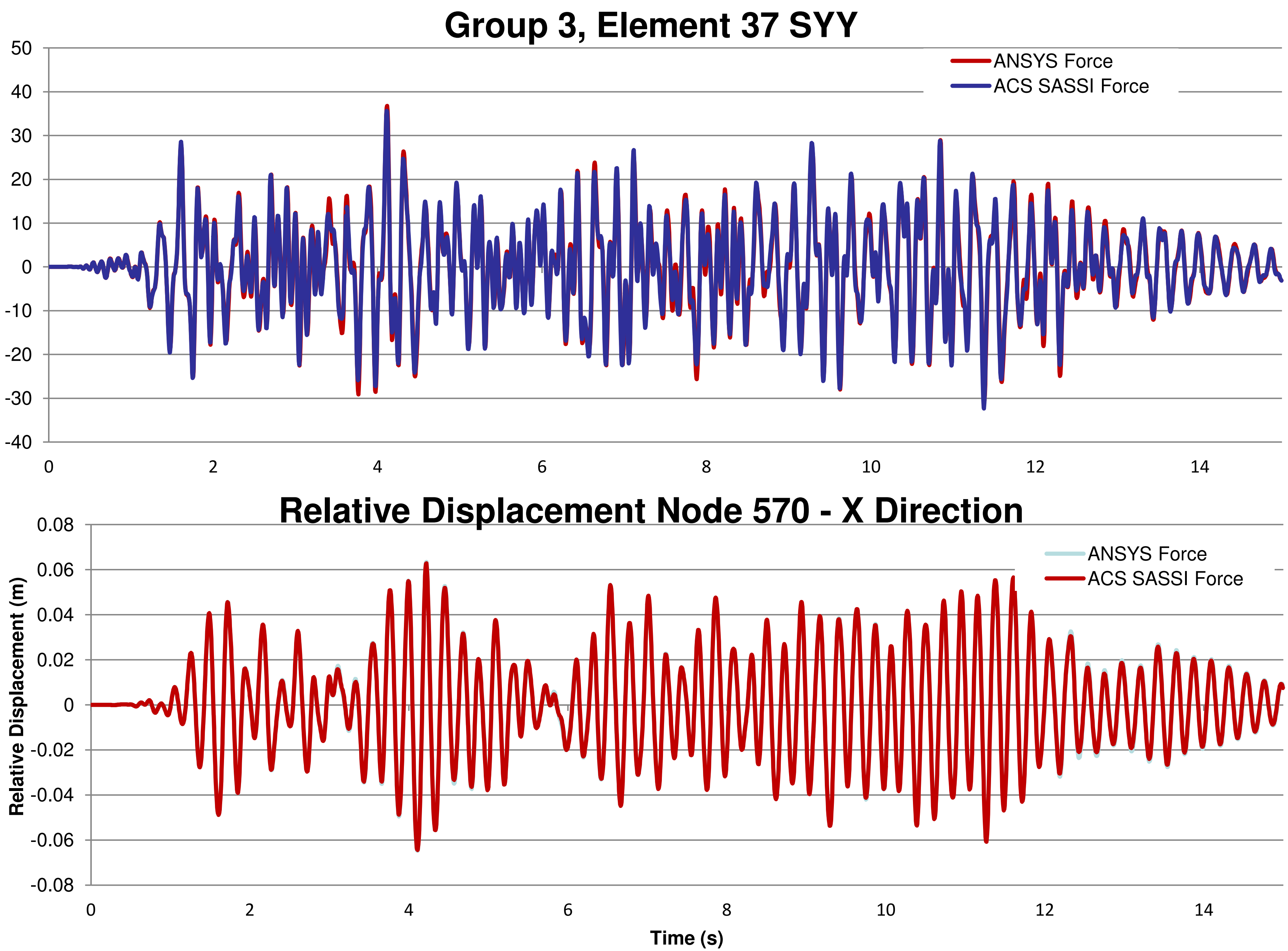
Generating all frame files of TF, RS and TH for all nodes

Computing Relative Displacements (RELDISP)



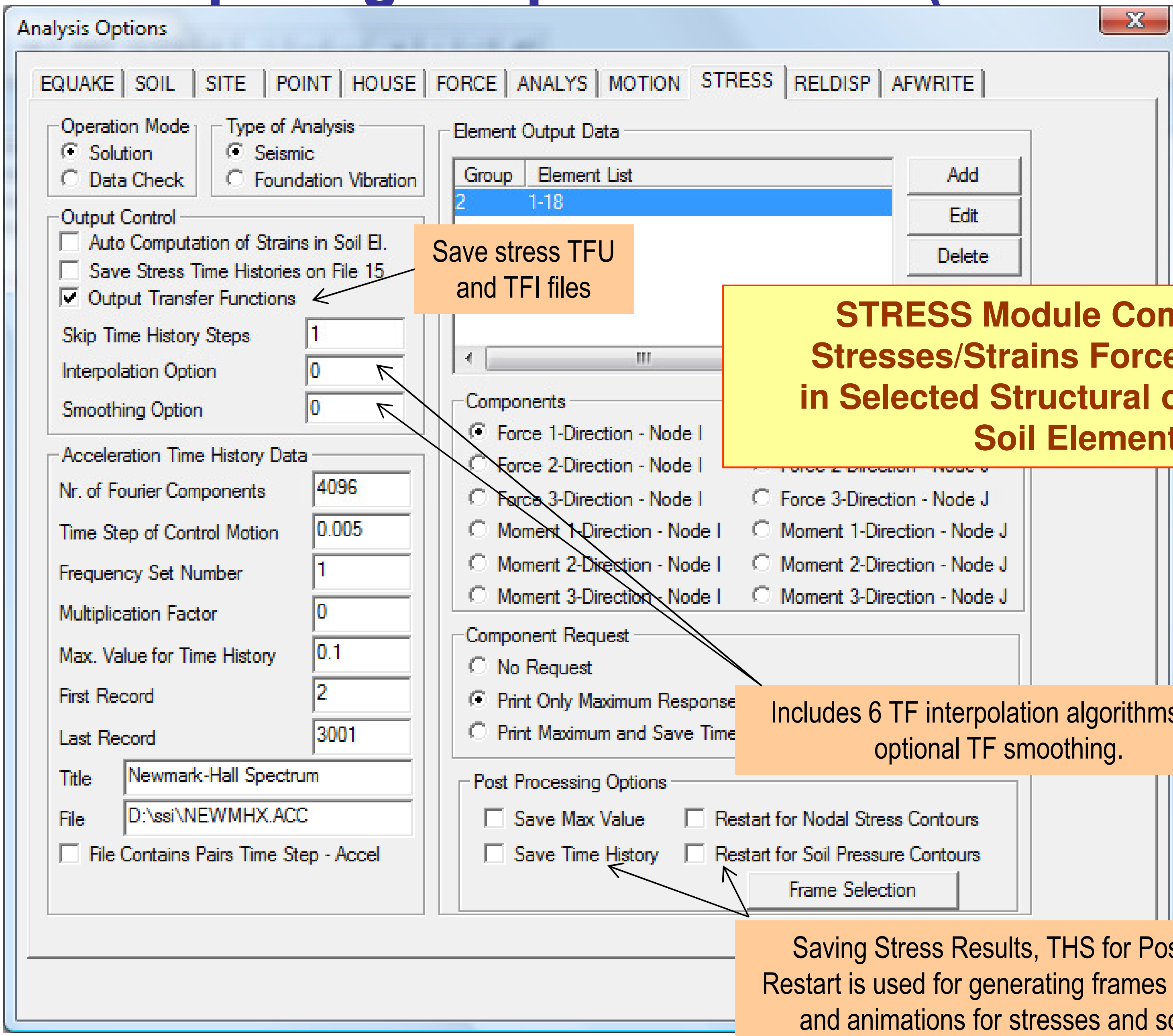
RELDISP Module computes transfer functions, TFD files, and motions, THD files for relative displacements.

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



Group 3, Element 37 SYY
ANSYS Force
ACS SASSI Force
50
40
30
20
10
0
-10
-20
-30
-40
0
2
4
6
8
10
12
14
Relative Displacement Node 570 - X Direction
ANSYS Force
ACS SASSI Force
Relative Displacement (m)
0.08
0.06
0.04
0.02
0
-0.02
-0.04
-0.06
-0.08
0
2
4
6
8
10
12
14
Time (s)

Computing Output Stresses (STRESS)



Save stress TFU and TFI files

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

Includes 6 TF interpolation algorithms and optional TF smoothing.

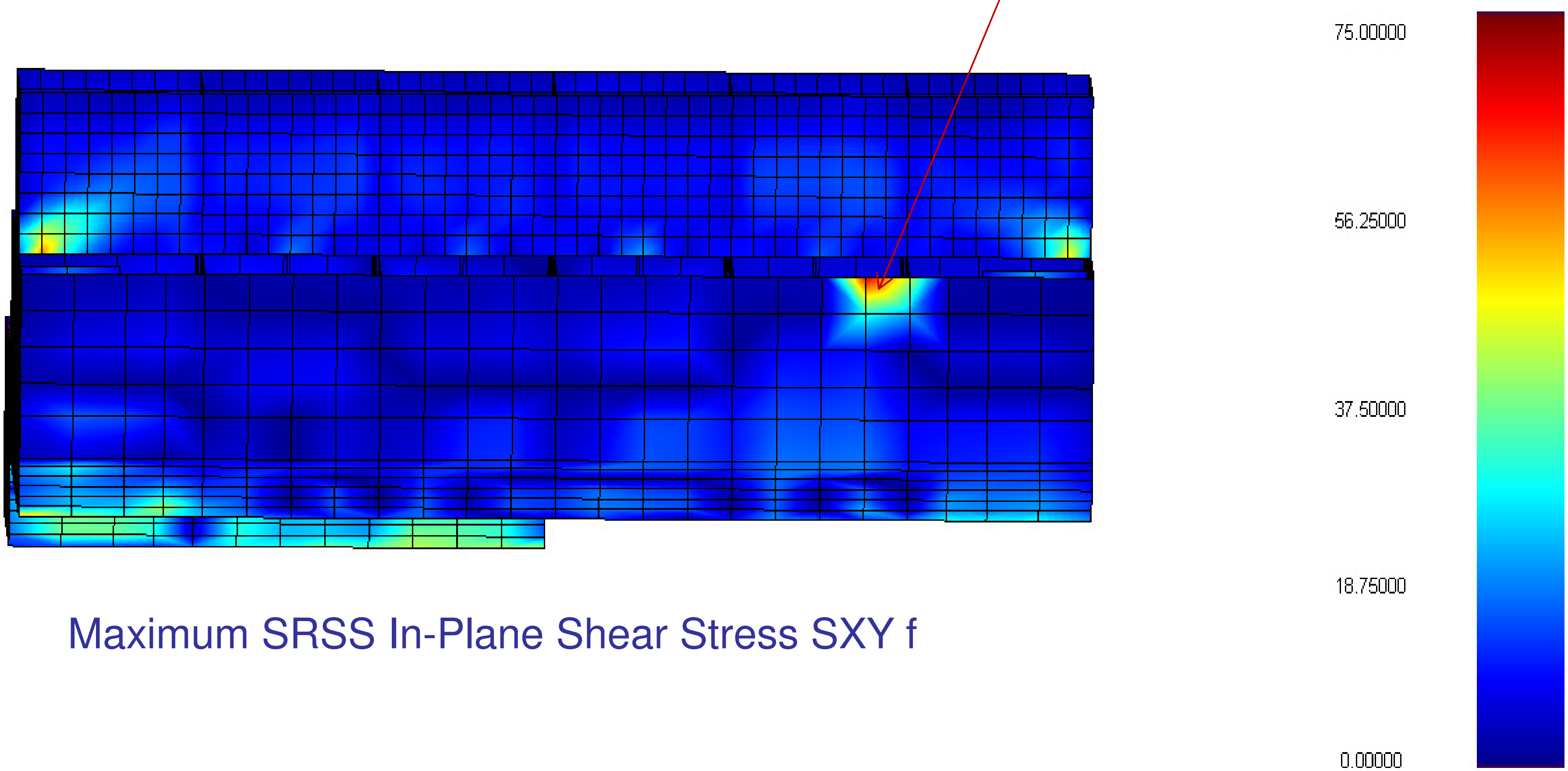
Saving Stress Results, THS for Post-processing. Restart is used for generating frames for contour plots and animations for stresses and soil pressures.

STRESS TF INTERPOLATION ISSUES

Longitudinal View #1

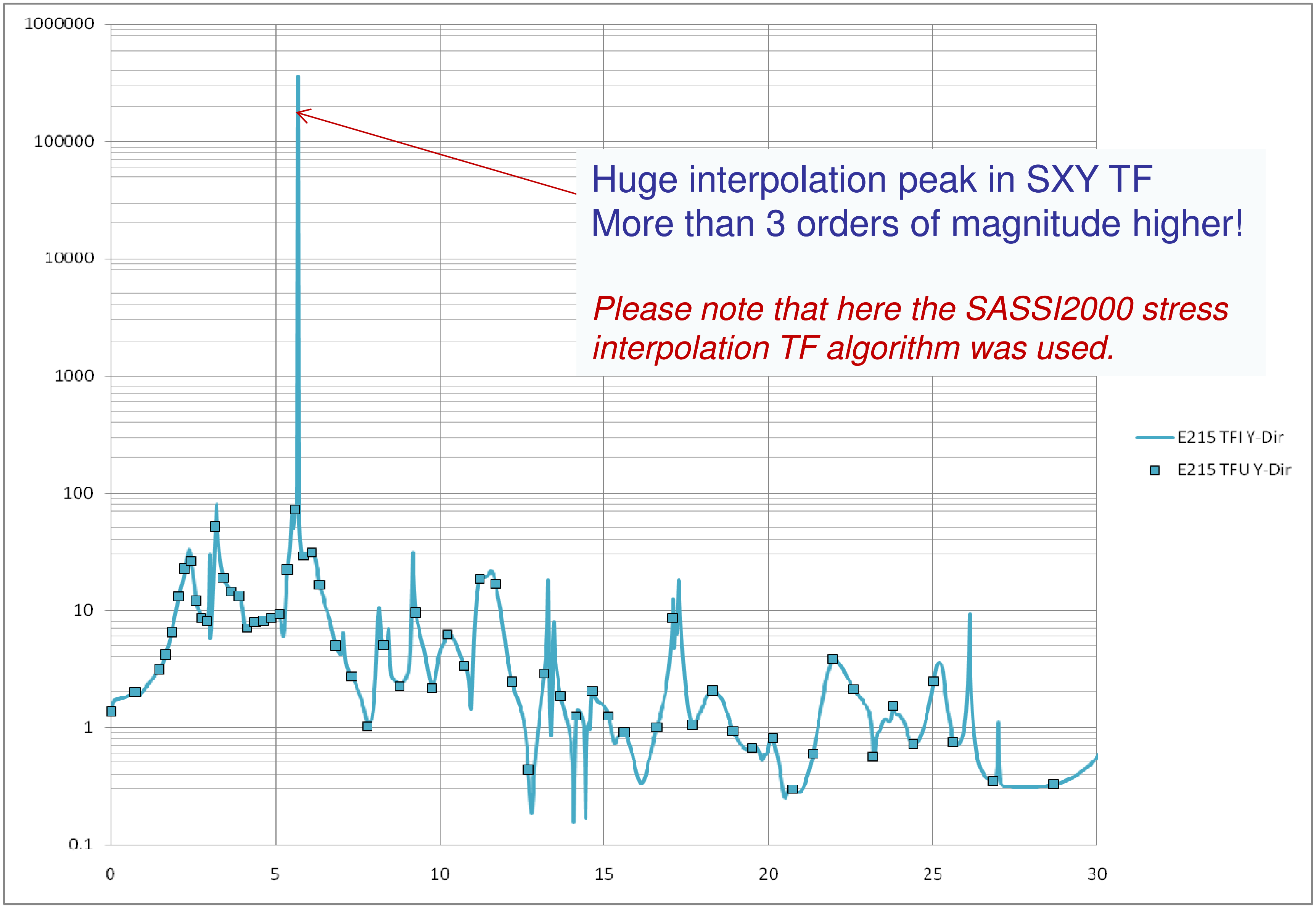
Spurious Interpolation Peak
in SXY Amplitude TF
(red spot) ELEMENT 215

Mean Soil - SXY - Group 33 - Subgroup 1

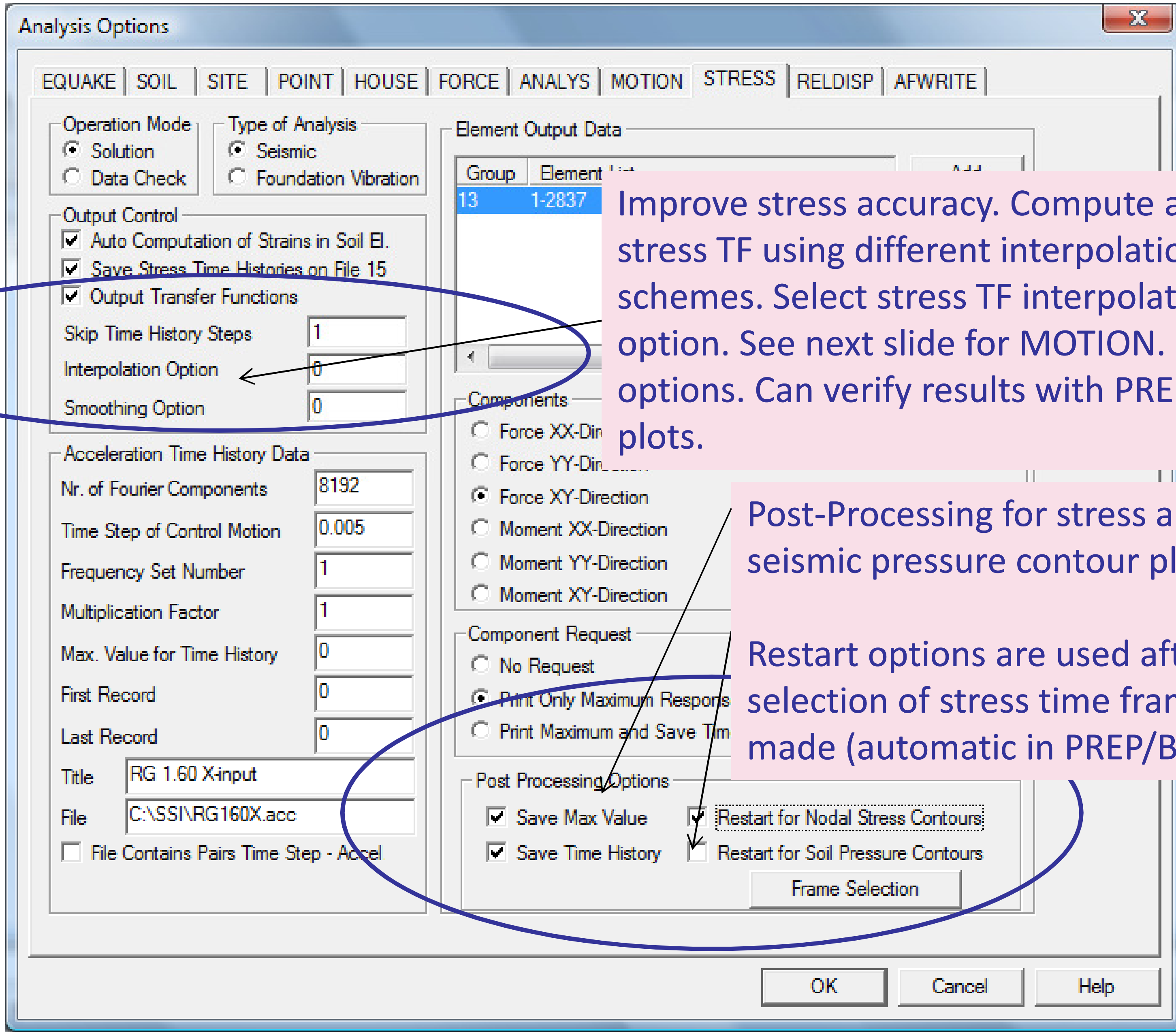


Maximum SRSS In-Plane Shear Stress SXY f

In-Plane Shear Stress in ELEMENT 215
(markers are for computed TF values and line is for interpolated TF values)



New Stress Computation and Plotting Options

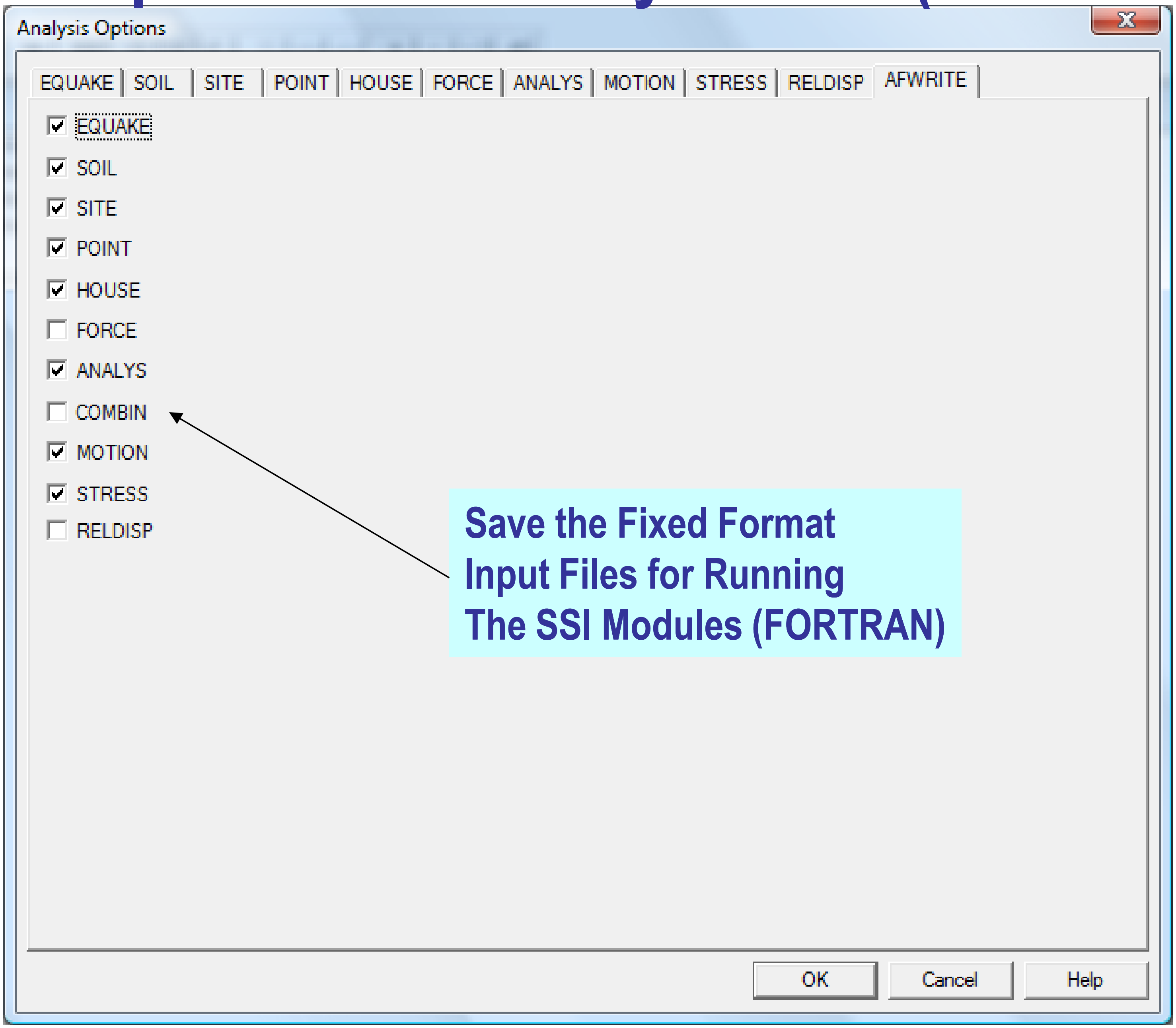


Improve stress accuracy. Compute and save stress TF using different interpolation schemes. Select stress TF interpolation option. See next slide for MOTION. Same options. Can verify results with PREP/TFU-TFI plots.

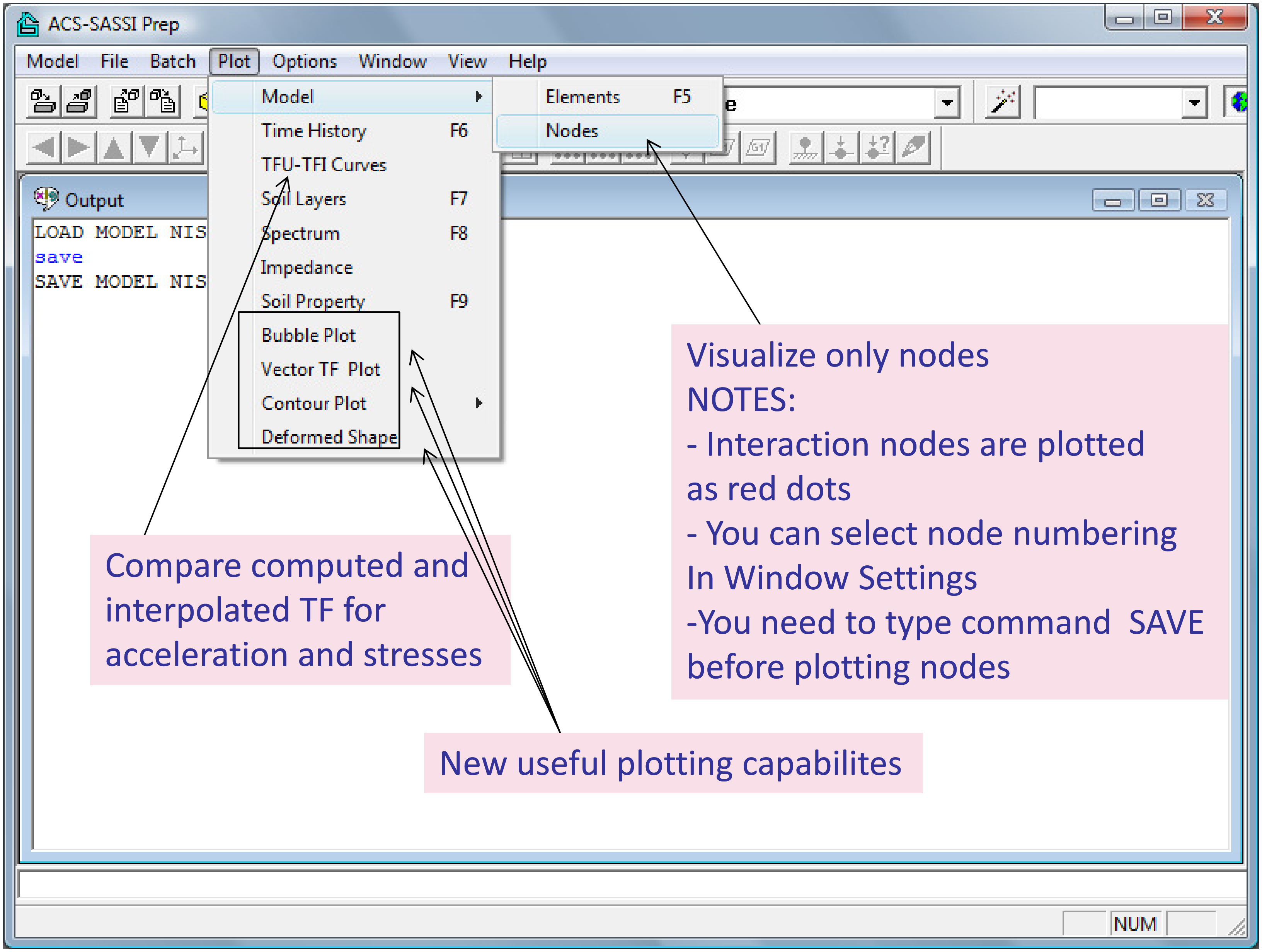
Post-Processing for stress and seismic pressure contour plots

Restart options are used after a selection of stress time frames is made (automatic in PREP/Batch)

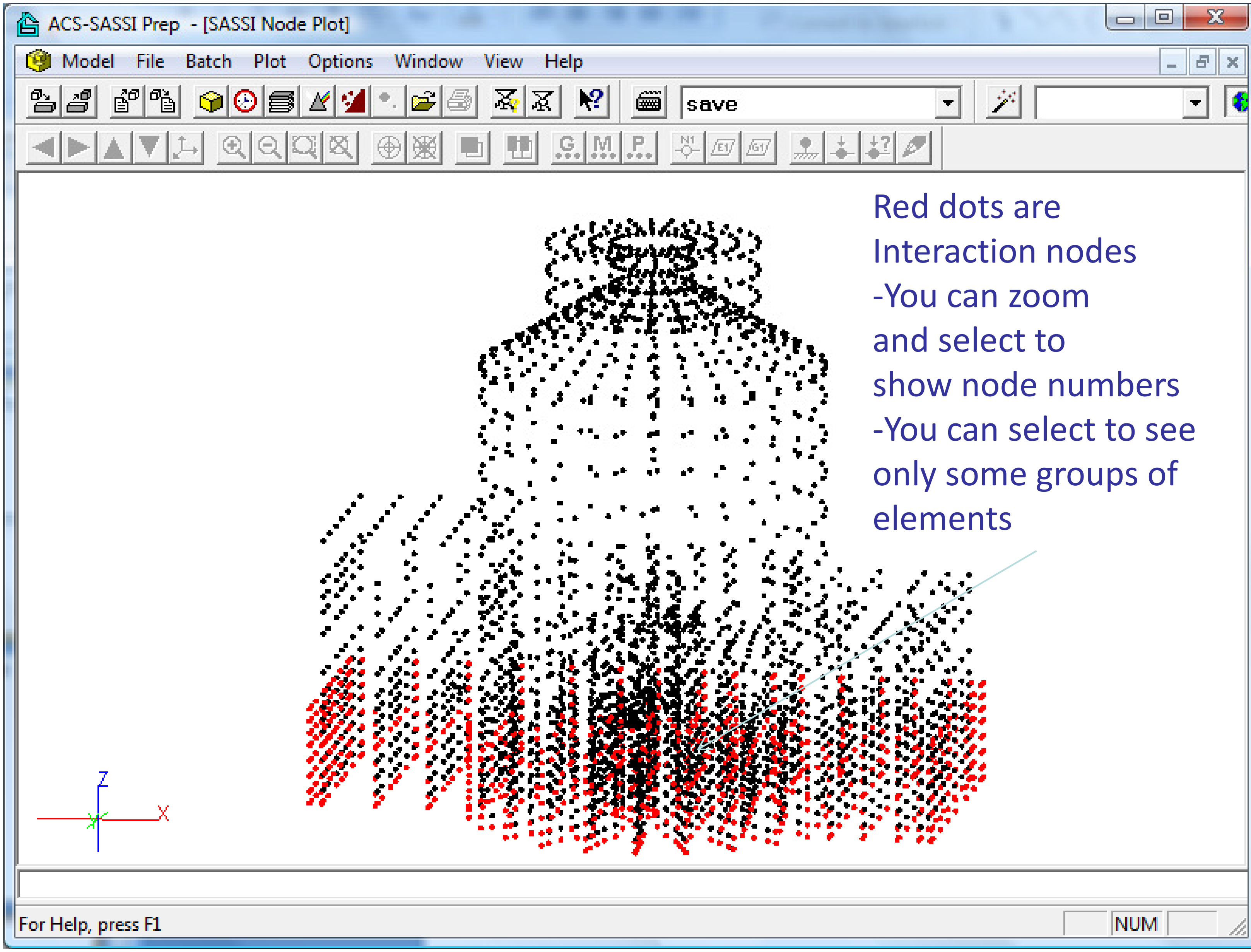
Save Inputs for SSI Analysis Run (AFWRITE)



ACS SASSI Post Processing Capabilities



Checking SSI Interaction Nodes



Red dots are Interaction nodes
-You can zoom and select to show node numbers
-You can select to see only some groups of elements

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	

Seismic SSI Response Structural Plotting Options

Bubble Plots – Static – Node Plots

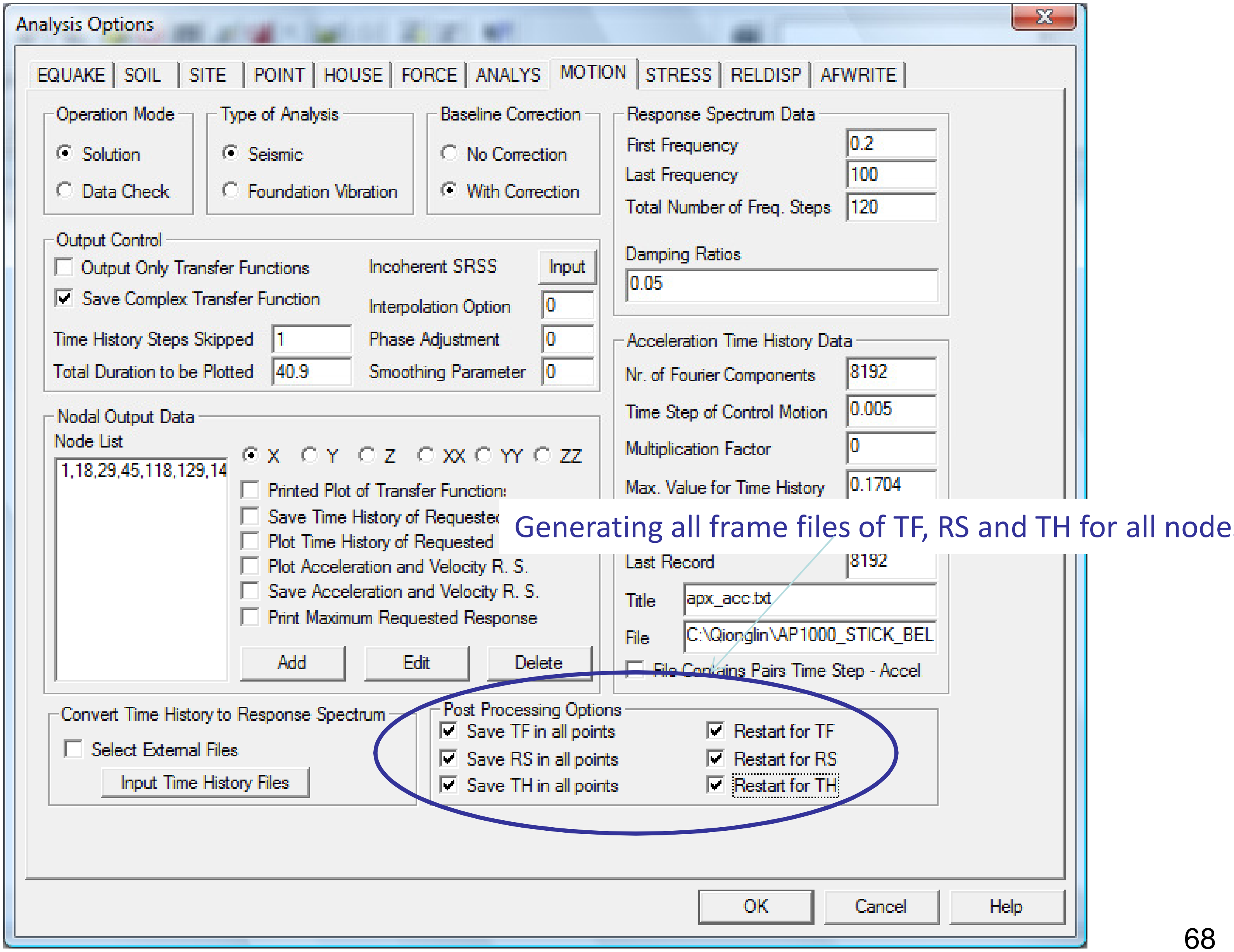
Complex ATF Vector Plots – Animated – Node Plots

Contour Plots – Static or Animated – Element Plots with Hidden Lines – Time Sequence or Selected Time Frames

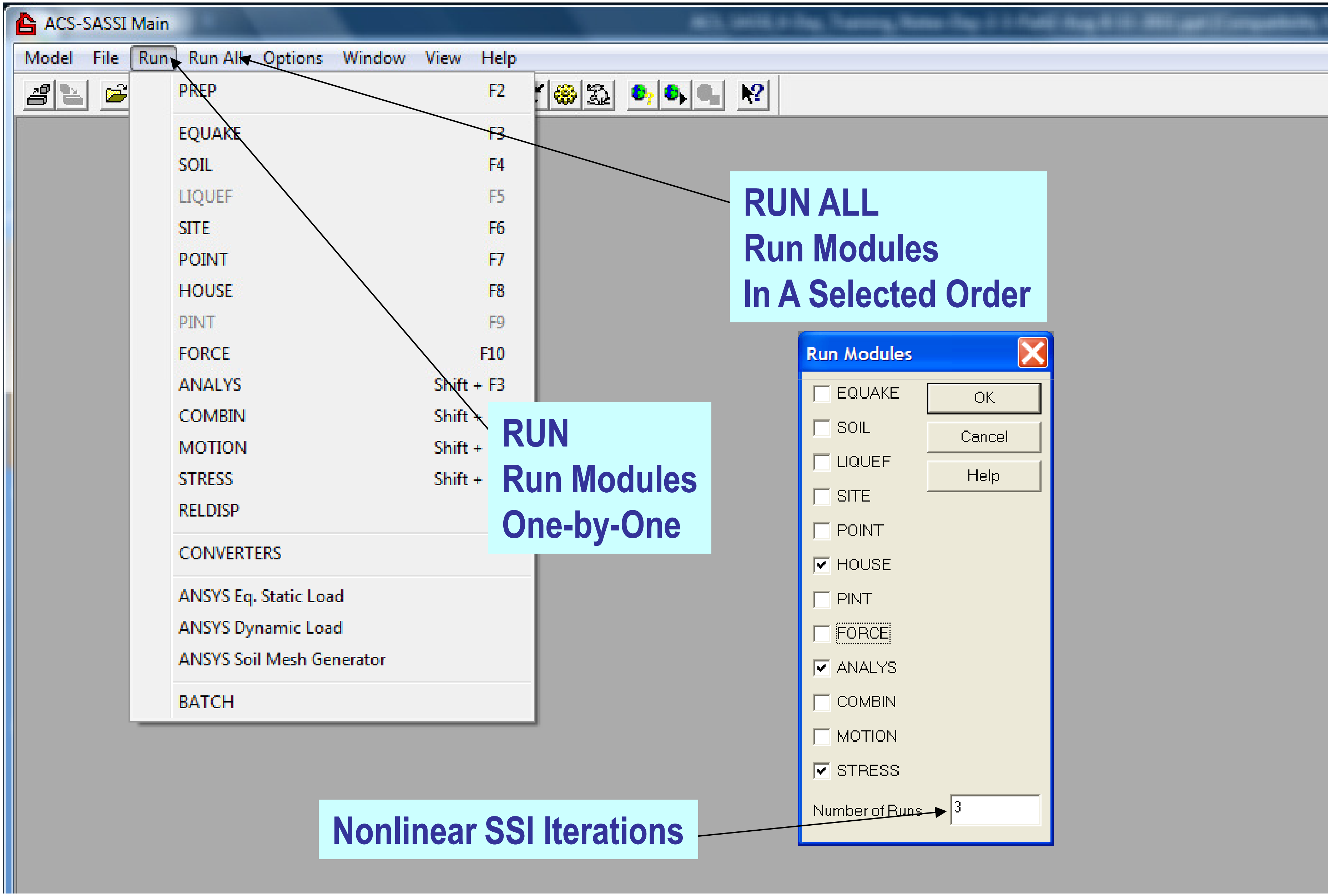
Deformed Shape – Animated – Element Plots with Hidden Lines – Time Sequence or Selected Time Frames

Show some real time examples…..

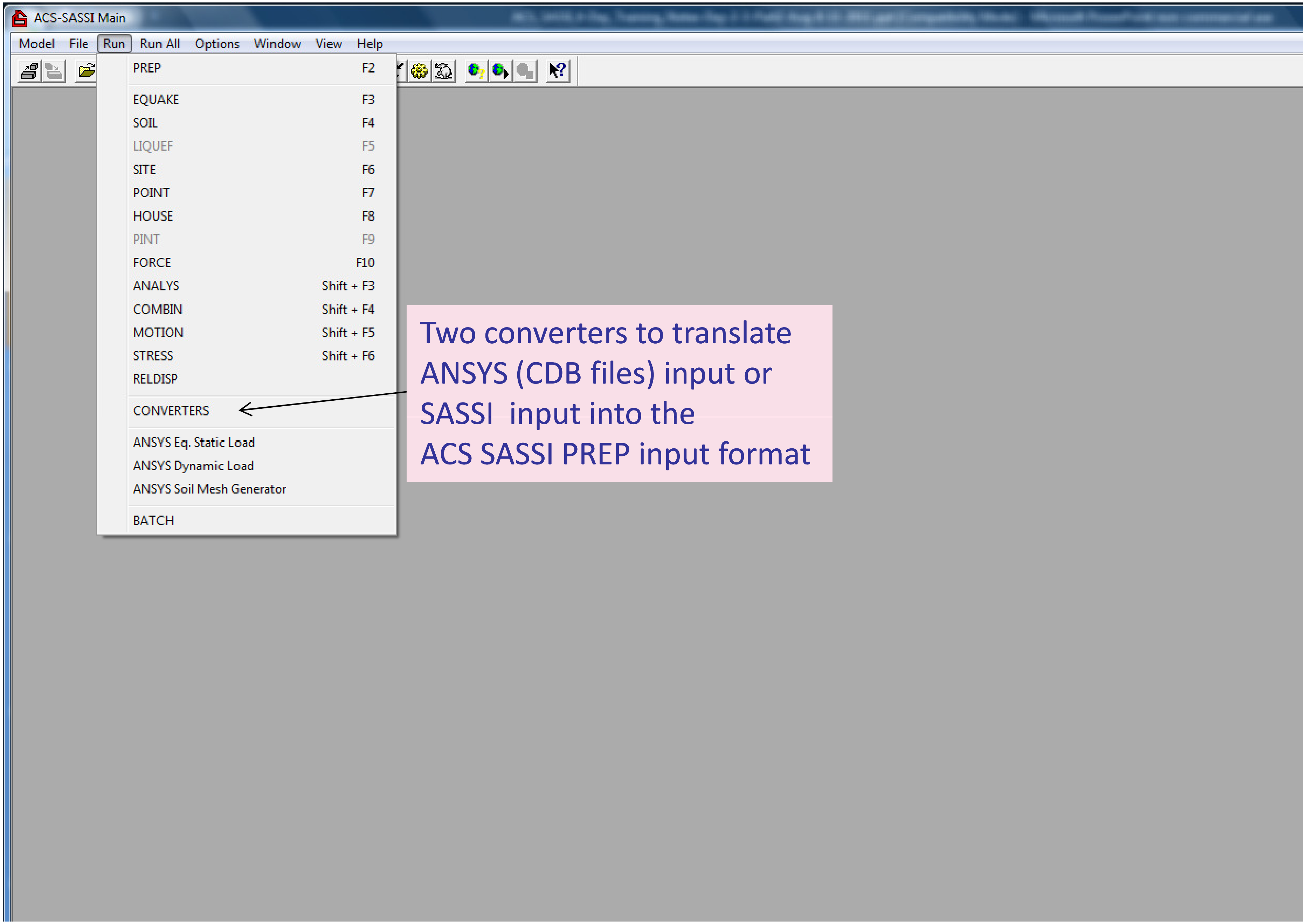
Generating Frame Files of TF, RS and TH Options



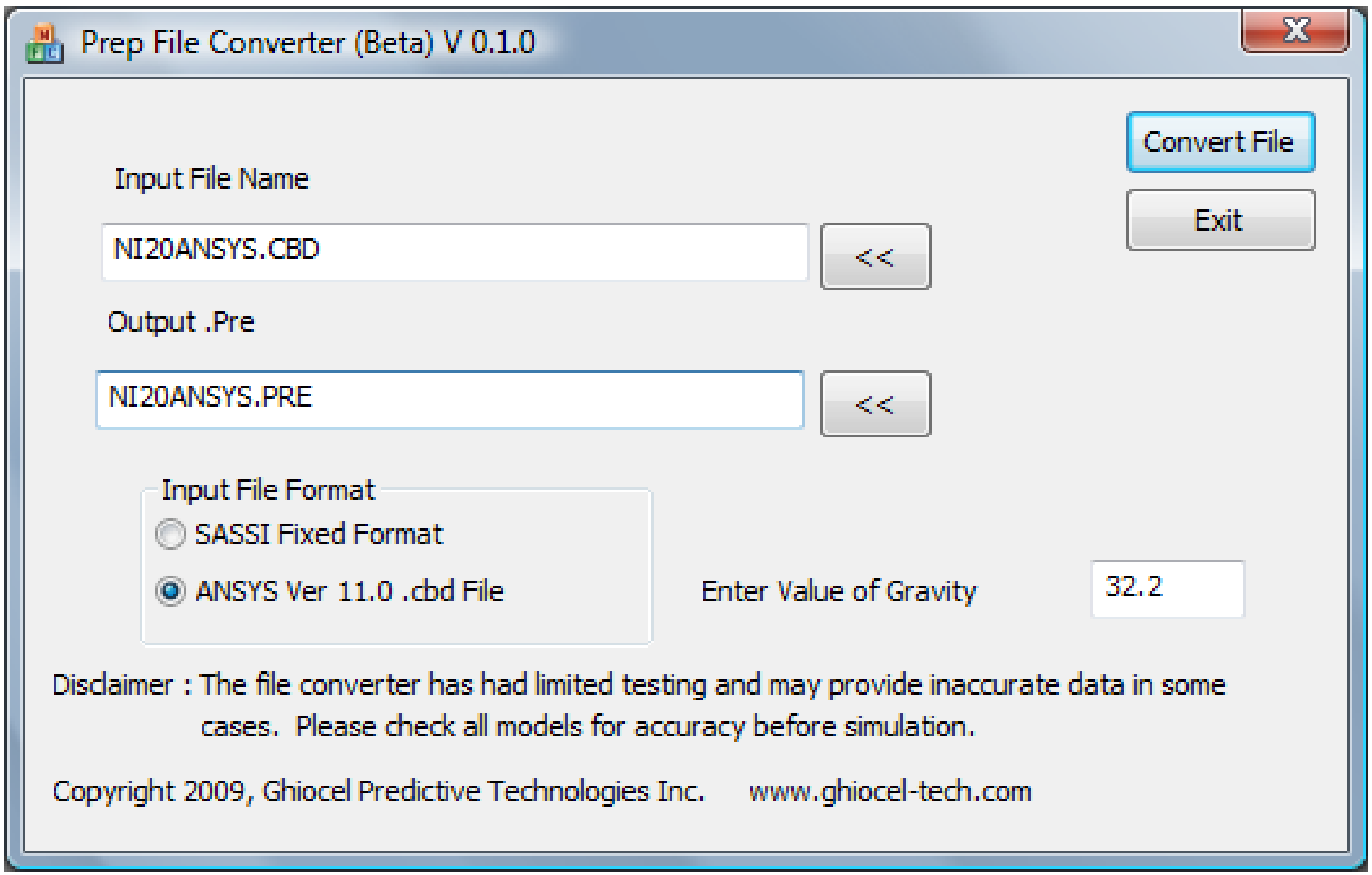
ACS MAIN Menu for Managing SSI Module Runs



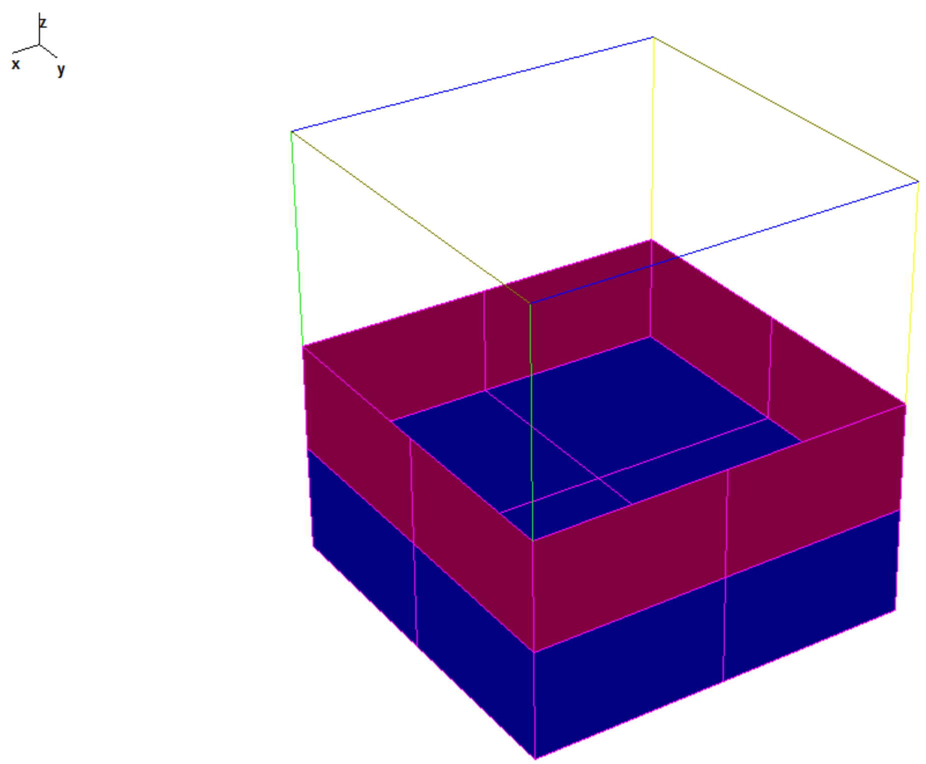
ACS SASSI MAIN Input File Converters



ANSYS (cdb) or SASSI2000 Input (.hou,.sit,.poi) Converter to ACS SASSI Input



ANSYS CDB file to ACS SASSI PRE file Converter



FE Model

The converter program will work with the following elements only

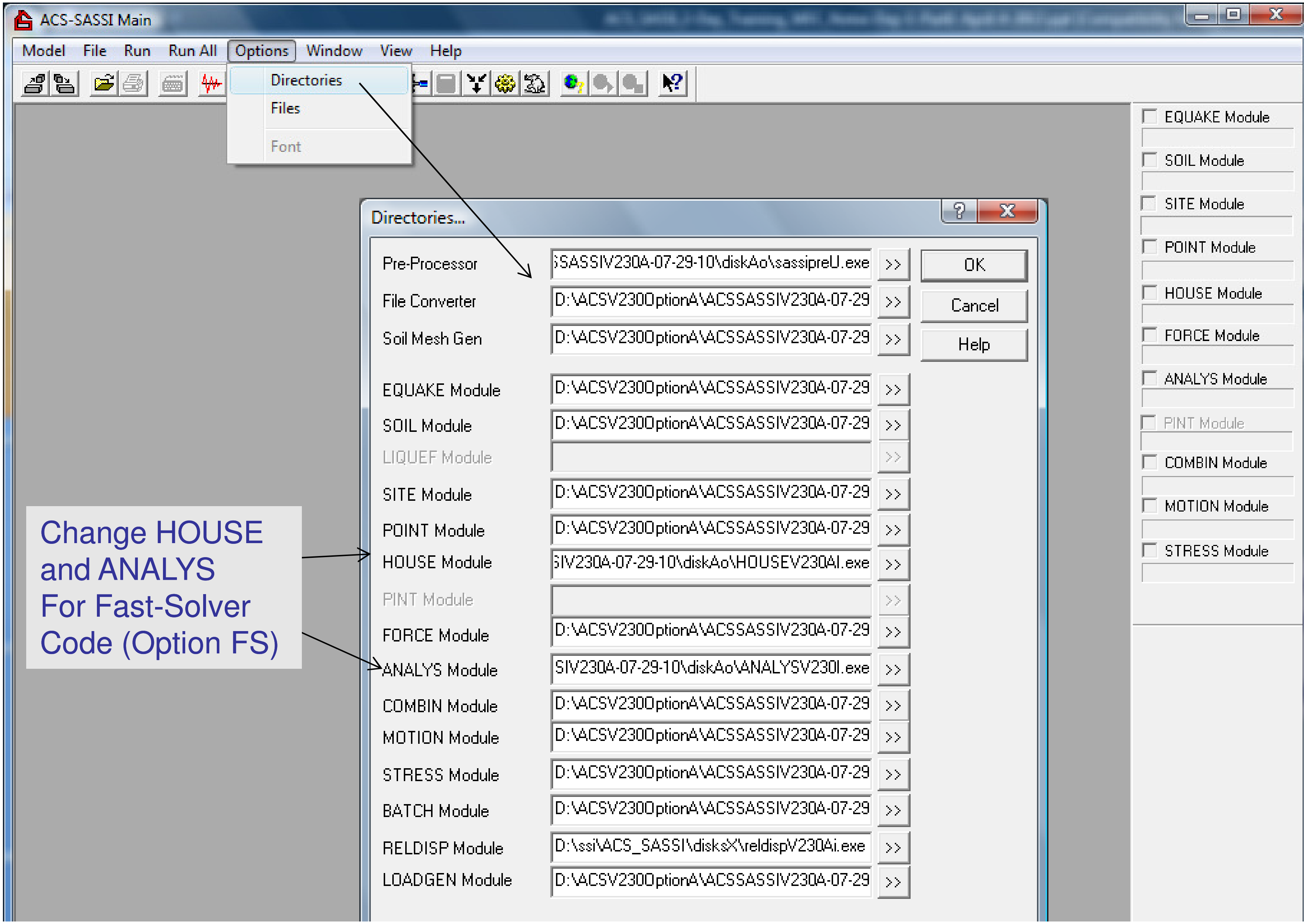
- BEAM4
- COMBIN14
- BEAM44
- SOLID45
- SHELL63
- MASS21

For BEAM4 and BEAM44 elements, the I, J, and K nodes must be defined.

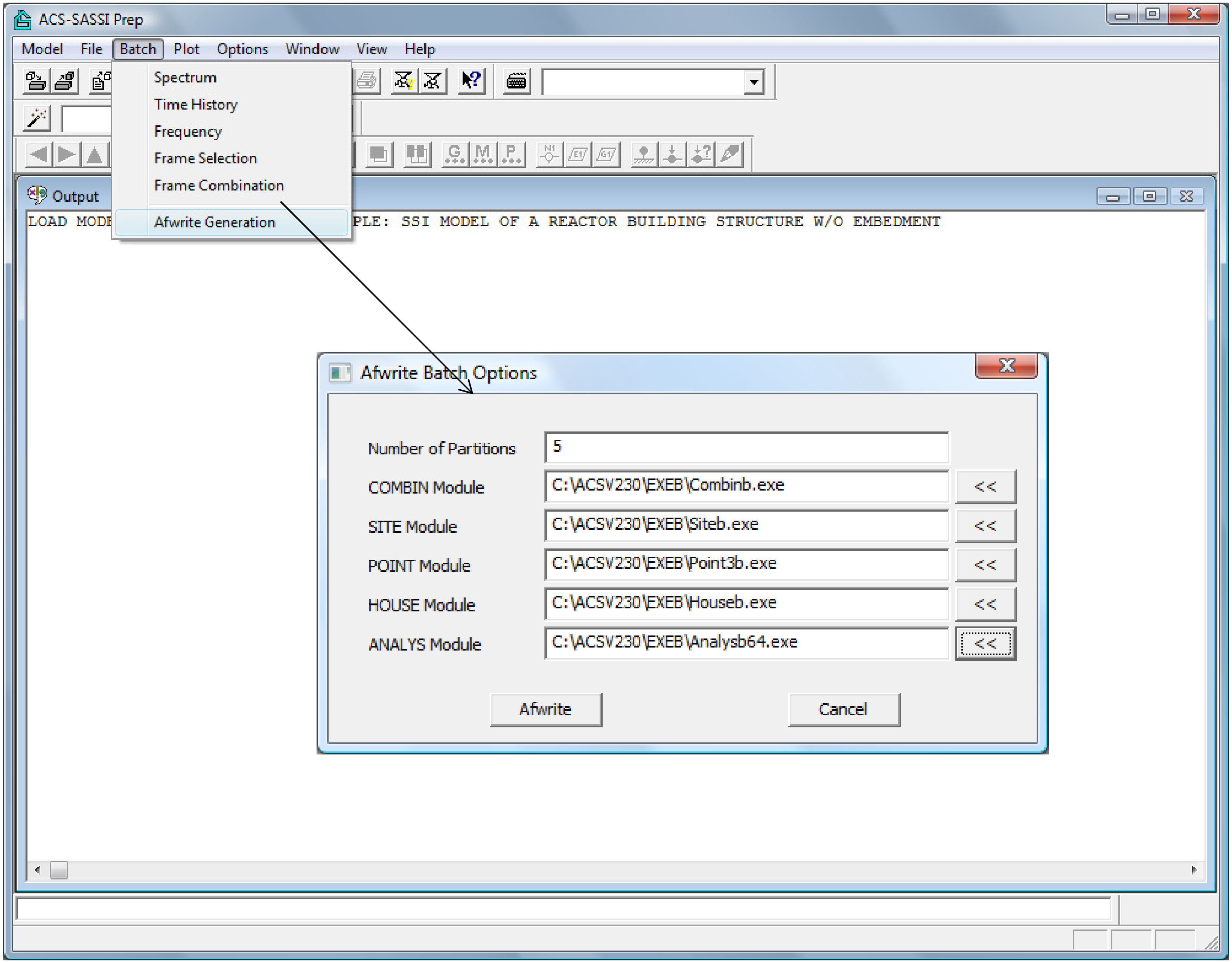
For COMBIN14 elements, the spring direction must be set using KEYOPT(2) and KEYOPT(1) must be 0.

The material properties need to be changed after the model is converted. ANSYS uses density for materials, while ACS SASSI uses specific weight. The material data from the converter output file must be multiplied by gravity to get the correct material property for the SSI analysis.

Define SSI Module Names and Paths



ACS SASSI PREP Used to Prepare Batch Runs



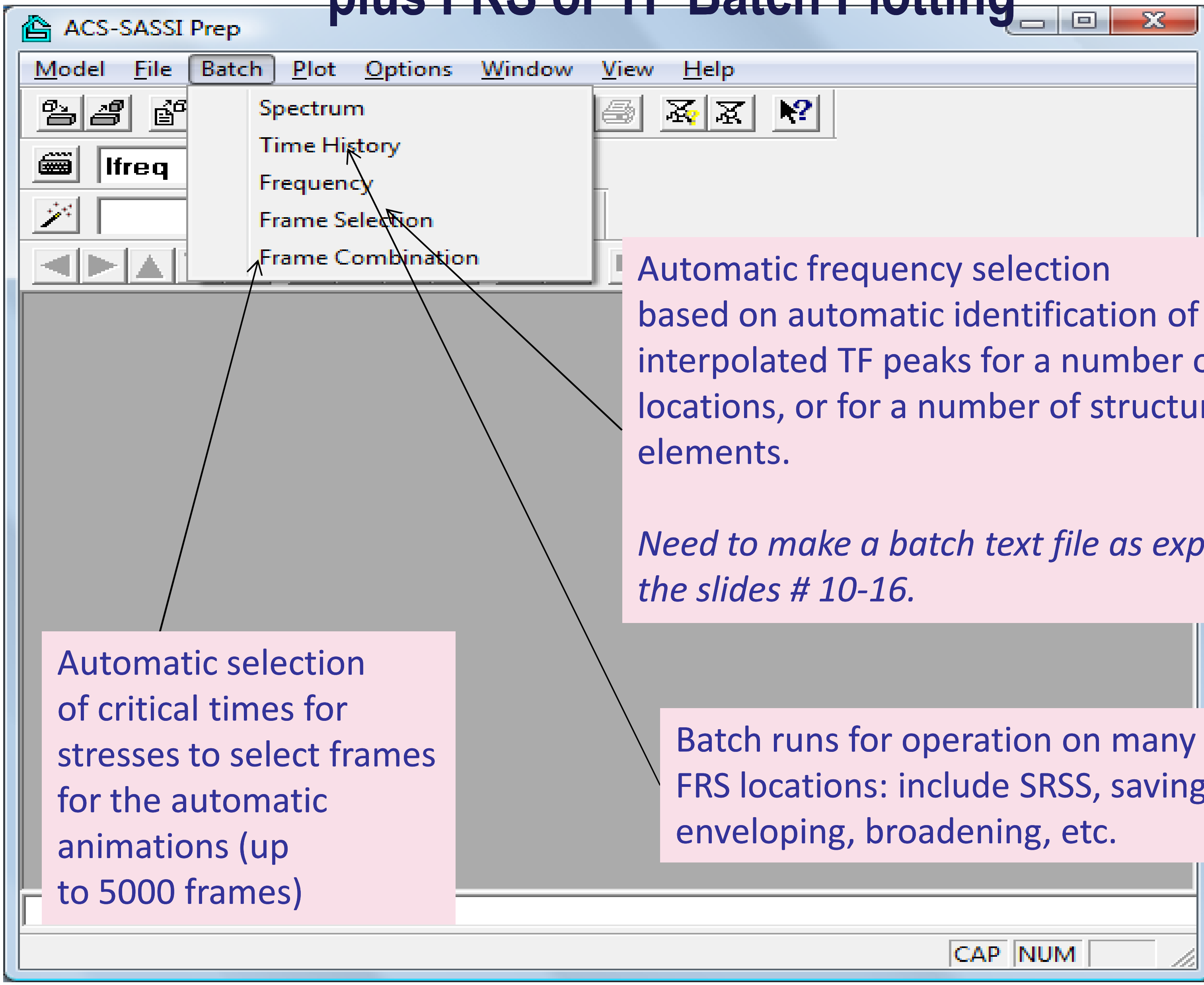
runbatch.bat

```
@echo off

set mname=RBX

mkdir .\work
cd .\work

echo %mname% > site.inp
echo %mname%.sit >> site.inp
echo %mname%_site.out >> site.inp

echo %mname% > point.inp
echo %mname%.poi >> point.inp
echo %mname%_point.out >> point.inp

echo %mname% > house.inp
echo %mname%.hou >> house.inp
echo %mname%_house.out >> house.inp

echo %mname% > analys.inp
echo %mname%.anl >> analys.inp
echo %mname%_analys.out >> analys.inp

for %%j in (X Y Z) do (
   if %%j NEQ Y (
      copy ..\%mname%_%%j.sit %mname%.sit
      C:\ACSV230\EXEB\Siteb.exe < site.inp

      copy ..\%mname%_%%j.poi %mname%.poi
      C:\ACSV230\EXEB\Point3b.exe < point.inp
   )

   if %%j EQU X (
      copy ..\%mname%_%%j.hou %mname%.hou
      C:\ACSV230\EXEB\Houseb.exe < house.inp
   )

   copy ..\%mname%_%%j.anl %mname%.anl
      C:\ACSV230\EXEB\Analysb.exe < analys.inp
   mkdir ..\%%jDIR
   copy FILE8 ..\%%jDIR\FILE8_%%j
   move *.out ..\%%jDIR
)

del *.* /Q
cd ..
rmdir .\work
```

runbatch.bat | Combine.bat

```
@echo off

set ipath=..
set opath=.\Combine_FILE8

mkdir %opath%
cd %opath%

for %%i in (X Y Z) do (
    copy %ipath%\Set1\%%iDIR\FILE8_%%i FILE81
    for %%j in (2 3 4 5 ) do (

        copy %ipath%\Set%%j\%%iDIR\FILE8_%%i FILE82
        C:\ACSV230\EXEB\Combinb.exe
        del FILE81
        del FILE82

        ren FILE8 FILE81

    )
    ren FILE81 FILE8_%%i
)
```

Automatic Frequency and Stress Selections, plus FRS or TF Batch Plotting



Automatic frequency selection based on automatic identification of interpolated TF peaks for a number of FRS locations, or for a number of structural elements.

Need to make a batch text file as explained in the slides # 10-16.

Batch runs for operation on many FRS locations: include SRSS, saving images, enveloping, broadening, etc.

Automatic selection of critical times for stresses to select frames for the automatic animations (up to 5000 frames)

Batch Post-Processing Response Spectrum Curves

Compute Average of Several Spectral Curves (up to 15 curves per operation)

This batch file is used to compute the average of three FRS or ATF inputs. Output file name is 00566TR_X01.RS .

Number of Curves

Average

Batch text file:

```
1
3 4 ./00566TR_X01.RS
Blank Line
./XDIR/00566TR_X01.RS
./YDIR/00566TR_X01.RS
./ZDIR/00566TR_X01.RS
```

Enveloping and Broadening Several Spectral Curves (up to 15 curves per operation)

This batch file is used to compute the broaden of six inputs. Output file name is 00565TR_X01_BRD.RS .

Number of Curves

Enveloping & Broadening

Batch text file:

```
1
6 1 ./00565TR_X01_BRD.RS
0 15
./00565TR_X01_SLB.RS
./00565TR_X01_SM.RS
./00565TR_X01_SUB.RS
./00565TR_X01_BLB.RS
./00565TR_X01_BM.RS
./00565TR_X01_BUB.RS
```

Creating Images of Several Spectral Curves (up to 15 curves per operation)

This batch file is used to plot three curves. Output image file name is 00566TR_X01_SUB.BMP. This image title is “Original Inputs”.

Number of Curves

Batch text file:

Capture Image

```
1
3 0 ./00566TR_X01_SUB.BMP
1 0 1 1  Original Inputs
./XDIR/00566TR_X01.RS
./YDIR/00566TR_X01.RS
./ZDIR/00566TR_X01.RS
```

Title

Ticks and Log

Compute SRSS for Co-Directional Spectra (3 curves per operation)

This batch file is used to compute SRSS of three inputs. Output file name is 00566TR_X01_SUB.RS.

Batch text file:

Number of Curves

SRSS

1
3 2 ./00566TR_X01_SUB.RS
Blank Line
./XDIR/00566TR_X01.RS
./YDIR/00566TR_X01.RS
./ZDIR/00566TR_X01.RS

Batch Processing of Time Histories

This Example Combines 3 time histories by addition and saves the result in New_Timehist.th

Time History Batch Text

1
3 0 New_Timehist.th
00001TR_X.acc
00046TR_X.acc
00052TR_X.acc

Number of Files

Algebraic Addition

Batch Automatic Selection of SSI Frequencies

Batch Frequency Selection Option

Input File

```
5 95 30
00778TR_X
01057TR_X
02098TR_X
02190TR_X
04970TR_X
```

The first number in the header line is the number of files to use to find additional frequencies.

The second number in the header is the tolerance on the difference between the TFI and TFU.

The third number in the header line is the percent below the max for which peaks are ignored .

After the header line, the files sets to be checked are listed without the file extension.

See Excel file

Automatic Selection of Frames for Vector TF Plotting

Vertical TF Frame Selection Option

Input File: *.tfani

```
1 20 1
C:\AP1000_STICK\XDIR\TFU
TFU_000.02_00001
TFU_000.17_00002
TFU_000.34_00003
TFU_000.49_00004
TFU_000.59_00005
TFU_000.68_00006
TFU_000.78_00007
TFU_000.88_00008
TFU_001.00_00009
TFU_001.10_00010
TFU_001.20_00011
TFU_001.29_00012
TFU_001.42_00013
TFU_001.49_00014
TFU_001.59_00015
TFU_001.68_00016
TFU_001.78_00017
TFU_001.88_00018
TFU_002.00_00019
TFU_002.10_00020
```

The first line is the number of files to create the frame: files from 1 to 20 by increments of 1

The second line is the directory of the 20 files.

20 files' name are listed.

Automatic Selection of Frames for Deformed Shape Plotting of Response Spectra

Frame RS Selection Option

Input File: *.rsani

```
1 20 1
C:\AP1000_STICK\XDIR\RS
RS01_000.20_00001
RS01_000.21_00002
RS01_000.22_00003
RS01_000.23_00004
RS01_000.25_00005
RS01_000.26_00006
RS01_000.27_00007
RS01_000.29_00008
RS01_000.30_00009
RS01_000.32_00010
RS01_000.34_00011
RS01_000.36_00012
RS01_000.37_00013
RS01_000.39_00014
RS01_000.42_00015
RS01_000.44_00016
RS01_000.46_00017
RS01_000.49_00018
RS01_000.51_00019
RS01_000.54_00020
```

The first line is the number of files to create the frame: files from 1 to 20 by increments of 1

The second line is the directory of the 20 files.

20 files' name are listed.

Automatic Selection of Frames for Deformed Shape Plotting of Time History

Time History Frame Selection Option

Input File: *.thani

```
1 20 1
C:\AP1000_STICK\XDIR\ACC
ACC_00.000_00001
ACC_00.005_00002
ACC_00.010_00003
ACC_00.015_00004
ACC_00.020_00005
ACC_00.025_00006
ACC_00.030_00007
ACC_00.035_00008
ACC_00.040_00009
ACC_00.045_00010
ACC_00.050_00011
ACC_00.055_00012
ACC_00.060_00013
ACC_00.065_00014
ACC_00.070_00015
ACC_00.075_00016
ACC_00.080_00017
ACC_00.085_00018
ACC_00.090_00019
ACC_00.095_00020
```

The first line is the number of files to create the frame: files from 1 to 20 by increments of 1

The second line is the directory of the 20 files.

20 files' name are listed.

Frame Plotting and Combination Examples For MOTION TFU, RS, ACC and RELDISP THD Files

Frame_Combine.txt

```
17602
3 1 .\Coherent\Combined\ACC\ACC_00.000_00001
.\Coherent\XDIR\ACC\ACC_00.000_00001
.\Coherent\YDIR\ACC\ACC_00.000_00001
.\Coherent\ZDIR\ACC\ACC_00.000_00001
3 1 .\Coherent\Combined\ACC\ACC_00.005_00002
.\Coherent\XDIR\ACC\ACC_00.005_00002
.\Coherent\YDIR\ACC\ACC_00.005_00002
.\Coherent\ZDIR\ACC\ACC_00.005_00002
3 1 .\Coherent\Combined\ACC\ACC_00.010_00003
.\Coherent\XDIR\ACC\ACC_00.010_00003
.\Coherent\YDIR\ACC\ACC_00.010_00003
.\Coherent\ZDIR\ACC\ACC_00.010_00003
3 1 .\Coherent\Combined\ACC\ACC_00.015_00004
.\Coherent\XDIR\ACC\ACC_00.015_00004
.\Coherent\YDIR\ACC\ACC_00.015_00004
.\Coherent\ZDIR\ACC\ACC_00.015_00004
3 1 .\Coherent\Combined\ACC\ACC_00.020_00005
.\Coherent\XDIR\ACC\ACC_00.020_00005
.\Coherent\YDIR\ACC\ACC_00.020_00005
.\Coherent\ZDIR\ACC\ACC_00.020_00005
```

ACC_Combined.thani

```
1 3000 1
.\Incoherent\Combined\ACC
ACC_00.000_00001
ACC_00.005_00002
ACC_00.010_00003
ACC_00.015_00004
ACC_00.020_00005
ACC_00.025_00006
ACC_00.030_00007
ACC_00.035_00008
ACC_00.040_00009
ACC_00.045_00010
ACC_00.050_00011
ACC_00.055_00012
ACC_00.060_00013
ACC_00.065_00014
ACC_00.070_00015
ACC_00.075_00016
ACC_00.080_00017
ACC_00.085_00018
ACC_00.090_00019
```

Frame Selection for Contour Stress Plots for STRESS THS Files

Frames.txt

```
3000
   1
   2
   3
   4
   5
   6
........
 2995
 2996
 2997
 2998
 2999
 3000
3 5000
8 9 10
```

Stress Frames Step j

Soil Pressure Frames Ngrp, Nmax Groups j, 1

*.contani

```
1 3000 1
.\Combined\
stress_00.000_00001_sig
stress_00.005_00002_sig
stress_00.010_00003_sig
stress_00.015_00004_sig
stress_00.020_00005_sig
stress_00.025_00006_sig
stress_00.030_00007_sig
stress_00.035_00008_sig
stress_00.040_00009_sig
stress_00.045_00010_sig
stress_00.050_00011_sig
stress_00.055_00012_sig
stress_00.060_00013_sig
stress_00.065_00014_sig
stress_00.070_00015_sig
stress_00.075_00016_sig
stress_00.080_00017_sig
stress_00.085_00018_sig
stress_00.090_00019_sig
```

Batch Automatic Selection of Animation Frames for Contour Stress or Soil Pressure Plotting

Batch Frame Selection Option

Input File

```
20      99
SHELL_013_01374_SXX.THS
SHELL_013_02276_SXX.THS
SHELL_013_01337_SXX.THS
SHELL_013_00576_SXX.THS
SHELL_013_01645_SXX.THS
SHELL_013_01891_SXX.THS
SHELL_013_01920_SXX.THS
SHELL_013_02674_SXX.THS
SHELL_013_02185_SXX.THS
SHELL_013_02092_SXX.THS
SHELL_013_02458_SXX.THS
SHELL_013_02811_SXX.THS
SHELL_013_01430_SXX.THS
SHELL_013_01785_SXX.THS
SHELL_013_02249_SXX.THS
SHELL_013_01273_SXX.THS
SHELL_013_01488_SXX.THS
SHELL_013_00487_SXX.THS
SHELL_013_00372_SXX.THS
SHELL_013_00621_SXX.THS
```

The first number in the header line is the number of files to use to find critical frames.

The second number in the header line is the percent of the node or element maximum used to identify the critical frames.

After the header line, the files sets to be checked are listed.

Batch Processing for the Combination of Frames

This Example Combines Three frames by summation and SRSS and saves the results to Test_Combin_Frame.out AND Test_Combin_Frame2.out

Frame Combination Text:

```
2
3 1 Test_Combin_Frame.out
.\THout_00001
.\THout_00002
.\THout_00003
3 0 Test_Combin_Frame2.out
.\THout_00001
.\THout_00002
.\THout_00003
```

Algebraic Sum

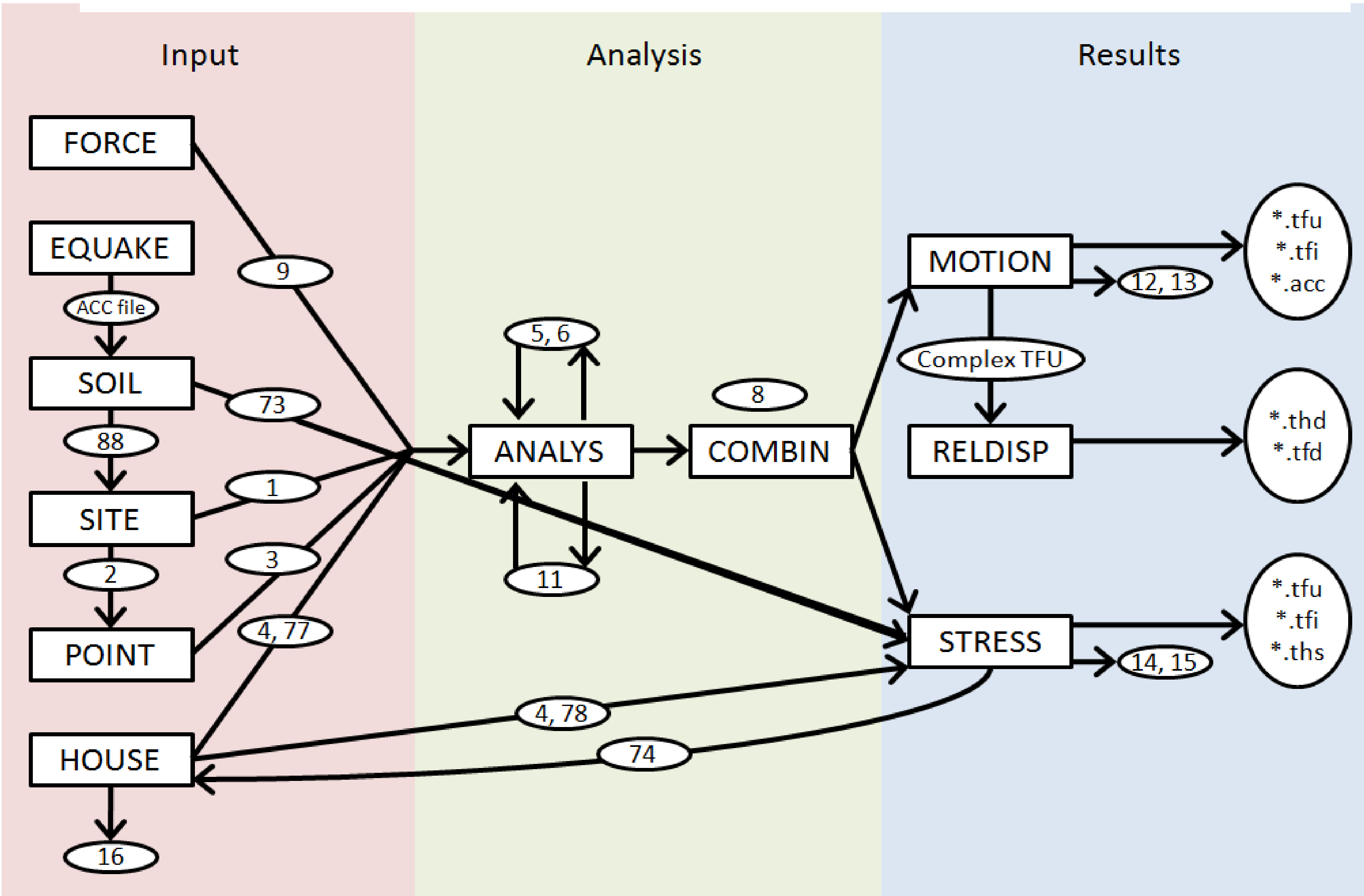
Number Of Files

SRSS

C. ACS SASSI Configuration and Use

- Modular Configuration
- Restart SSI Analysis Runs
- Building A Seismic SSI Analysis Model
- Hands-on Session …

ACS SASSI Modular Configuration



Description of SSI Modules

1. EQUAKE

The EQUAKE module generates earthquake accelerograms that are compatible with given ground response spectra. The input file has extension .equ and it is created by the ACS SASSI PREP AFWRITE command. A time-varying correlation can be specified for the horizontal components.

The user can also use recorded accelerograms to control the phasing for the generated three-component accelerograms. The generated accelerograms are then be input in the site response analysis and SSI analysis through SOIL, MOTION and STRESS modules. For details on this, see also the V&V problem # 29 in the Verification Manual of the NQA version.

The NQA version EQUAKE in addition to the requested output files will produce a file with extension .psd that is a comparison of the calculated PSD versus target PSD for the RG 1.60 spectrum as defined in SRP 3.7.1.

.

2. SOIL

The SOIL module performs a nonlinear site response analysis using an equivalent linear model for soil hysteretic nonlinear behavior. The input file has extension .soi and it is created by the ACS SASSI PREP AFWRITE command.

The SOIL module is based on the SHAKE code methodology with some additional programming improvements done over years. The computed equivalent soil properties can be sequentially used in the SSI analysis. In addition to the output file, SOIL produces also other text files with extension .TH that are response time histories for plotting purposes. The TH files include time histories for accelerations (ACCxxx), soil layer strains (prefix SNxxx) and stresses (prefix Ssxxx). The xxx notations refers to free-field soil layer number (numbering is done from the ground surface to the depth).

SOIL also produces the text file File73 that contains the material soil curves that are used for the nonlinear SSI analysis by the STRESS module, and File88 with the iterated, equivalent linear or effective soil properties that are used by SITE is nonlinear SSI option is selected by the user.

4. Module SITE

The SITE module solves the site response problem. The input file has extension .sit and it is created by the ACS SASSI PREP AFWRITE command.The control point and wave composition of the control motion has to be defined in the input files . The information needed to compute the free-field displacement vector used is computed and saved on disk in File1. The program also stores information required for the transmitting boundary calculations in File2. The actual time history of the control motion is not required in this program module, but later in the MOTION module. The soil motion incoherency is introduced elsewhere, in the HOUSE module. In addition to the output and binary files File1 and File2, SITE also produces the text file IncohDirection file that contains a flag for the HOUSE module that is used when the incoherent SSI analysis option is selected.

5. Module POINT3 (or POINT2)

The POINT module consists of two subprograms, namely POINT2 and POINT3 for two- and three-dimensional problems, respectively. The input file has extension .poi and it is created by the ACS SASSI PREP AFWRITE command.The POINT module computes information required to form the frequency dependent flexibility matrix. The results are saved on File3. File2 created by program module SITE is required as input. Thus, the SITE module must be run before the POINT2 or POINT3 module.

6. Module HOUSE

The HOUSE module forms the mass and stiffness matrices of all the elements used in discretized model are determined and stored in File4. The input file has extension .hou and it is created by the ACS SASSI PREP AFWRITE command. The discretized model may include only the structure or also the irregular soil zone. The random field decomposition for incoherent motions is performed in this module. The HOUSE results for incoherent SSI are stored in File77 to be used by ANALYS. If the user wants to check the accuracy of the coherence kernel decomposition, HOUSE produces the text file File16. File 16 could be a very large size file. Therefore, we suggest select the coherence decomposition accuracy checking option only when it is very needed and justified.

HOUSE also produces the text file File78 that is a non-empty file only if nonlinear SSI analysis option is used. File78 is used by STRESS during the SSI nonlinear iterations.

The HOUSE module can be executed independent of SITE and POINT modules if the coherent SSI analysis option is used. If incoherent SSI analysis option is selected, then HOUSE has to be run after SITE.

HOUSE also incorporated an optimizer for node numbering. If the node renumbering option is selected a new HOUSE input text file with extension .hownew is saved in the working directory. This file contains the new optimized SSI model. This file will be used by ANALYS for computing the SSI solution for the optimized SSI model. This node numbering optimization can reduce significantly the SSI analysis run time especially for large-size SSI models with significant embedment that require very large run times of several thousand seconds per each SSI frequency.

9. Module ANALYS

The ANALYS module computes the problem solution for the required frequency steps. The input file has extension .anl and it is created by the ACS SASSI PREP AFWRITE command. Files1, File3 and File4 are always required as input files. For external load cases File9, and for incoherence analysis Files77 are also required as input.

ANALYS performs the following computational steps:

- Forms the flexibility matrix for the discretized model.
- Computes the impedance matrix for the discretized model.
- Determines the external load or seismic load vectors, including incoherency effects
- Solves the equation system for each frequency step, using triangularization and back-substitution algorithms and obtains transfer functions for each degree of freedom.

The solution output computed by the ANALYS module contains the complex transfer functions which depending on the option required are from the control motion to the final motions or from external loads to total displacements.

In either case, the SSI TF results are stored in File8 that is used by MOTION and STRESS for computing SSI responses. File5 and File6 are unformatted SSI solution database files with large sizes. These files are useful to be saved if repeated SSI reanalysis are needed; for example if the coherent SSI analysis is performed for a number of acceleration input time histories; or nonlinear SSI is used; or if the incoherent SSI analysis is done using the stochastic simulation or SRSS approach.

If the global, rigid body impedance analysis option is selected, ANALYS also produces File11 that is a quite large size file (this option selection is to be avoided if rigid body impedances are not needed by the user).

Interpolation of transfer functions in frequency domain and further output requirements are handled by the modules described below.

10. Module MOTION

The MOTION module reads the transfer functions from File8, and performs an efficient frequency domain interpolation using a complex domain scheme based on the 2 DOF complex transfer function model that has five parameters to be determined. The input file has extension .mot and it is created by the ACS SASSI PREP AFWRITE command. The interpolated transfer functions are then, used to compute the SSI response motions at a set of nodes selected by the user.

Acceleration, velocity, or displacement response spectra may be requested in different location points and degrees of freedom. The MOTION module requires only File8 as input. If baseline correction is used (this is a much more approximate solution to get relative displacements in a structure than using the RELDISP module), the nodal point motions are saved on File13 which a formatted file.

In addition to the output file that could be often very large size (if time histories are saved), MOTION produces specific text files for post-processing. These text files include the extension .TFU, .TFI, .ACC, .RS files that contain nodal SSI responses for the three translation DOF, respectively, the computed TF (TFU), interpolated TF (TFI), acceleration time histories (ACC) and the in-structure response spectra (RS) for selected damping values.

These text file names are xxxxxTR_y.ext, where xxxxx is the node number, y is the DOF that can be X, Y or Z, and .ext is the extension that can be TFU, TFI or ACC. For response spectra files, the names are xxxxxTR_yzz.RS, where zz is the order number of the damping ratio value (for example, 01 and 02 for two selected values of the damping ratio of 0.02 and 0.05). See Table 1 for more details on the SSI response text files.

If the MOTION post-processing restart option is used, then additional text files for post-processing are generated in the \TFU, \RS and \ACC subdirectories. These frame text files contain the SSI response values computed for all active nodal DOF at each frequency step or time step. These frame files are used by the ACS SASSI PREP module to create structural bubble plots, TF vector plots, contour plots, or deformed shape animations. See Table 2 for more details on frame text files.

11. Module RELDISP

The RELDISP module uses the acceleration complex TF computed by MOTION (TFI files) to compute analytically the relative displacements at different selected nodes. The input file has extension .rdi and it is created by the ACS SASSI PREP AFWRITE command. RELDISP produces and output file with the computed maximum nodal relative displacements. It also produce extension .TFD and .THD files that contain the nodal relative displacement complex TF and the relative displacement time history. Their names are similar to extension .TFU and .ACC files produced by MOTION. See Table 1 for more details on the SSI response text files.

If the RELDISP post-processing restart option is used, then additional text files for post-processing are generated in the \THD subdirectory. These frame text files contain the SSI response values computed for all active nodal DOF at each time step. These frame files are used by the ACS SASSI PREP module to create structural deformed shape animations. See Table 2 for more details on frame text files.

12. Module STRESS

The STRESS module computes requested stress, strain, and force time histories and peak values in the structural elements. The input file has extension .str and it is created by the ACS SASSI PREP AFWRITE command. The module STRESS requires File4 and File8 as inputs. Stress time histories are saved on File15, and the computed transfer functions of stresses or forces and moments are saved on File14. File15 and File 14 are text files. In addition to these text files, STRESS also produces File74, if the nonlinear SSI analysis option is employed. For nonlinear SSI, STRESS also uses File78 produced by HOUSE as an input.

In addition to the output file STRESS produces also some specific text files useful for post-processing. These text files include the extension .TFU,.TFI and .THS that contain structural element stress responses in each selected element, respectively, the computed TF (TFU), interpolated TF (TFI) and stress time histories (THS). These text file names have the format etype_gnum_enum_comp plus extension; for example, BEAMS_003_00045_MXJ that contains the MX moment at node J for the BEAM element number 45 that belongs to Group 3. See Table 1 for more details on SSI response text files.

The STRESS module in addition to the above files also generates an important text file named ELEMENT_CENTER_ABS_MAX_STRESSES.TXT that contains the maximum element stress components (calculated by STRESS) for all the selected elements by the user.

If the STRESS post-processing restart option is used, then additional text files for post-processing are generated in the \NSTRESS subdirectory. These frame files are used by the ACS SASSI PREP module to create structural node stress contour plots, static (for a selected time or for maximum stress values) or animated. The STRESS post-processing handles only SOLID and SHELL elements for 3D SSI models. If the SSI model contains both SOLID and SHELL elements, the frames include only average node stresses for the membrane stresses. For the SHELL elements only, separate frames are generated for the average node bending stresses (the file extension include letters bd from bending). See Table 2 for more details on frame text files.

f the SSI model includes near-field soil elements that are adjacent to the foundation walls, then the soil pressure frames can be generated. The soil pressure frames are saved in \SOILPRES subdirectory. In addition to the seismic soil pressures frames at each time step, a single frame with maximum soil pressures is also generated. The user can also create total soil pressure frames including the static bearing pressures plus the computed seismic pressures. The static pressure text file is named STATIC_SOIL_PRESSURES.TXT and is generated when the soil pressure frames are requested. When it is generated the first time by the STRESS restart analysis for soil pressure option, the static pressure file has only zero values

. Then, if the user inputs the non-zero static pressure values and runs again the STRESS post-processing restart for soil pressure option, these non-zero static pressures are added to the seismic pressures values using algebraic summation and the total soil pressures are saved in the soil pressure frames stored in the \SOILPRES subdirectory.

If the soil pressure restart option is used, then, other two text files are generated, namely pres_max_ele and pres_max_nod files. They contain the maximum element soil pressures (calculated by STRESS) and the average nodal soil pressures (approximate values to be used only for plotting purpose) in the SOLID elements that model the adjacent near field soil.

NOTE: It should be noted that the STRESS frame files contain average nodal stresses and average nodal pressures to be used only for plotting purposes. The element nodal stresses and soil pressures were computed directly from the SOLID element center stresses or pressures (normal stress to the solid element face). The element nodal stress was assumed to be equal to element center stress that introduce a certain level of approximation of the nodal stresses (no shape functions are used). In addition, the nodal averaging process could produce stresses and pressures could produce values that are difficult to interpret and use.

The accurate stress and soil pressure values to be used by the analyst for the SSI calculations and seismic design are the computed values in the element centers (that are provided in the STRESS outputs, or the text files called ELEMENT_CENTER_ABS_MAX_STRESSES.TXT and pres_max_ele), not the nodal average values. However, the average nodal stress and soil pressure add invaluable information for understanding the SSI model seismic behavior and for identifying the critical stress zones, or critical pressure areas on the foundation walls and mat.

For the nonlinear SSI analysis option, STRESS generates the File74 after each SSI iteration. File74 is then used by HOUSE for the next SSI iteration.

12. Module COMBIN

The COMBIN module combines results computed for different frequencies from two ANALYS runs. This module is useful when after the solution was obtained it is found that some additional frequencies are needed to be included. The COMBIN module requires as input two solution files of File8 type, renamed File 81 and File 82. The output file of this module is a new File8 obtained by combining the two old solution files.

Batch SSI Analysis 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 CD-ROM 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 PREP AFWRITE command.

Restart SSI Analyses

The restart analyses imply that large files (File 5 and File 6) were saved. The following changes of problem parameters need different levels for the restart analyses:

1. **Change in the Control Motion**
   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. **Change in Seismic Environment**
   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.

   If the incoherency of seismic motion is changed, then the HOUSE module has to be re-executed also for creating a new File 77 for ANALYS input.

Restart SSI Analyses (cont.)

3. **Change in Structure or Near-Field Soil**
   If changes are made in the superstructure or in the motion incoherency characteristics, the HOUSE, ANALYS and MOTION modules have to be re-executed. Only File5 is needed for restart.

In general, we recommend the application of the FI-EVBN method that provides both numerical accurate and reasonable computational speed when compared with the reference FV method. The FI-EVBN method is several times faster than the FV method and only few times slower than the FI-FSIN method.

For the application of the FI-FSIN method for soil sites, we always recommend a preliminary sensitivity study to check it against the FV tor FI-EVBN methods, especially for situations with foundation excavation in very soft soils (or backfill soils). The FI-FSIN could be sometime numerically unstable in the higher frequency range depending on the surrounding soil stiffness and the excavation volume configuration. For stiffer soil sites or rock sites, the FI-FSIN method is expected to provide highly accurate results coincident with the FV and FI-EVBN method results.

It should be also noted that the FV method is typically more robust to excavation volume horizontal mesh size than the FI methods. The FI-FSIN is especially sensitive to horizontal mesh variation in excavation volume. FI-FSIN becomes unstable in higher frequency ranges much faster than FI-EVBN.

.

2010 ACS SASSI-ANSYS Integration for Refined Seismic Structural Stress Analysis and Soil Pressure Computations

ACS SASSI-ANSYS Interface for Seismic Soil-Structure Interaction Analysis of Nuclear/Critical Facility Structures

ACS SASSI-ANSYS Interface provides new SSI analysis capabilities through ANSYS:

For structural stress analysis:

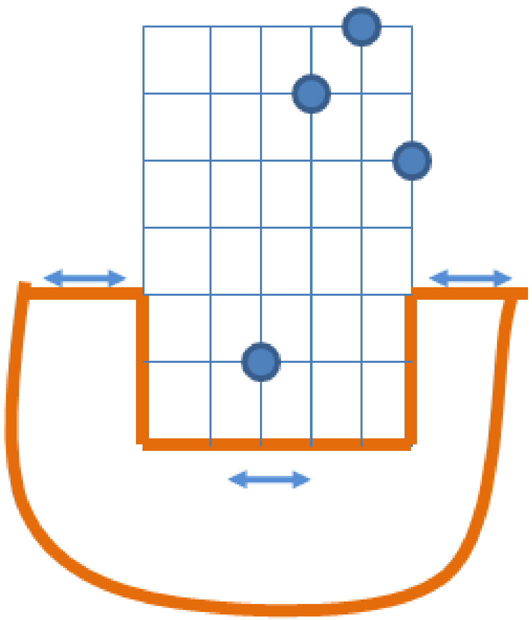
- *ANSYS Equivalent-Static Seismic SSI Analysis* Using Refined Mesh FE Models
- *ANSYS Dynamic Seismic SSI Analysis* Using Nonlinear or More Refined FE Models (including refined mesh, element types including local nonlinearities, nonlinear materials, contact elements, etc.)

For soil pressure computation:

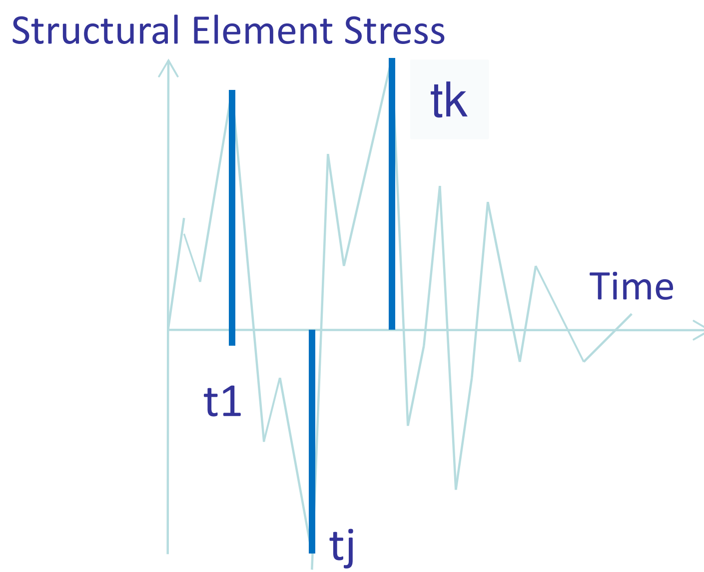
- *ANSYS Equivalent-Static Seismic Soil Pressure Computation* Including Soil-Foundation Separation Effects

ACS SASSI Seismic SSI Analysis

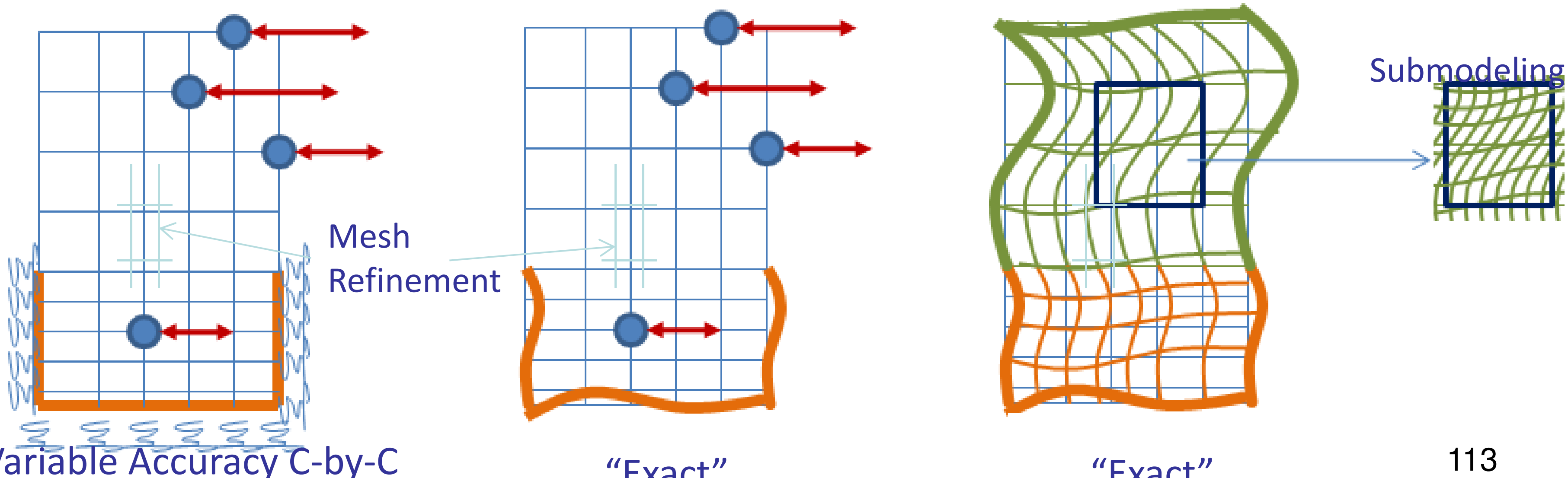
Computing Structural Stress/Forces



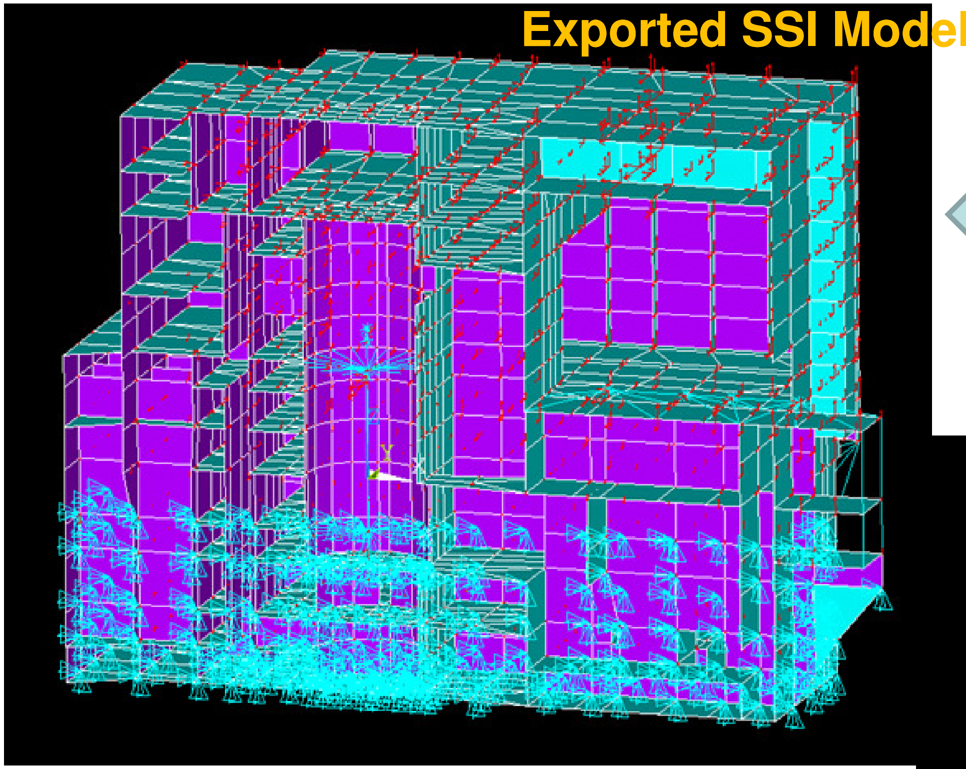
Selected Critical Time Steps for Maximum Stresses
To be Used for Equivalent Static Structural Analysis



SSI Solution Time Frames As Equivalent Static Structural Loading at Critical Time Steps

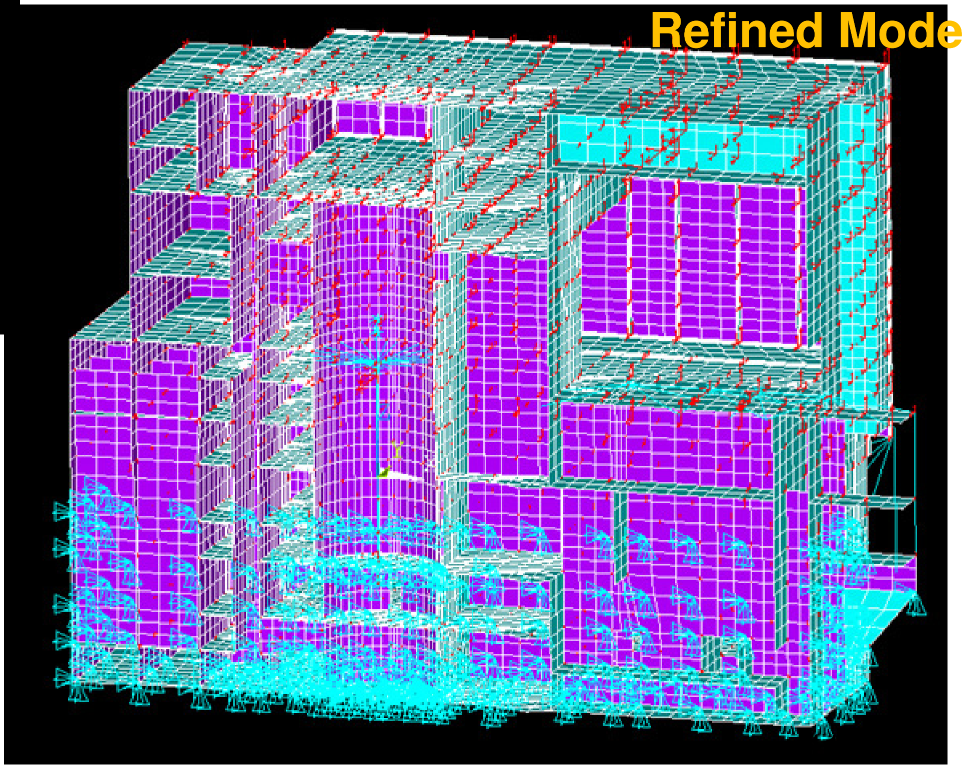


ACS SASSI – ANSYS Interface for Refined Seismic Stress Analysis



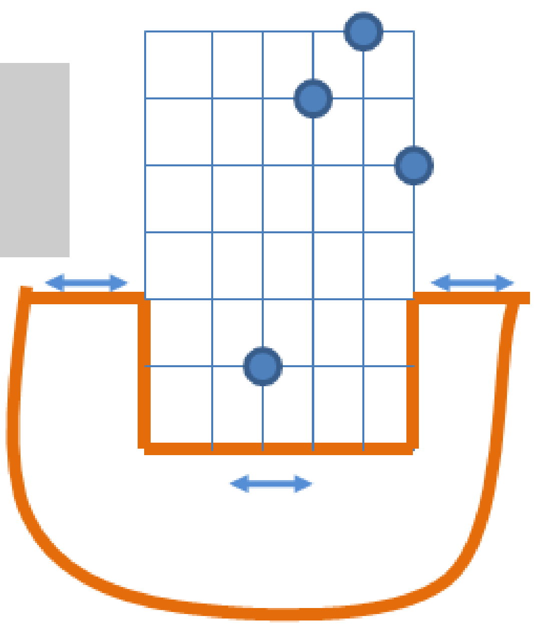
ANSYS Structural Model Automatically Converted From ACS SASSI Using PREP Module

ANSYS Refined Structural Model Using EREFINE command or ANSYS GUI (rank 1-6)

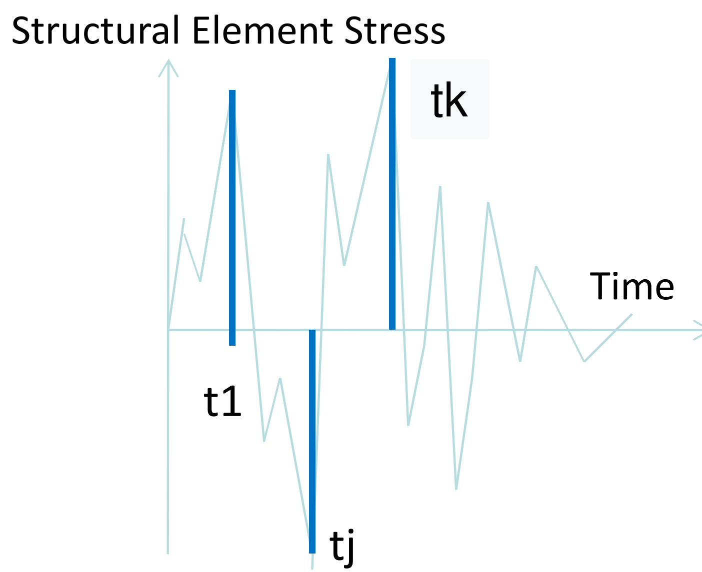


ACS SASSI Seismic SSI Analysis

Computing Seismic Soil Pressures

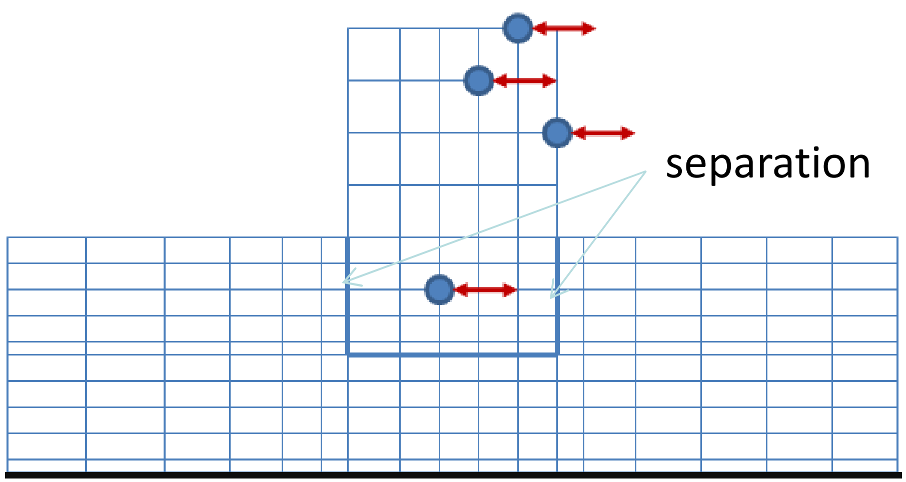


Selected Critical Time Steps for Maximum Stresses
To be Used for Equivalent Static Structural Analysis

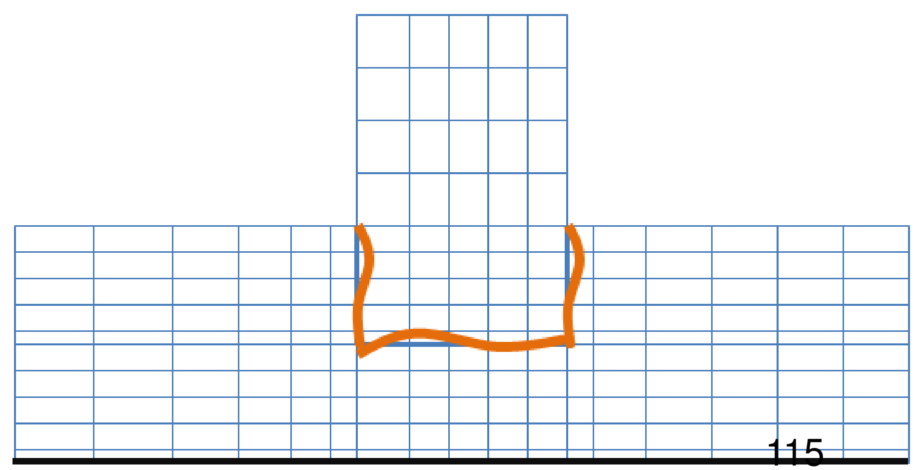


SSI Solution Time Frames As Equivalent Static Loading at Critical Time Steps

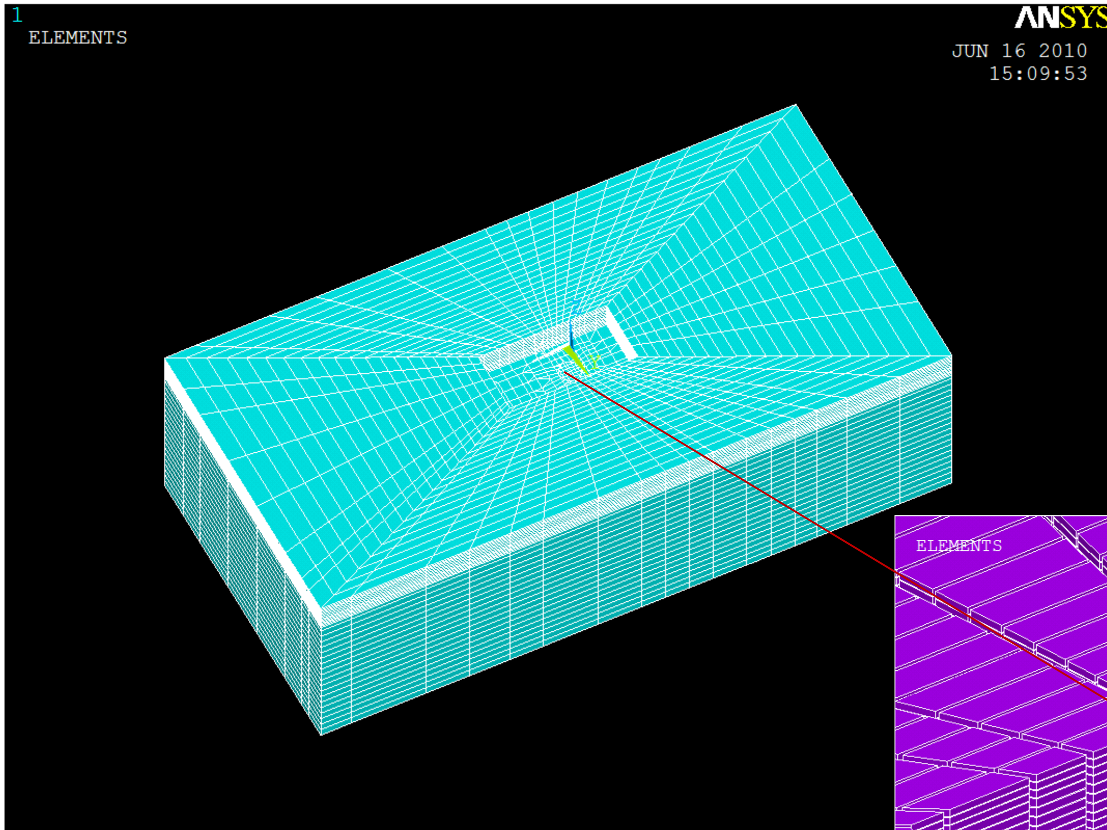
EQS Forces – Linear & Nonlinear



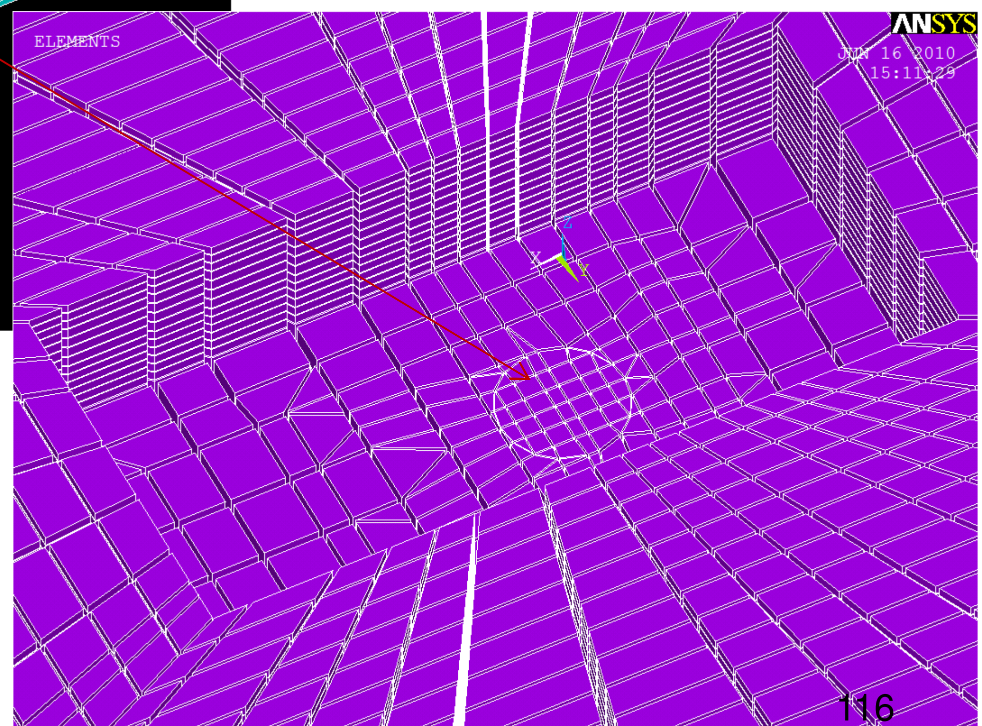
EQS Relative Displacements – Linear (Welded)



ACS SASSI – ANSYS Interface for Seismic Soil Pressure Analysis

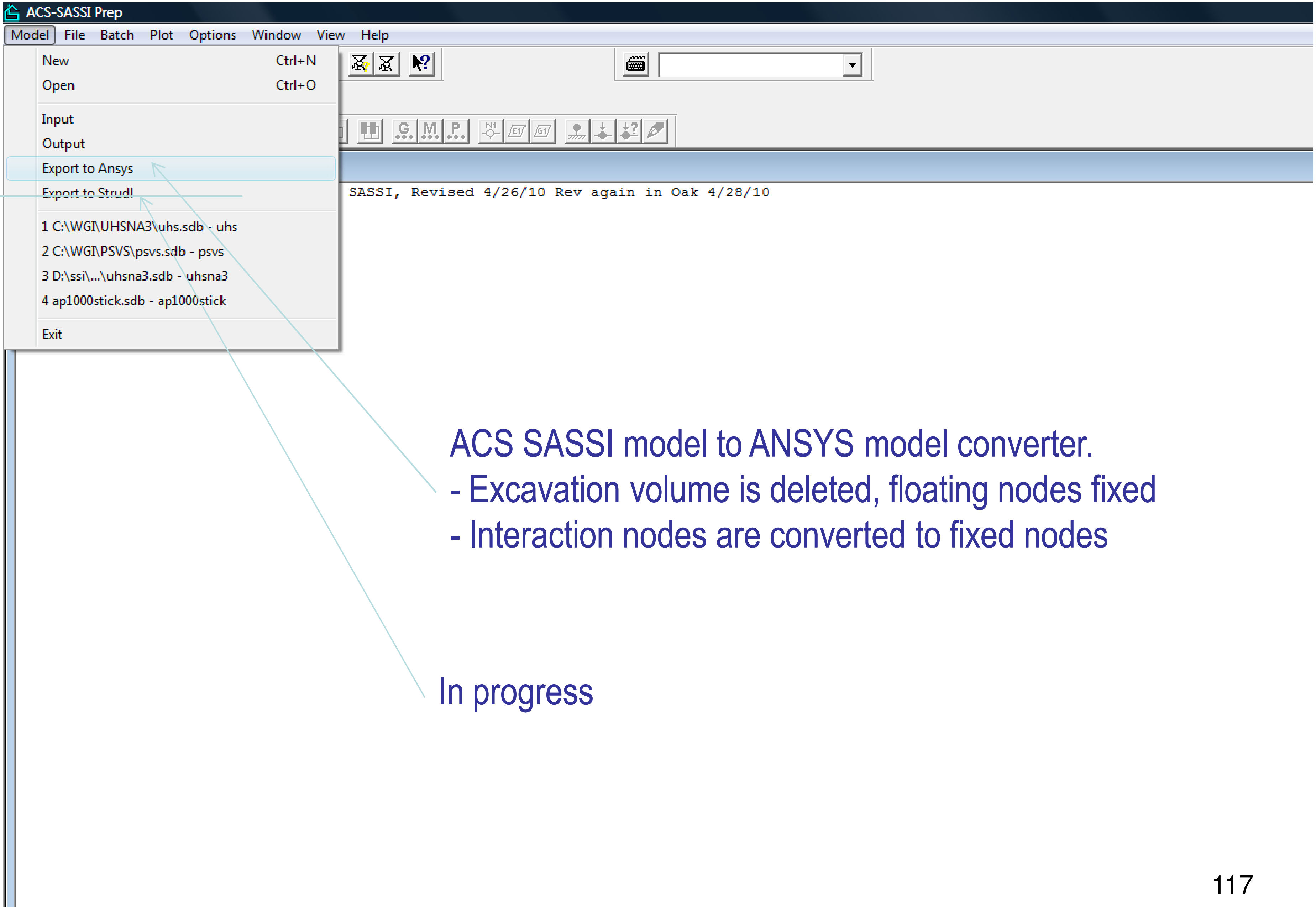


ANSYS Soil FE Model
Is Automatically Generated
by SOILMESH Module



Embedment mesh is extended.
User controls extension size and
mesh density. Can use EREFINE.
Contact surfaces automatically added
By ACS SASSI SOILMESH module.

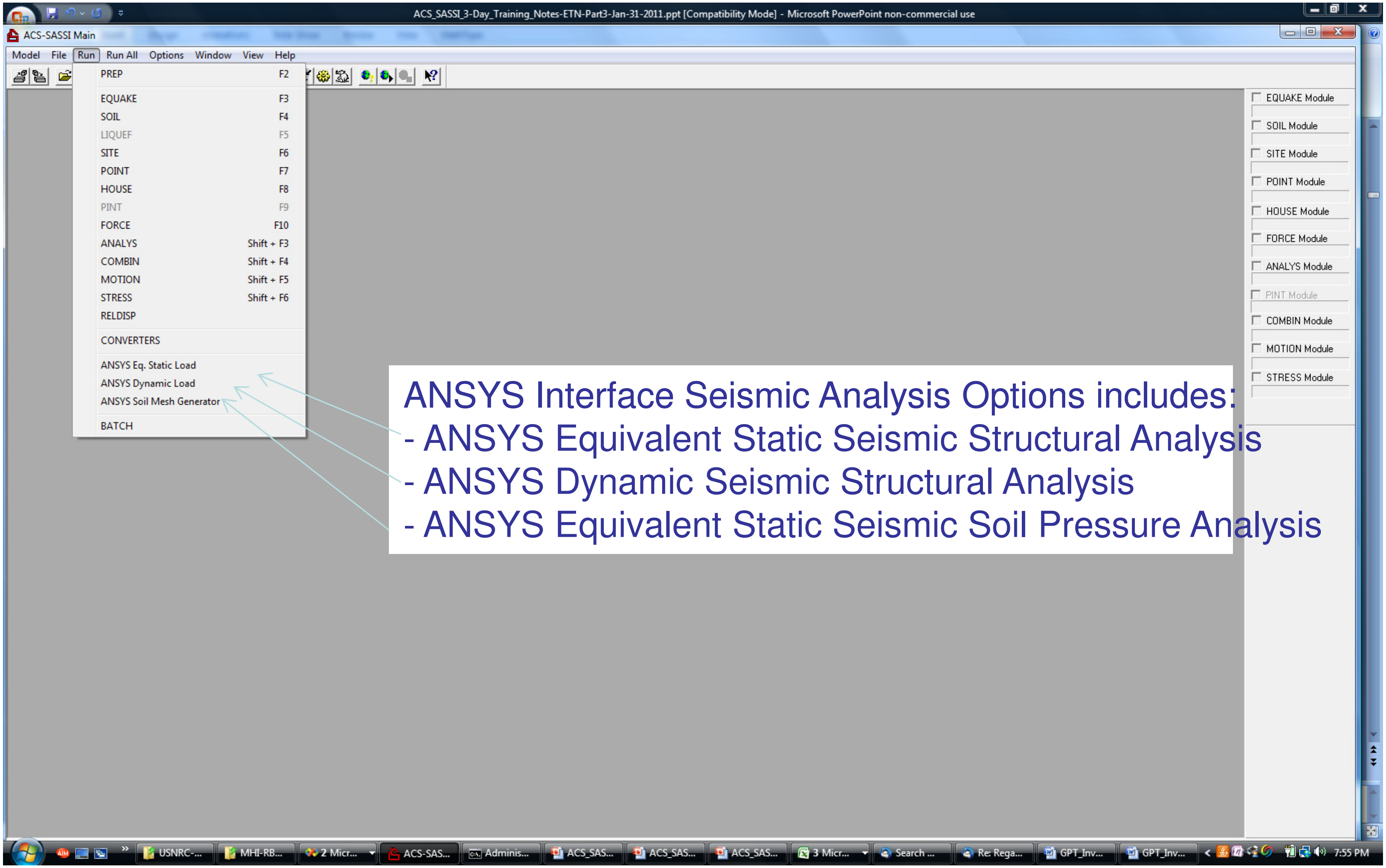
New Structural Model Converter from ACS SASSI to ANSYS



ACS SASSI model to ANSYS model converter.
- Excavation volume is deleted, floating nodes fixed
- Interaction nodes are converted to fixed nodes

In progress

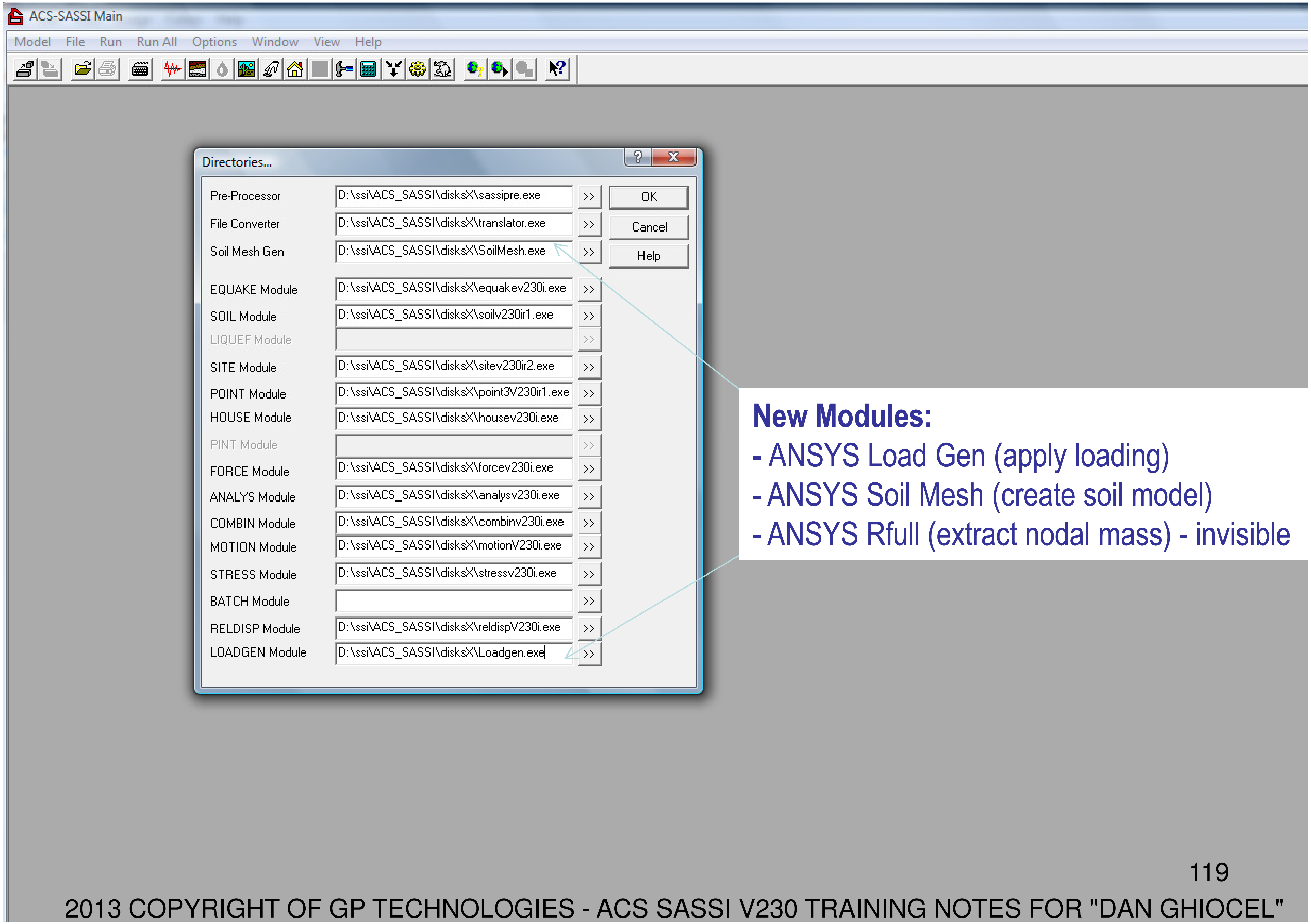
ACS SASSI – ANSYS Interface for Refined Seismic Stress Analysis



ANSYS Interface Seismic Analysis Options includes:

- ANSYS Equivalent Static Seismic Structural Analysis
- ANSYS Dynamic Seismic Structural Analysis
- ANSYS Equivalent Static Seismic Soil Pressure Analysis

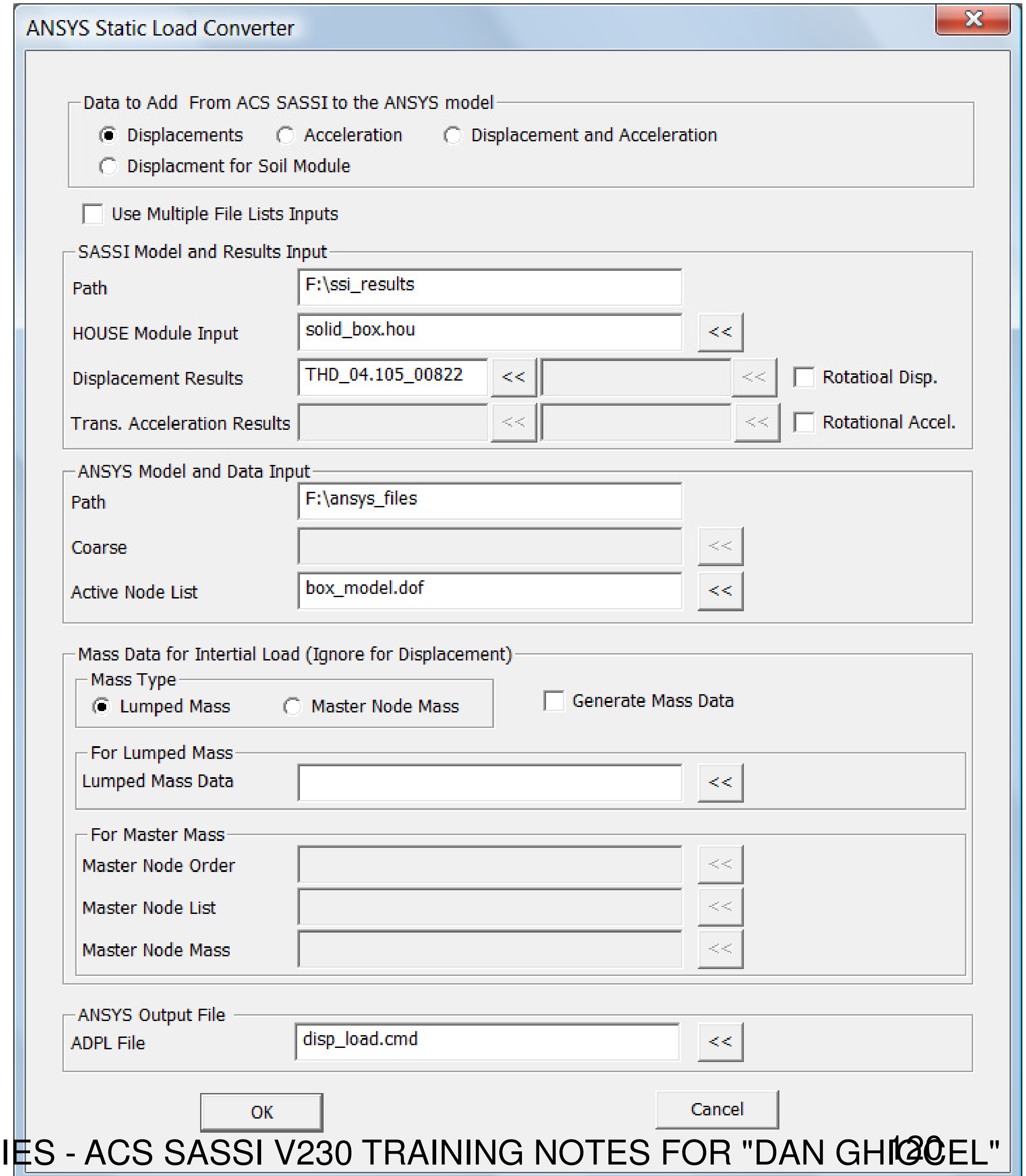
New ACS SASSI Modules for ANSYS Interface – for Structure & Soil



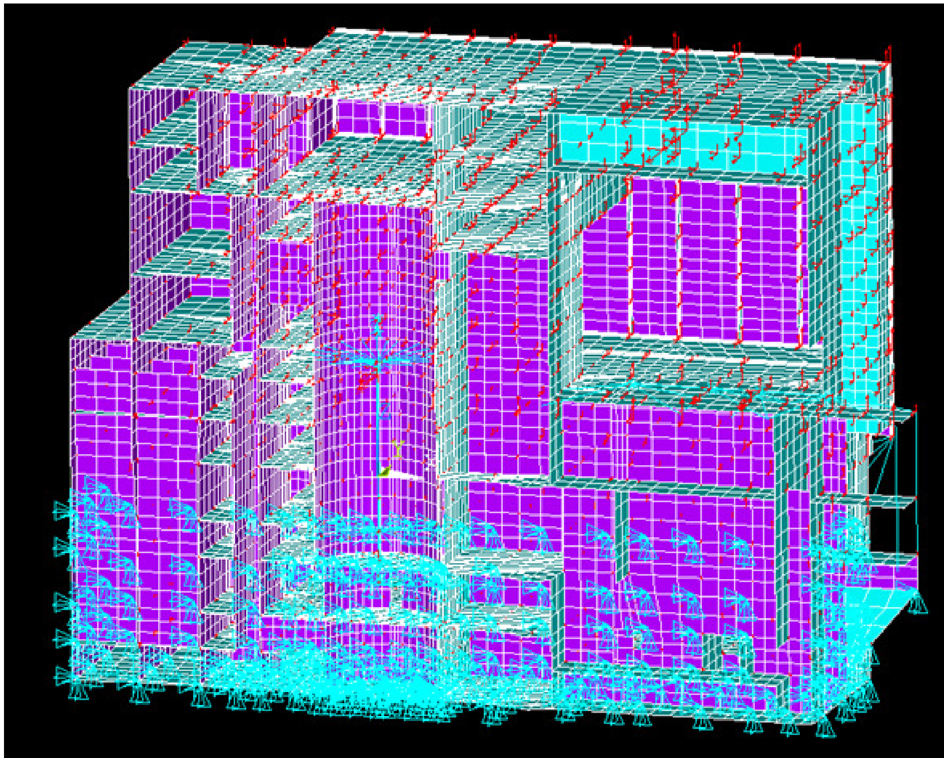
New Modules:
- ANSYS Load Gen (apply loading)
- ANSYS Soil Mesh (create soil model)
- ANSYS Rfull (extract nodal mass) - invisible

Exporting Equivalent Static Loads to ANSYS

- From ACS SASSI-MAIN select “ANSYS Static Load” from the Run menu
- Fill in the appropriate boxes as described in the documentation
- ANSYS APDL input files are created containing the load data are created when the user clicks “OK”

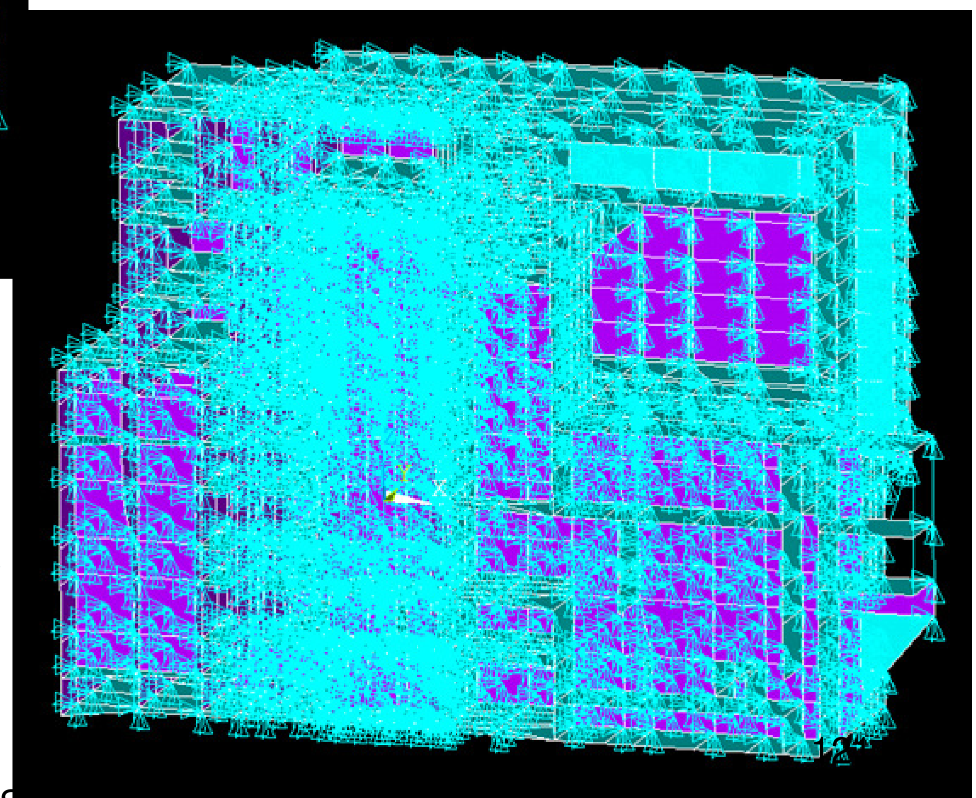


Exporting Equivalent Static Loads to ANSYS

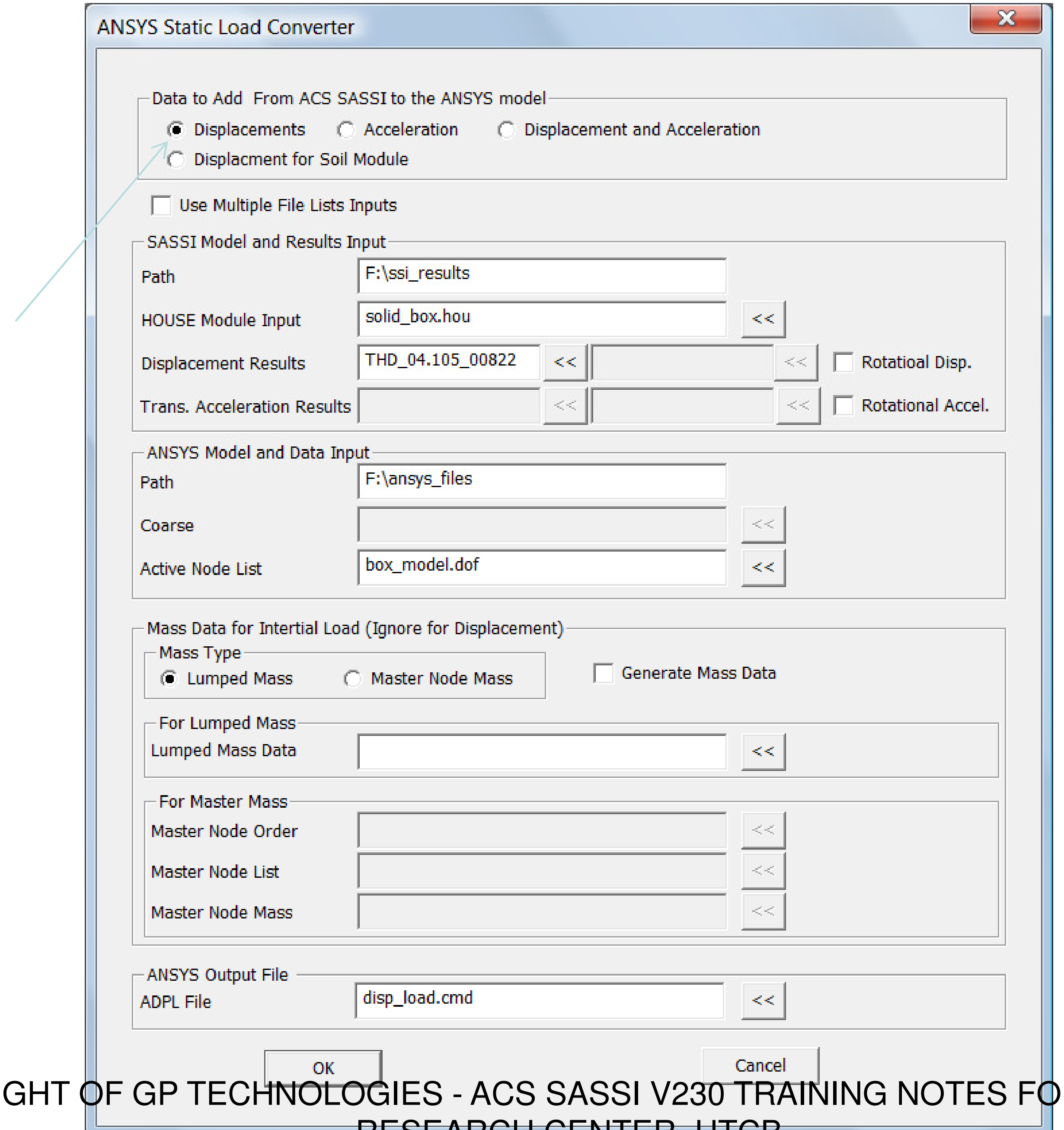


Acceleration & Displacements BC
(Uses ANSYS Refined Model Solut
- Accurate for Refined Stress Anal

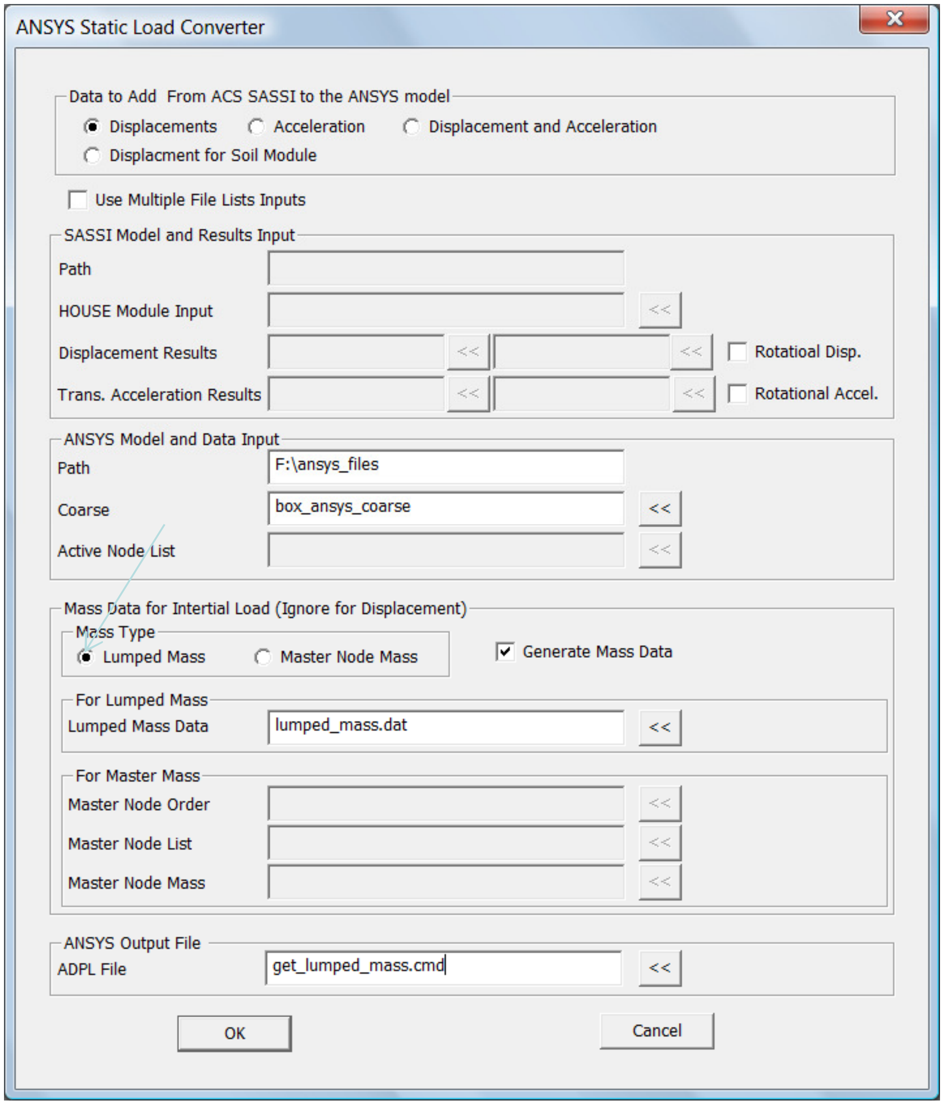
ANSYS Displacement BC
(Uses ACS SASSI Model Solution)
- Less Accurate for Refined Models
- Useful for subsystem input



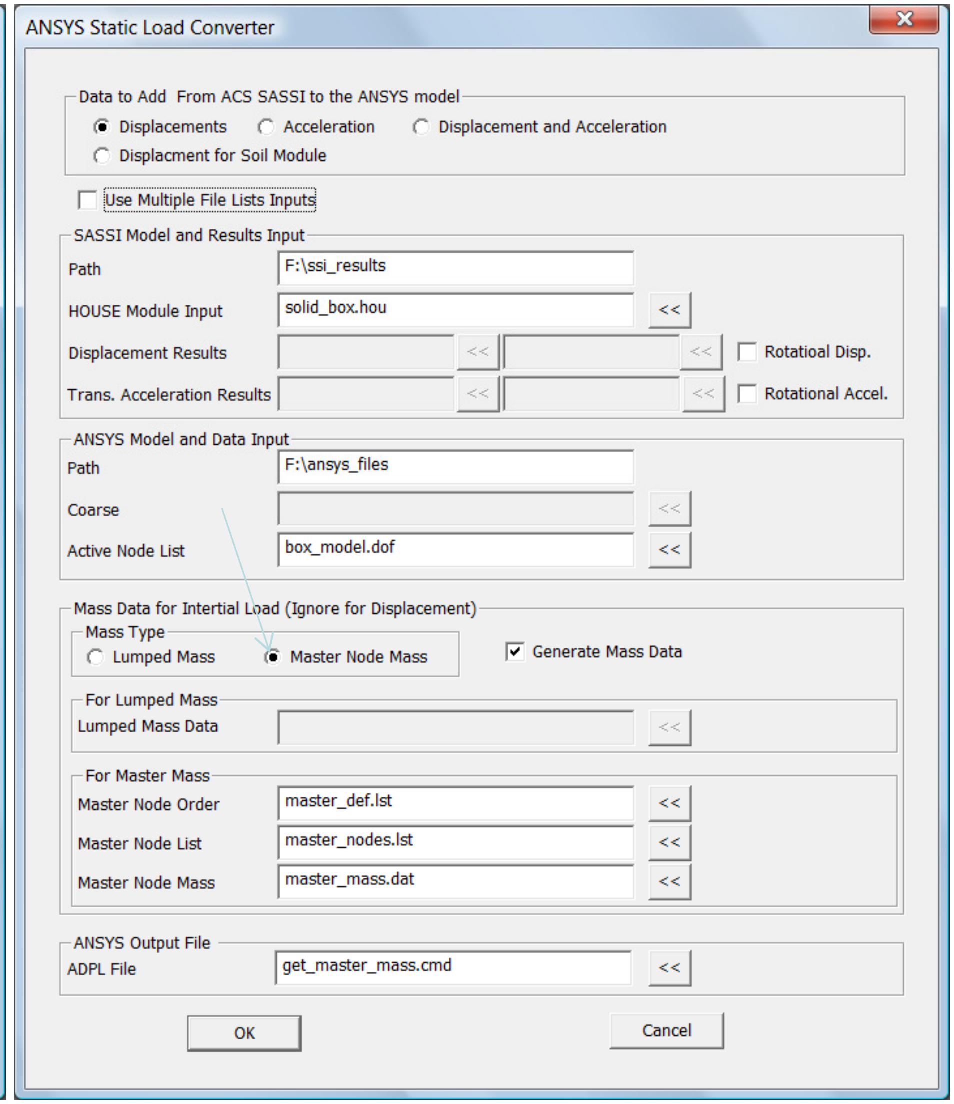
Displacement Option – Use SSI Model Solution



Acceleration Option – Select Nodal Mass Type

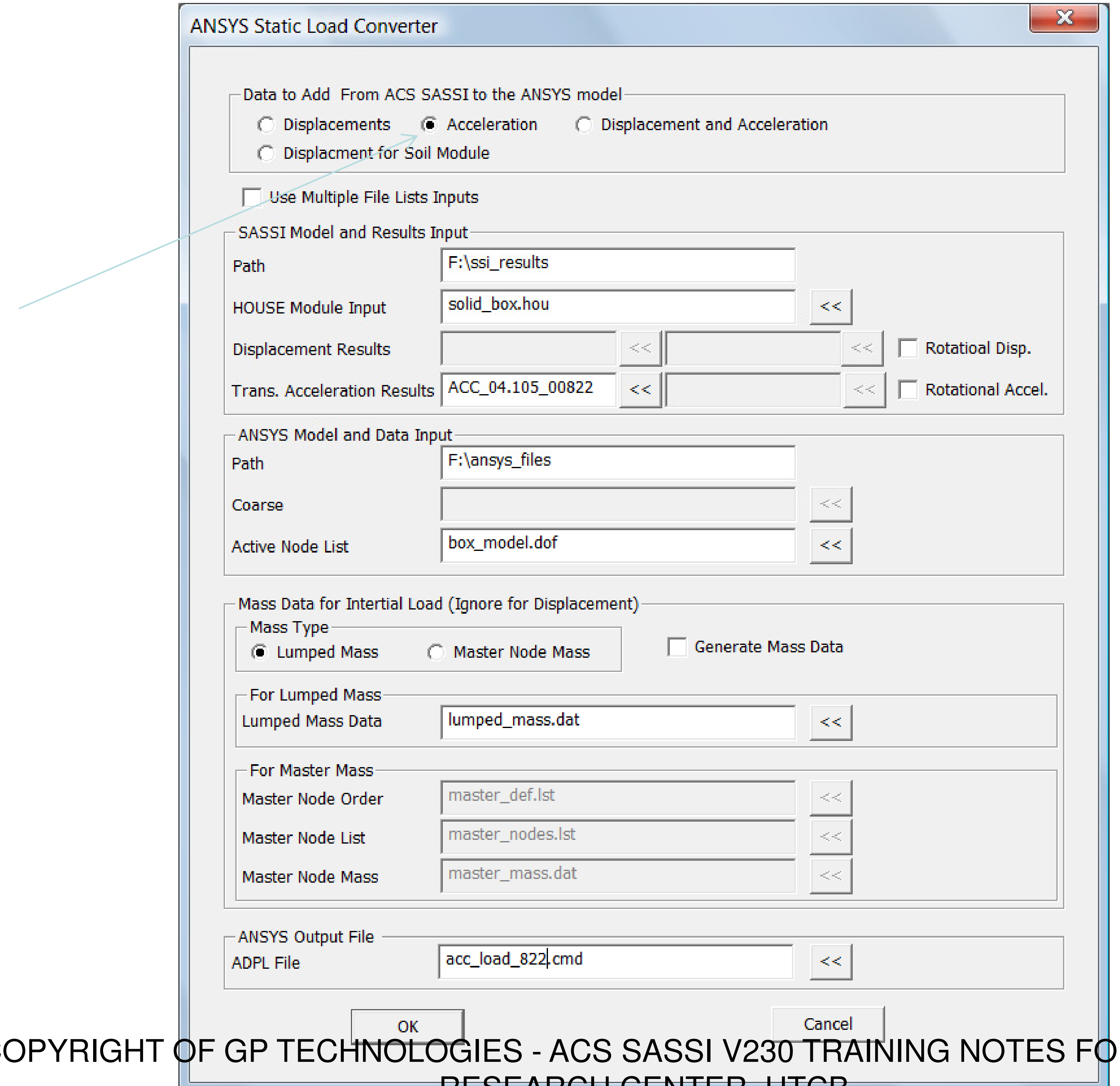


Lumped Mass/Coarse

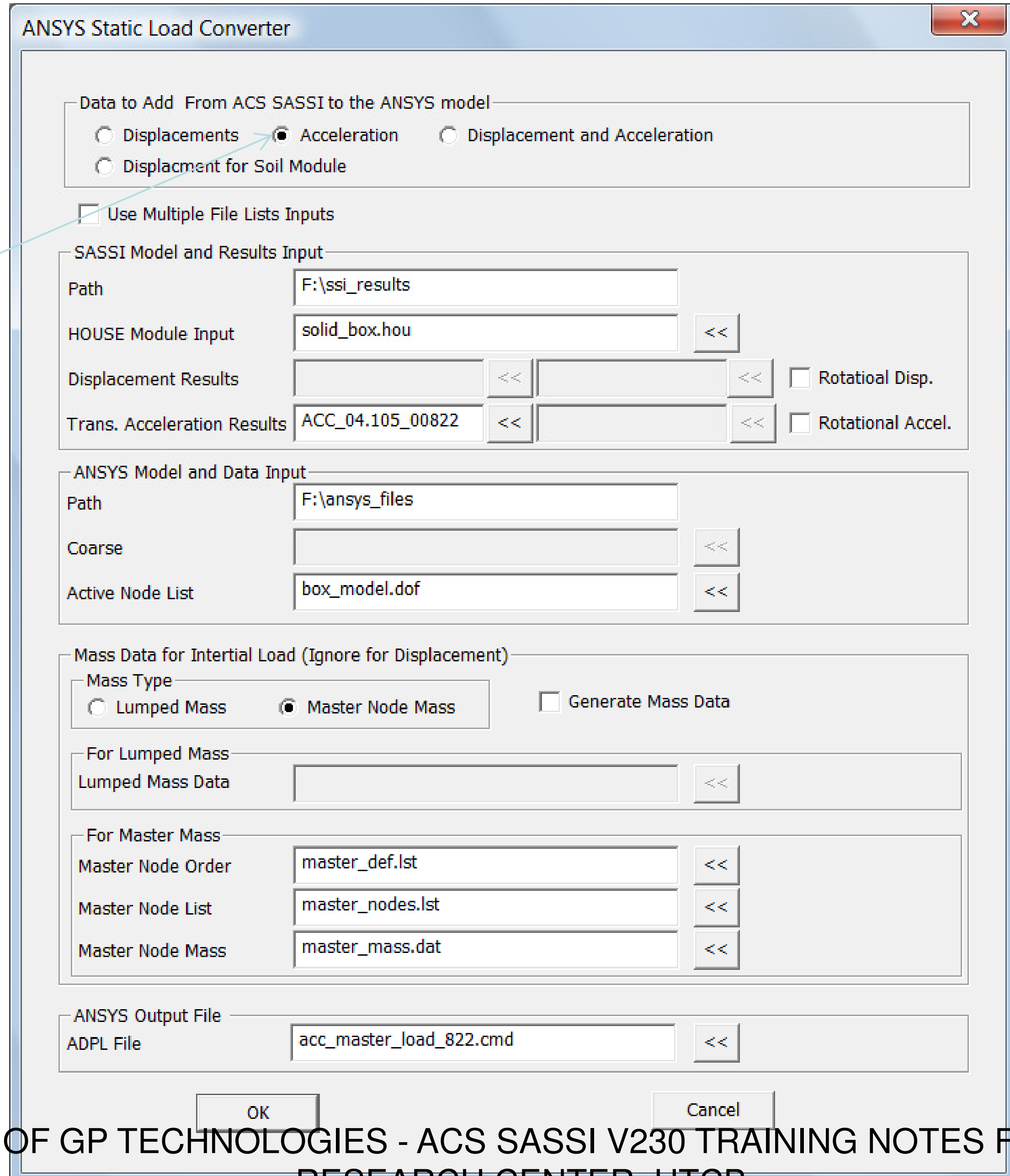


Master DOF Masses/Refined

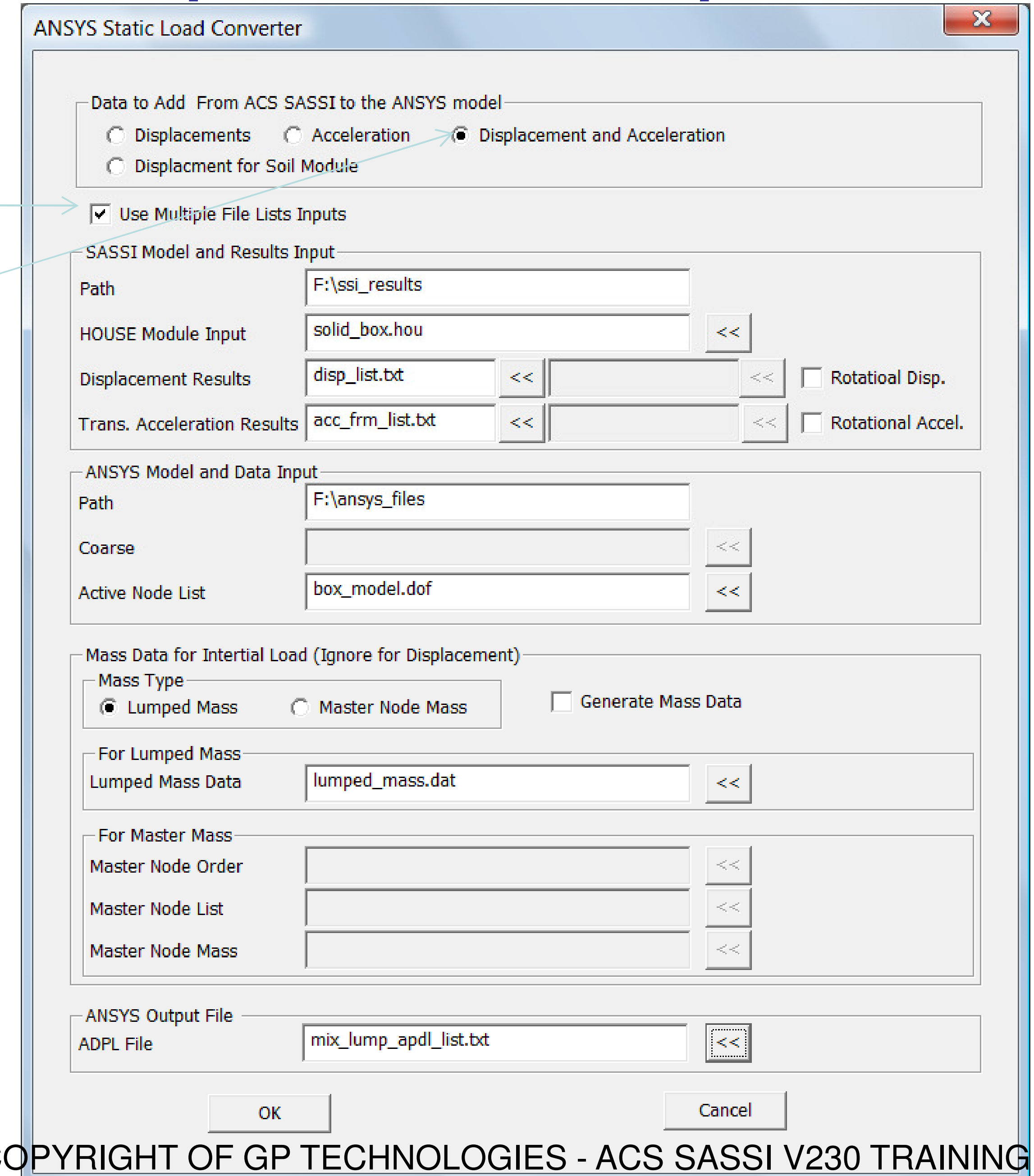
Acceleration Option – With Nodal Lumped Masses



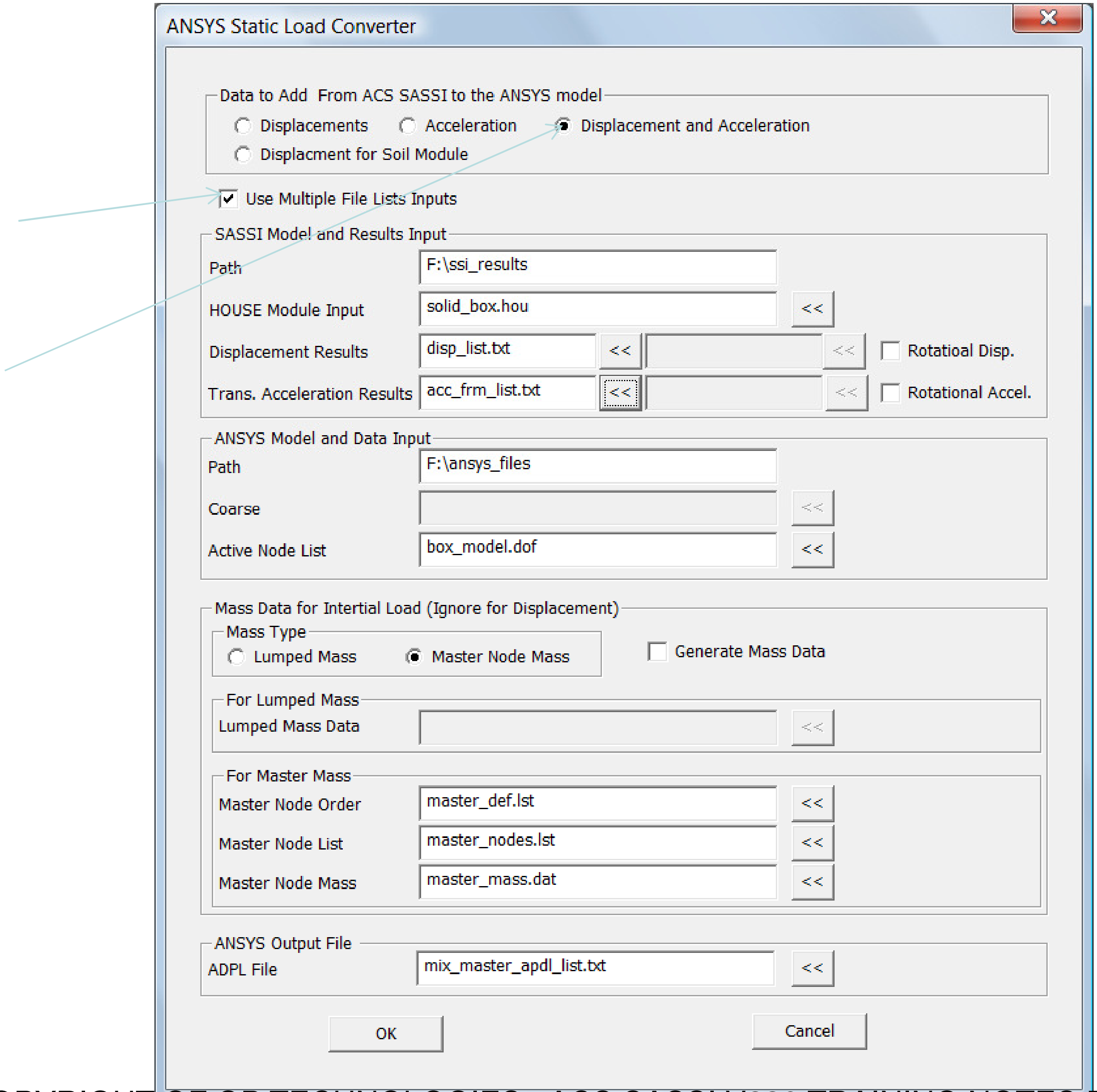
Acceleration Option – With Nodal Master DOF Masses



Mixed Option – With Lumped Masses



Mixed Option – With Master DOF Masses



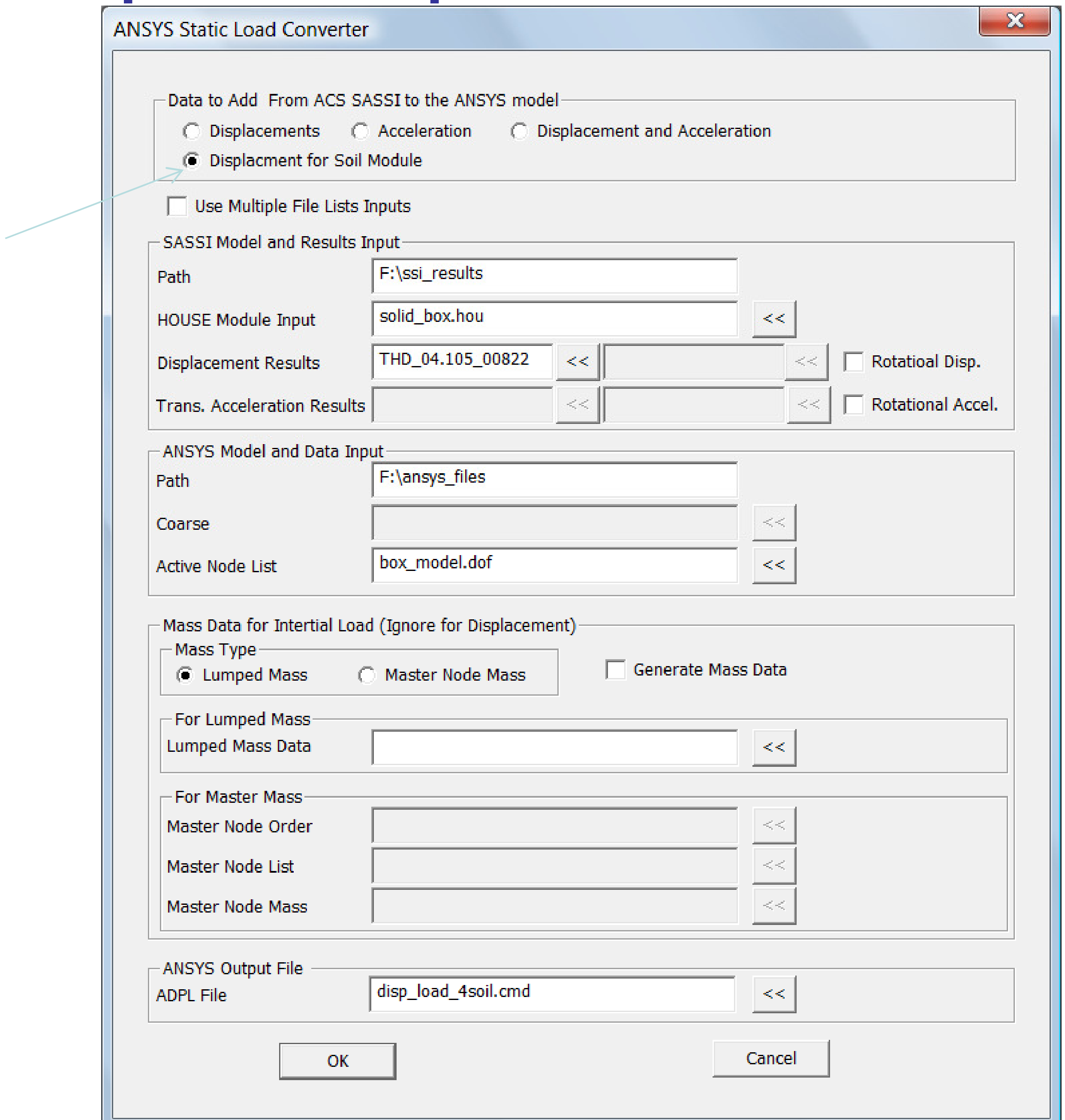
Example of Equivalent Static APDL File Created

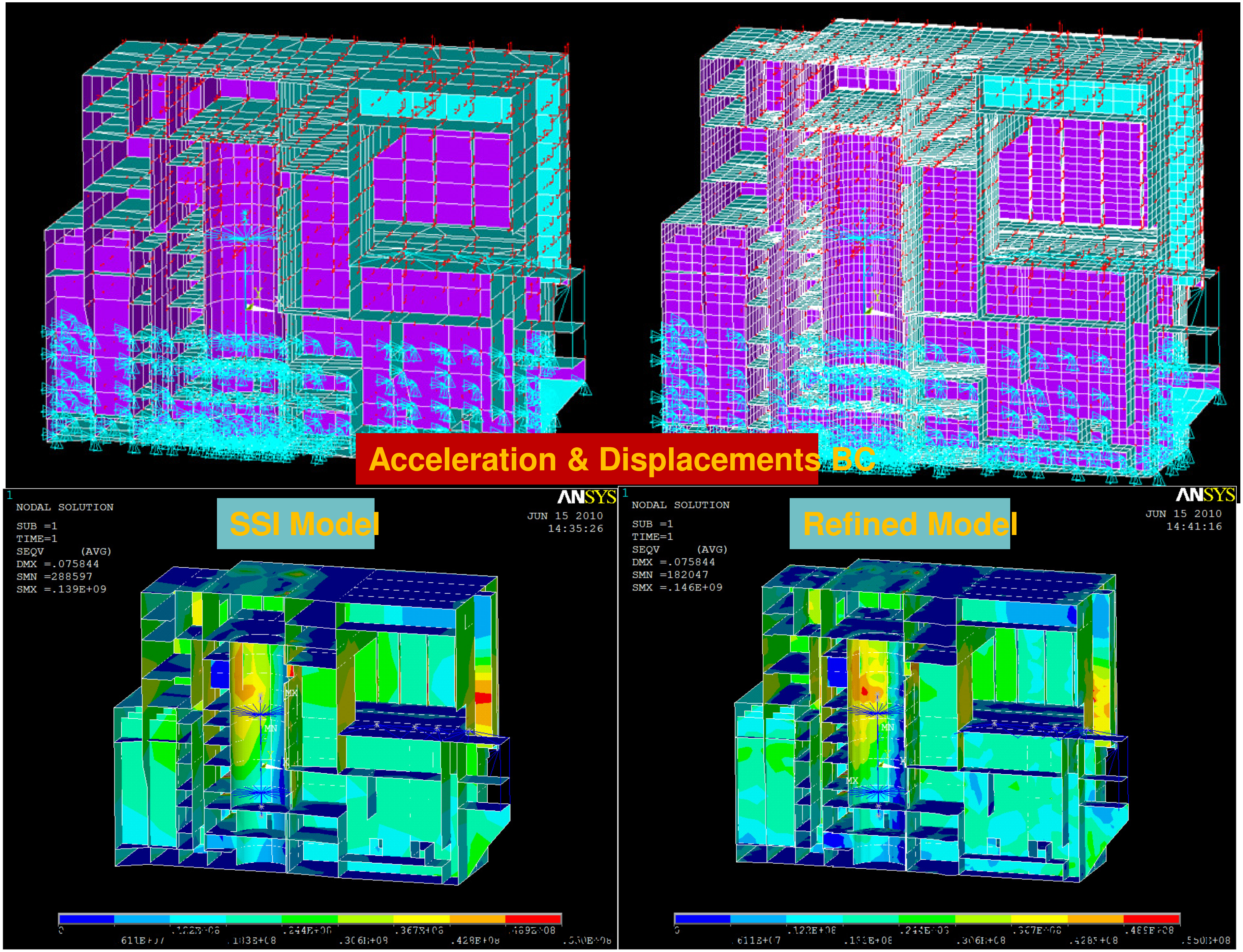
forces.inp - Notepad

File Edit Format View Help

```
FINIsh
/prep7
Node_Id=Node( 0.300000000E+01, 0.300000000E+01, 0.530000000E+02)
F,Node_Id,FX, 0.120690430E+03
F,Node_Id,FY, 0.179244520E+02
F,Node_Id,FZ, 0.259900690E+02
Node_Id=Node( 0.300000000E+01, 0.600000000E+01, 0.530000000E+02)
F,Node_Id,FX, 0.133931875E+03
F,Node_Id,FY, 0.197587250E+02
F,Node_Id,FZ, 0.250601330E+02
Node_Id=Node( 0.300000000E+01, 0.130000000E+02, 0.530000000E+02)
F,Node_Id,FX, 0.187532156E+03
F,Node_Id,FY, 0.221239760E+02
F,Node_Id,FZ, 0.247980250E+02
Node_Id=Node( 0.300000000E+01, 0.230000000E+02, 0.530000000E+02)
F,Node_Id,FX, 0.265551468E+03
F,Node_Id,FY, 0.163200870E+02
F,Node_Id,FZ, 0.259055440E+02
Node_Id=Node( 0.300000000E+01, 0.330000000E+02, 0.530000000E+02)
F,Node_Id,FX, 0.313659073E+03
F,Node_Id,FY, 0.795919600E+01
F,Node_Id,FZ, 0.262560410E+02
Node_Id=Node( 0.300000000E+01, 0.430000000E+02, 0.530000000E+02)
F,Node_Id,FX, 0.328311039E+03
F,Node_Id,FY, 0.000000000E+00
F,Node_Id,FZ, 0.258664210E+02
Node_Id=Node( 0.300000000E+01, 0.530000000E+02, 0.530000000E+02)
F,Node_Id,FX, 0.313659073E+03
F,Node_Id,FY,-0.795919600E+01
F,Node_Id,FZ, 0.262560410E+02
Node_Id=Node( 0.300000000E+01, 0.630000000E+02, 0.530000000E+02)
F,Node_Id,FX, 0.265551468E+03
F,Node_Id,FY,-0.163200870E+02
F,Node_Id,FZ, 0.259055440E+02
Node_Id=Node( 0.300000000E+01, 0.730000000E+02, 0.530000000E+02)
F,Node_Id,FX, 0.187532156E+03
F,Node_Id,FY,-0.221239760E+02
F,Node_Id,FZ, 0.247980250E+02
Node_Id=Node( 0.300000000E+01, 0.800000000E+02, 0.530000000E+02)
F,Node_Id,FX, 0.133931875E+03
F,Node_Id,FY,-0.197587250E+02
F,Node_Id,FZ, 0.250601330E+02
Node_Id=Node( 0.300000000E+01, 0.830000000E+02, 0.530000000E+02)
F,Node_Id,FX, 0.120690430E+03
F,Node_Id,FY,-0.179244520E+02
F,Node_Id,FZ, 0.259900690E+02
Node_Id=Node( 0.600000000E+01, 0.300000000E+01, 0.530000000E+02)
F,Node_Id,FX, 0.121983904E+03
F,Node_Id,FY, 0.360173100E+01
```

Soil Model Option – Displacements at Interaction Nodes





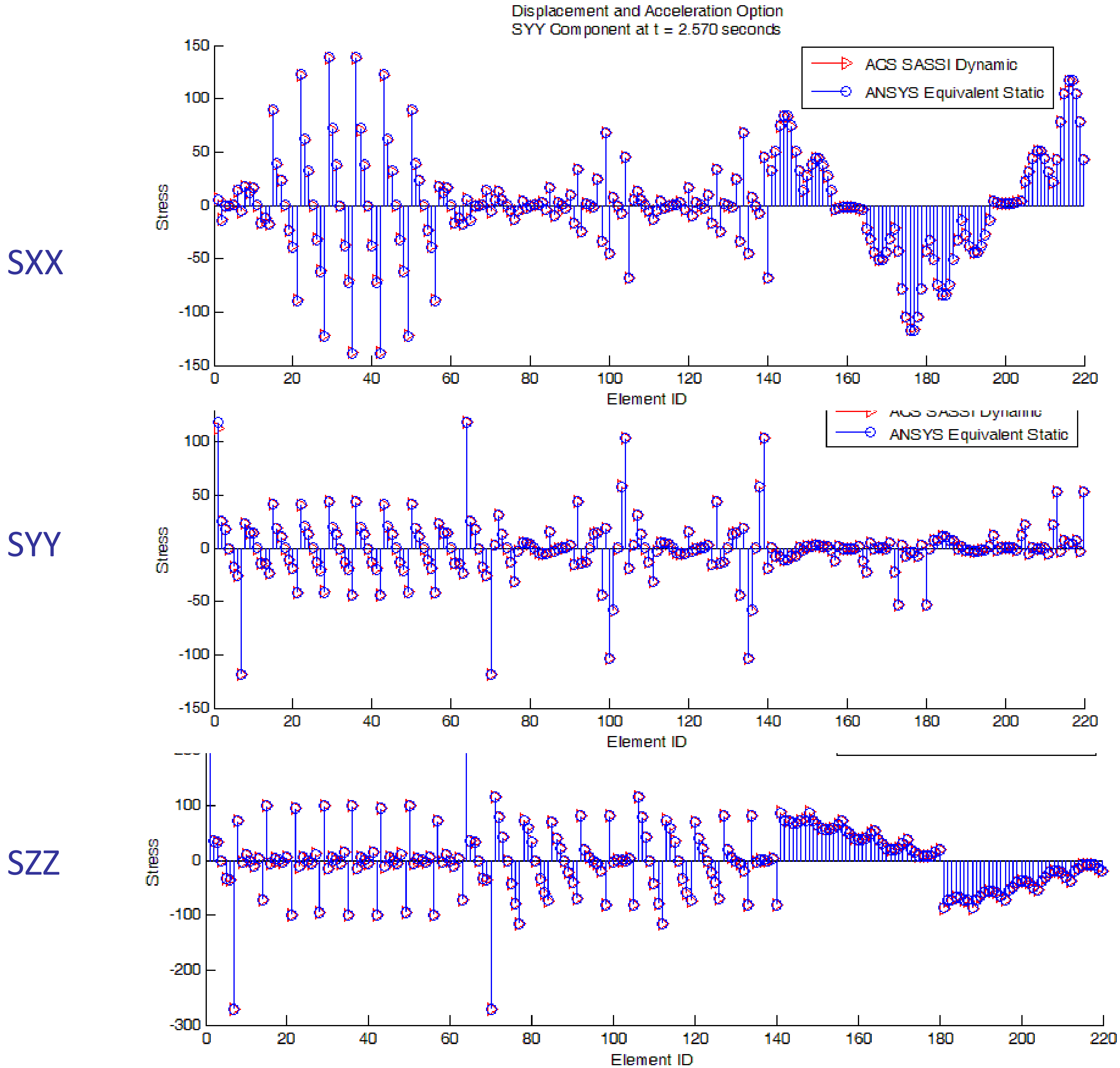
Acceleration & Displacements BC
SSI Model
NODAL SOLUTION
SUB =1
TIME=1
SEQV (AVG)
DMX =.075844
SMN =288597
SMX =.139E+09
ANSYS
JUN 15 2010
14:35:26
Refined Model
NODAL SOLUTION
SUB =1
TIME=1
SEQV (AVG)
DMX =.075844
SMN =182047
SMX =.146E+09
ANSYS
JUN 15 2010
14:41:16

ANSYS Equivalent-Static vs. ACS SASSI

SSI Analysis

Surface Concrete Box
SOLID Elements

Soil Vs=1000fps


Displacement and Acceleration Option
SYY Component at t = 2.570 seconds
ACS SASSI Dynamic
ANSYS Equivalent Static
SXX
SYY
SZZ
Stress
Element ID

ANSYS Equivalent-Static vs. ACS SASSI

SSI Analysis

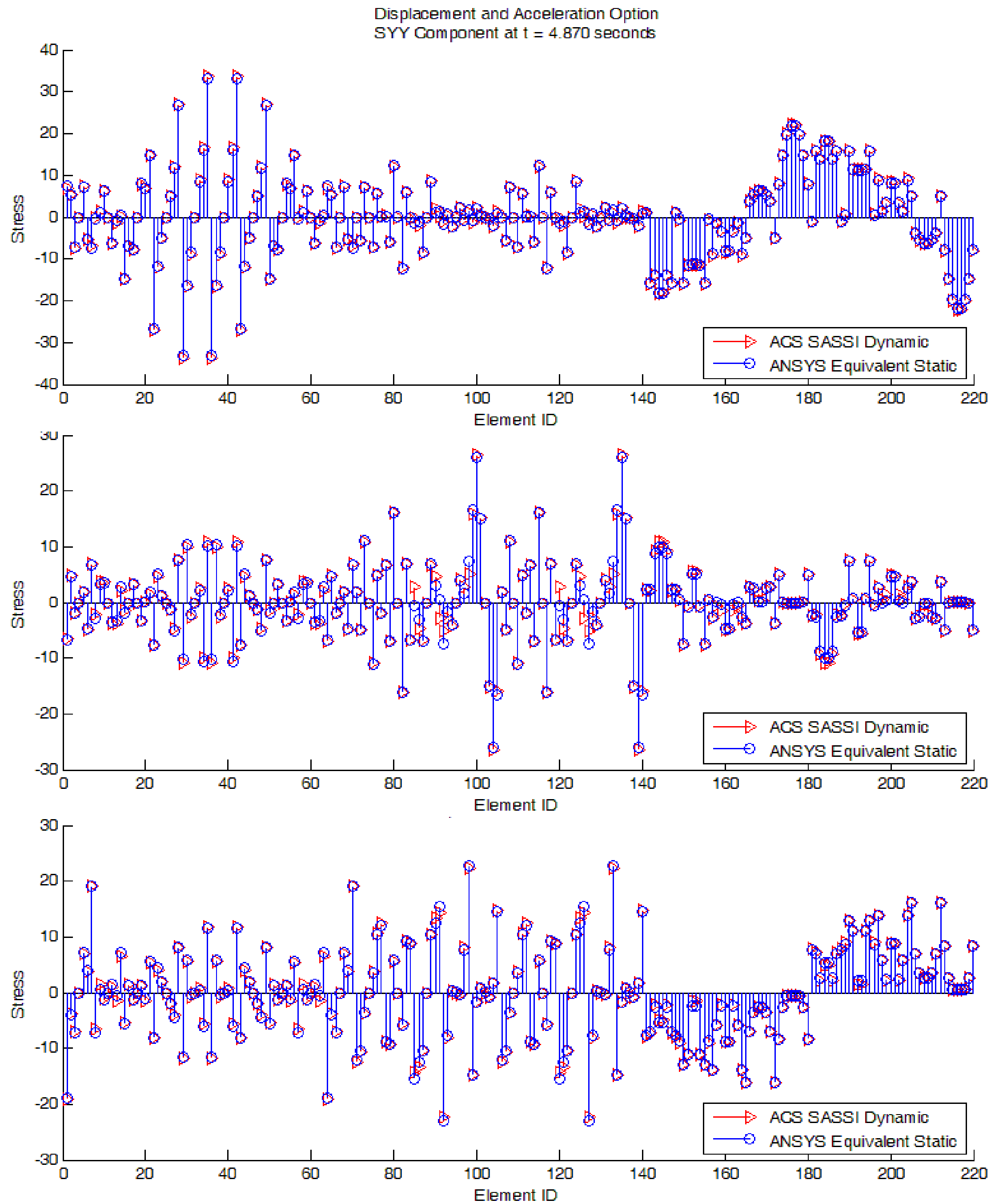
Deeply Embedded Concrete Box
SOLID Elements

Soil Vs=1,000 fps

SXX

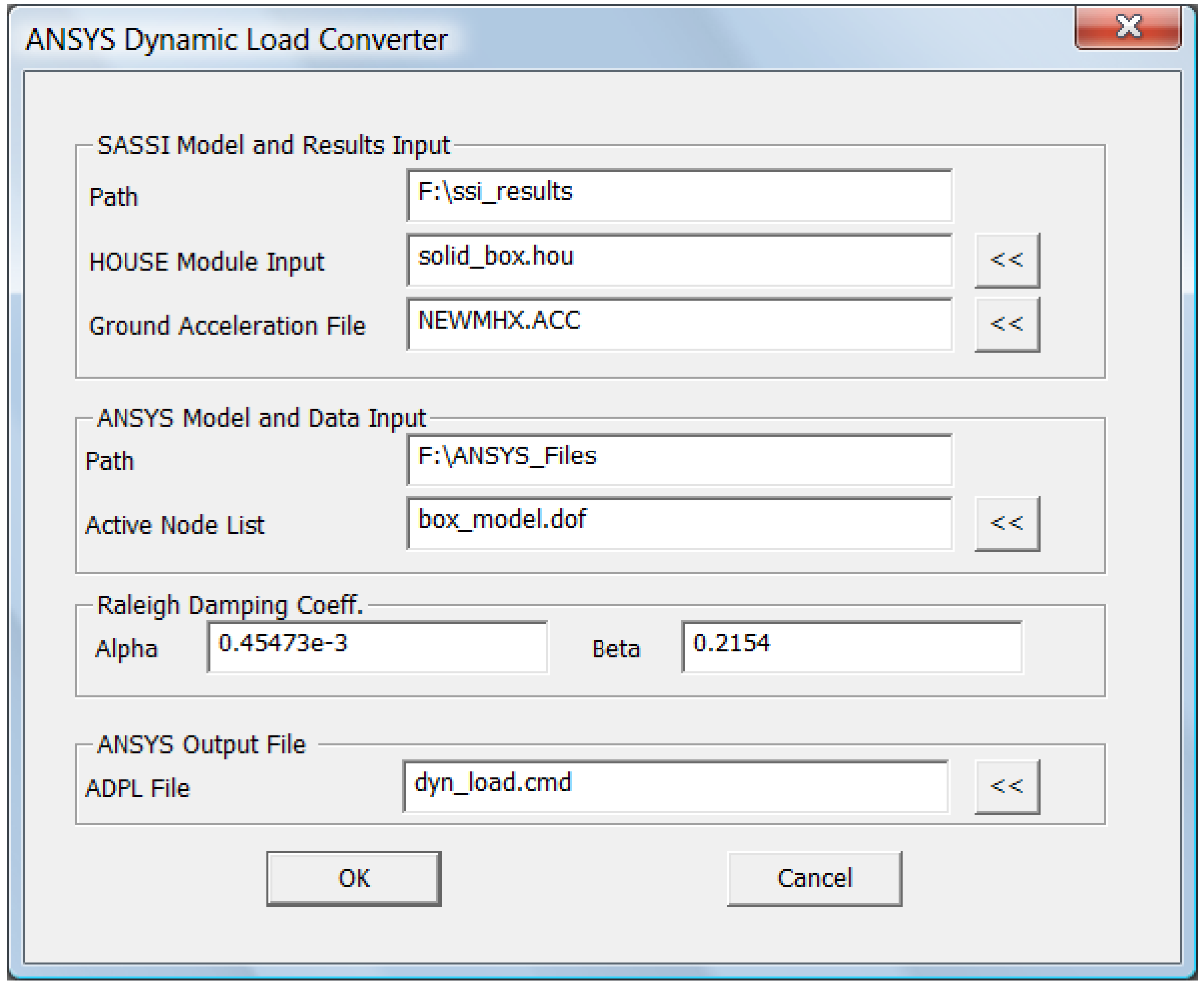
SYY

SZZ


Displacement and Acceleration Option
SYY Component at t = 4.870 seconds
Stress
Element ID
ACS SASSI Dynamic
ANSYS Equivalent Static

ANSYS Dynamic Load Generation from ACS SASSI Frames

- From ACS SASSI-MAIN select “ANSYS Dynamic Load” from the Run menu
- Fill in the appropriate boxes as described in the documentation
- ANSYS APDL input files are created containing the load data are created when the user clicks “OK”



ANSYS Dynamic Load APDL File Created

dynamic.inp - Notepad

File Edit Format View Help

```
FINISh
/Config,NRes,   6000
/SOLU
ANTYPE,TRANS
TRNOPT,FULL
 ALPHAD, 0.45473e-3
 BETAD, 0.2154
 DELTIM 0.005
!
 /INPUT,disp_gacc_load_00001
 TIME,   0.0001
 KBC,1
 OUTRES,ALL,LAST
 SOLVE
!
 /INPUT,disp_gacc_load_00002
 TIME,   0.0050
 KBC,1
 OUTRES,ALL,LAST
 SOLVE
!
 /INPUT,disp_gacc_load_00003
 TIME,   0.0100
 KBC,1
 OUTRES,ALL,LAST
 SOLVE
!
 /INPUT,disp_gacc_load_00004
 TIME,   0.0150
 KBC,1
 OUTRES,ALL,LAST
 SOLVE
!
 /INPUT,disp_gacc_load_00005
 TIME,   0.0200
 KBC,1
 OUTRES,ALL,LAST
 SOLVE
!
 /INPUT,disp_gacc_load_00006
 TIME,   0.0250
 KBC,1
 OUTRES,ALL,LAST
 SOLVE
!
 /INPUT,disp_gacc_load_00007
 TIME,   0.0300
 KBC,1
 OUTRES,ALL,LAST
```

ANSYS Dynamic vs. ACS SASSI

SSI Analysis

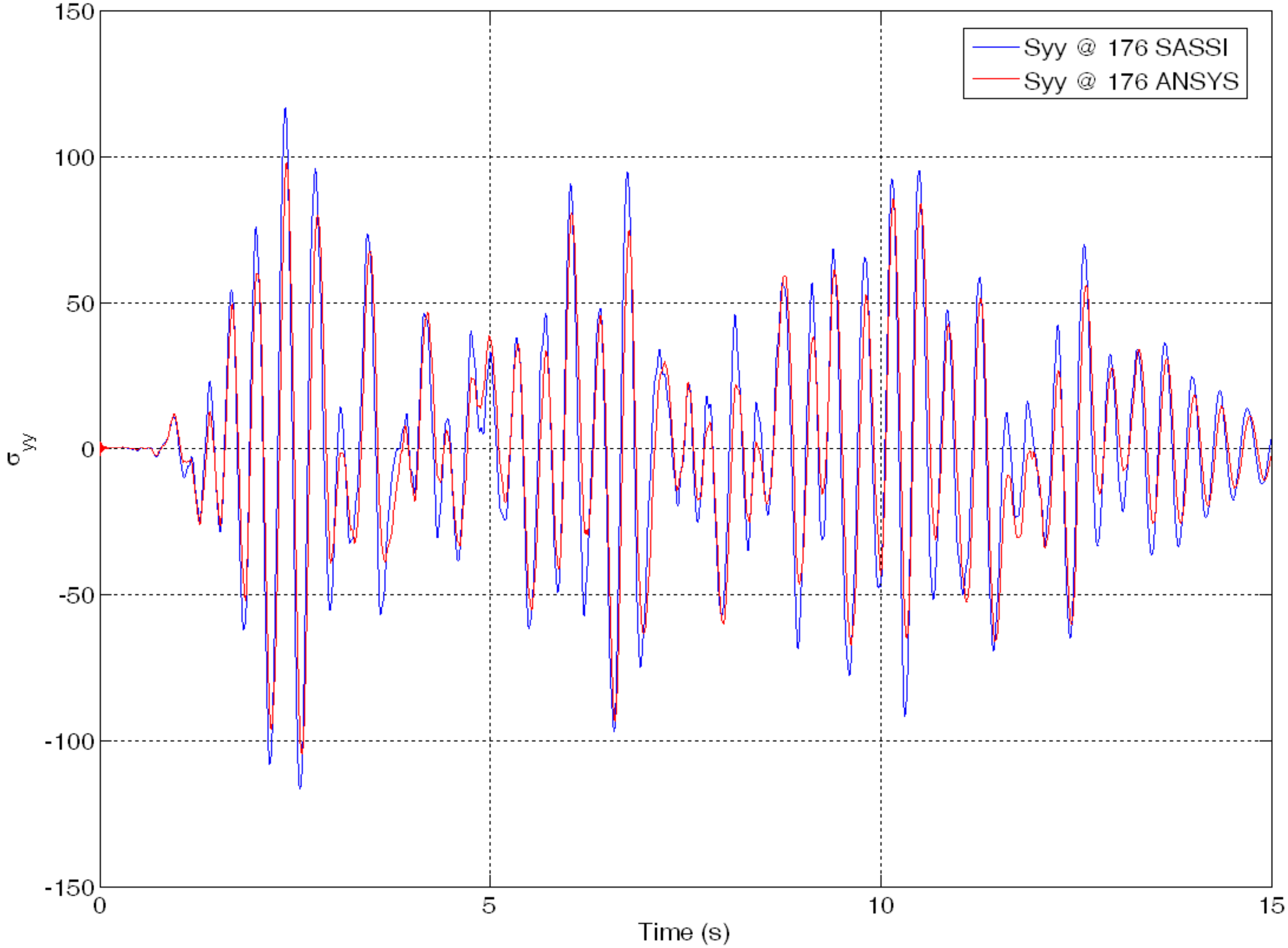
Surface Concrete Box
SOLID Elements
Soil Vs=1,000 fps

Seismic Loading for ANSYS:
Ground Acceleration Histories and Relative Displacement Histories wrt Free-Field Surface Motion

ANSYS Dynamic vs. ACS SASSI – Surface SSI Model

Above Ground Surface

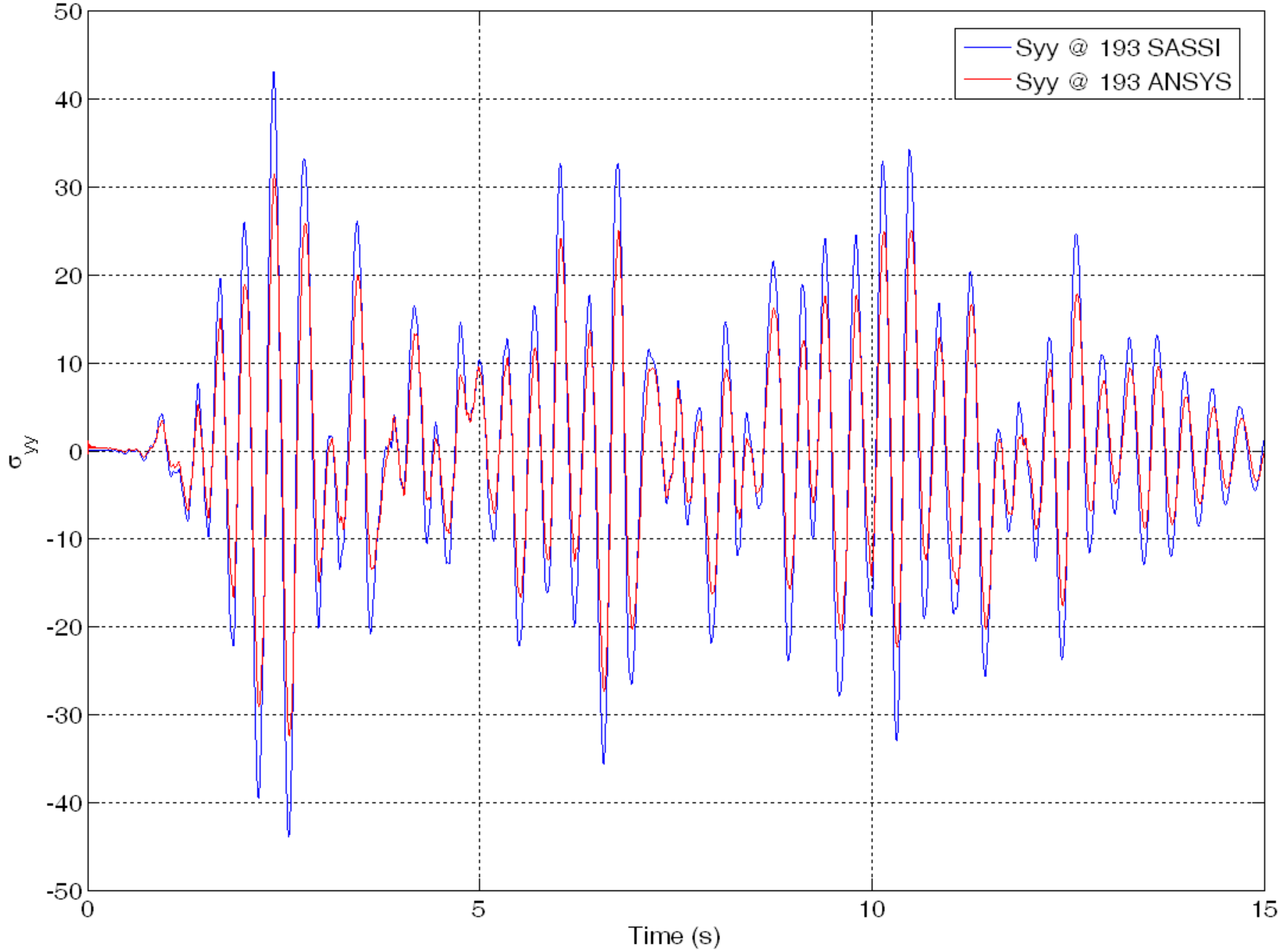
SYY



ANSYS Dynamic vs. ACS SASSI – for Surface SSI Model

Below Ground Surface

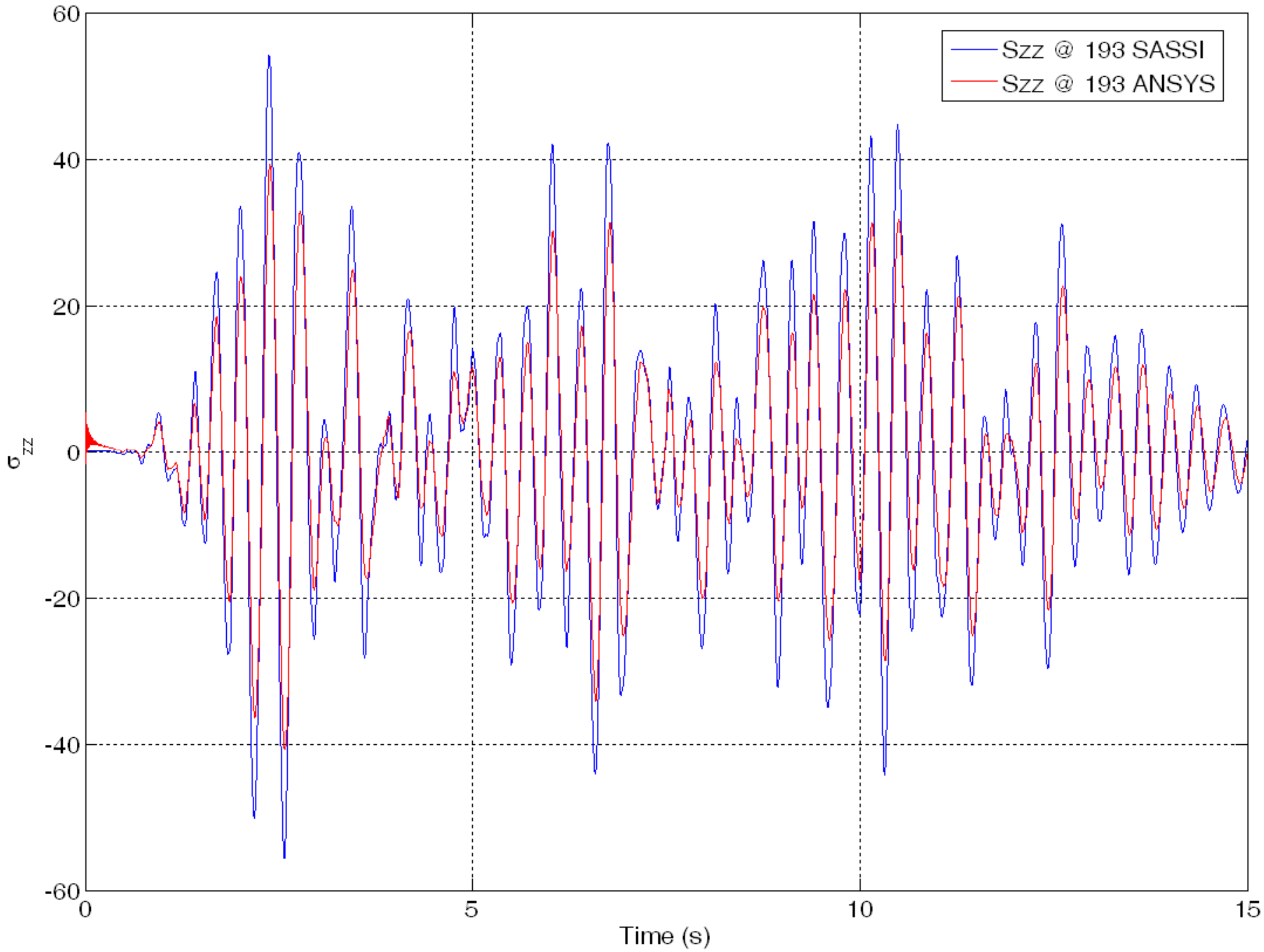
SYY



ANSYS Dynamic vs. ACS SASSI – for Surface SSI Model

Below Ground Surface

SZZ



ANSYS Dynamic vs. ACS SASSI

SSI Analysis

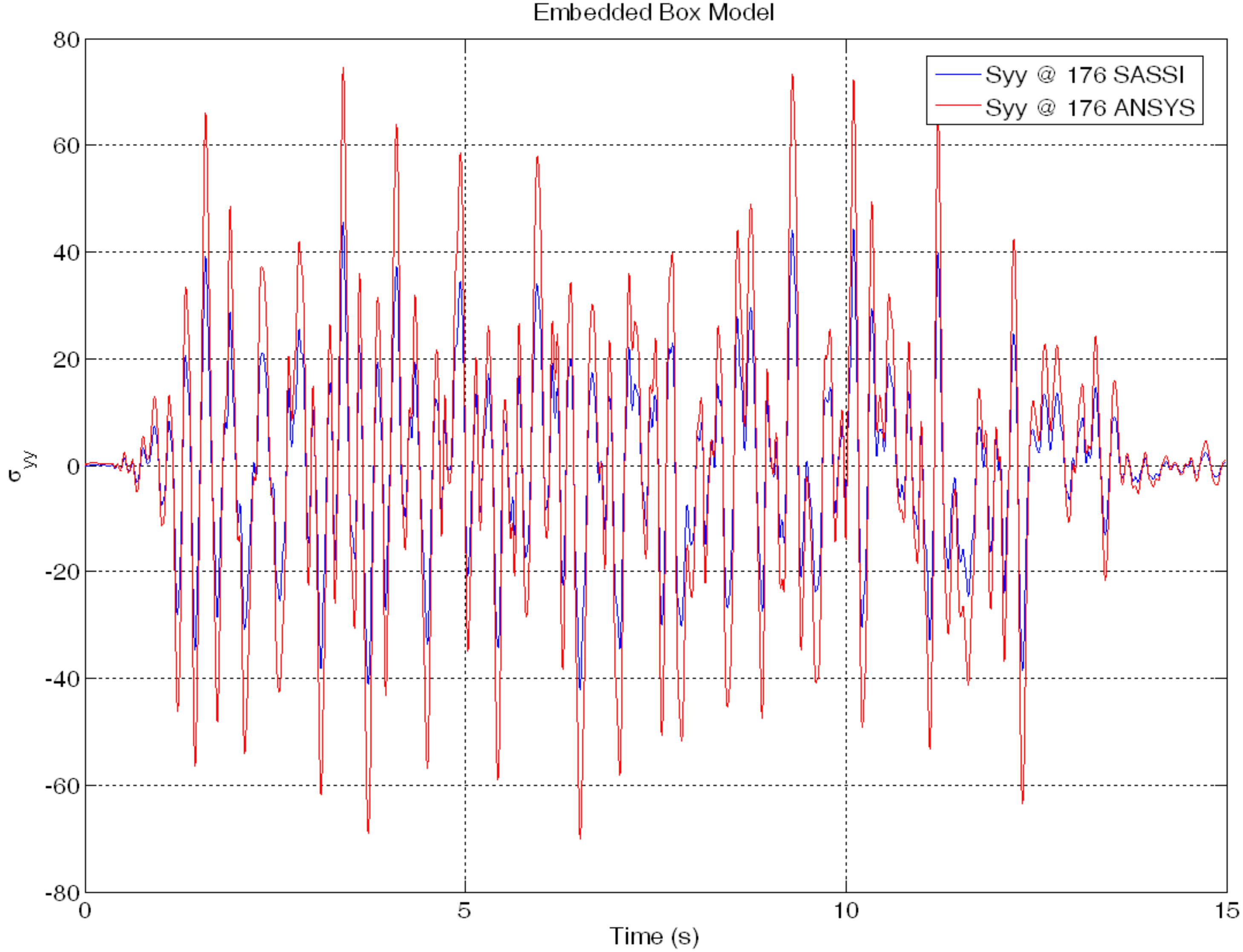
Deeply Embedded Concrete Box
SOLID Elements
Soil Vs=1,000 fps

Seismic Loading for ANSYS:
Ground Acceleration Histories and Relative Displacement Histories wrt Free-Field Surface Option
(No kinematic SSI included)

ANSYS Dynamic vs. ACS SASSI – Deeply Embedded SSI Model

Above Ground Surface

SYY

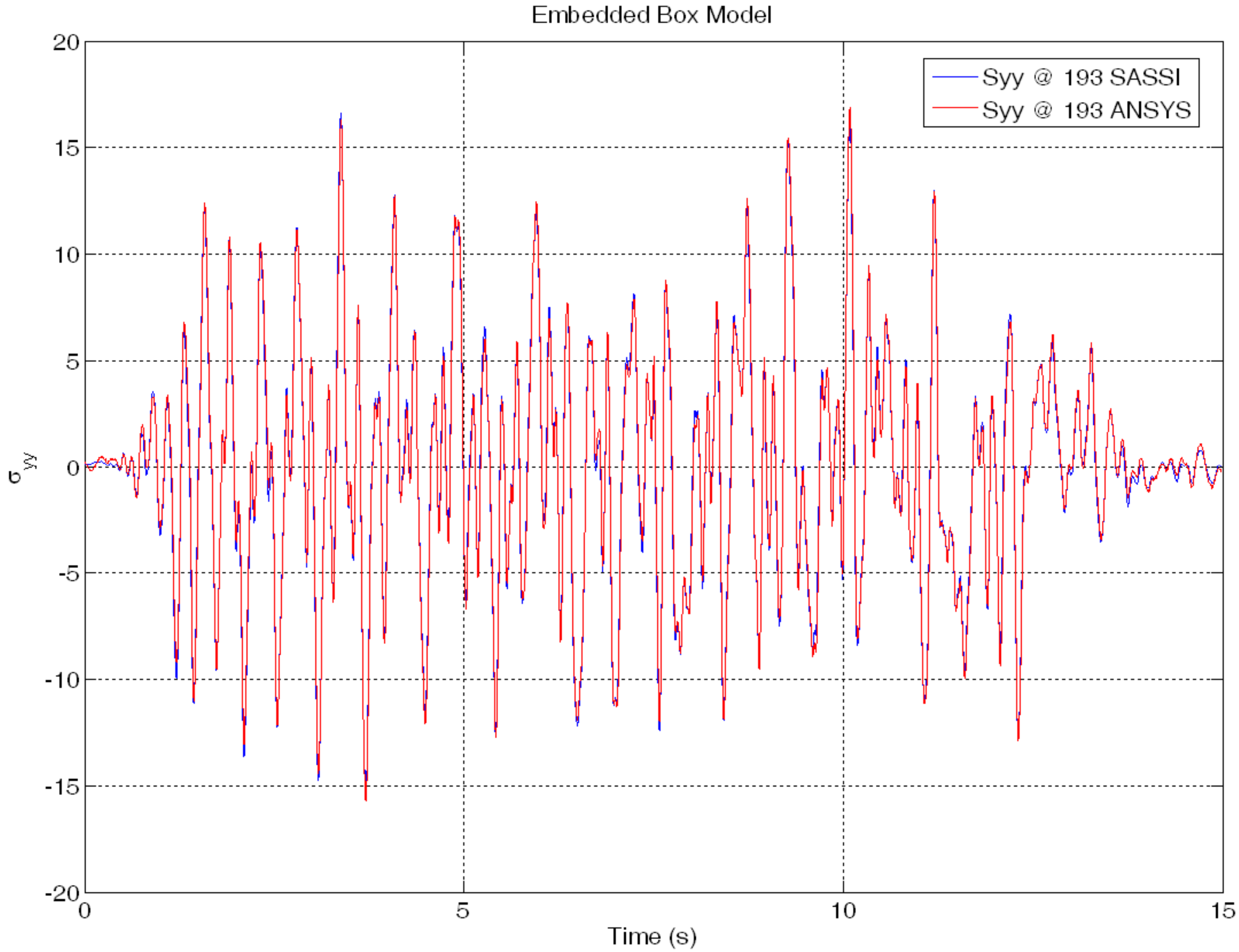


NOTE: In this example ANSYS results does not include kinematic SSI effects on accel

ANSYS Dynamic vs. ACS SASSI – Deeply Embedded SSI Model

Below Ground Surface

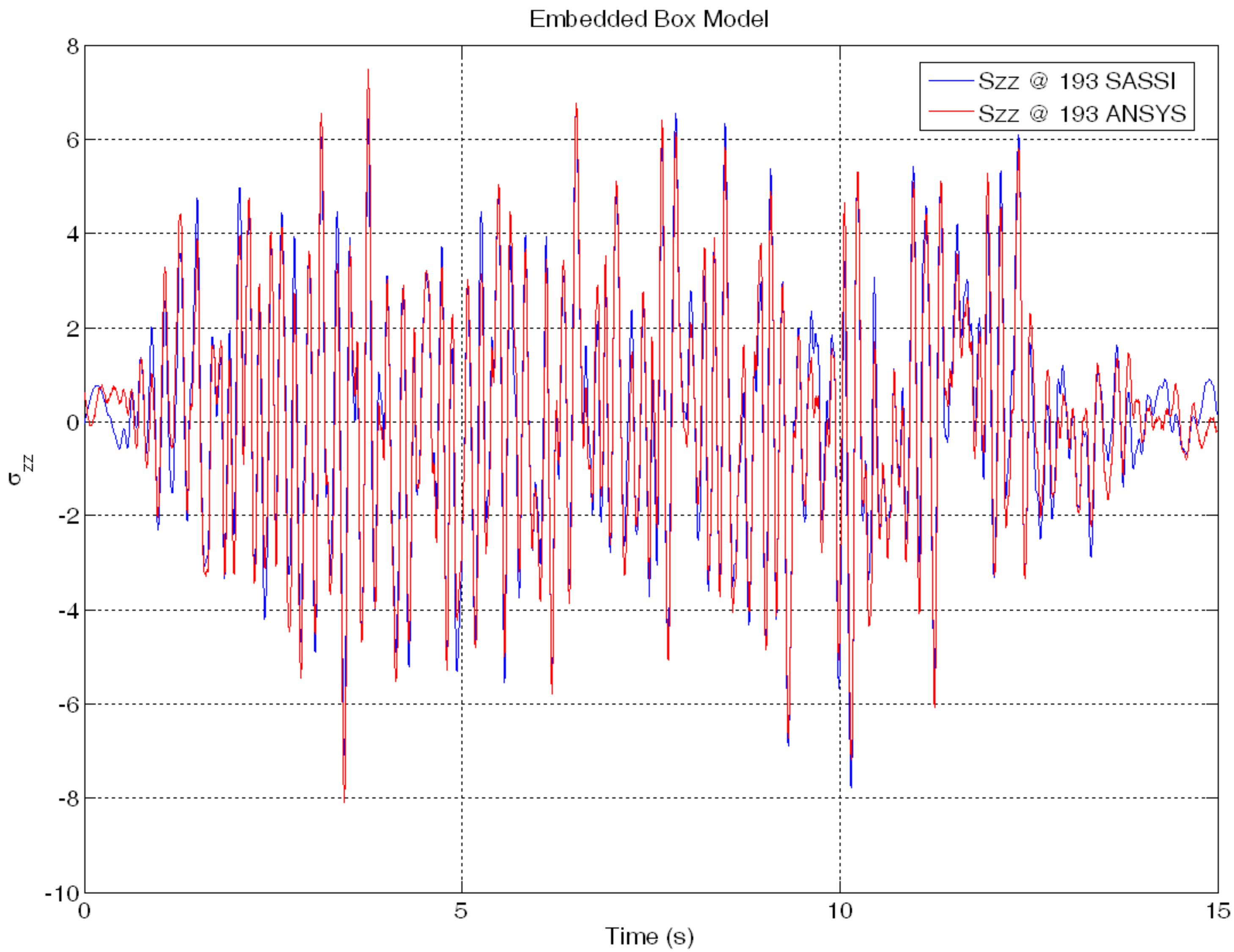
SYY



ANSYS Dynamic vs. ACS SASSI – Deeply Embedded SSI Model

Below Ground Surface

SZZ



New SOILMESH Module for Soil Pressure Computation

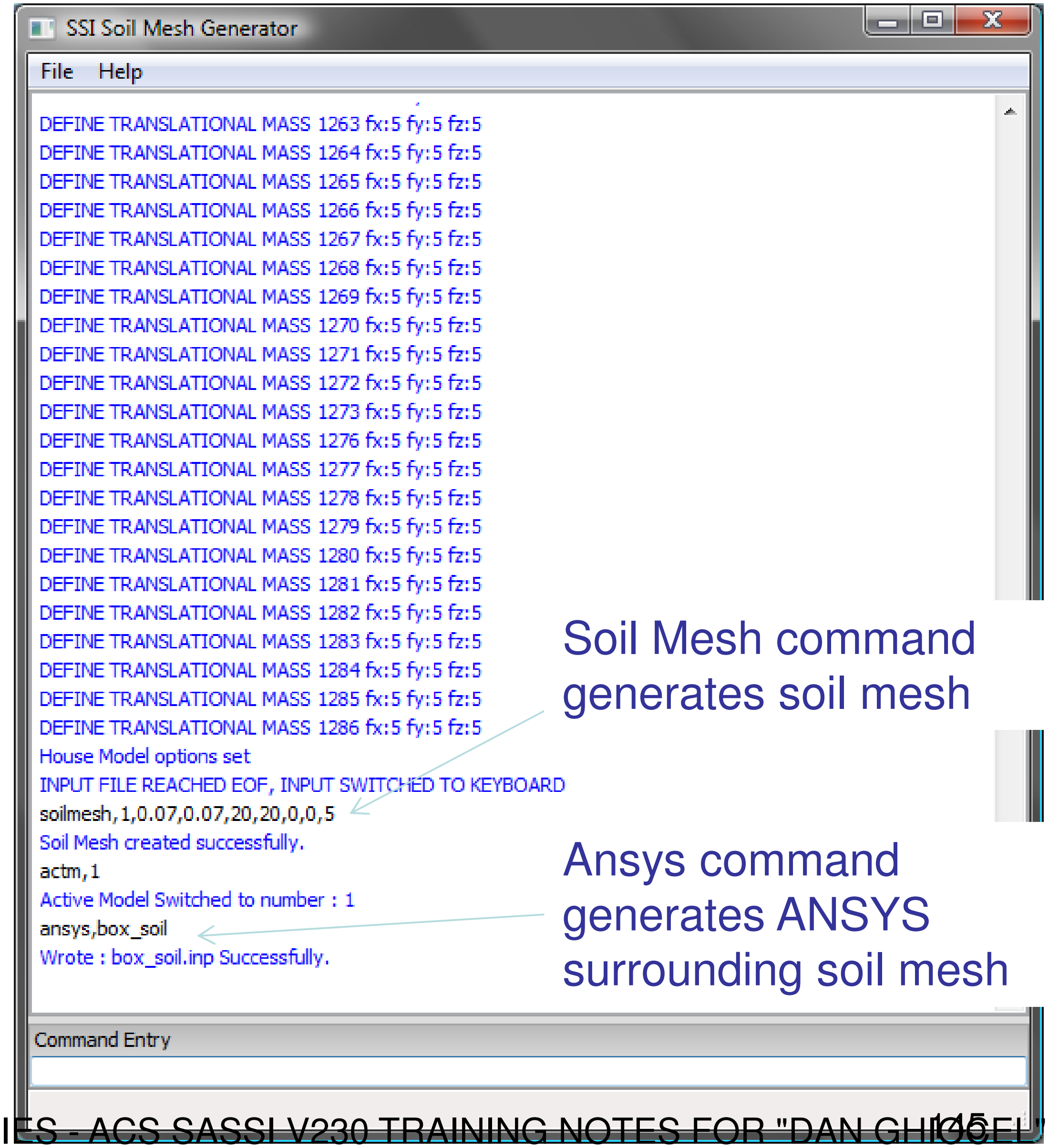
- Input .pre file with SSI model data
- Generates a soil FE model for soil pressure analysis using the "soilmesh" command
- Can export either structural or soil FE model to ANSYS APDL input file
- Computes seismic soil pressures produced using either

  i) the foundation seismic forces pushing on surrounding soil, or

  ii) the relative motion of the foundation wrt to the free-field soil motion.

Soil is assumed to be at rest. Soil stiffness is not frequency dependent. The new implementation produces "approximate" seismic soil pressures.

Significant analysis improvement in comparison with the current practice.



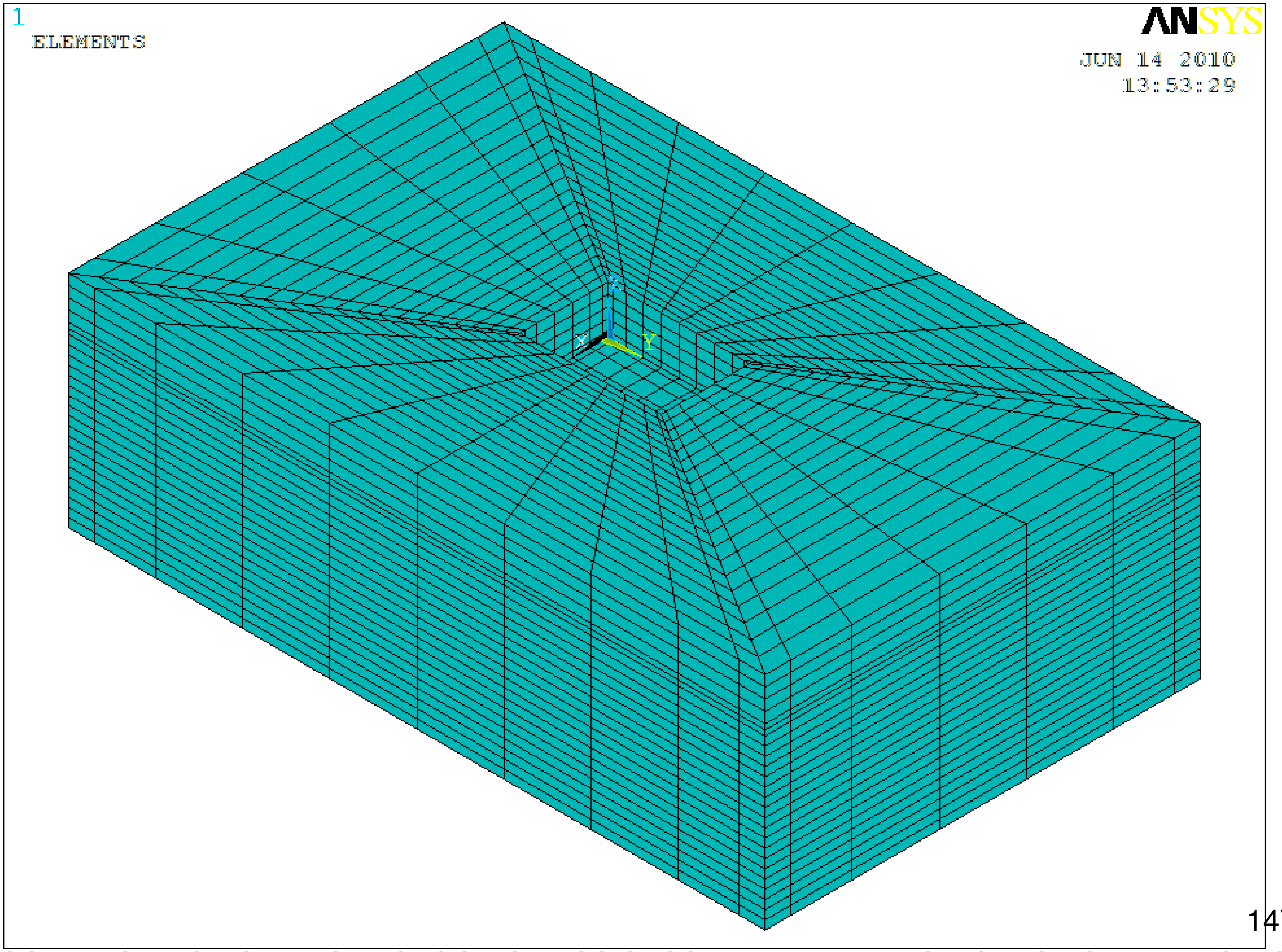
Example of APDL file for Soil FE Model

box_soil.inp - Notepad

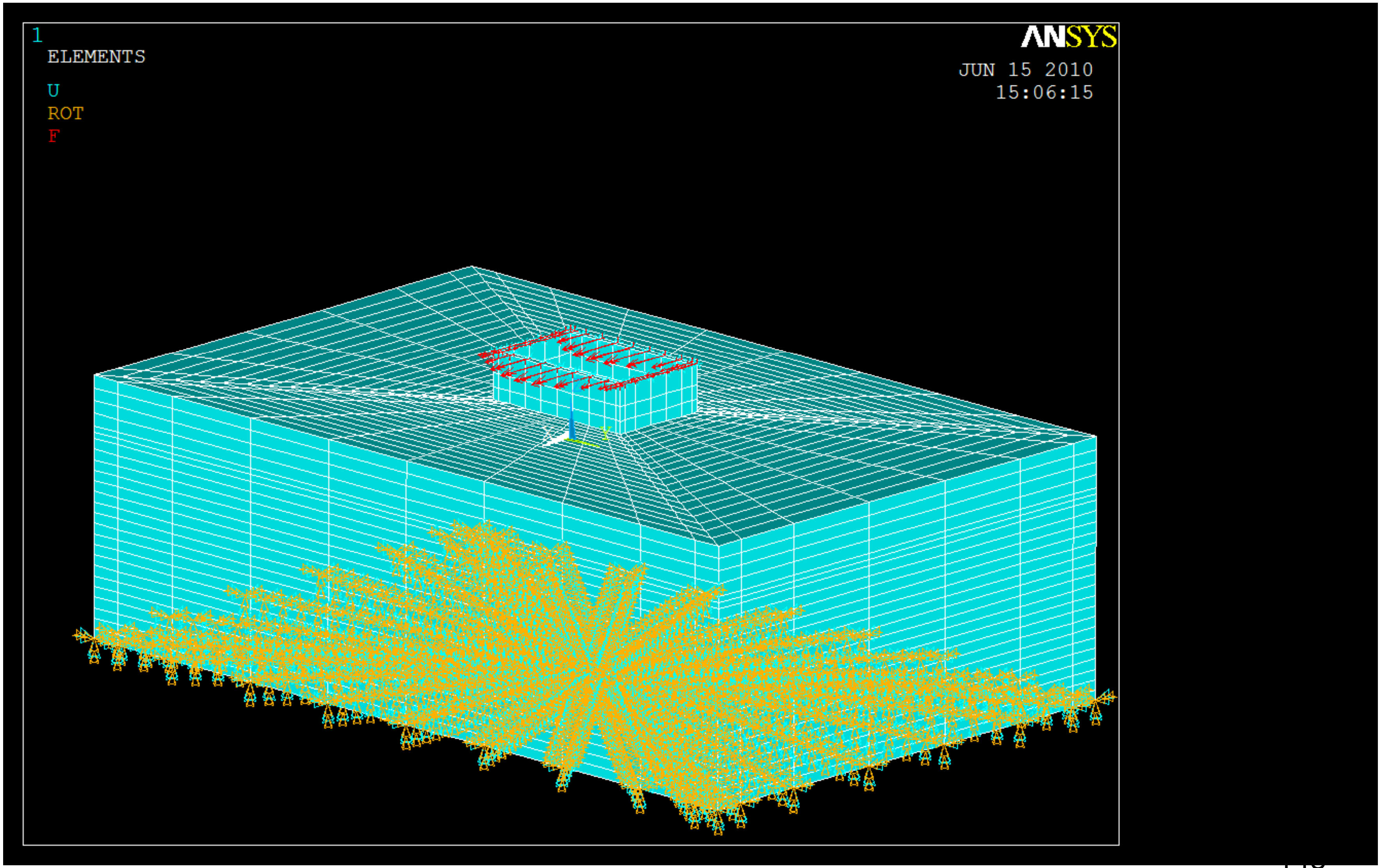
File Edit Format View Help

```
/PREP7
! Element Type
ET,101,CONTA173
ET,102,TARGE170
ET,103,SOLID45
! Nodes
N,145,3,3,3
N,146,3,6,3
N,147,3,13,3
N,148,3,23,3
N,149,3,33,3
N,150,3,43,3
N,151,3,53,3
N,152,3,63,3
N,153,3,73,3
N,154,3,80,3
N,155,3,83,3
N,158,6,3,3
N,159,6,6,3
N,160,6,13,3
N,161,6,23,3
N,162,6,33,3
N,163,6,43,3
N,164,6,53,3
N,165,6,63,3
N,166,6,73,3
N,167,6,80,3
N,168,6,83,3
N,171,13,3,3
N,172,13,6,3
N,173,13,13,3
N,174,13,23,3
N,175,13,33,3
N,176,13,43,3
N,177,13,53,3
N,178,13,63,3
N,179,13,73,3
N,180,13,80,3
N,181,13,83,3
N,184,23,3,3
N,185,23,6,3
N,186,23,13,3
N,187,23,23,3
N,188,23,33,3
N,189,23,43,3
N,190,23,53,3
N,191,23,63,3
N,192,23,73,3
N,193,23,80,3
```

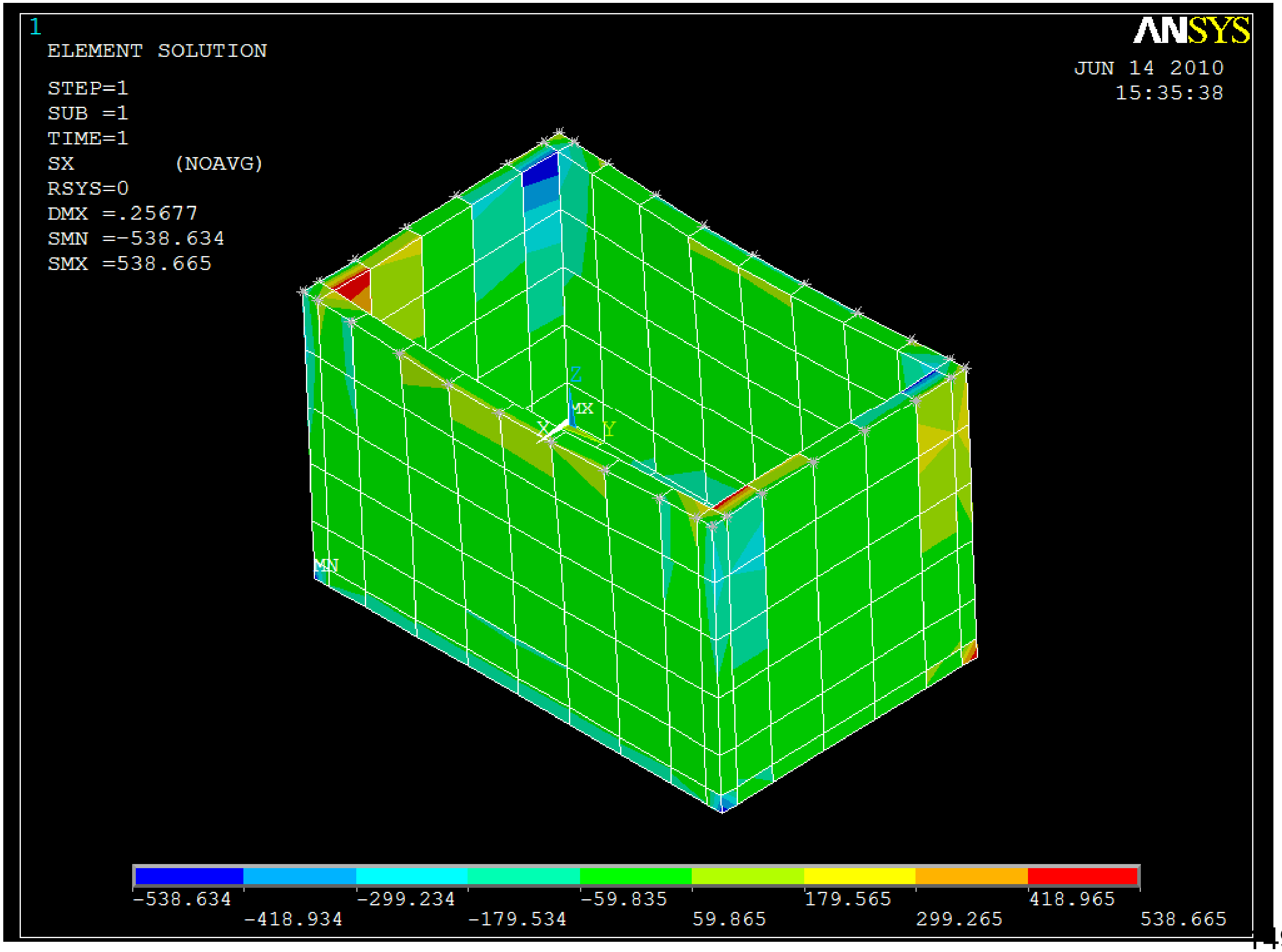
Example of Soil FE model Created Automatically by New SOILMESH Module for A Box Structural Model



Equivalent-Static Stress Analysis for Structure-Soil System Model (Generated by SOILMESH)



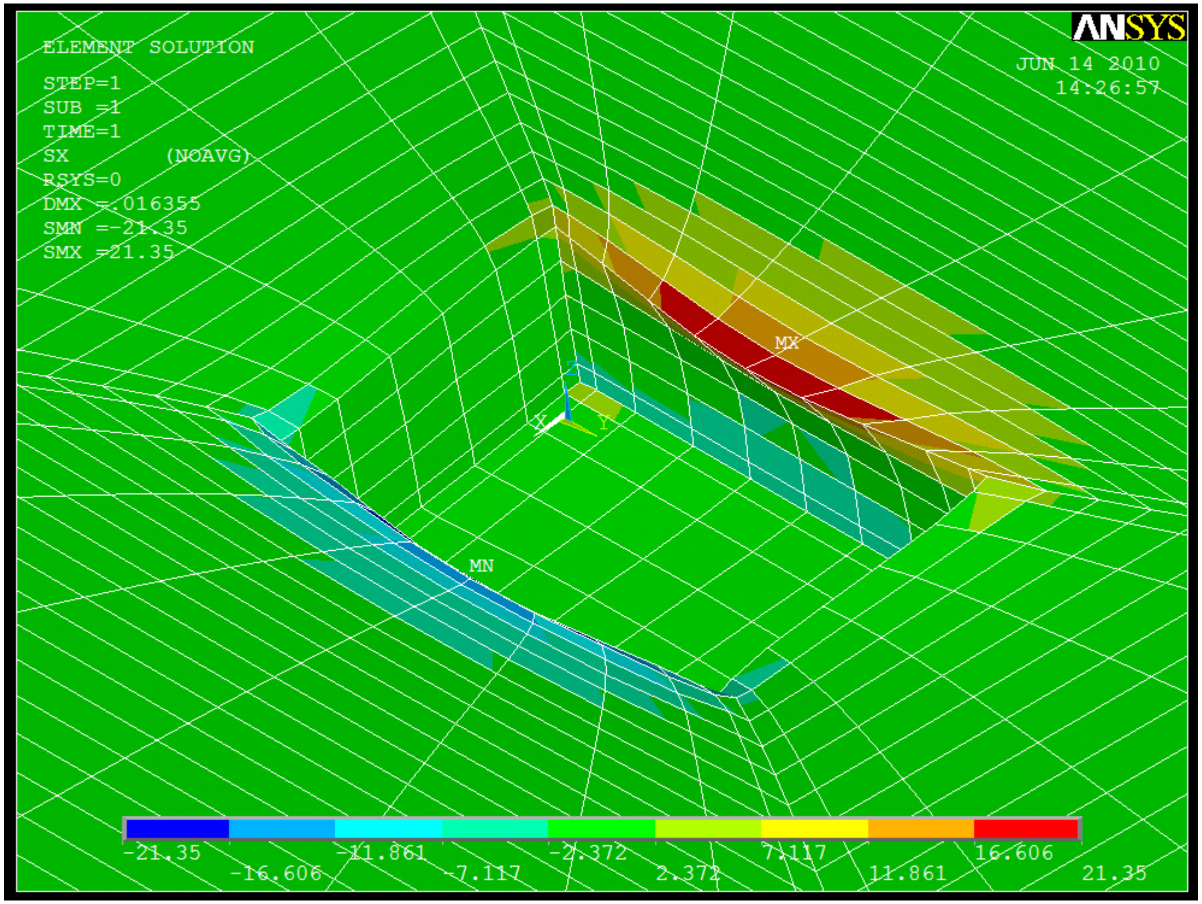
Computed Structural Displacements



Linear Seismic Soil Pressure Analysis

LINEAR (WELDED)

- This option provides for a basic soil pressure analysis assuming there is no separation possible between the structure and the soil
- Displacements from the interaction nodes of the structure are applied directly to the soil FE model. The structural FE model is not required for this case



Nonlinear Seismic Soil Pressure Analysis

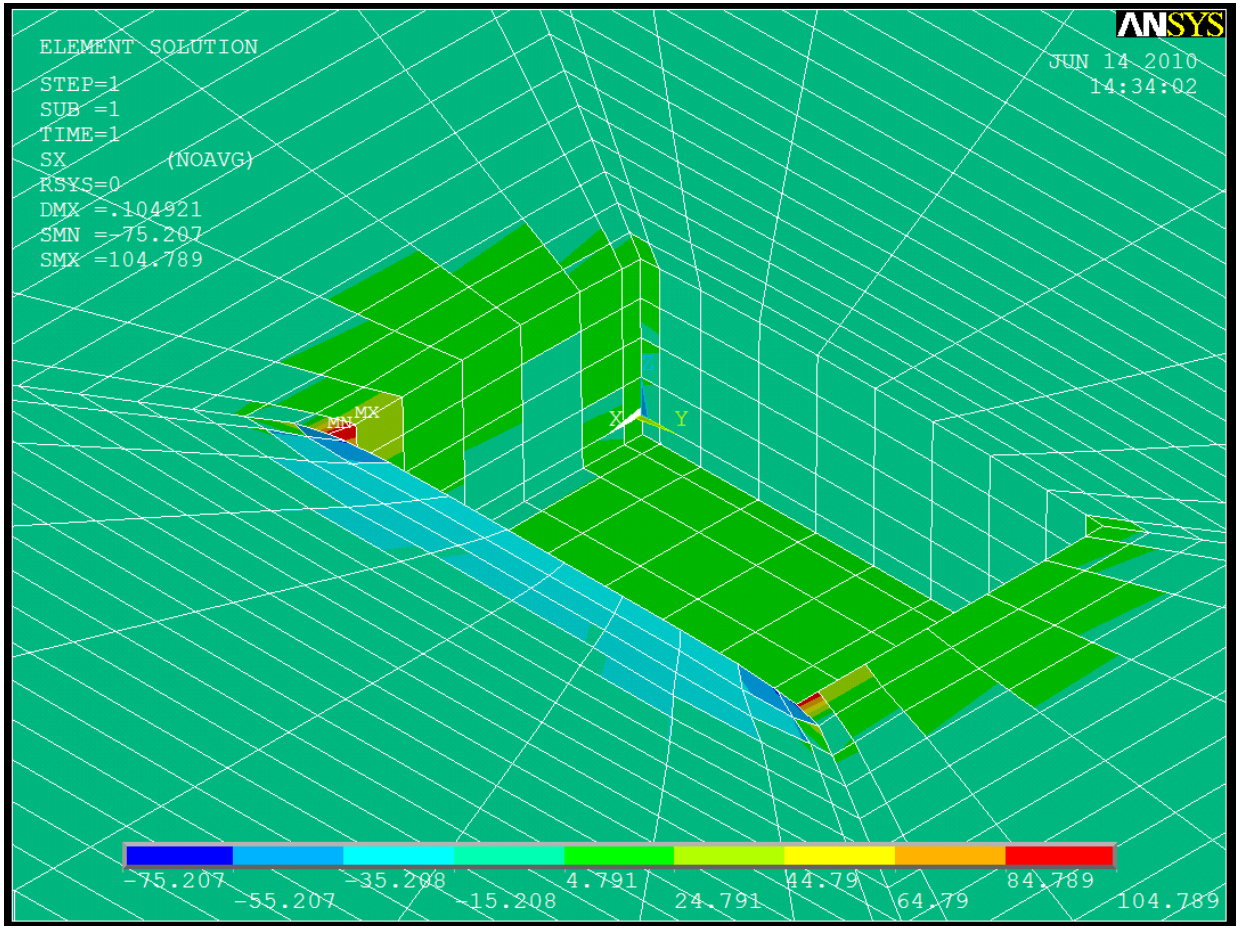
NONLINEAR CONTACT (FOUNDATION SEPARATION)

- This option allows for the structure to separate from the soil using surface to surface contact elements in ANSYS

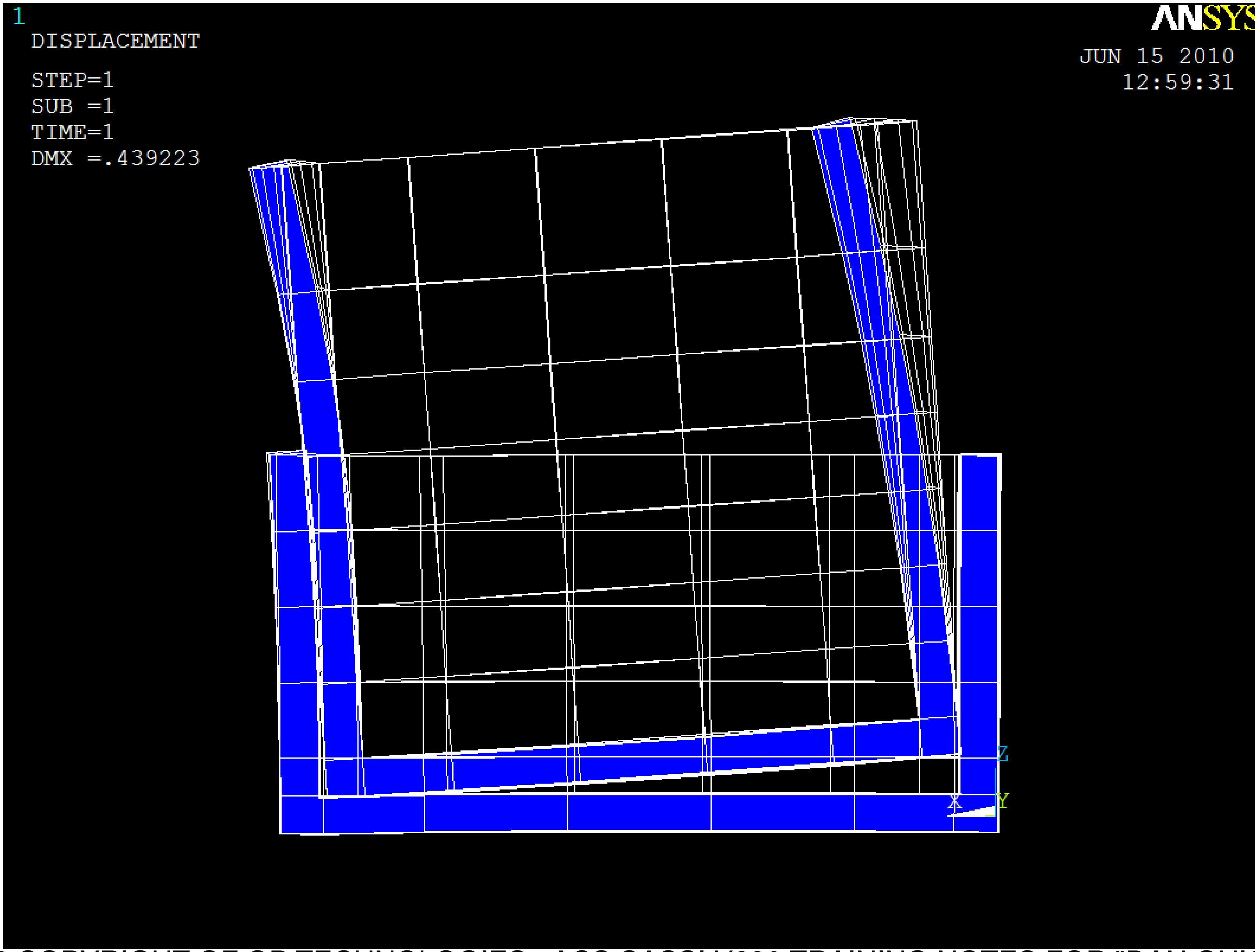
- Both the structural elements and the soil elements are required. Both APDL files written from SOILMESH must be loaded into ANSYS.

-Inertial Force should be applied to the structure.

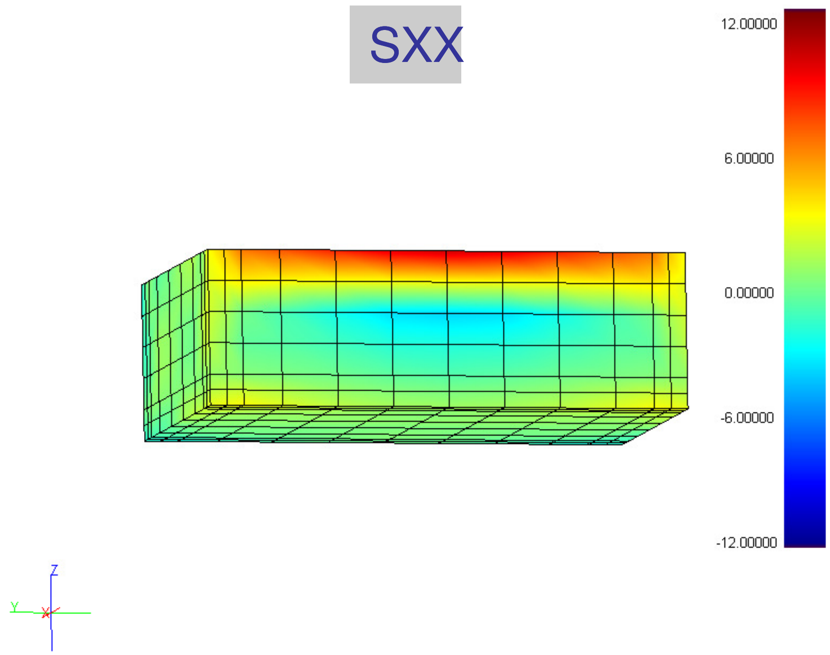
- Contact and target surfaces are included in the soil FE model

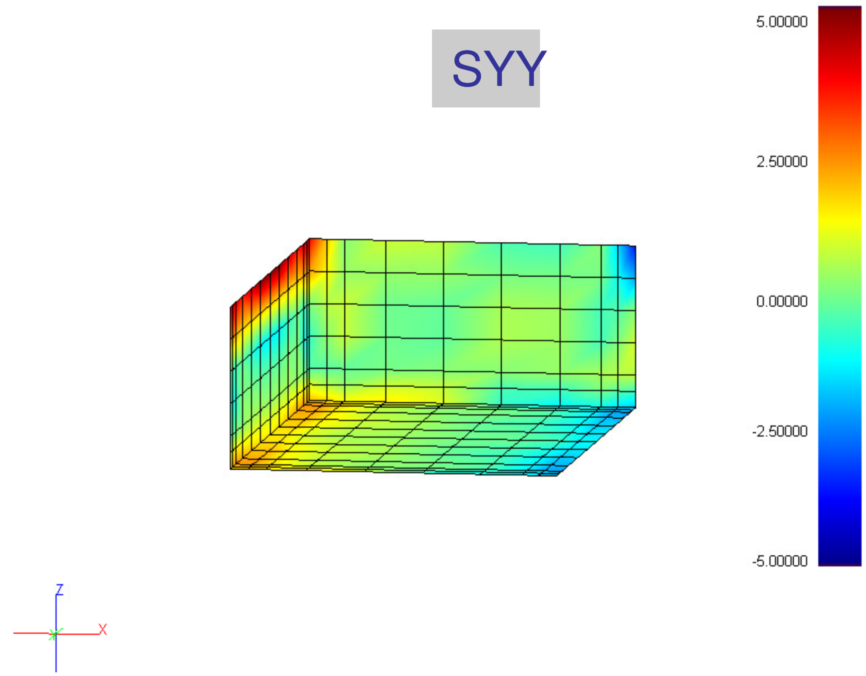


Nonlinear Seismic Soil Pressure Analysis

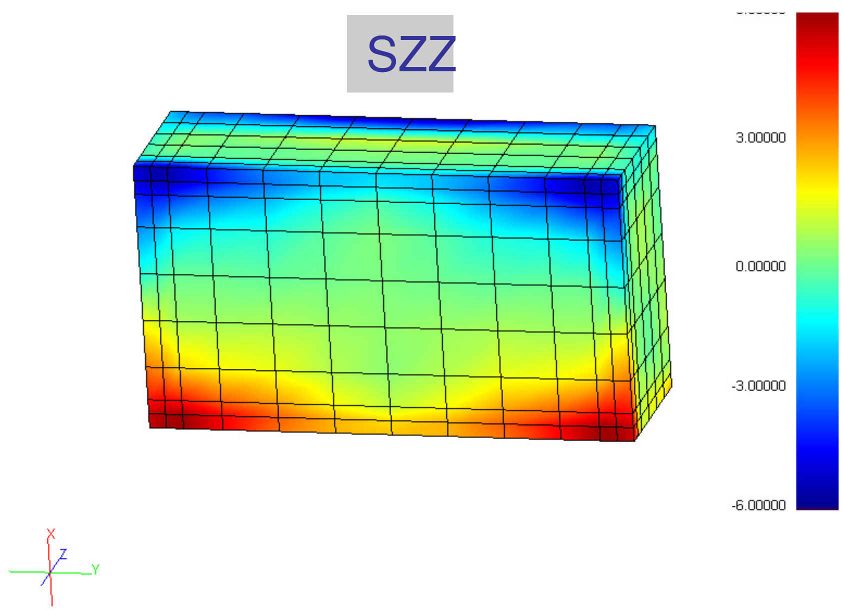


ACS SASSI Seismic Soil Pressures for X-Input (Frame 903)

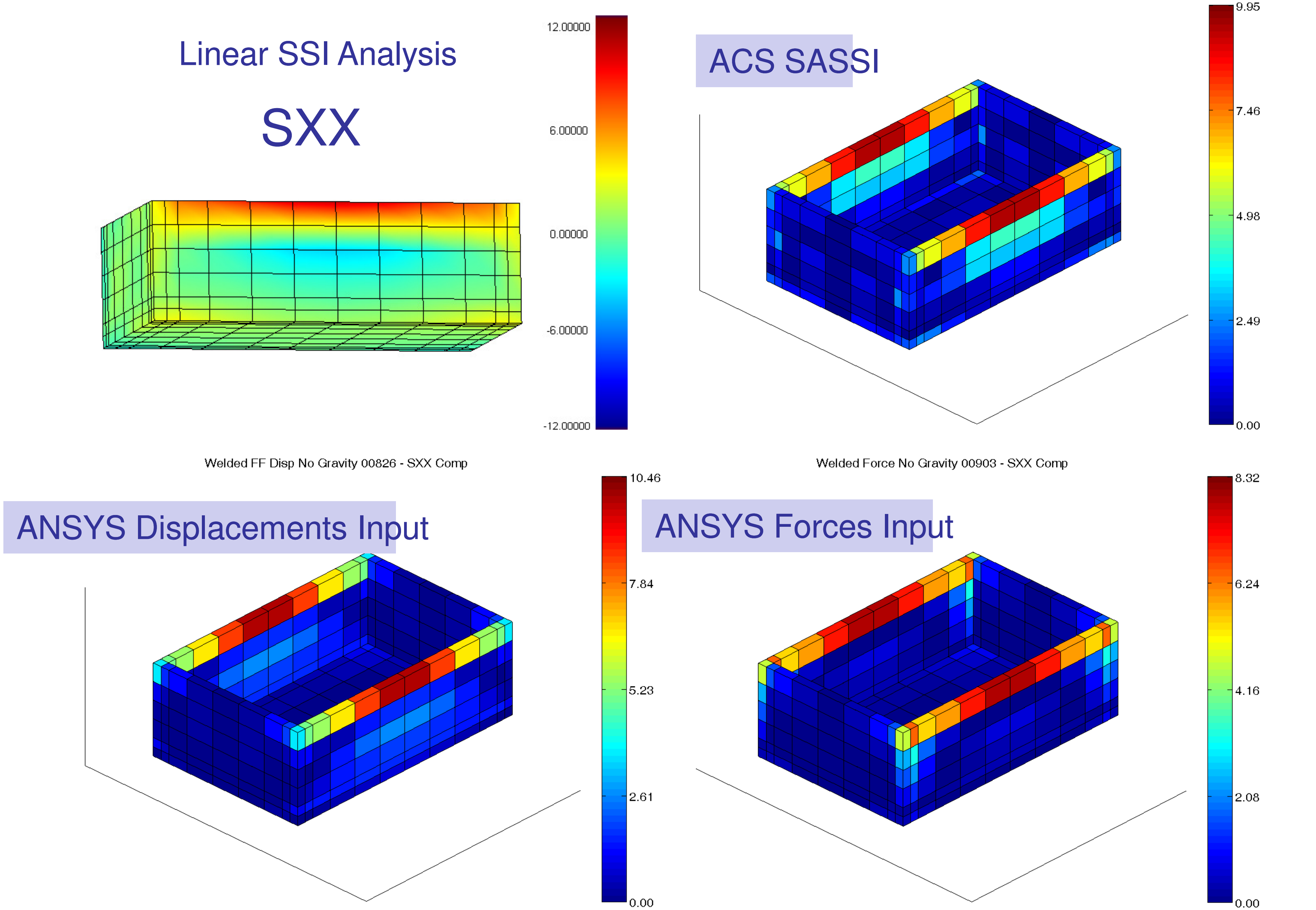




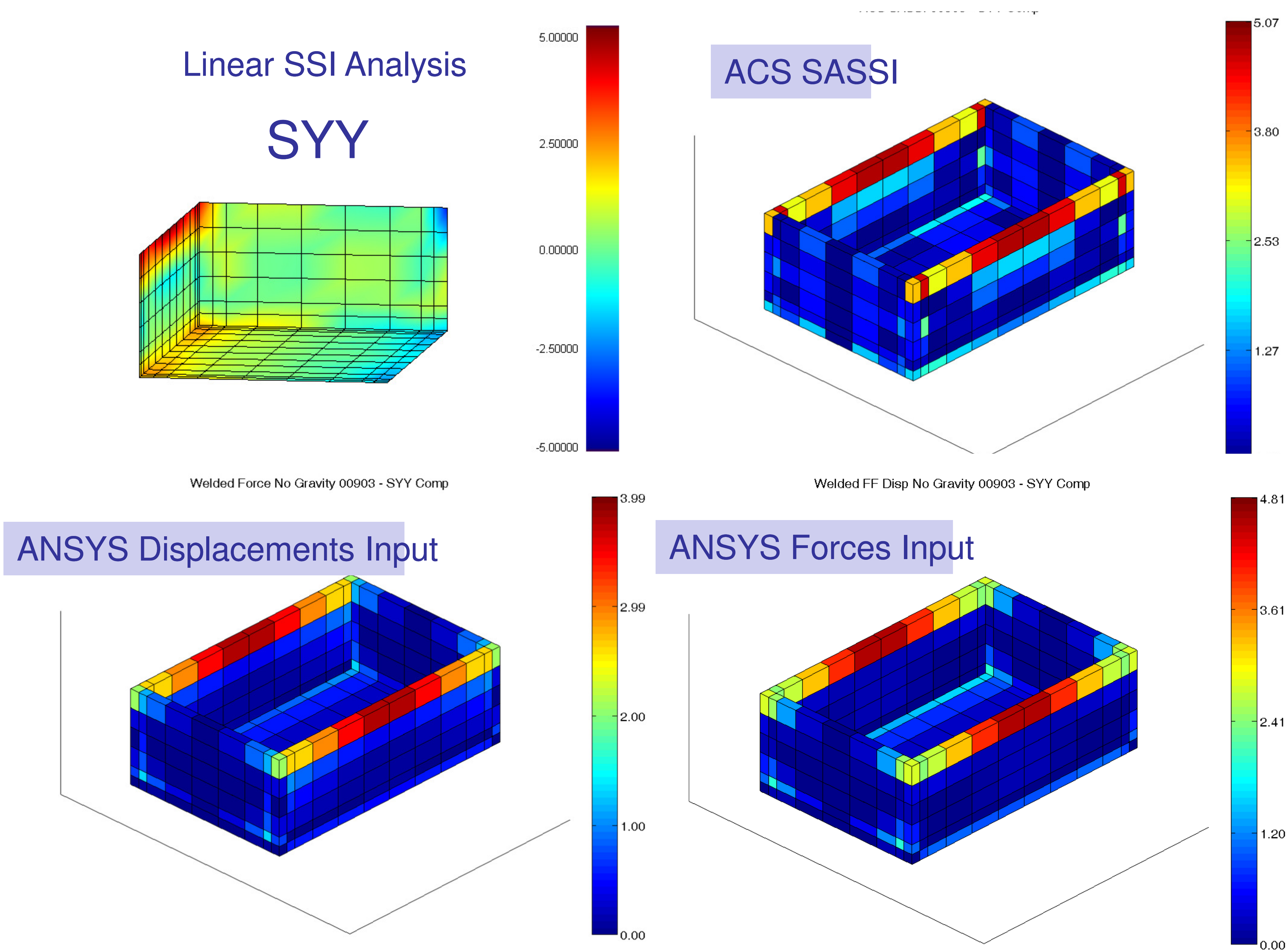
Linear SSI Analysis



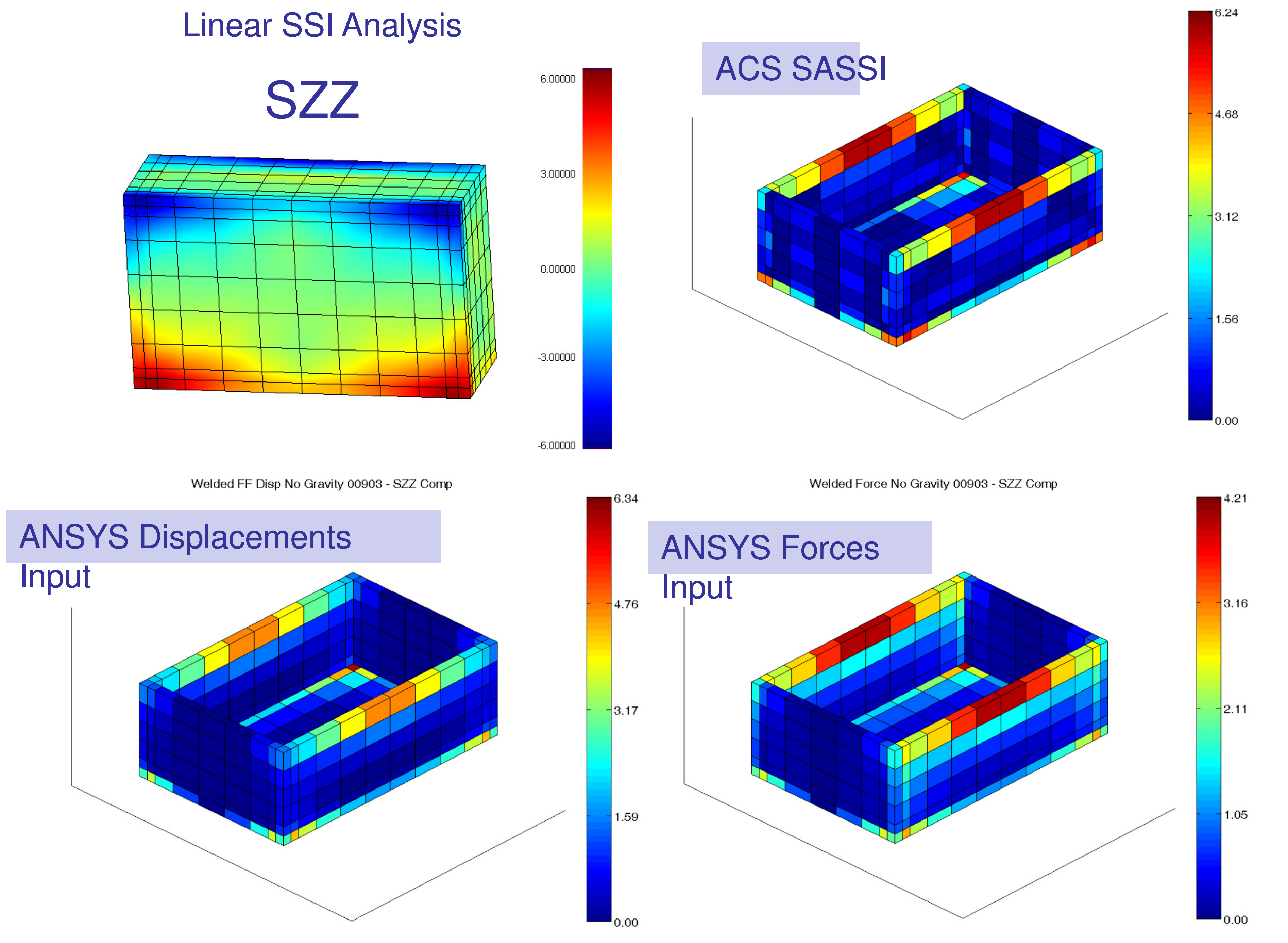
ACS SASSI and ANSYS Element Stresses for X-Input (Frame 903)



ACS SASSI and ANSYS Element Stresses for X-Input (Frame 903)



ACS SASSI and ANSYS Element Stresses for X-Input (Frame 903)



2011 ACS SASSI ACS SASSI NQA V230 Fast-Solver Code (Option FS)

ACS SASSI NQA V230 Fast-Solver Code (Option FS)

The ACS SASSI "Fast-Solver Option" offers a much faster SSI analysis solution than the baseline code, especially for complex, larger-size FE models. The fast-solver code takes advantage of the newer Windows 64-bit multicore-processor PC architecture.

The ACS SASSI fast-solver code provides much faster solutions that are tens and hundreds of times faster than the standard SASSI software based on the skyline per block solution algorithm. The fast-solver code is much faster than the standard code, especially for larger-size SSI problems that include hundreds of thousands of equations, up to 100,000 nodes and 600,000 equations.

The larger the SSI model is, the more efficient the new fast-solver is in comparison with the standard SASSI solver based on the skyline per block algorithm. The required hard-drive storage by the new fast-solver version for the SSI initiation runs for large size models is only a small fraction of that required by the standard SASSI code versions.

SSI Analysis Runtime Comparison for Standard SASSI Algorithm vs. ACS SASSI Fast-Solver Algorithm Using A Windows 7 16 GB RAM PC with IC-7 3820 Processor

SSI Model Description	Total Number of Nodes	Number of Interaction Nodes	SSI Analysis Runtime/Frequency	
			Standard Algorithm	Fast-Solver Algorithm
TB Complex Surface Model	13152	1312	25 min	4 min
RB Complex Surface Model	18532	1389	47 min	5.5 min
RB Complex ** Embedded Model FV (7 layers)	27883	11200	Not Runnable, Too slow	408 min
RB Complex Embedded Model (4 layers) FI-EVBN	23715	3216	567 min	20 min
SSSI Standard Plant Surface Model	81120	7386	Non Runnable, Too slow	82 min
SSSI Standard Plant* (4 layers) Embedded Model FI-EVBN	71145	9648	Not Runnable, Too slow	244 min
NI Complex Embedded Surface Model	48082	3625	Not Runnable, Too slow	42 min
NI Complex Embedded Model (7 Layers) FI-EVBN	57246	7720	Not Runnable, Too slow	105 min
RB Complex Embedded Model (8 layers) FV*	35680	9310	Not Runnable, Too slow	135 min
SSSI RB-TB Surface Model	33690	2780	1210 min	25 min

*there runs were done on 32 GB RAM PC with IC-7 3820 Processor

**these runs were done on 64 GB RAM PC with IC-7 3820 Processor

Automatic SSI Model Checking Tools

The ACS SASSI fast-solver code has some unique features for SSI model checking. The fast-solver code automatically checks the SSI solution based on the computed complex acceleration transfer functions (ATF).

Zero-Frequency Checking: For the 1st frequency (it should be frequency number equal to one), the fast-solver ANALYS module checks the ATF amplitudes for all the translation dofs in the principal direction. If the computed ATF amplitudes vary with more than 5% from unity, then a warning message is displayed and also printed in the ANALYS output. A list of the equations for which the ATF amplitude values deviate from 1.00 with more than 5% is provided in FILE66. Node number dofs could be identified using the node-equation mapping available in the HOUSE output. In these cases, the SSI model should be carefully revised by the user.

*All Frequency Checking: For all selected SSI frequencies, the fast-solver ANALYS module code*checks the ATF amplitudes for abnormally large values. These could occur most likely due to poor FE modeling or SSI analysis input errors.

Automatic Checking for Potential Unstable Solutions

If the ATF amplitudes for translation dofs indicate apparent spurious peaks/valleys, a message “potential unstable frequencies” will be displayed and printed in ANALYS output. A list of these potential unstable frequencies with apparent spurious solutions will be printed.

In addition to FILE8, there is a FILE80 that contains the complex ATF solution for the original SSI frequency set, and a FILE8C that contains the complex ATF solution for all the stable SSI frequencies. In FILE8C, the potentially unstable frequencies are not removed.

Building SSI Models
Demo Problems

ACS SASSI NQA Verification & Validation

Verification Manual includes 37 Selected SSI Problems (more than 100 subproblems, 5,800 files, 480MB) to cover most of the ACS SASSI functionalities:

- 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 (80 layers/2009, FV vs. FI methods/2010)

ACS SASSI Version 2.3.0 Problem Size Limitations

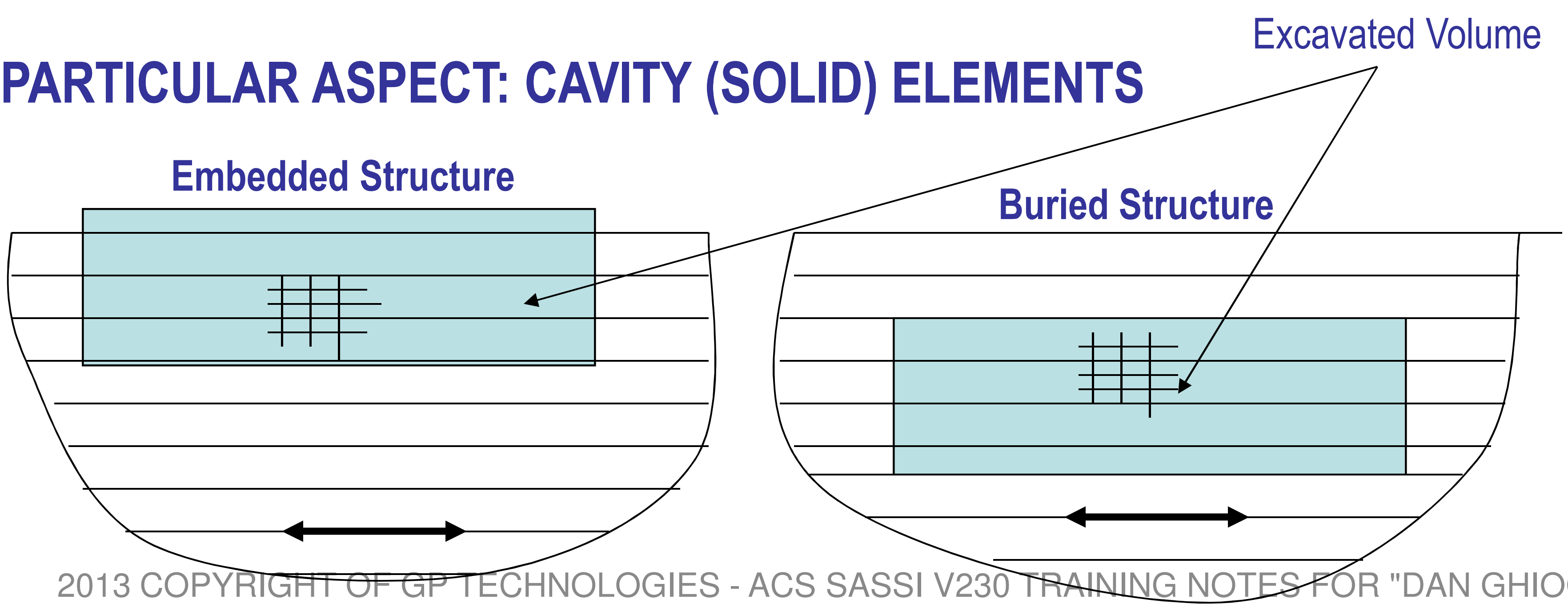
The most important ACS SASSI Version 2.3.0 *SSI problem size limitations*:

- Maximum number of SSI frequencies is 500 (1500 for MOTION, STRESS)
- Maximum number of soil layers is 125
- Maximum number of half-space layers is 20
- Maximum number of the time steps/Fourier frequency points is 16384
- Maximum number of damping ratios for response spectra computation is 5
- Maximum number of all SSI model nodes is limited by the hardware
- Maximum number of interaction nodes for global impedance analysis is 10,000
- Maximum of 5,000 interaction nodes per embedment level for incoherent SSI analysis.
- Maximum number of elements per group 500
- Maximum number of structural embedment node layers (sets with interaction nodes with different Z coordinates) for seismic motion incoherency analysis is 50

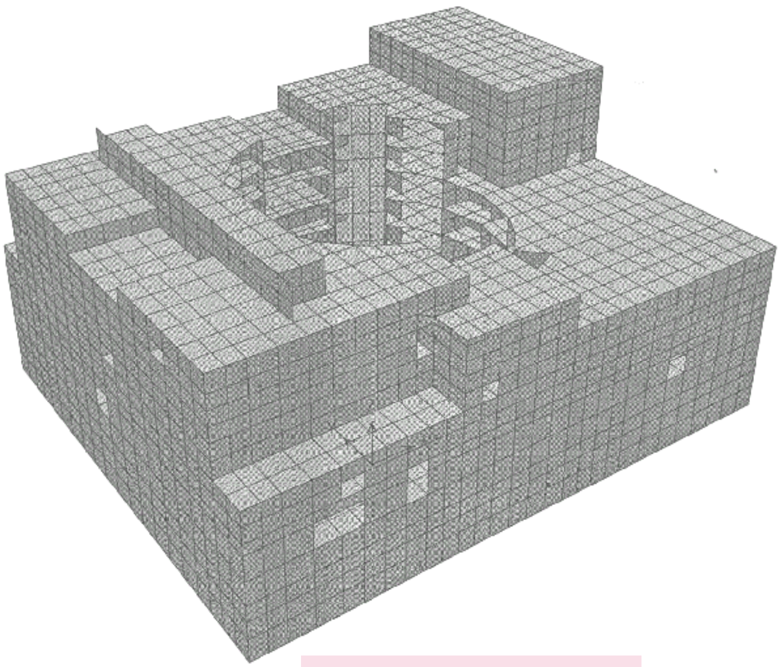
Building A SSI Analysis Models

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

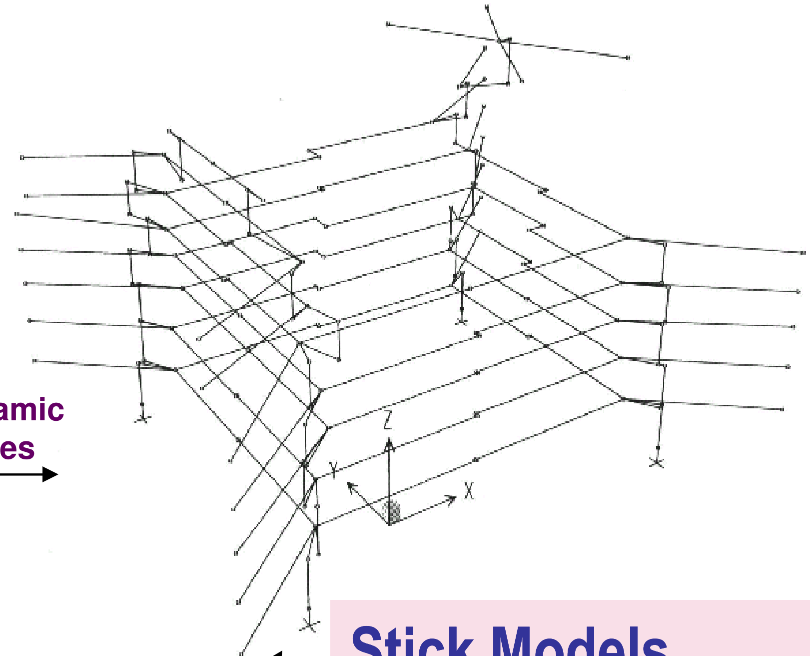
PARTICULAR ASPECT: CAVITY (SOLID) ELEMENTS

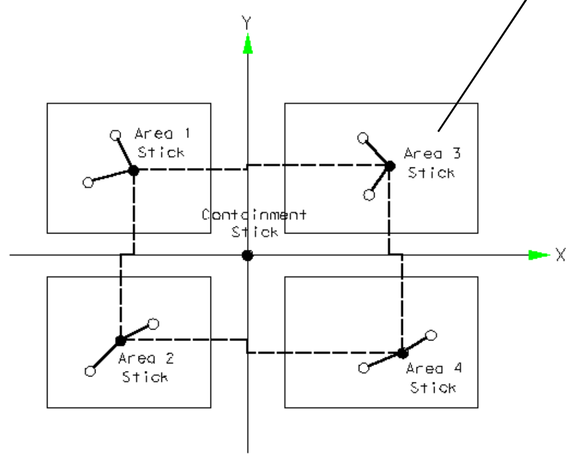


Stick Models vs. FEA SSI Models



Same Dynamic Properties





Detailed 3D Structural FE Model

Stick Models.
Labor Intensive
Calibration
Hidden Problem:
STICKS COULD BE NUMERICALLY SENSITIVE

Specific SSI Model Building Requirements

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
- Excavation volume layers to be numbered from ground surface to baserock
- Excavation volume elements to be numbered from ground surface to baserock

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 by email at dan.ghiocel@ghiocel-tech.com.

ACS SASSI Session
for Describing the .Pre File Structure

Example of the .Pre File for A Embedded Rigid Cylinder

.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:

NGEN command

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

Generates additional nodes from a pattern of nodes.

ITIME, *INC*

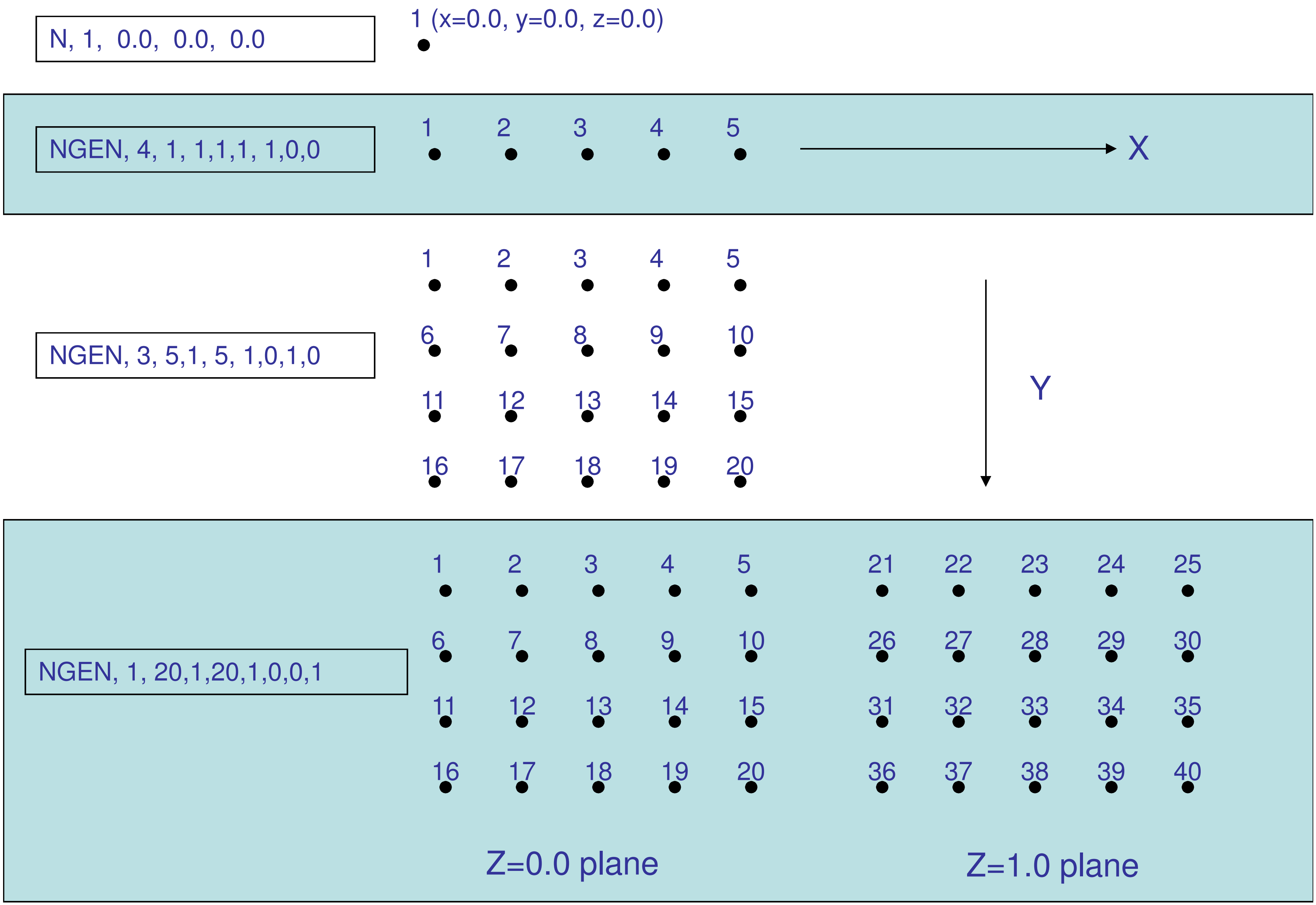
Do this generation operation a total of *ITIME* times, incrementing all nodes in the given pattern by *INC* each time after the first. *ITIME* must be > 1 for generation to occur.

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
1 (x=0.0, y=0.0, z=0.0)
NGEN, 4, 1, 1,1,1, 1,0,0
1 2 3 4 5
X
NGEN, 3, 5,1, 5, 1,0,1,0
1 2 3 4 5
6 7 8 9 10
11 12 13 14 15
16 17 18 19 20
Y
NGEN, 1, 20,1,20,1,0,0,1
1 2 3 4 5
6 7 8 9 10
11 12 13 14 15
16 17 18 19 20
21 22 23 24 25
26 27 28 29 30
31 32 33 34 35
36 37 38 39 40
Z=0.0 plane
Z=1.0 plane

.PRE File Structure

Input constrained displacement by “D” command:

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

Input interaction nodes by “INT” command:

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

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

Input material properties by “M” command:

.PRE File Structure

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 Structure

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

Input solid element by “E” command:

EINT,1,440,1,1

EINT command for solid element

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

MSET command for solid element

.PRE File Structure

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" command for element group 2

```
ETYPE,1,440,1,1
```

"EINT" command for element group 2

```
EINT,1,440,1,2
```

MSET command for beam element

```
MSET,1,440,1,2
```

EGEN Command

- **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.

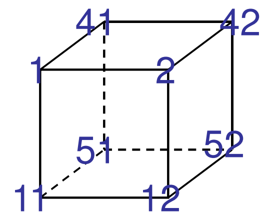
IEL1, *IEL2*, *IEINC*

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

MINC

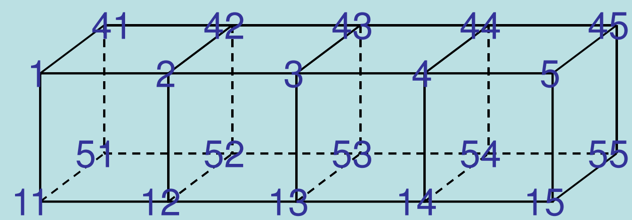
Increment material number of all elements in the given pattern by *MINC* each time after the first.

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



Element #1

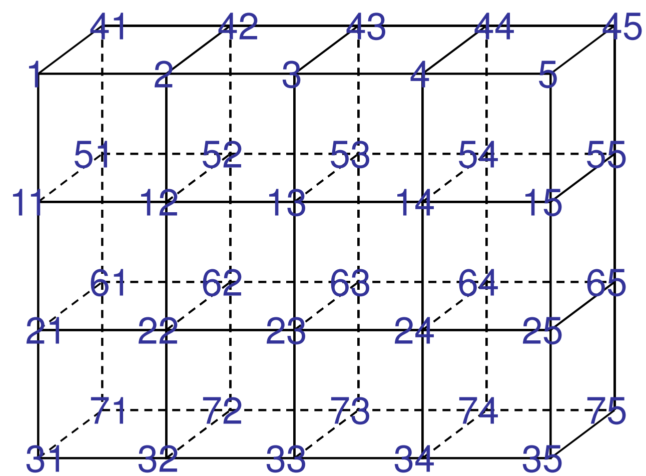
EGEN,3,1,1,1,1



E #1-4

(old 1) (new 2-4)

EGEN,2,10,1,4,1



E #1-12

(old 1-4, new 5-12)

.PRE File Structure

EQTIT,

“EQTIT” command

RSIN,1,
RSIN,2,
RSIN,3,

“RSIN” command

RSOUT,1,
RSOUT,2,
RSOUT,3,

“ROUT” command

ACCIN,1,
ACCIN,2,
ACCIN,3,

“ACCIN” command

.PRE File Structure

THFILE

“THFIL” command

STRESS,0,1,1,1

“STRESS” command

* Frequencies
FREQ,1,1,10,20,30,40,50,60,80,100,120
FREQ,2,1,10
FREQ,3,1,10,20,30,40,50

“FREQ” command

Hands-on Session follows....