Engineering Advances Implemented in ACS SASSI Version 3.0 for Seismic SSI Analysis of Nuclear Structures

Ghiocel Predictive Technologies Inc.

Dr. Dan M. Ghiocel
Email: dan.ghiocel@ghiocel-tech.com
Ghiocel Predictive Technologies Inc.
http://www.ghiocel-tech.com

PART 1: New SSI Analysis Capabilities

Tokyo Station Convention Center

April 14, 2015

Scope of this Presentation:

This presentation will discuss the new 2015 ACS SASSI Version 3.0 capabilities for seismic SSI analysis of nuclear structures.

Some of the new developed SSI capabilities are related to the new ASCE 04-2015 recommendations and the new USNRC SRP 3.7.1 and 3.7.2 requirements.

New 2015 ACS SASSI Version 3.0 SSI Capabilities.

A. Improvements in Fast-Solver Version (Available Now)

1) Simulation of Spectrum Compatible Input Accelerations
2) Fast-Flexible Volume (FFV) for deeply embedded structures
3) Incoherent SSI Analysis for evaluating SSI and SSSI effects
4) 3 times faster SSI solution with single run for all X,Y and Z input directions

B. Major Developments (Available Now or in Next 3 Months)

DETERMINISTIC SSI ANALYSIS:

1) Section-Cuts capability for shearwall structures (NEW, now)
2) ACS SASSI-ANSYS integration in *Option AA* (NEW, now)
3) Nonlinear Structure SSI Analysis in *Option NON* (NEW, July 15)
4) Random Vibration Theory SSI in *Option RVT* (NEW, May 15)

PROBABILISTIC SSI ANALYSIS (NEW, May 15)

1) Probabilistic Site Response Analysis (PSRA) in *Option PRO*
2) Probabilistic SSI Analysis (PSSIA) in *Option PRO*

A. Improvements in Fast-Solver Version

1) *Improved simulation of spectrum compatible input acceleration time histories* (EQUAKE) in full compliance with USNRC SRP 3.7.1 criteria

2) Included 2D and symmetric models (HOUSE, ANALYS)

3) Improved motion incoherency modeling (HOUSE)

- Include both generic, isotropic (radial) and new *site-specific, directional (anisotropic)* Abrahamson plane-wave incoherency models for rock and soil
- Include *user defined plane-wave coherency models* for X, Y and Z.
- Perform a *multilevel incoherency analysis* for deeply embedded structures, as SMRs, or for nonuniform geometry basements
- Included capability to input *incoherent motions with nonuniform amplitude spatial variations in horizontal plane (different motions at interaction nodes)* (HOUSE).

4) *No specific software limitation for the number of interaction nodes* rather than 100,000 nodes limitation for all FE model nodes. The practical limitation is the amount of RAM available for the SSI runs (ANALYS).

5) *SSI solution much faster than Version 2.3.0*. The SSI analysis is about 3 times faster; no need to do restart analyses for Y and Z input components (ANALYS)

6) *Develop SUBMODELER GUI module with new commands for*
   - SSI model building, checking and improving its numerical condition.
   - Section-Cut calculations for concrete shearwalls and
   - Option A-AA capability for converting and merging ANSYS models including solid and fluid elements.

7) *Number of Fourier frequencies* up to 32,768 frequencies (SOIL, MOTION, RELDISP, STRESS).

8) *Up to 200 soil layers* for SSI analysis for deep soft soil deposits.(SITE)

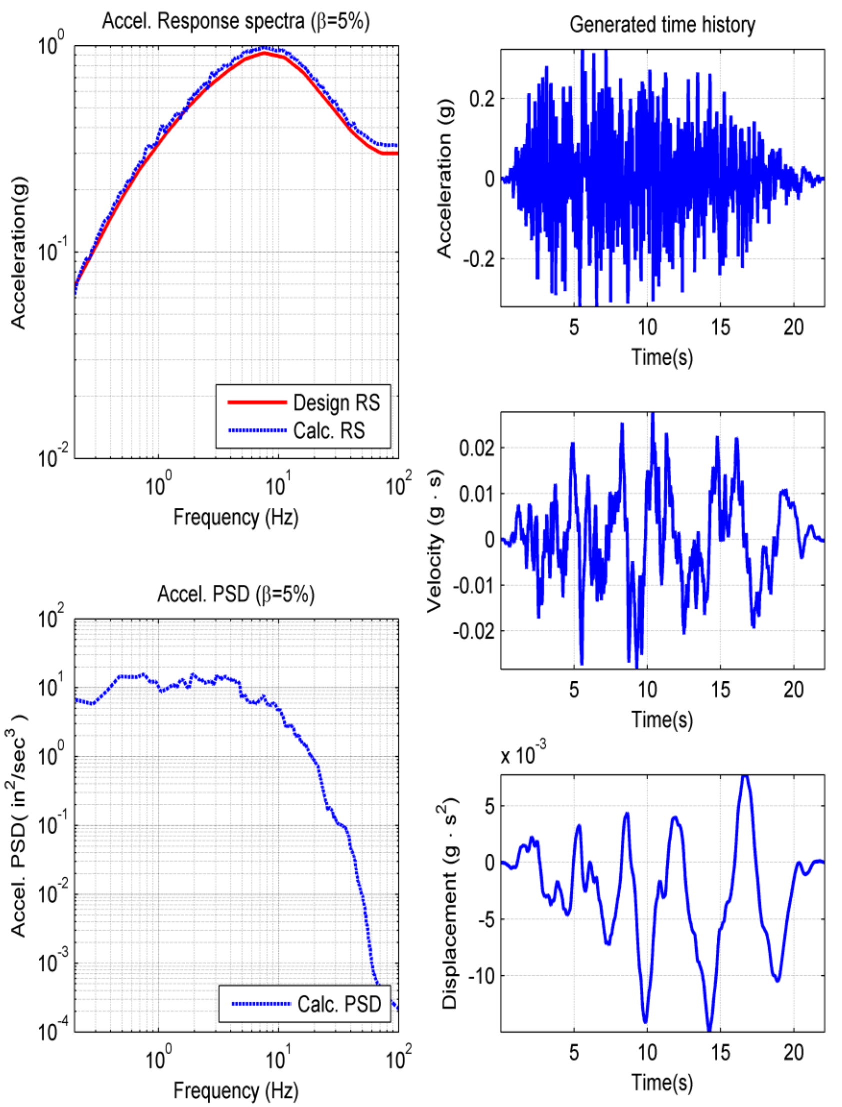
9) *Fast FV approach* for deeply embedded structures. Automatic interaction nodes generation included. (SUBMODELER)

12) *Improved interpolation scheme for ATF and STF.* New interpolation scheme is based on a complex bicubic spline function that is highly effective for performing incoherent SSI analysis using stochastic simulation with a larger number of SSI frequencies. It should be applied without smoothing.(MOTION and STRESS)

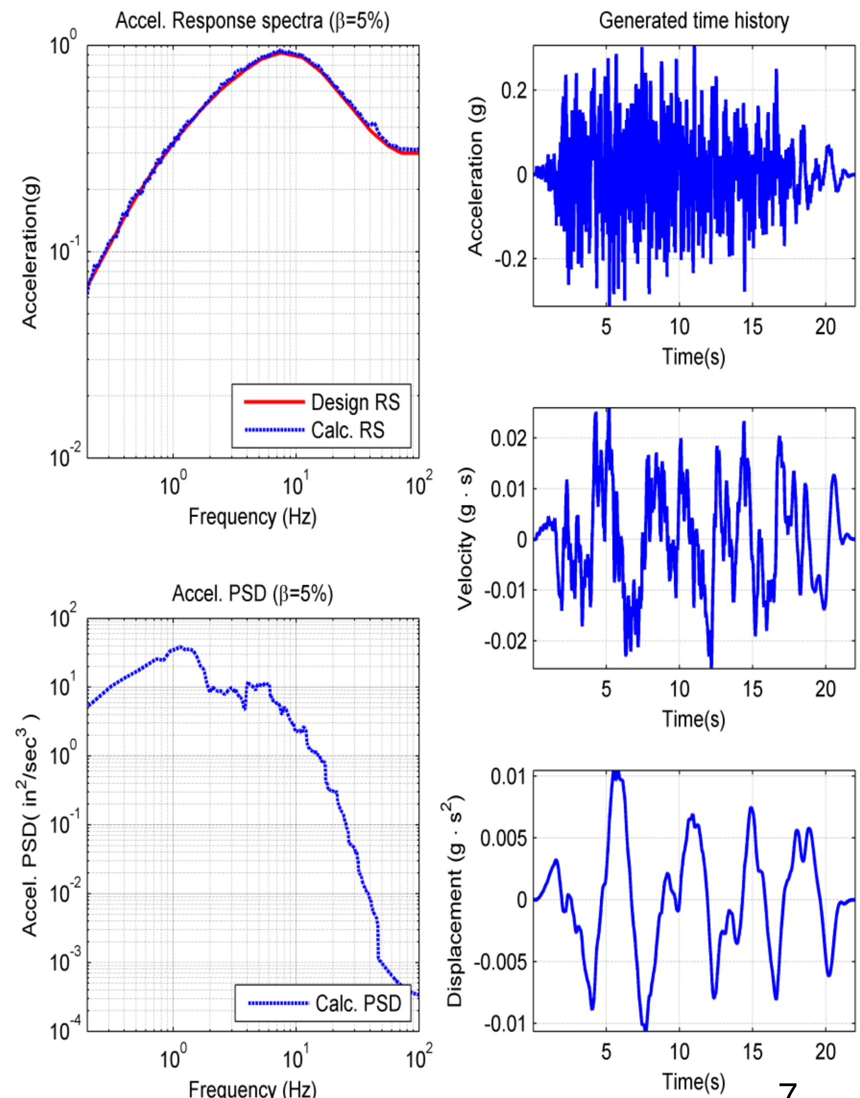
10) *New ACS SASSI-ANSYS interfacing* that uses ANSYS FE model K, C and M matrices directly for SSI analysis. Includes beam, shell, solid, spring, pipes and fluid elements, and also MPC Rigid Beam and Link, and CP and CE commands *(SUBMODELER, HOUSE, ANALYS).*

EQUAKE New Capabilities; Compute Vel, Dis and PSD

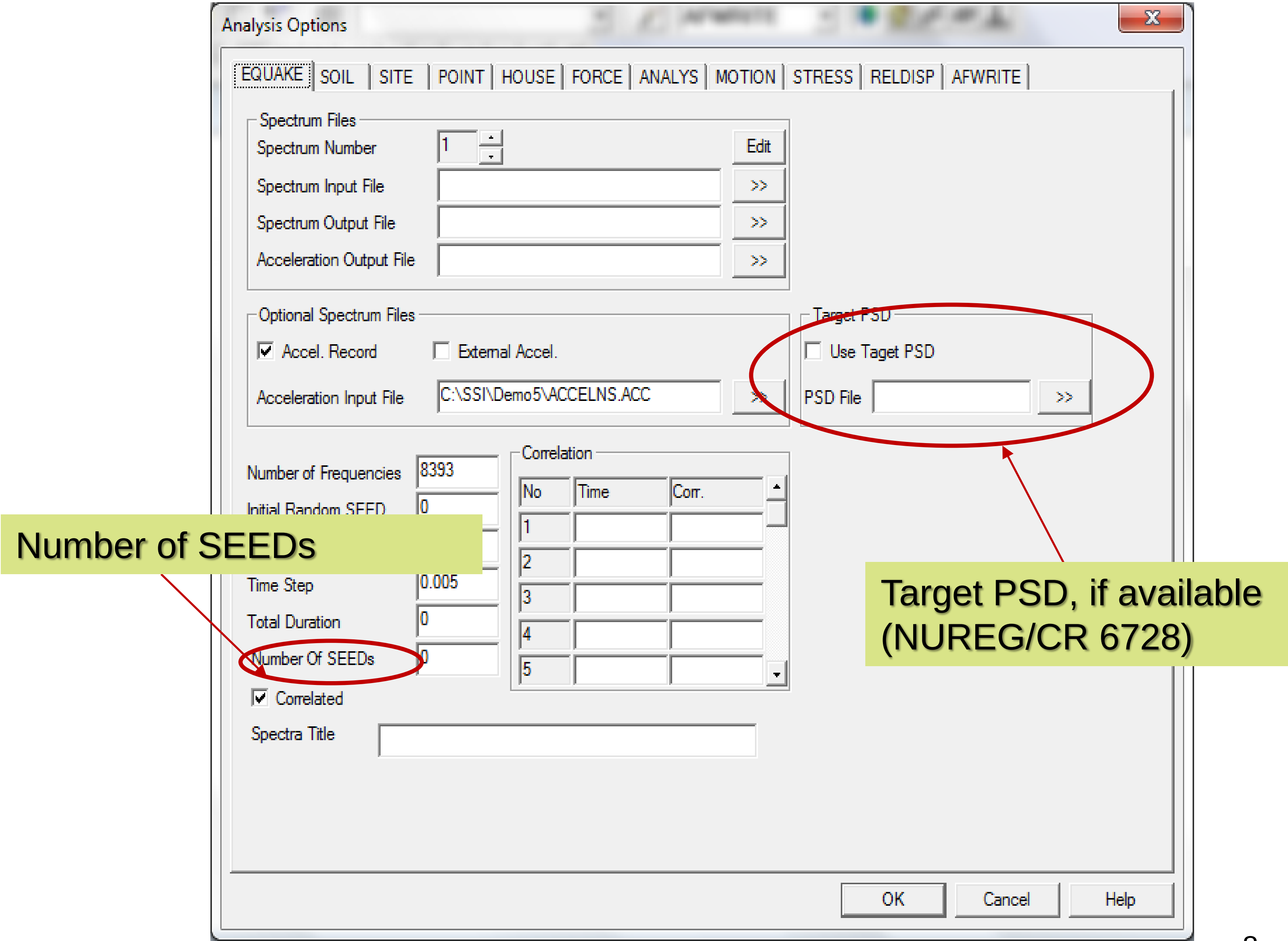
WITHOUT SEED RECORDS



WITH SEED RECORDS



EQUAKE New GUI in PREP/SUBMODELER



HOUSE New Incoherent SSI and Option AA (ANSYS Integration) Analysis Capabilities

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

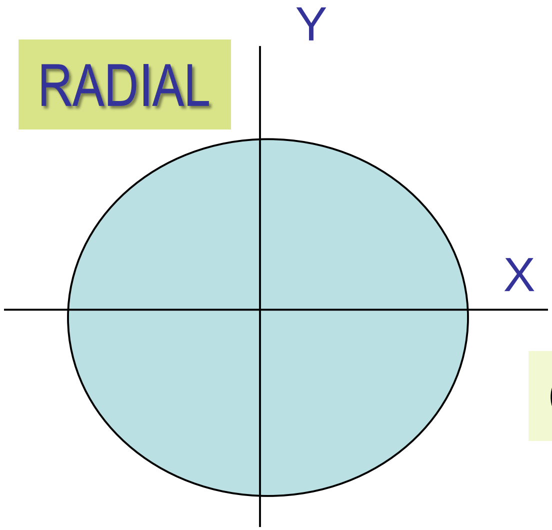
1) 1986 Luco-Wong model (theoretical, unvalidated, geom anisotropic)
2) 1993 Abrahamson model for all sites and surface foundations
3) 2005 Abrahamson model for all sites and surface foundations
4) 2006 Abrahamson model for all sites and embedded foundations
5) 2007 Abrahamson model for hard-rock sites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

REMARKS:

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

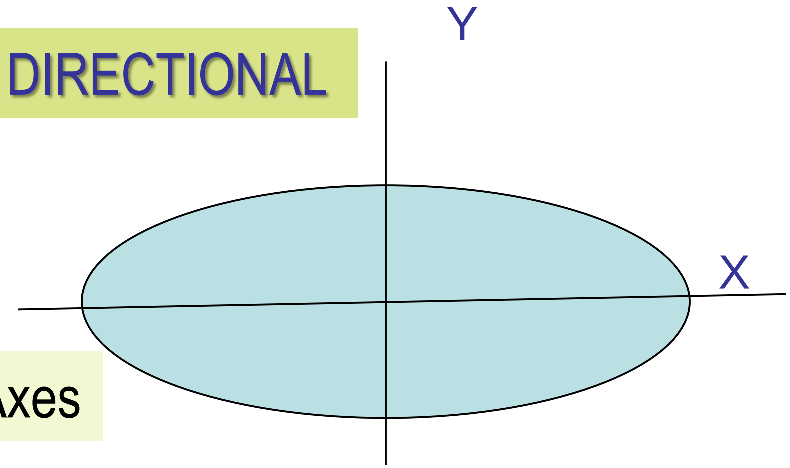
Radial vs. Directional Motion Coherency Models

RADIAL



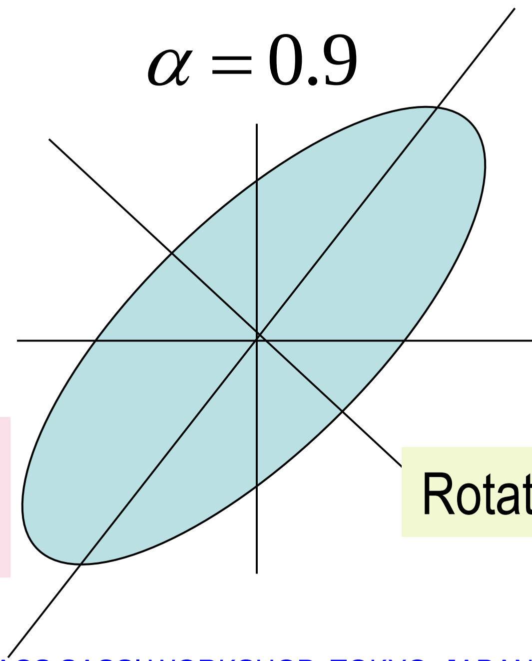
$$\alpha = 0.5$$

DIRECTIONAL



Global Axes

$$\alpha = 0.9$$

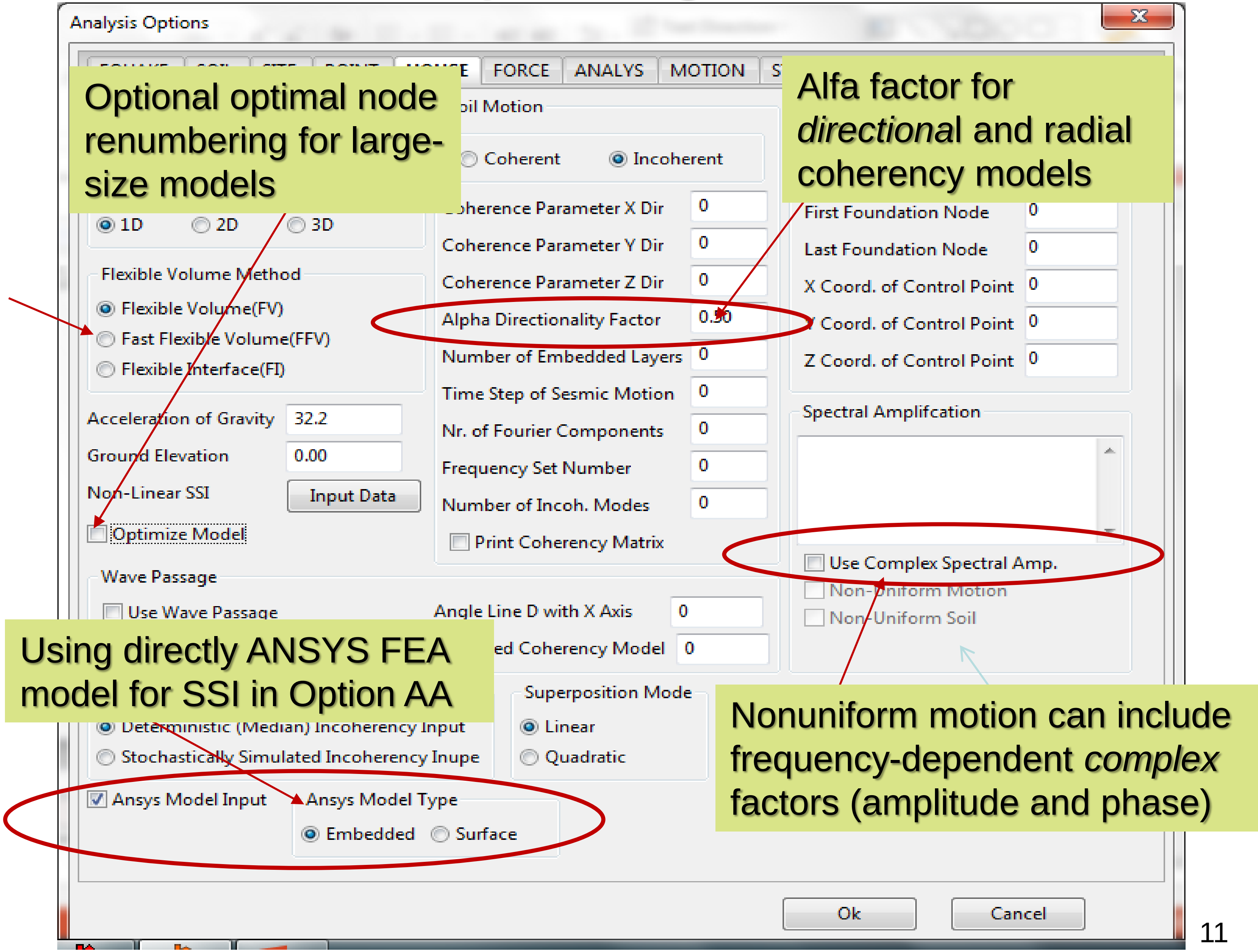


Rotated Axes

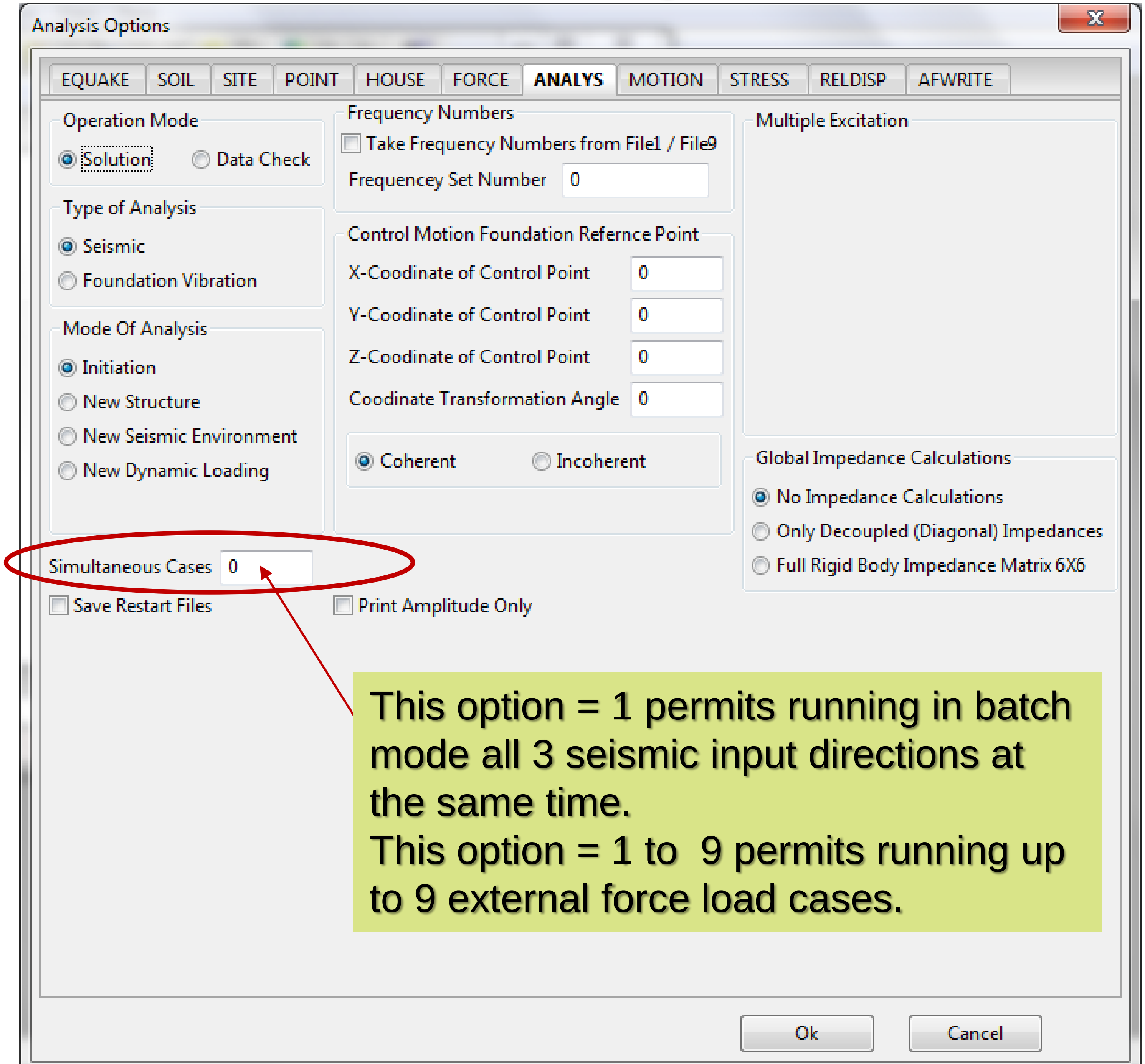
$$D\verb|^|2 = 2[(1-\alpha)Dx\verb|^|2 + \alpha Dy\verb|^|2]$$

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

HOUSE New GUI Input Using SUBMODELER Module

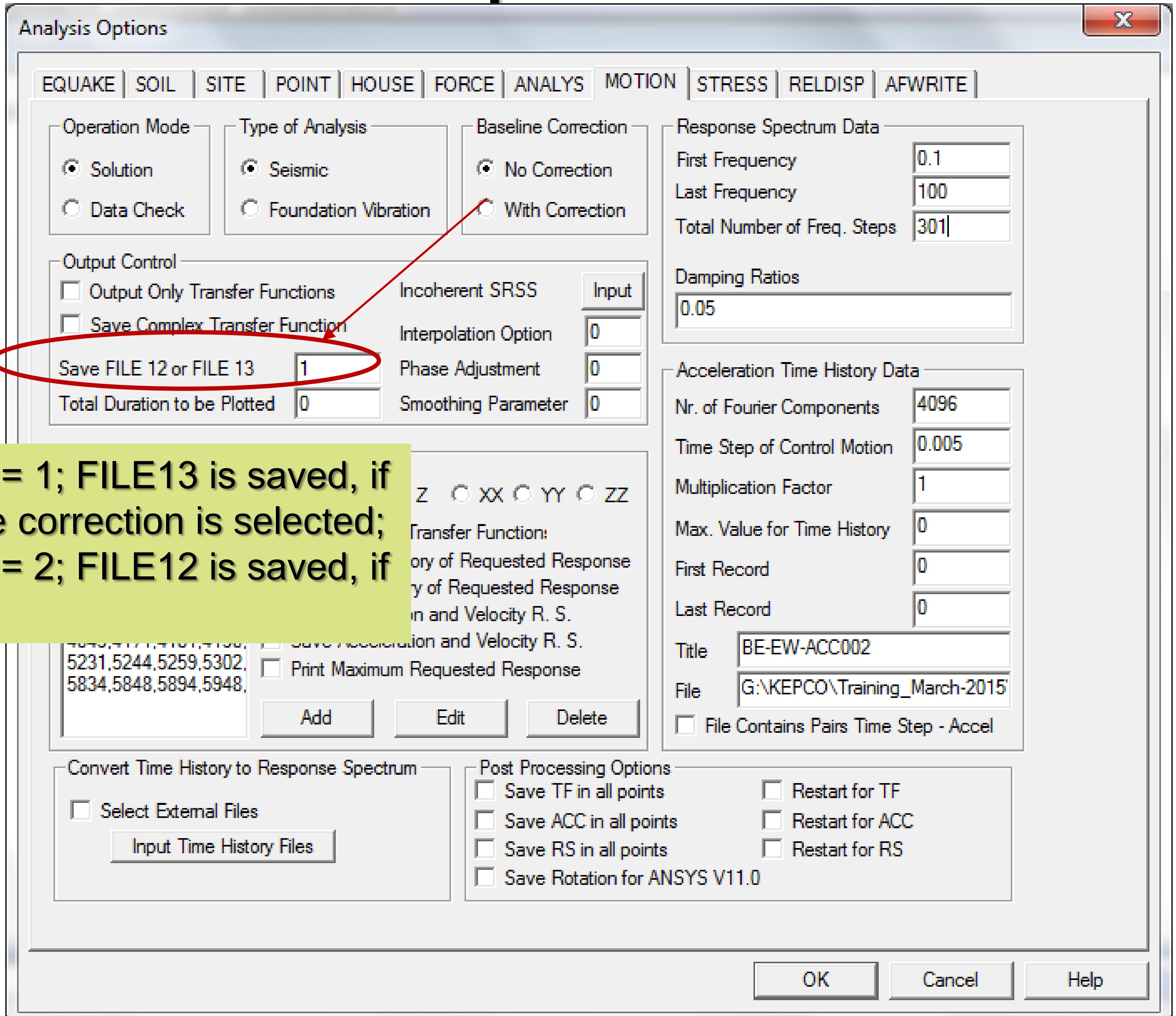


ANALYS New GUI Input Using SUBMODELER Module



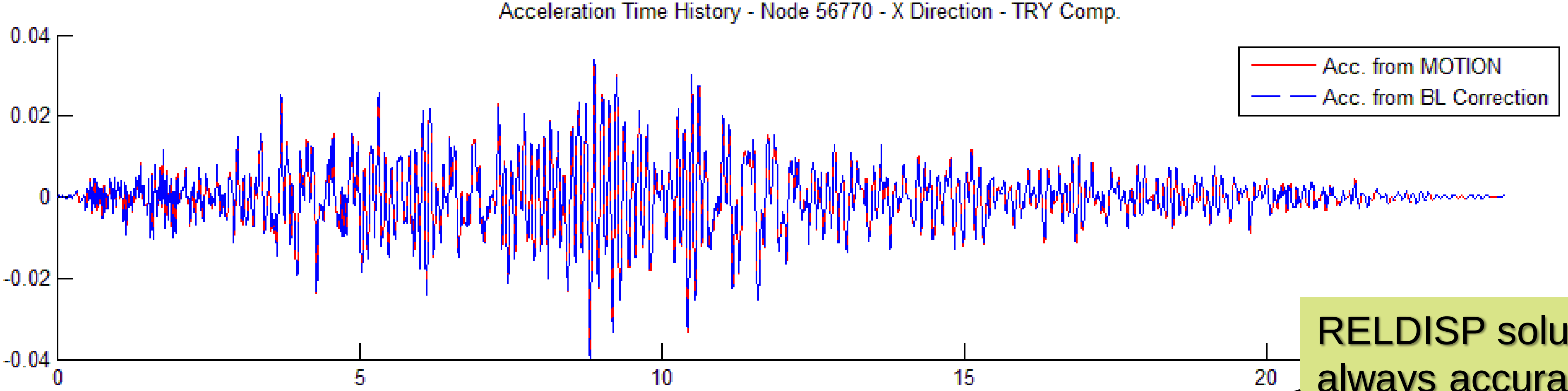
This option = 1 permits running in batch mode all 3 seismic input directions at the same time.
This option = 1 to 9 permits running up to 9 external force load cases.

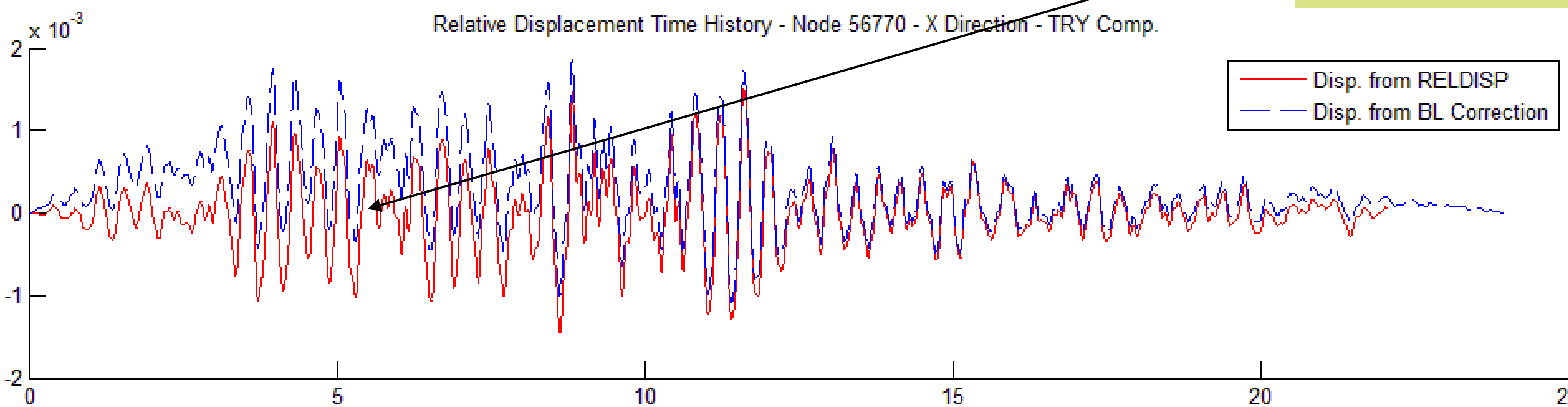
MOTION New GUI Inputs in PREP/SUBMODELER



Save… = 1; FILE13 is saved, if baseline correction is selected;
Save… = 2; FILE12 is saved, if needed

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



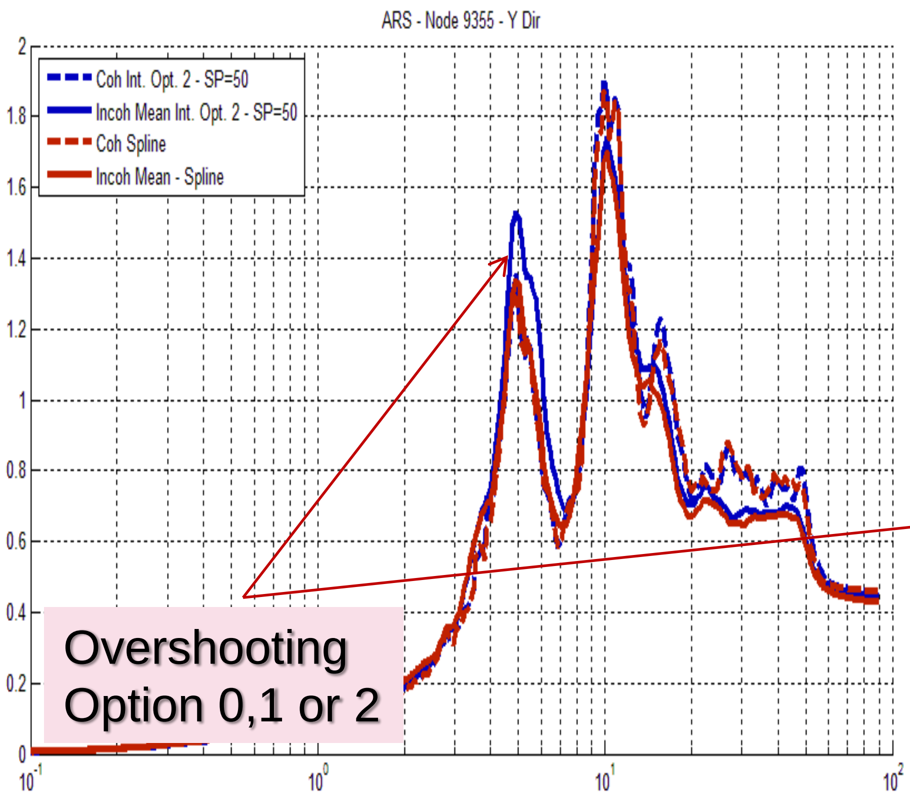


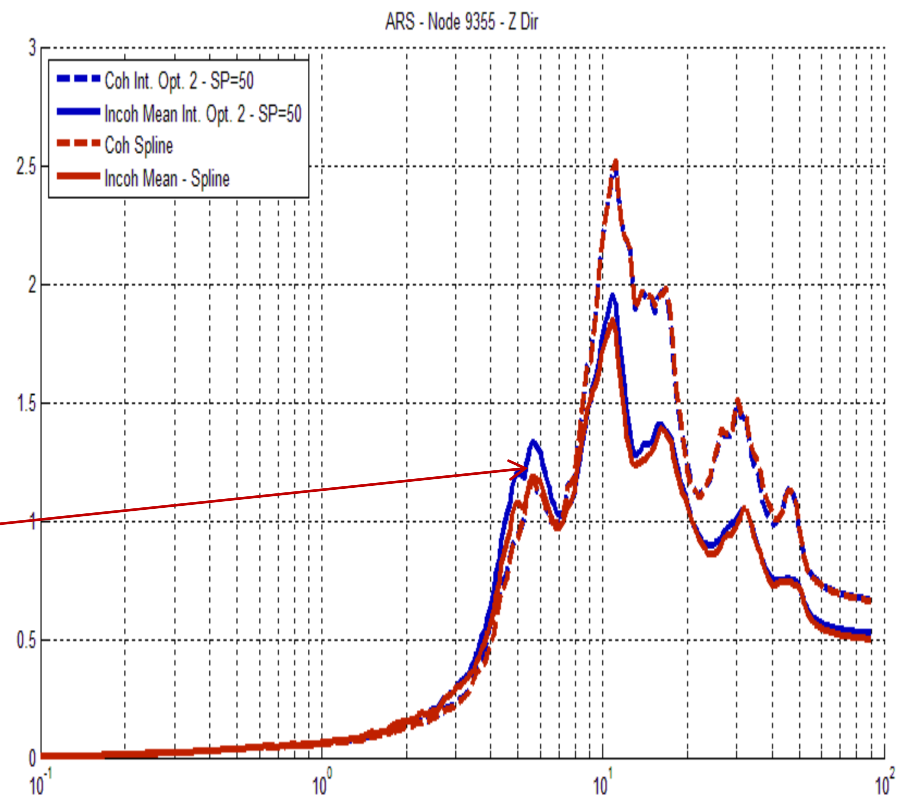
RELDISP solu
always accura
Use RELDISP

Spline Interpolation Good for Incoherent SSI; Eliminates Overshooting for Low-Frequency RS Peaks

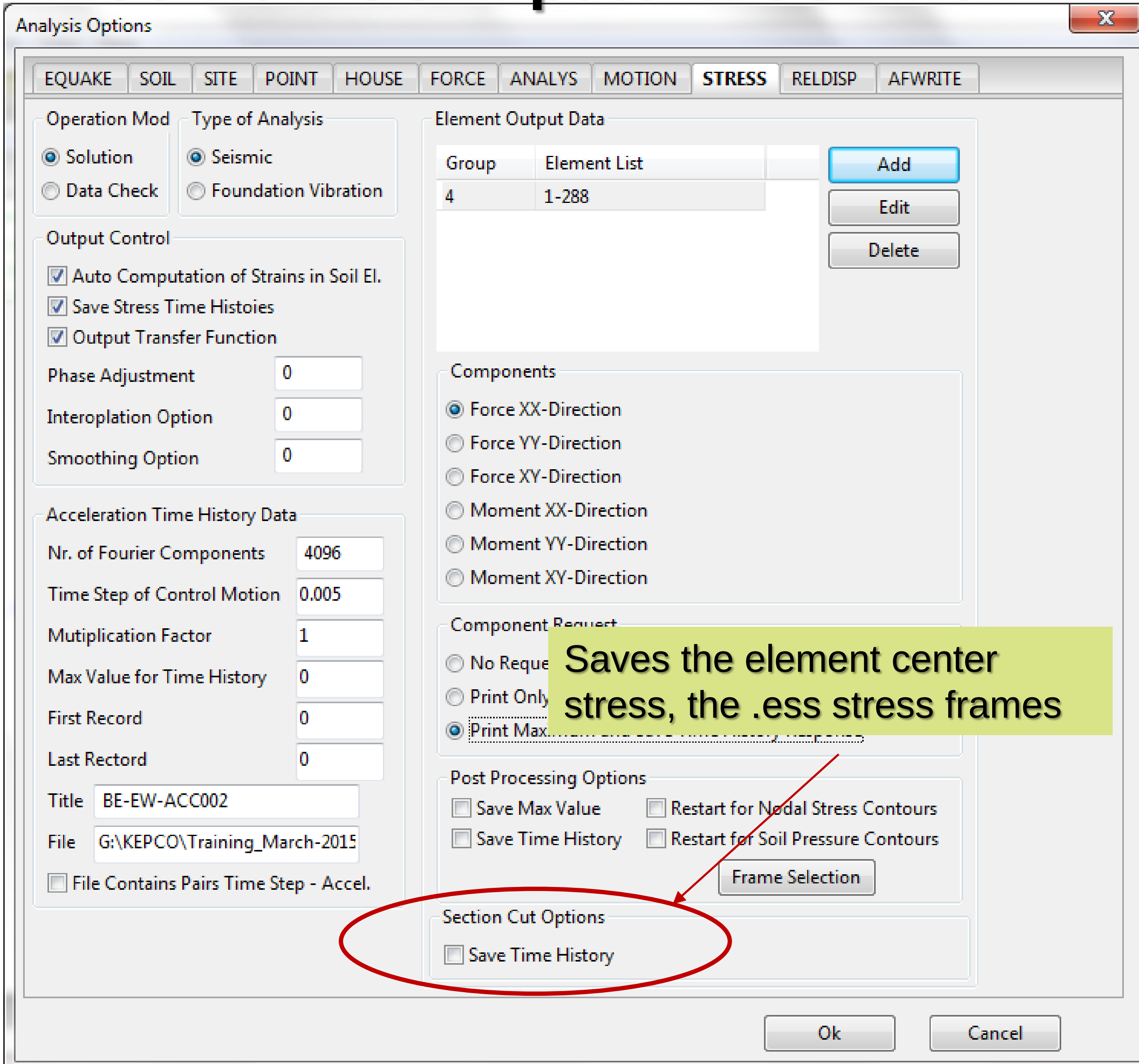
ATF in Y-Direction

ATF in Z-Direction

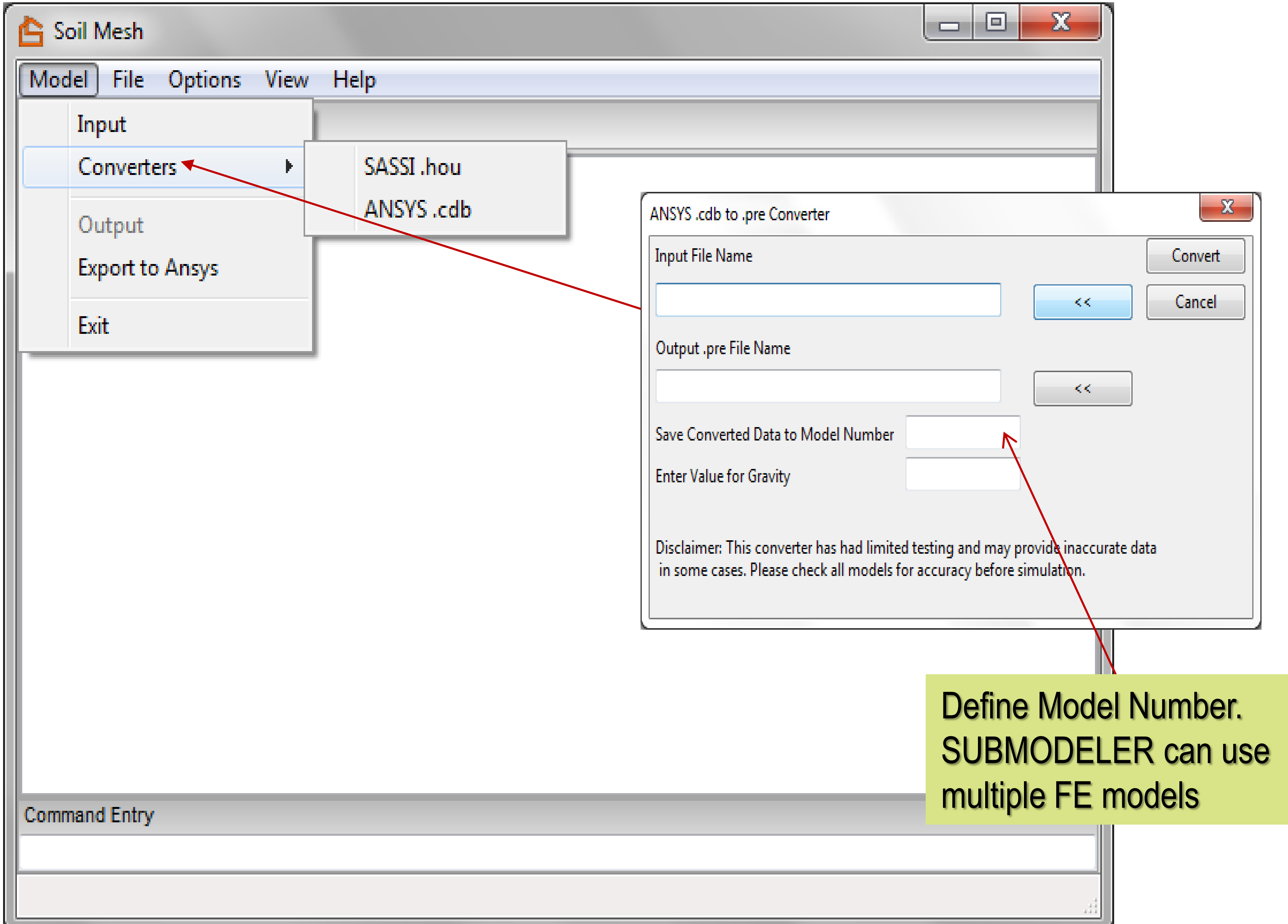




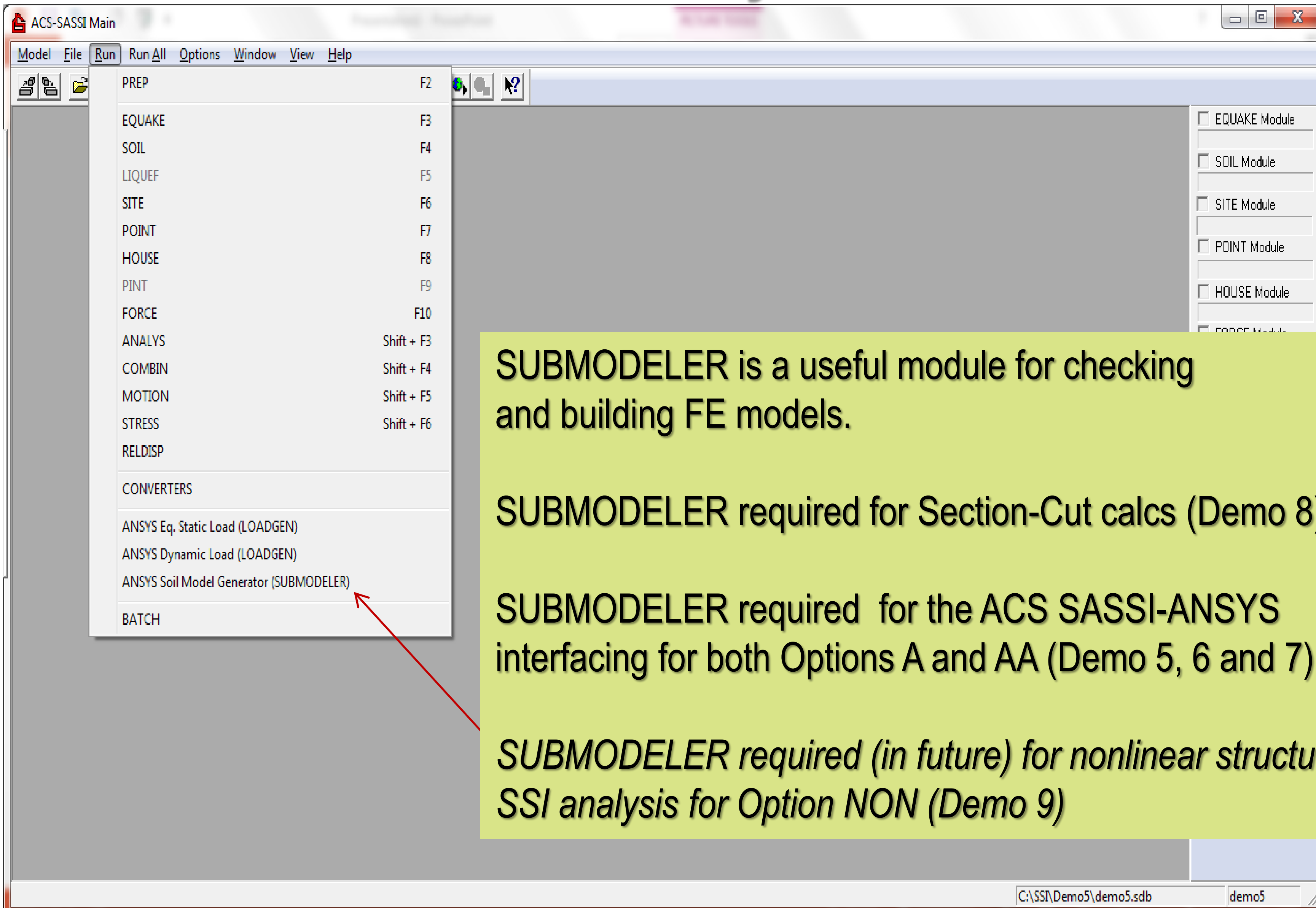
STRESS New GUI Inputs in SUBMODELER



SUBMODELER ANSYS .cdb Converter Includes FLUID80



SUBMODELER Has Many New Commands



SUBMODELER is a useful module for checking and building FE models.

SUBMODELER required for Section-Cut calcs (Demo 8)

SUBMODELER required for the ACS SASSI-ANSYS interfacing for both Options A and AA (Demo 5, 6 and 7)

SUBMODELER required (in future) for nonlinear structure SSI analysis for Option NON (Demo 9)

Using SUBMODELER for FE Model Checking and Improving Its Numerical Condition

Commands for building and checking SSI and SSSI models:
MERGESOIL (AA), EXCAV, EXTRACTEXCAV, INTGEN, FIXEDINT, HINGED

Improving the FEA model numerical condition and speed/storage:
FIXROT, FIXSHELL, FIXSOLID, FIXSPRING

Section-Cut Commands:
CUTVOL, SLICE, CSECT, CALCPAR, CALCSECTHIST, etc.
(see Demo 8)

SUBMODELER commands described in "ACS SASSI-ANSYS Integration Capability" manual

The **MERGESOIL** command takes 2 models in memory and merges the models together. This command is used to combine models from ANSYS in Option AA. The 2 ANSYS models can come from any location.

Example code to merge an ANSYS structure model and ANSYS excavation model for ACS SASSI analysis in Option AA.

Actm,1
Convert,ansys,struct.cdb,32.2
** Adjust Model before the merge*
Actm,2
Convert, ansys,Soil.cdb,32.2
** Excavation has to have elements of type 2*
etypegen,2
Actm,3
MergeSoil,1,2,1,,,,mappingfile.txt

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

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

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

EXTRACTEXCAV command copies excavation elements from the current model to the model specified by the user. The original model is not changed.

Example code for extracting the excavation model in a separate model

```
Actm,1
INP, Example_model.pre
*Create excavation model in Model 2
Extractexcav,2
Actm,2
*Saving the Excavation model 2 in a .pre file
Write, Example_Excavation.pre
```

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

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

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

NOTE: Requires same horizontal meshes at different levels in basement

FIXROT to automatically add the needed soft rotational springs to improve numerical conditioning for detailed flat shell models (for the Kirckhoff plate element the drilling degree of freedom has no stiffness associated with it, and therefore could produce poorly conditioned or unstable numerical models), FIXROT,<Stiff>.

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

```
Actm,1
Inp, Example_Model.pre
* Add soft springs with overall stiffness 10 to oblique shell nodes;
FixRot,10
```

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

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

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

B. Major Developments

DETERMINISTIC SSI ANALYSIS:

1) Section-Cuts capability for shearwall structures (NEW, now)
2) ACS SASSI-ANSYS integration in *Option AA* (NEW, July 14)
3) Nonlinear Structure SSI Analysis in *Option NON* (NEW, July 15)
4) Random Vibration Theory SSI in *Option RVT* (NEW, May 15)

PROBABILISTIC SSI ANALYSIS (NEW, May 15)

1) Probabilistic Site Response Analysis (PSRA) in *Option PRO*
2) Probabilistic SSI Analysis (PSSIA) in *Option PRO*

NOTE: 2015 ACS SASSI NQA Version 3.0 will include only fast-solver version. Separate, additional capabilities will include Option A-AA, PRO, RVT and NON.

1) SUBMODELER Section-Cuts Capability (now)

The SUBMODELER Section-Cut capability has two options:

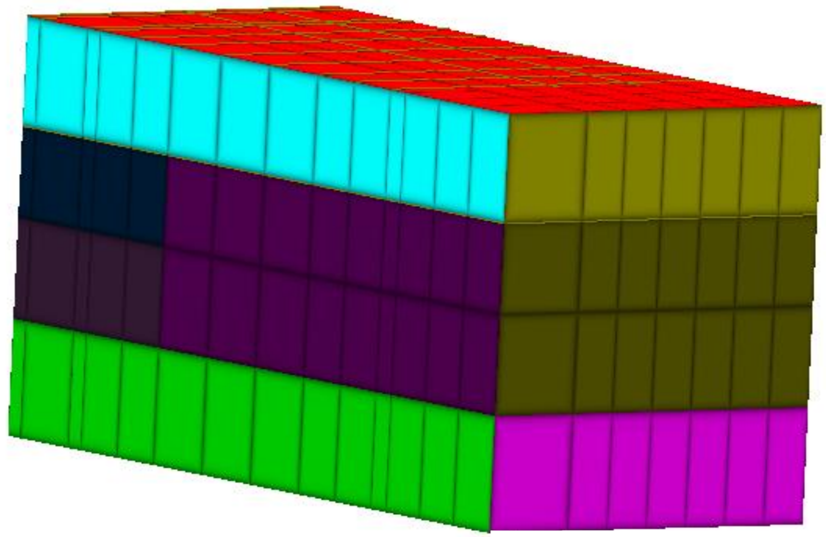
1) Uses *a single frame of stress data* (single .ess frame file) to compute the section-cut forces and moments on a cross-section at a specific time step.

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

Section-Cut Commands:
CUTVOL, SLICE, CSECT, CALCPAR, CALCSECTHIST, etc

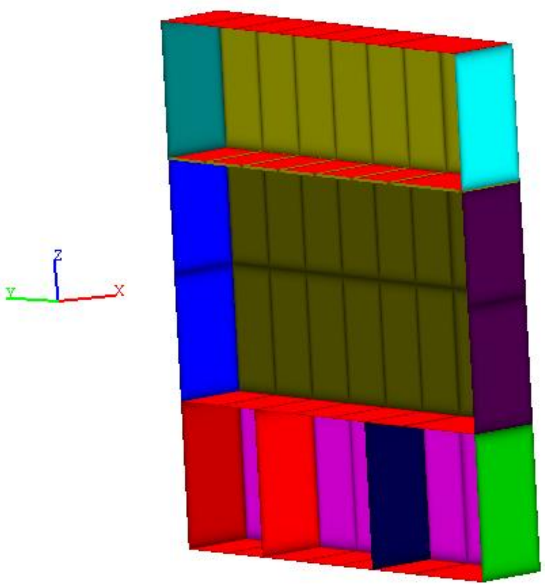
Demo 8

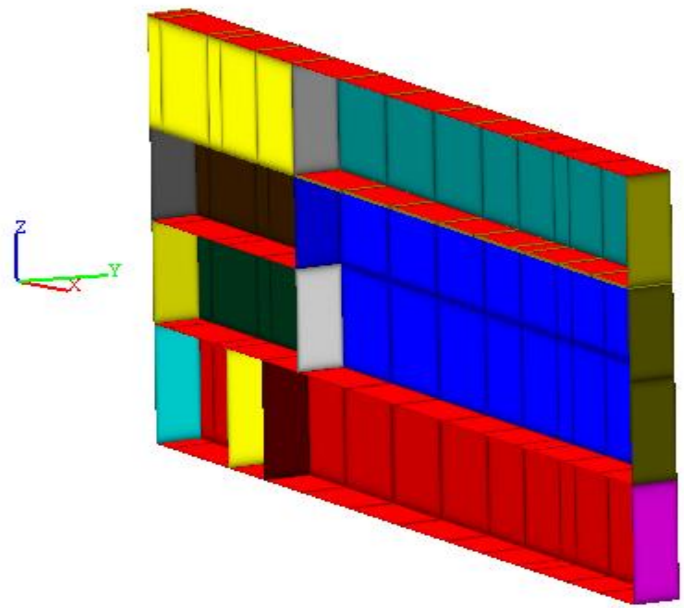
SUBMODELER Section-Cut Models



Transverse Wall Cut Using CUTVOL Command

Longitudinal Wall Cut Using CUTVOL Command





SUBMODELER Section-Cut for Single Stress Frame

* Read element center stress frame

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

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

CUTVOL,1,132.4

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

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

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

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

Section-Cut CALCPAR Command Results

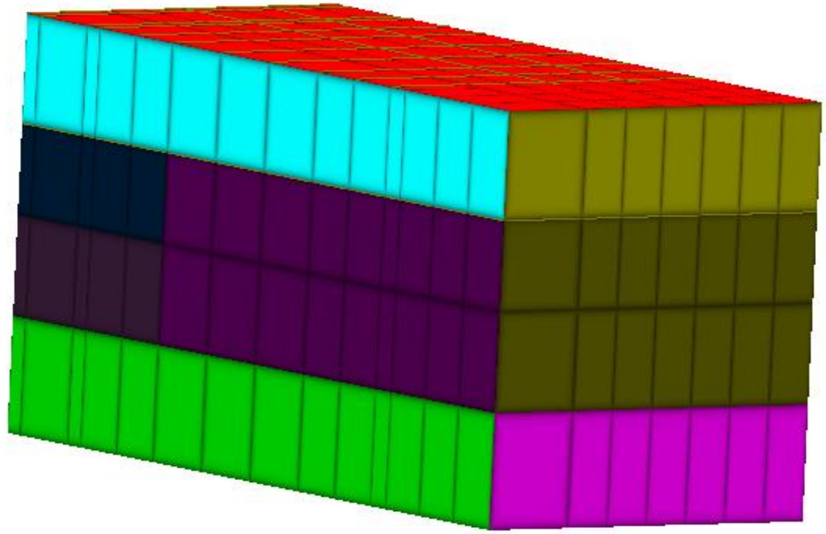
```
Model Parameters
Centroid X =145.443 Y =-149.003 Z = 15.8 Area = 342
Ixx = 305990 Iyy = 5183.71 Izz = 311174
Fx = -28.0657 Fy = 11456.9 Fz = 109.184
Mx = -323054 My = 124.862 Mz = 97618.6
```

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

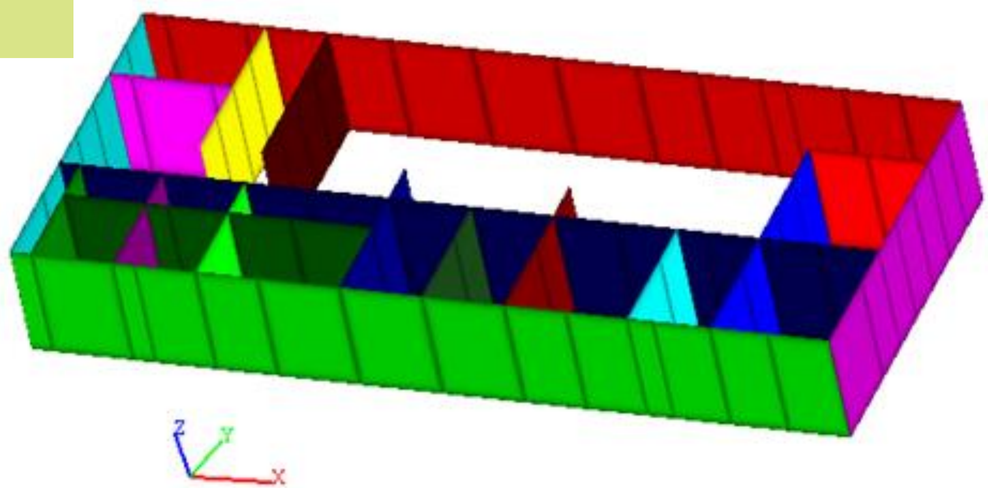
SUBMODELER CALCPAR Command Example

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

SUBMODELER Section-Cut for Multiple Stress Frames



Section-Cut Model Using SLICE Command



SUBMODELER CALCSECTHIST Command Batch Input

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

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

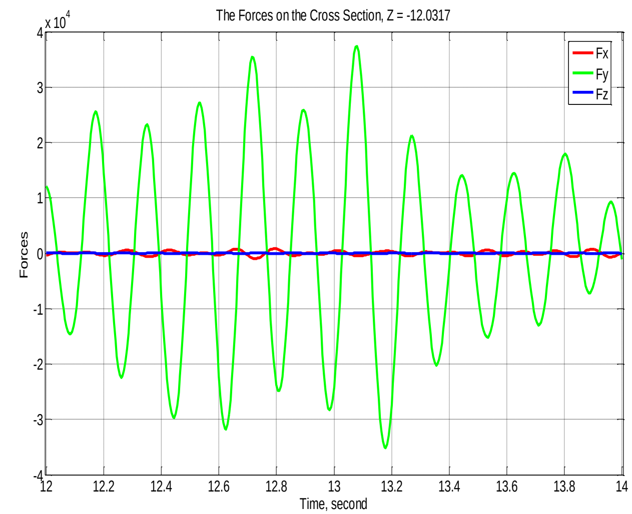
SUBMODELER CALCSECTHIST Command Example

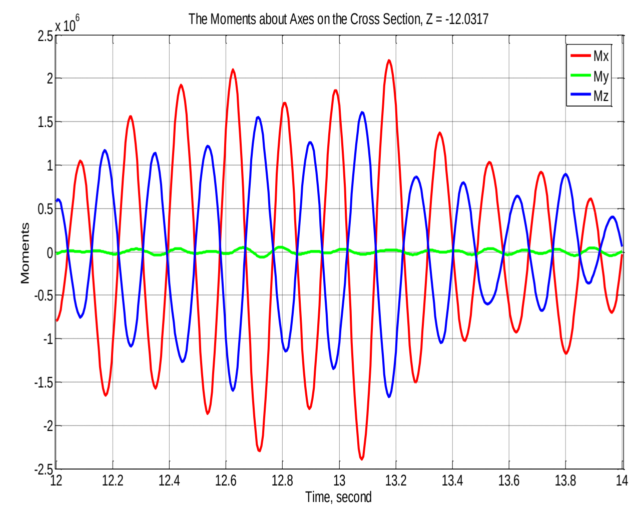
**Batch .pre input file of section cut for multiple frame data*
actm,0
**Replace Directory Path*
inp,demo8.pre,C:\DEMO_PROBLEMS\DEMO8
** Define structure component to be cut*
slice,1,0.0,0.0,-12.0317,0.0,0.0,1.0
** Cut the selected structure component using cutting plane*
** Calculate the parameters on it, and output to given file*
Calcsecthist,C:\DEMO_PROBLEMS\DEMO8\estr_frame_files.lst,1
,0.0,0.0,-
12.0317,0.0,0.0,1.0,1.0,0.0,0.0,1,.005,C:\DEMO_PROBLEMS\DE
MO8\frc_mmt_on_cut02.txt
** output cross sections for visualization with PREP (optional)*
cut2sub,1,1
actm,1
write,Slice.pre,C:\DEMO_PROBLEMS\DEMO8

Section-Cut CALCSECTHIST Command Results

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

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





Demo 8

2) ACS SASSI Version 3.0 Two-Step SSI Analysis Using ANSYS Interfacing (Options A and AA)

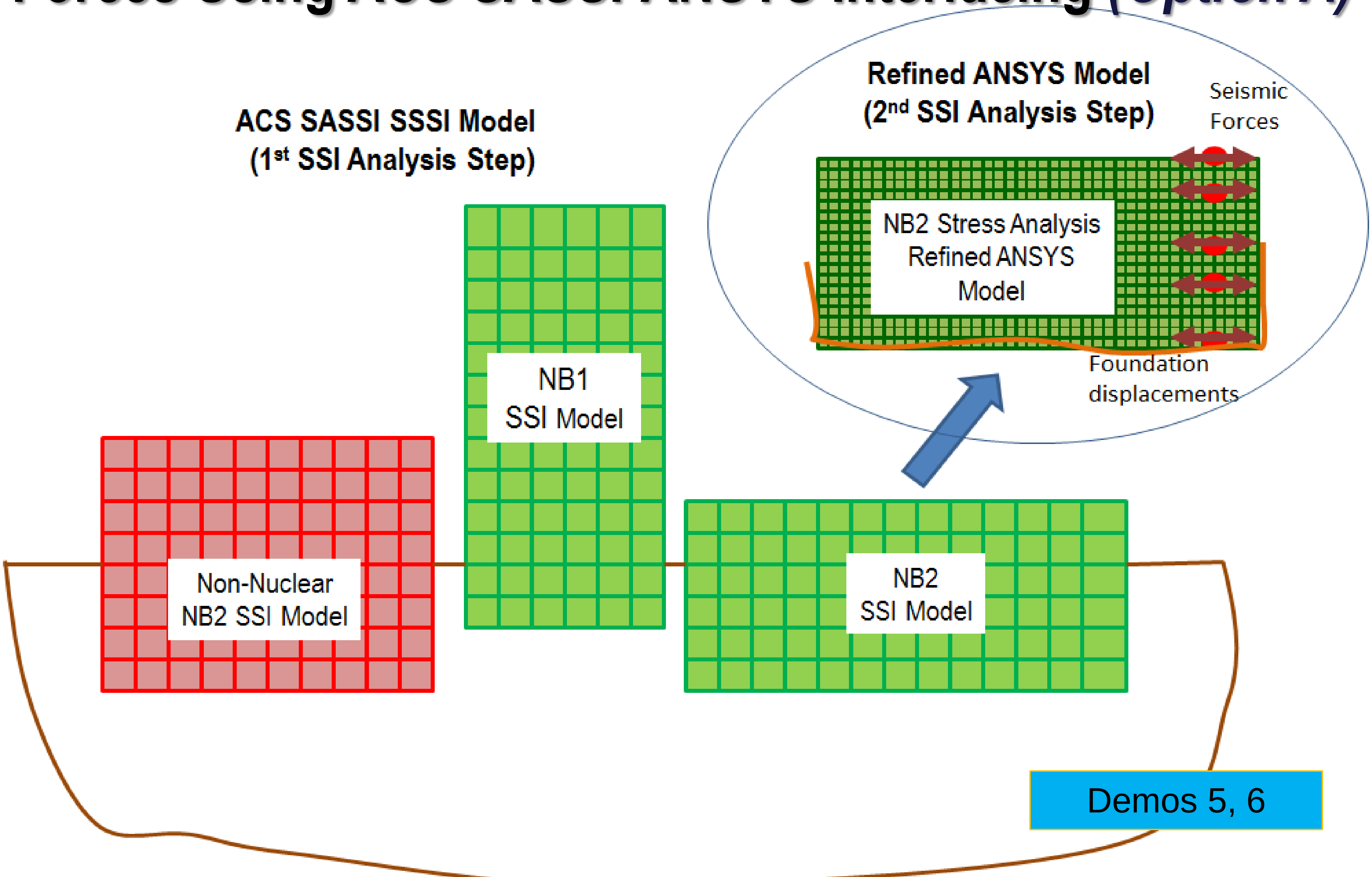
Two engineering analysis options that combines ACS SASSI with ANSYS:

i) ***One step SSI analysis*** *using ACS SASSI* for computing overall SSI responses motions, including ISRS, maximum accelerations and relative displacements within structure and structural forces and stresses ***(Option AA)***

ii) ***Two step SSI analysis*** *using ACS SASSI in 1st step and ANSYS in 2nd step* for computing forces and stresses in structure using a more refined structural FEA modeling via ANSYS. The *1st step is the overall SSI analysis* that is identical with the analysis above mentioned at item i). The 2nd step uses SSI responses as input BCs. *The 2nd step consists in an equivalent (quasi)static stress analysis using a much more refined FE mesh structural model* (via ANSYS static analysis). The 2nd step can be also a ANSYS transient analysis (no soil need to included in ANSYS model). ***(Option A)***

The ACS SASSI-ANSYS interface is extremely efficient, very easy to use.

Two-Step SSI/SSSI Approach for Computing Structural Forces Using ACS SASSI-ANSYS Interfacing *(Option A)*



Demos 5, 6

OPTION A: ACS SASSI-ANSYS Interface for SSI Analysis Using ANSYS Models (Demo 5 and 6)

ACS SASSI-ANSYS interfacing provides useful analysis capabilities:

For *structural stress analysis (Demo 5)*:

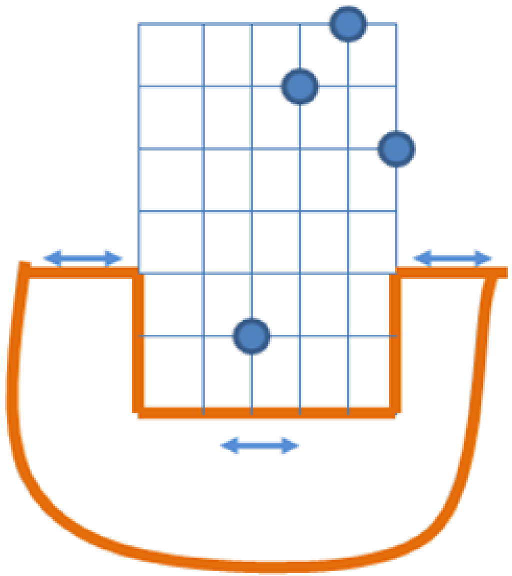
- *ANSYS Equivalent-Static Seismic SSI Analysis* Using Refined FE Models (including refined mesh, element types including local nonlinearities, nonlinear materials, contact elements, etc.)
- *ANSYS Dynamic Seismic SSI Analysis* Using More Refined FE Models (including refined mesh, element types including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

For *soil pressure computation (approximate) (Demo 6):*

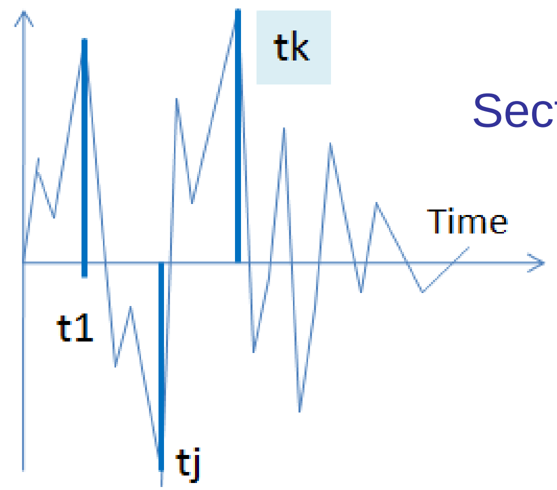
- *ANSYS Equivalent-Static Seismic Soil Pressure Computation* including Soil-Foundation Separation Effects

ACS SASSI Seismic SSI Analysis

Computing Structural Stress/Forces



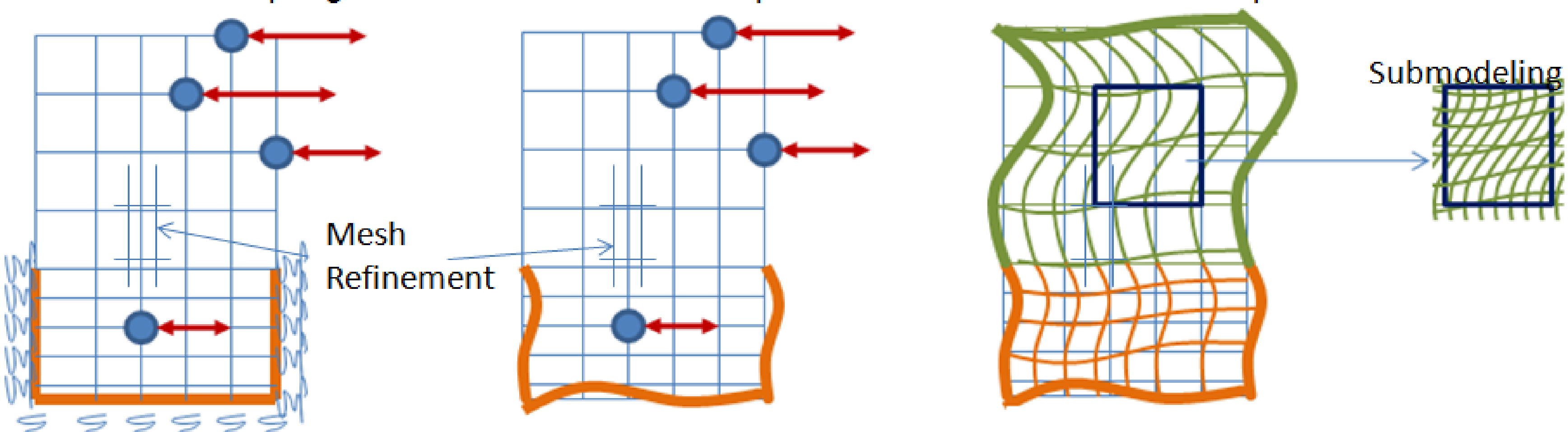
Selected Critical Time Steps for Maximum Stresses
To be Used for Equivalent Static Structural Analysis



Section-Cuts

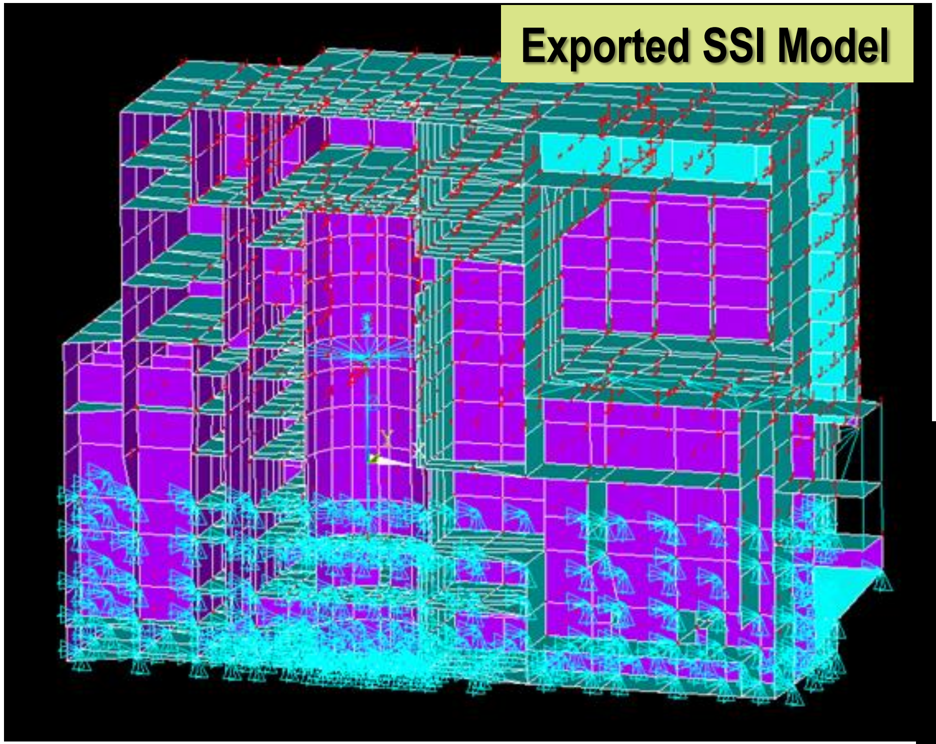
SSI Solution Time Frames As Equivalent Static Structural Loading at Critical Time Steps

EQS Forces + BC Springs — EQS Forces + BC Displacements — EQS Relative Displacements

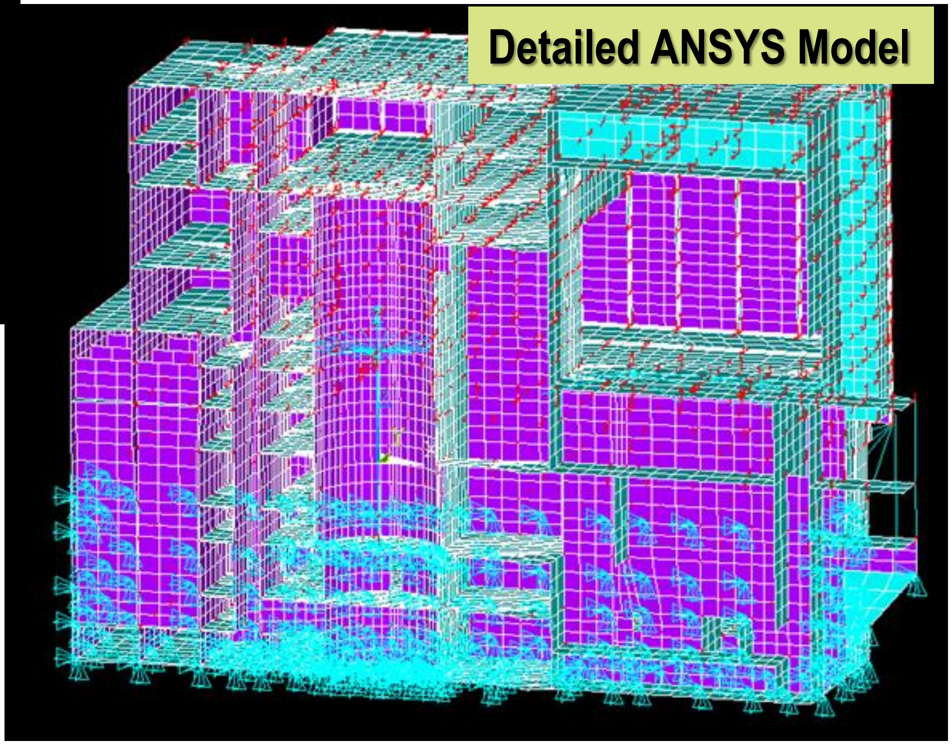


Variable Accuracy C-by-C — "Exact" — "Exact"

Option A for A Refined Seismic Stress Analysis (Demo 5)



ANSYS Structural Model Automatically Converted From ACS SASSI Using PREP Module

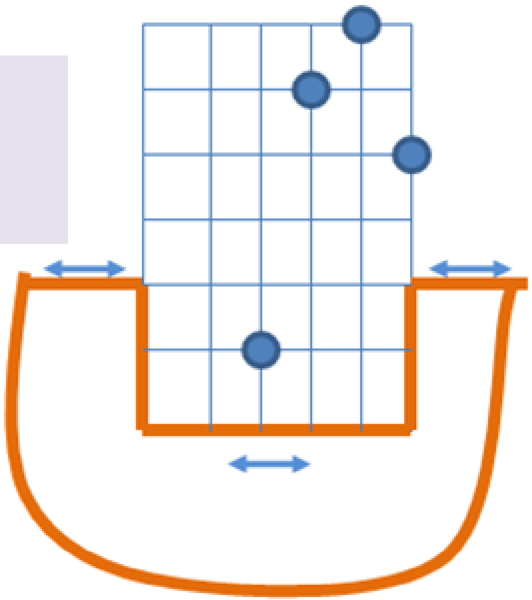


ANSYS Refined Structural Model Using EREFINE command or ANSYS GUI (rank 1-6)

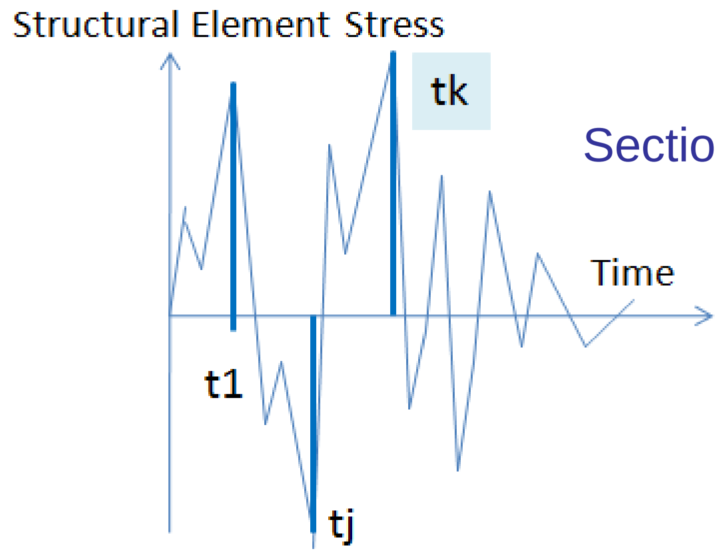
Demo 5

ACS SASSI Seismic SSI Analysis

Computing Seismic Soil Pressures



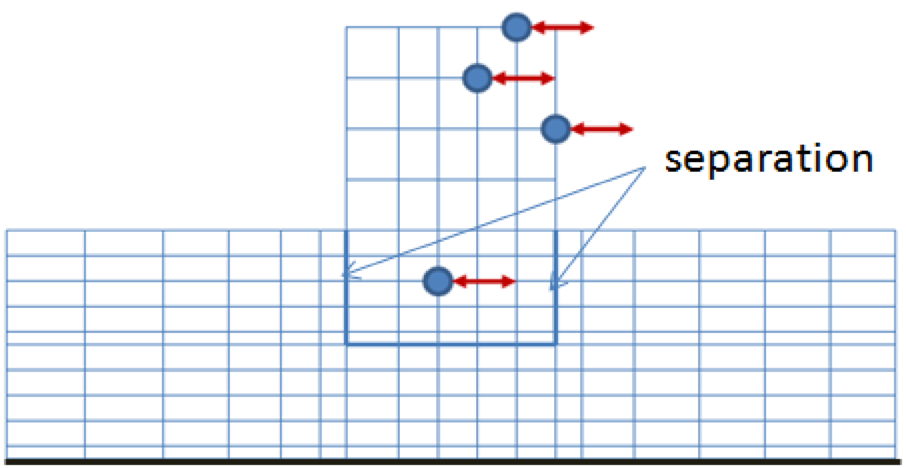
Selected Critical Time Steps for Maximum Stresses
To be Used for Equivalent Static Structural Analysis



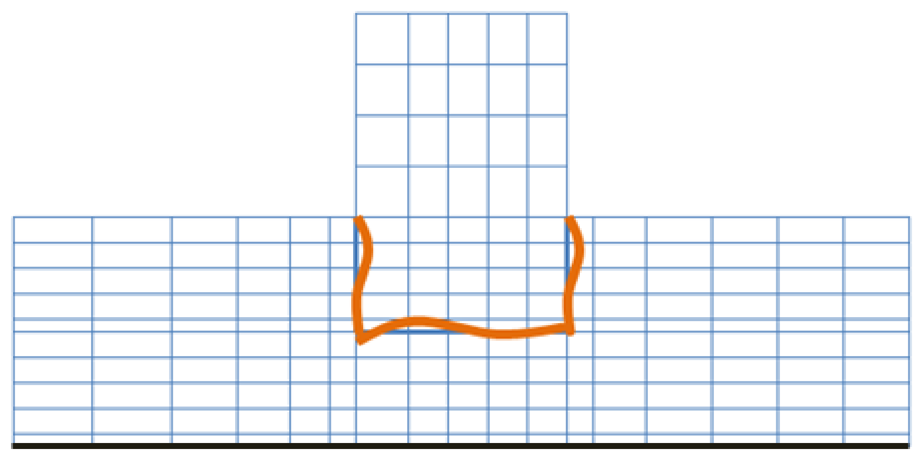
Section-Cuts

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

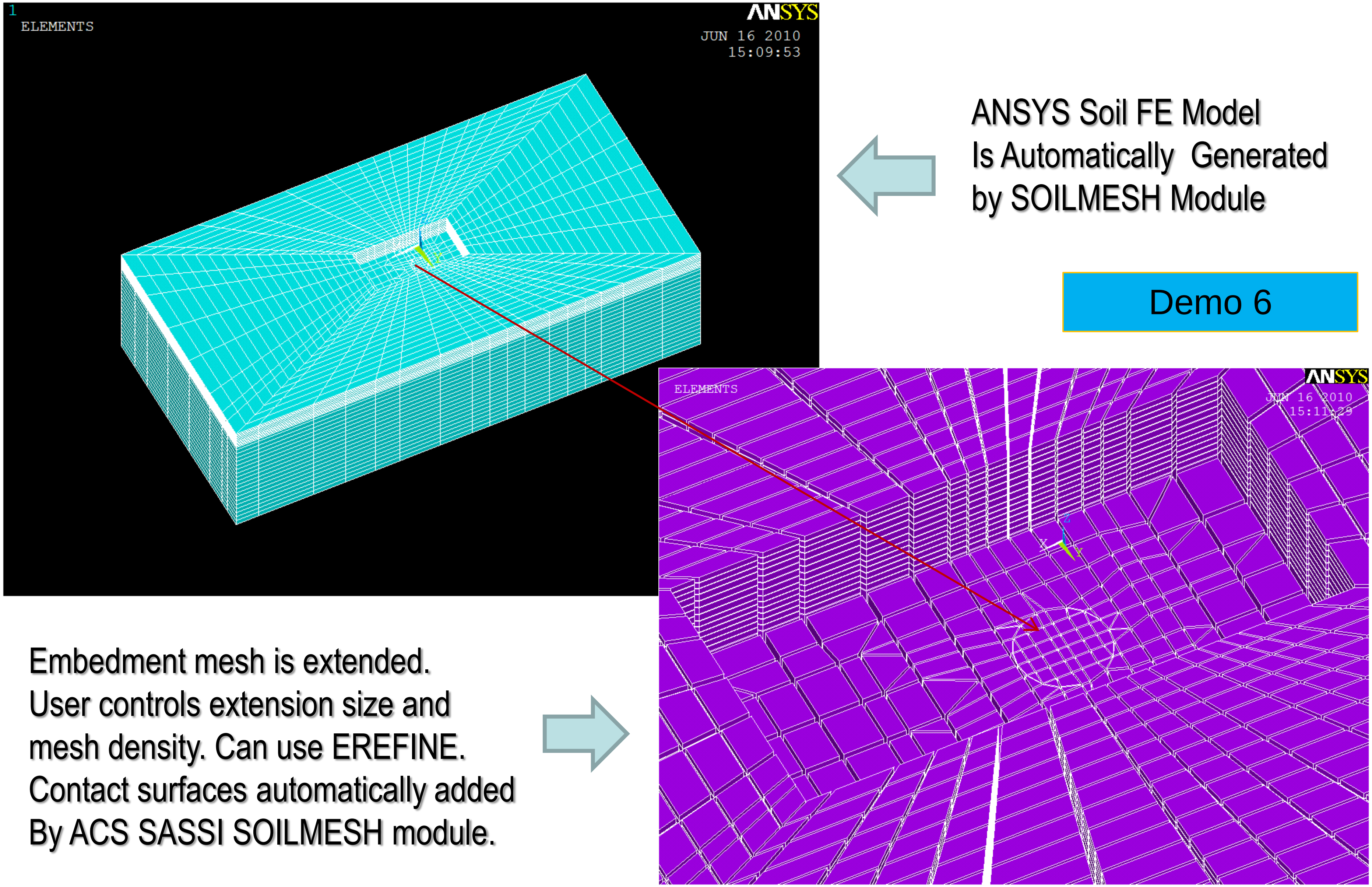
EQS Forces – Linear & Nonlinear



EQS Relative Displacements – Linear (Welded)



Option A for Seismic Soil Pressure Analysis (Demo 6)

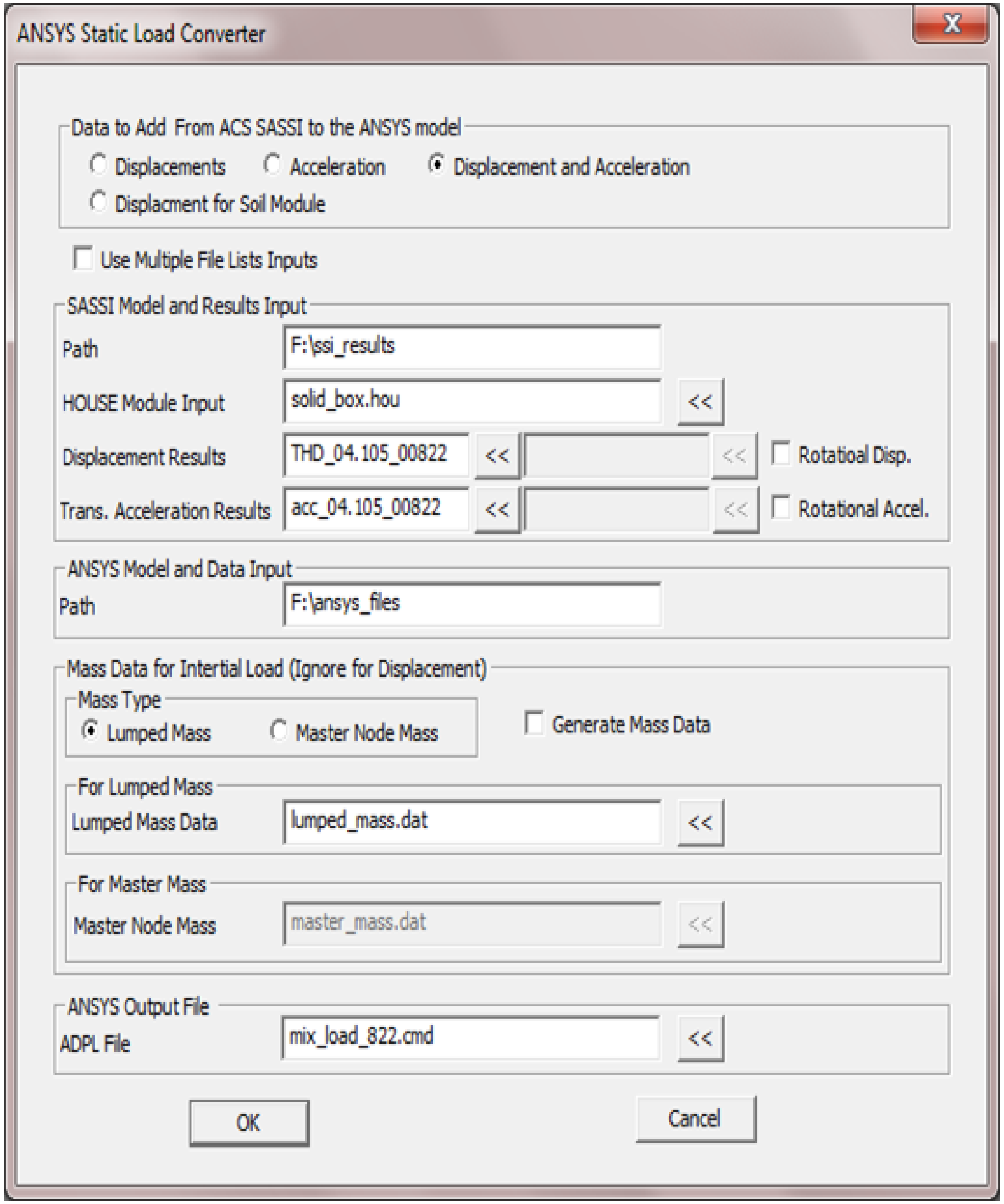


ANSYS Soil FE Model
Is Automatically Generated
by SOILMESH Module

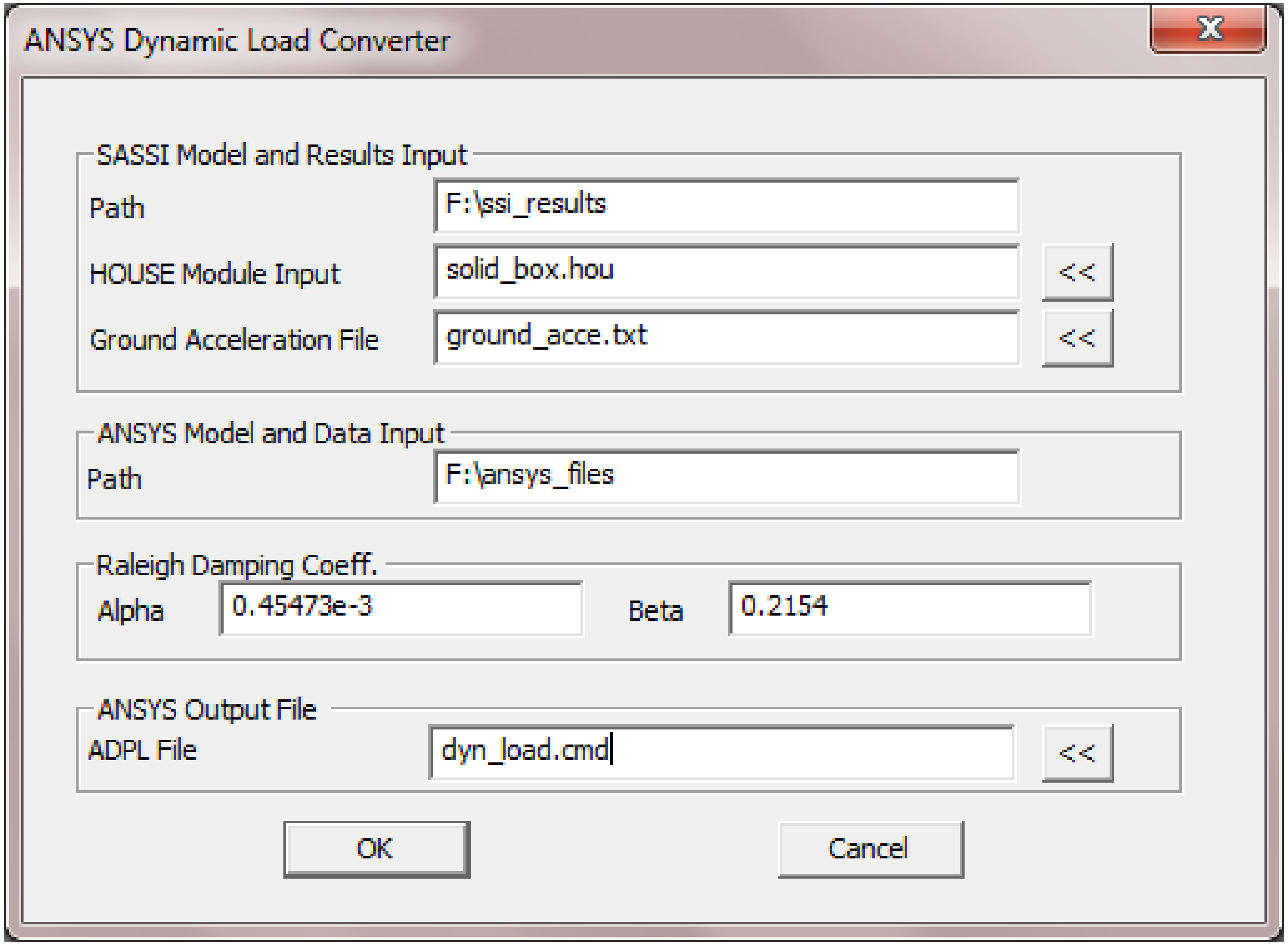
Demo 6

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

MAIN/LOADGEN Module for Equivalent Static Analysis



MAIN/LOADGEN Module GUI for Dynamics



Demo 5

OPTION AA: ACS SASSI-ANSYS Interface for SSI Analysis Using ANSYS Models (Demo 7)

OPTION AA uses directly ANSYS structural model for SSI analysis

Sequence of Steps:

1) Develop ANSYS structural FEA model with no modeling restrictions (any FE type, CP, CE, rigid links)
2) If embedded, develop also the ANSYS excavated soil FEA model
3) Using an ANSYS ADPL macro generate matrices K, M, C
4) Using ACS SASSI SUBMODELER GUI read ANSYS model .cdb for structure and excavation to convert the ANSYS model geometry configuration to ACS SASSI for post-processing
5) Merge Structure and Excavation models in SUBMODELER. Add interaction nodes automatically. And AFWRITE the SSI model to produce HOUSE input.

Demo 7

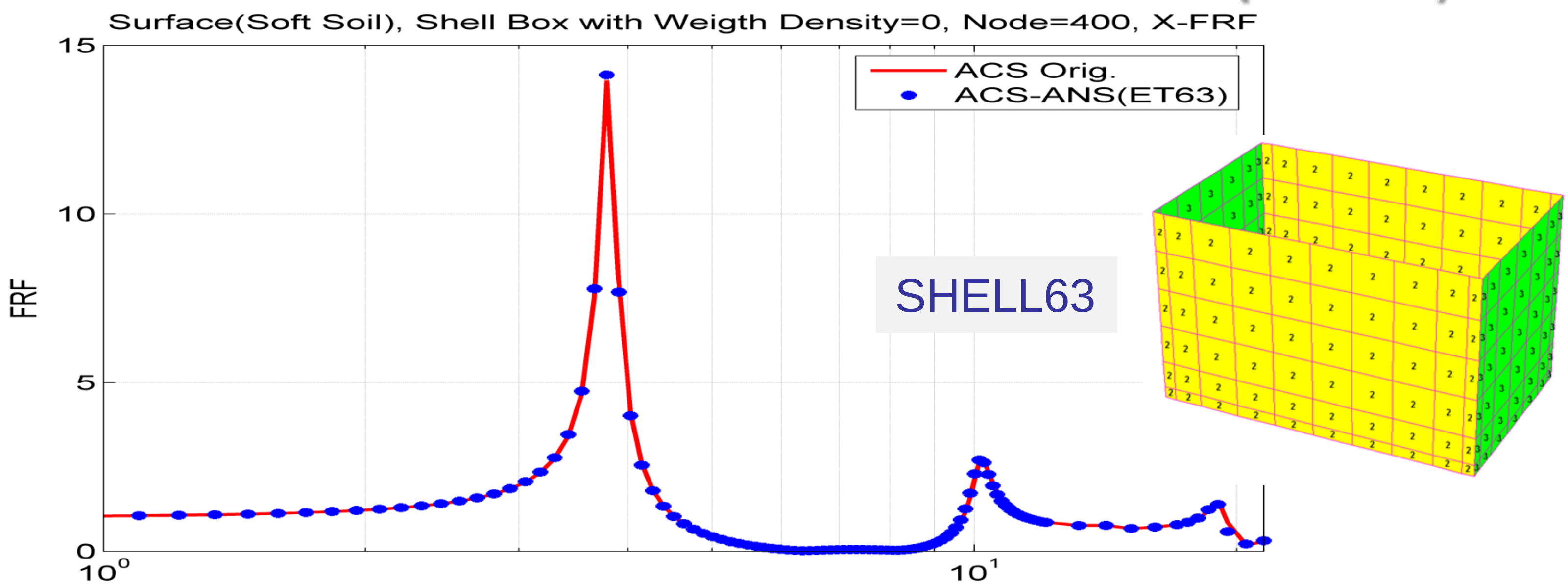
6) Finally, run HOUSEFSA module that reads and merge ANSYS K, M and C matrices and produce the complex K matrix and mixed M matrix for SSI analysis (COOSK and COOSM files)

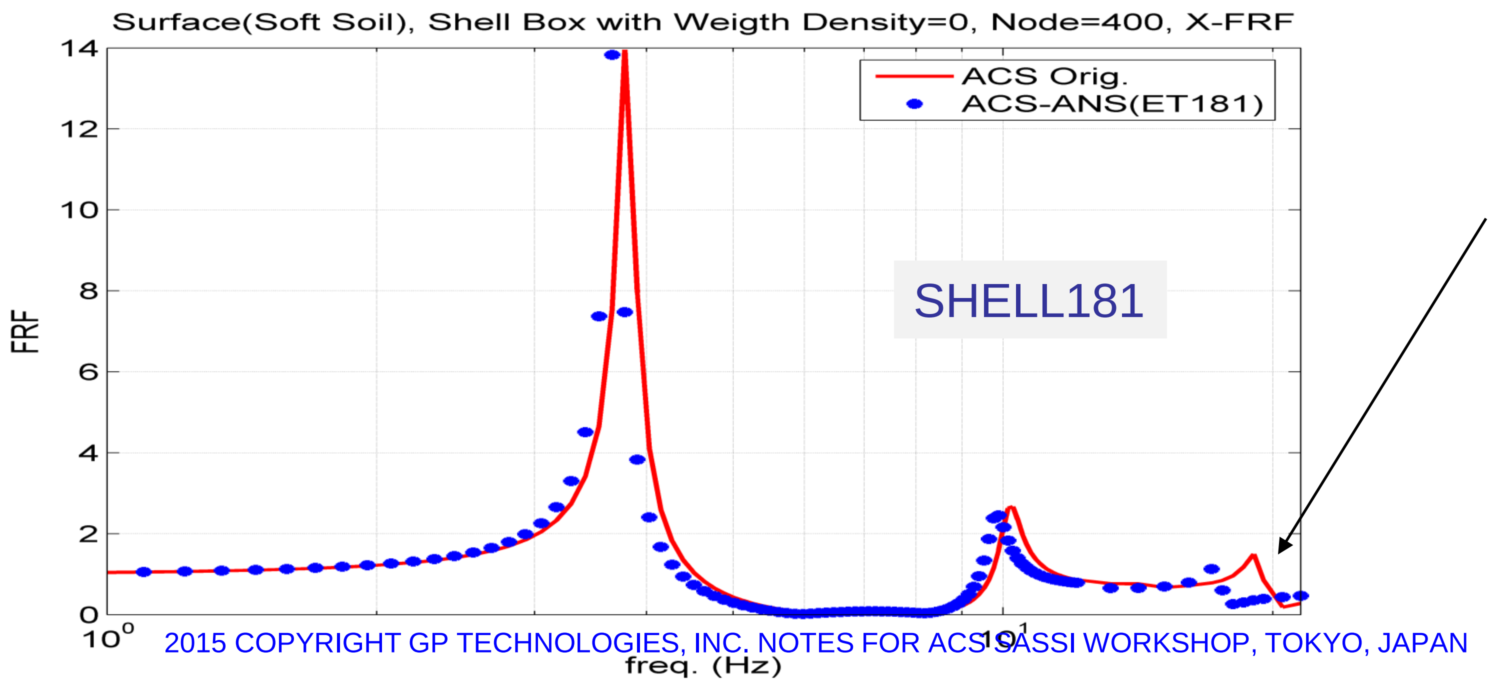
7) Perform SSI analysis with the same ANALYSFSA module.

ANSYS FE Types Acceptable for Option AA

- SOLID element types: SOLID45 and SOLID185;
- SHELL element types: SHELL63 and SHELL181;
- BEAM element types: BEAM44 and BEAM188;
- PIPE element types: PIPE288;
- COMBIN element types: COMBIN14;
- Couple nodes (CP command) and Constraint equations (CE command)
- Multipoint constraint element types: MPC184 Rigid Link and/or Rigid Beam
- Fluid element types: FLUID80

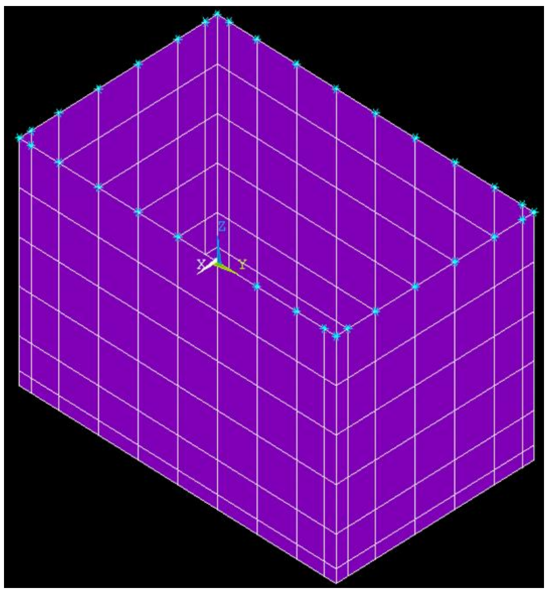
Concrete Pool ANSYS SHELL Model (Pb 45)



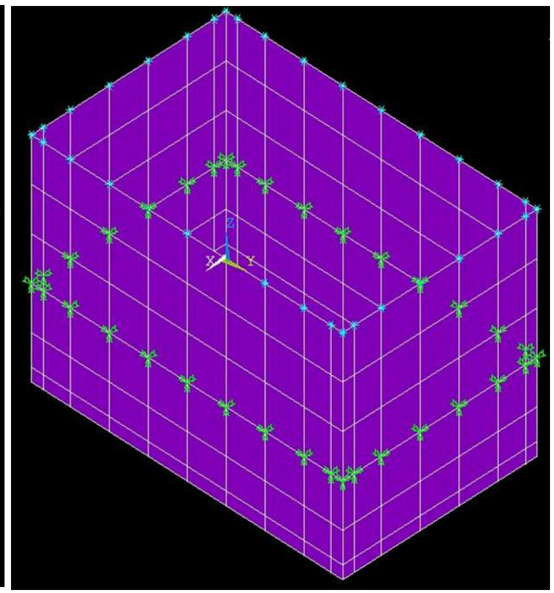


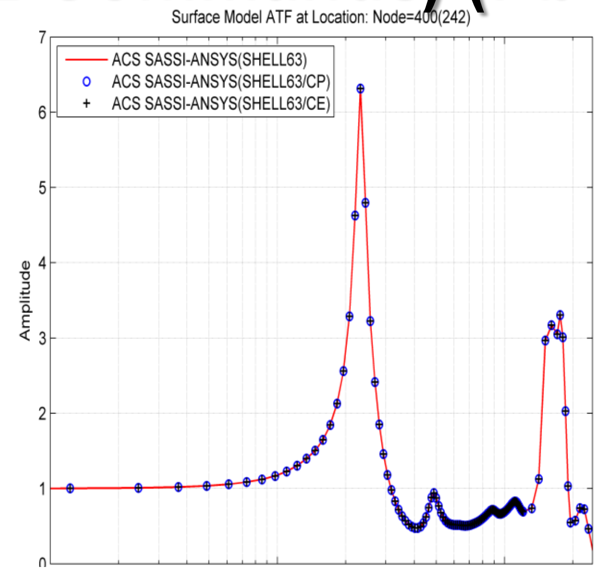
Using ANSYS Couple Nodes (CP Commands) and ANSYS Constrained Equations (CE Commands) (Pb 46)

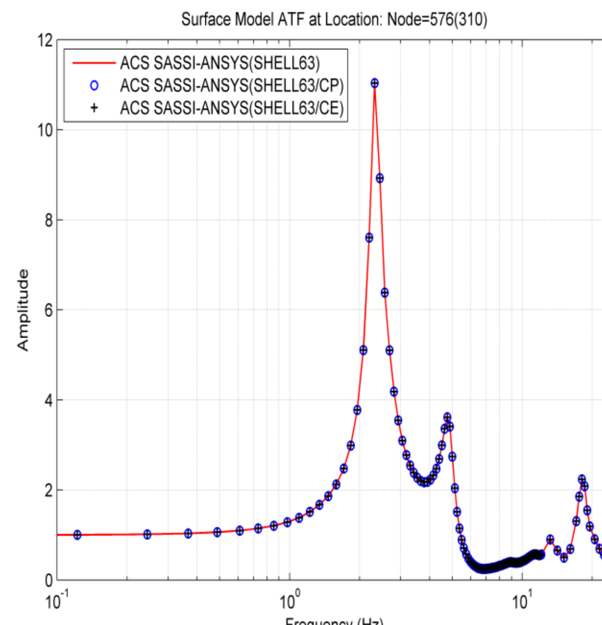
Continue Model



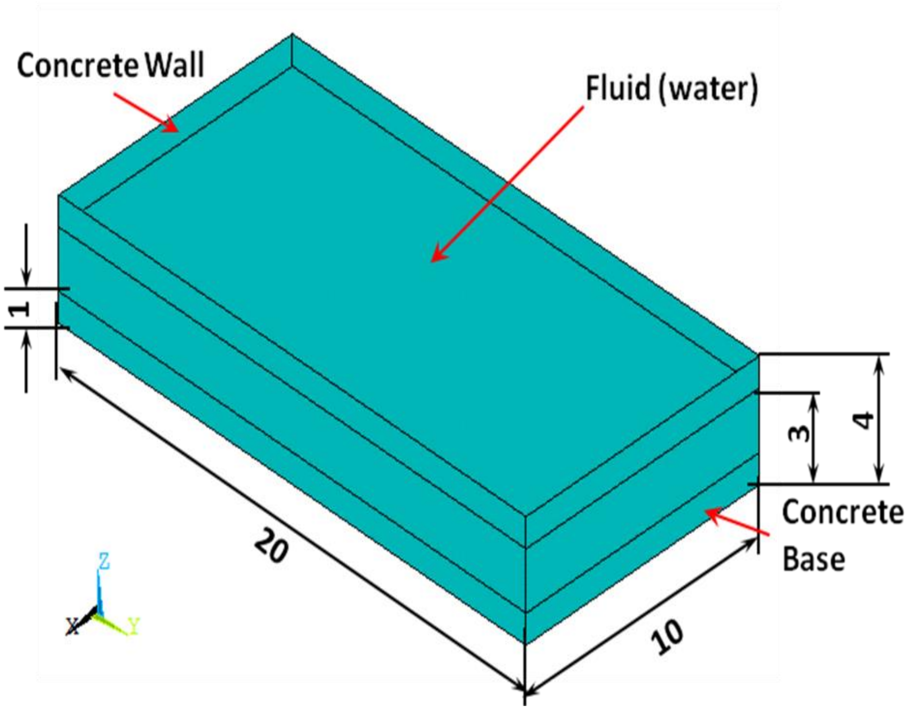
Cut Model

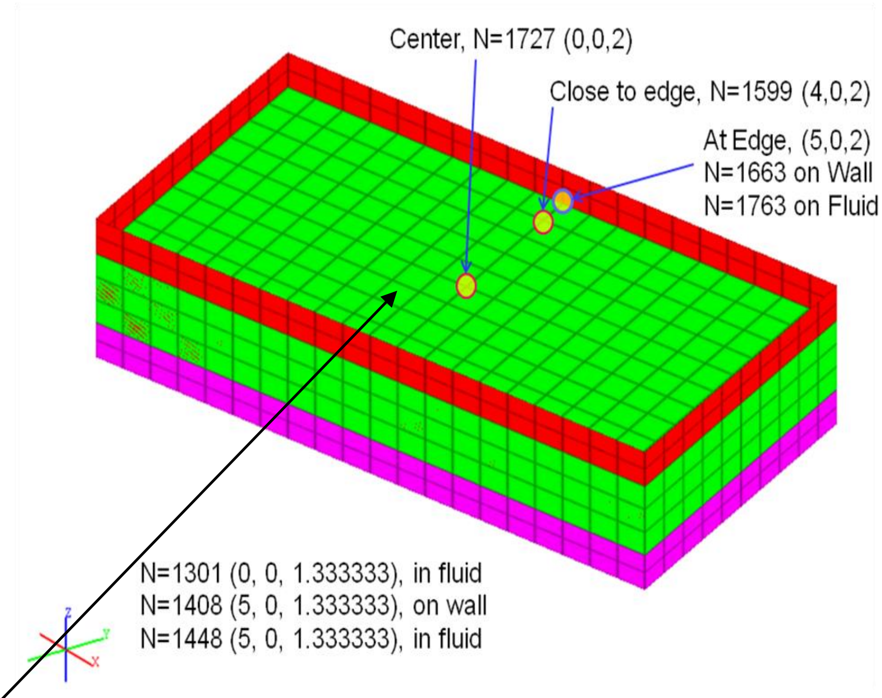






ANSYS FLUID80 Elements for A Fluid-Structure Interaction for A Concrete Pool Via Option AA (Pb 48)





Fluid elements (as solids) can be automatically generated using SUBMODELER FILLPOOL command

FILLPOOL, [Stiff],[Sensitivity],[EmptyLevels],[ShellArea],[Offset],[stiff2]

The pool is filled solid elements to fill the volume. The interface of the pool wall/water are connected by a set of springs with the stiffness of these springs determined by the user. The pool FE model should only contain the walls and floor of a single pool to be filled. The walls and floor must be made of either shells or solids.

<Stiff> - Stiffness of the water wall spring interface parallel to the normal (Default 106)

<Sensitivity> - allowable tolerance variation in Z coordinate on the same Z-level (Default 0)

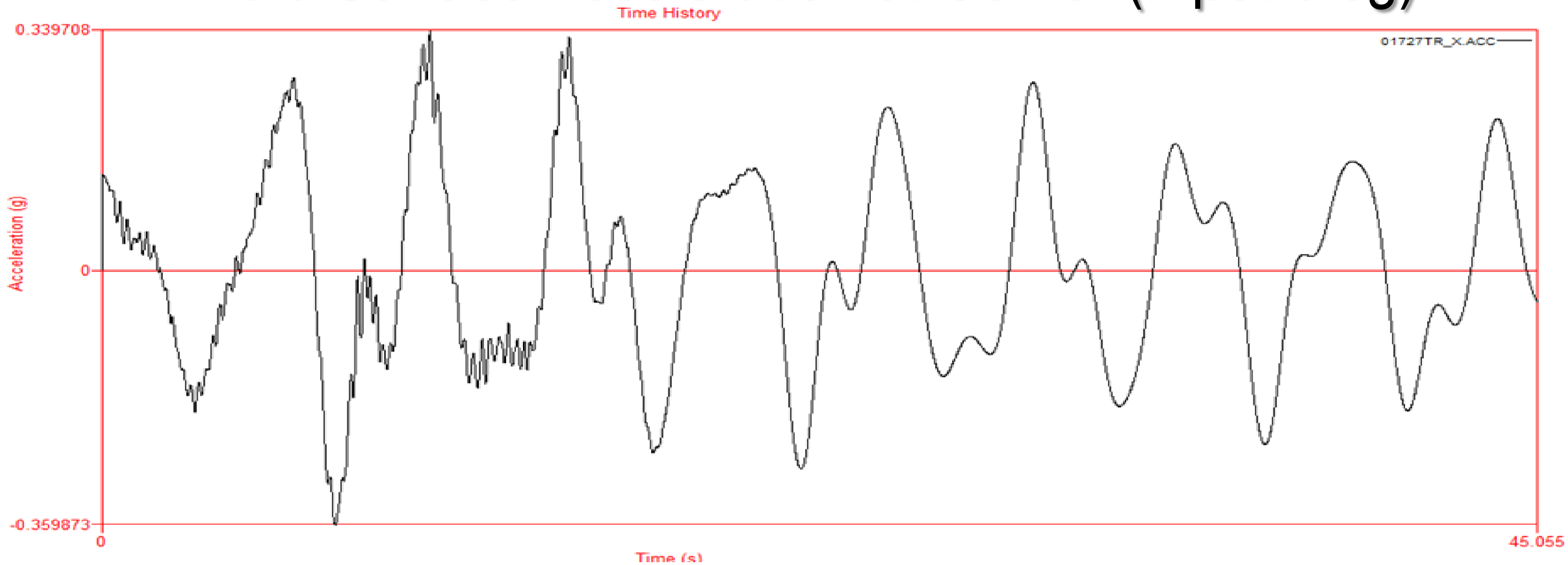
<EmptyLevels> - Number of Z-levels, starting at the highest level, not to be filled with water (Default is 0, that is pool is entirely filled with water)

<ShellArea> - This parameter should be skipped by leaving blank its field.

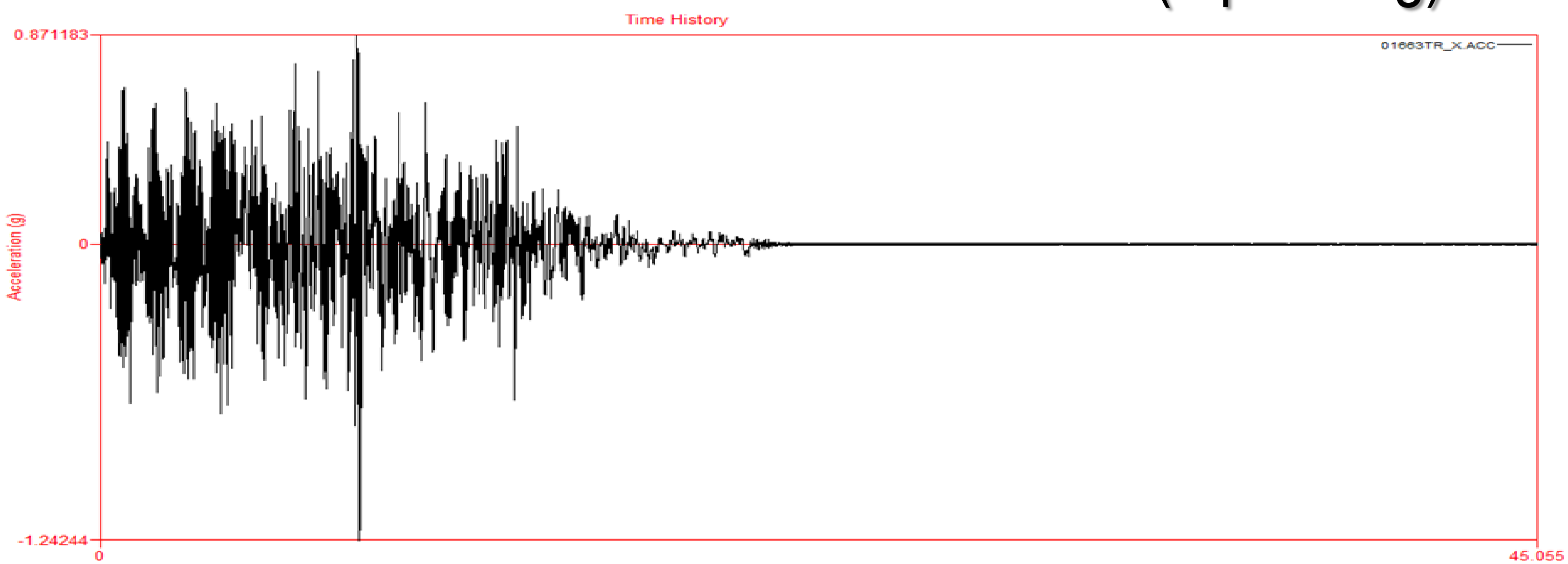
<offset> – the user-defined starting node number for numbering of the pool elements. This number should be greater than or equal to last node number from the original FE model if the user intends to import the water and spring group back into the original model. If the offset is less than or equal to 0 the pool wall node number maximum will be used (Default -1) an error will occur for any positive number that is less than the pool wall node number maximum.

<stiff2> - Stiffness of wall spring interface in tangential direction to the wall (Default 0)

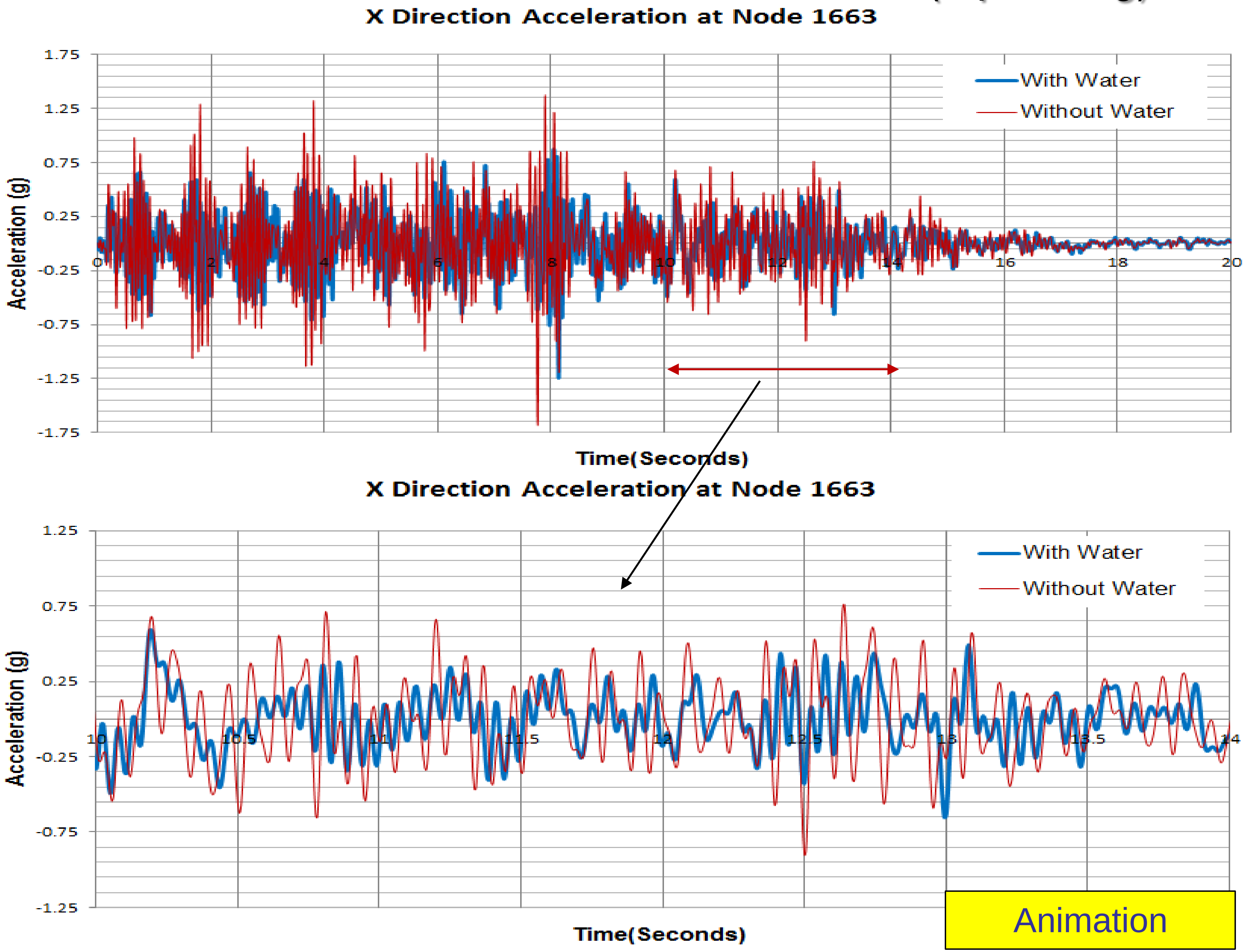
Fluid Surface Acceleration at Center (Input 0.3g)



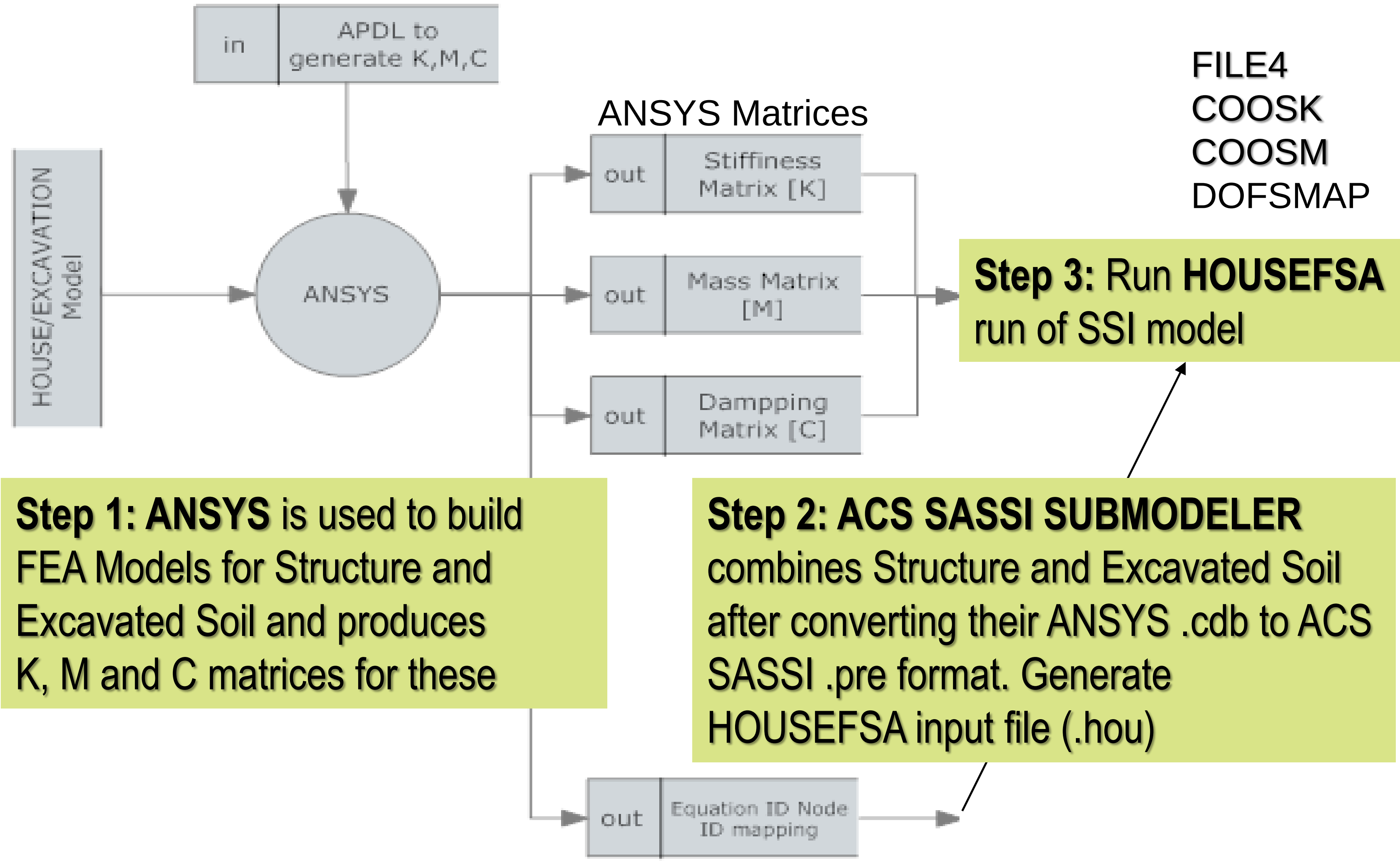
Wall Transverse Acceleration at Center (Input 0.3g)



Wall Transverse Acceleration at Center (Input 0.3g)



Steps for Running SSI analysis Using ANSYS Model



Basic Computational Steps in Option AA *(Demo 7)*

To use the Option AA capabilities, the following steps must be performed

1) Generate the ANSYS model mass, stiffness and damping matrices *using ANSYS with the gen_kmc.mac APDL macro inputfile.* It includes the execution of the installed *SSI2ANSYS.exe* program to generate the matrix and node-equation mapping files for ANSYS model .

2) Create the HOUSEFSA module input file (.hou file) using *ACS SASSI SUBMODELER module.*

   2.1 *Convert ANSYS structural model* (struct.cdb file), and *the excavation volume* model (excv.cdb).

   2.2 For embedded models, *merge structure and excavation models using MERGESOIL command* to get the overall SSI model

   2.3 After completing SSI model, *AFWRITE to get the .hou input file*

3) *Run ACS SASSI HOUSEFSA module with the input files that includes* the .hou input file (by SUBMODELER) and the matrix and the node-equation mapping files (produced by ANSYS and SSI2ANSYS in Step 1)

AA Step 1: Running ANSYS with gen_kmc. mac

FOR STRUCTURE ANSYS Model:

At the ANSYS command line input
gen_kmc,'.',0,'.'

APDL Macro produces the following files:
coosk_r, cooski_r, coosm_r, coosmi_r, coosc_r, coosci_r, and Node2Equ_Stru.map

FOR EXCAVATION ANSYS Model:

At the ANSYS command line input
gen_kmc,'.',1,'.'

APDL Macro produces the following files:
cooek_r, cooeki_r, cooem_r, cooemi_r, cooec_r, cooeci_r, and Node2Equ_Excv.map

Using ANSYS with gen_kmc.mac APDL Macro for Extracting Matrix and Mapping Structure and Excavation

cooec_r	7/25/2014 1:27 PM	File
cooeci_r	7/25/2014 1:27 PM	File
cooek_r	7/25/2014 1:27 PM	File
cooeki_r	7/25/2014 1:27 PM	File
cooem_r		
cooemi_r		
coosc_r	7/25/2014 11:06 AM	File
coosci_r	7/25/2014 11:06 AM	File
coosk_r	7/25/2014 11:06 AM	File
cooski_r	7/25/2014 11:06 AM	File
coosm_r	7/25/2014 11:06 AM	File
coosmi_r	7/25/2014 11:06 AM	File
Node2Equ_Excv.map		
Node2Equ_Stru.map		
sldbox45_excv.cdb		
sldbox45_struct.cdb		

ANSYS Model K, C and M Matrix Files (APDL macro

ANSYS Model Mapping Files (APDL macro)

ANSYS Model .cdb Files (ANSYS)

AA Step 2) Using ACS SASSI SUBMODELER for Converting and Merging ANSYS Models for Structure and Excavation

1) Launch SUBMODELER
3) Convert the .cdb file to a .pre file by using the SUBMODELER ANSYS Converter, by selecting Model Converters ANSYS.cdb
4) In the Converter Input box, locate the Structure.cdb file
5) In the Converter Output box, type structure.pre filename including path
6) In the Converter model box, save .pre in model number "1", and enter "32.2" in acceleration of gravity box [ft/s2 or m/s2] and close Converter
In the SUBMODELER command line,
7) Type "actm,1" to switch to model 1
8) Define ground elevation "GroundElev, Z-coordinate" and "Etypegen,1"
9) Switch to model 2 by typing "actm,2" in the command line
11) Convert the excavation volume .cdb file to a .pre file by using the SUBMODELER ANSYS Converter, by selecting Model Converters ANSYS.cdb
12) In the Converter Input box, locate the Excavation.cdb file
13) In the Converter Output box, type structure.pre filename including path

14) In the Converter model box, save .pre in model number "1", and enter "32.2" in acceleration of gravity box [ft/s2 or m/s2] and close Converter

In the SUBMODELER command line,

15) Type "actm,2" to switch to model 2

16) Define the ground elevation for this model as 20 [ft] by typing the command "GroundElev,Z-level" in the SUBMODELER command line.

17) Assign the excavation volume element type by typing "etypegen,2"

18) Activate model 3 for the combined SSI model by typing "actm,3" in the command line

19) Create SSI model by using the MergeSoil command as follows "Mergesoil,1,2,1,,,,full_path_modelname_excv.map"

This will produce the SSI model in model 3 and the modelname_excv.map file

Mapping filename is "model_name_excv.map"

20) Use the Intgen command to generate interaction nodes; "IntGen,1" for flexible volume

21) Use Mdl command to set the paths for the AFWRITE command files

Mdl, modelname, path

22) Type AFWRITE to produce the .hou file for the HOUSEFSA module run

SUBMODELER Code Example to Merge an ANSYS Structure and Excavation Models for SSI Analysis in Option AA.

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

Actm,1
Convert,ansys,struct.cdb,32.2
Etypegen, 1

Actm,2
Convert, ansys,Soil.cdb,32.2
Etypegen,2

** Create the ACS SASSI SSI model by combining Models 1 and 2 in Model 3*
Actm,3
MergeSoil,1,2,1,,,,mappingfile_excv.map
Groundelev, 0
Intgen, 1

AA Step 3: Run HOUSEFSA and SSI Analysis in Batch

```
@echo off
set bpath=C:\ACSV300\EXEB
set prb_name=modelname
echo %prb_name% > site.inp
echo %prb_name%.sit >> site.inp
echo %prb_name%_sit.out >> site.inp
echo %prb_name% > point.inp
echo %prb_name%.poi >> point.inp
echo %prb_name%_poi.out >> point.inp
echo %prb_name% > house.inp
echo %prb_name%.hou >> house.inp
echo %prb_name%_hou.out >> house.inp
echo %prb_name% > analys.inp
echo %prb_name%.anl >> analys.inp
echo %prb_name%_ana.out >> analys.inp
%bpath%\SITEB.exe < Site.inp
%bpath%\POINT3B.exe < Point.inp
%bpath%\HOUSEFSAB.exe < House.inp
%bpath%\ANALYSFSAB.exe < Analys.inp
```

Demo 7

3.0) Nonlinear SSI for Reinforced Concrete Shearwall Structures (Option NON, Demo 9, July 15)

The new nonlinear SSI approach can be used to perform *fast and accurate* nonlinear SSI analyses including sophisticated nonlinear hysteretic models at a small fraction of the runtime of a time domain nonlinear SSI analysis…

The nonlinear SSI in complex frequency *more robust and free-of numerical noise* in comparison nonlinear SSI in time domain.

Planned Release Schedule:

Release 1: Initially (by July 15) the new nonlinear SSI approach will be limited to low-rise concrete shearwall structures with plane walls (no curved walls).

Release 2: Next including curved walls, beam and columns. Also, we include nonlinear springs for simulating sliding and soil separation….(by Dec 2015)

ASCE 04-2015 Criteria for Concrete Cracking Effects

The ASCE 04-2014 standard recommends: "One method for determining best estimates of the stiffness of concrete shear walls for linear dynamic analysis of low aspect ratio reinforced concrete shear walls is to check the stress state in the wall as follows. 1) Develop an analytical model that is representative of the structure. 2) Analyze the structure using uncracked stiffness and damping properties for in-plane bending and shear of walls (i.e. 1.0*GA and 4%). 3) Post-process results and check the stress state in the walls to determine if they have cracked by comparing the average wall cross section shear stress to $\sqrt{3 f_c^{'}}$ and the flexural stress state to $\sqrt{7.5 f_c^{'}}$. If the stresses in the wall exceed these values the concrete has cracked significantly. If it is determined that the walls in the analysis have experienced extensive cracking, change the stiffness and damping values for those walls to cracked properties (i.e. 0.5GA and 7%) and use the uncracked properties for the walls that do not exceed that threshold and re-run the analysis. After running this second analysis that includes cracked properties for some or all walls, it is not necessary to recheck the wall stress state."

Thus, at least two iterative SSI analyses are required to establish the final cracked concrete pattern within the structure. Only after the cracked concrete pattern within structure is established, the cracked structure can be used for the SSI analysis production runs.

Nonlinear Structural Analysis Using Hybrid Approach. Applicable to Design Level and Beyond Design Level

The nonlinear SSI analysis is performed using an innovative, accurate and efficient frequency-time hybrid iterative method. *The hybrid approach uses in frequency-domain local equivalent linearized hysteretic models for the concrete shearwall panels based on their local nonlinear hysteretic behavior in time-domain*.

The runtime of a nonlinear SSI analysis is only about 2-3 times the runtime of linear SSI analysis.

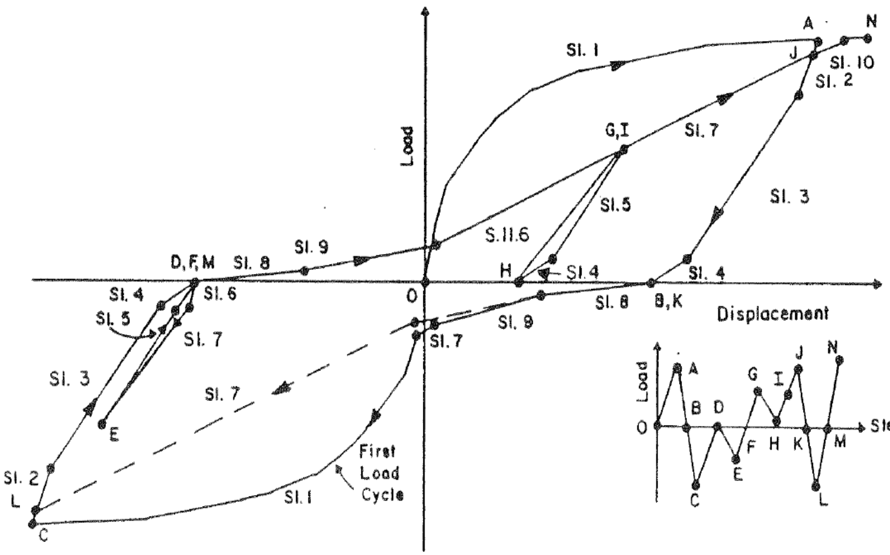
The ductilities and inelastic absorbtion factors for each panel are computed using nonlinear time domain solution after the last iteration.

Nonlinear Structural SSI Analysis Uses An Iterative Local Equivalent Linearization Procedure

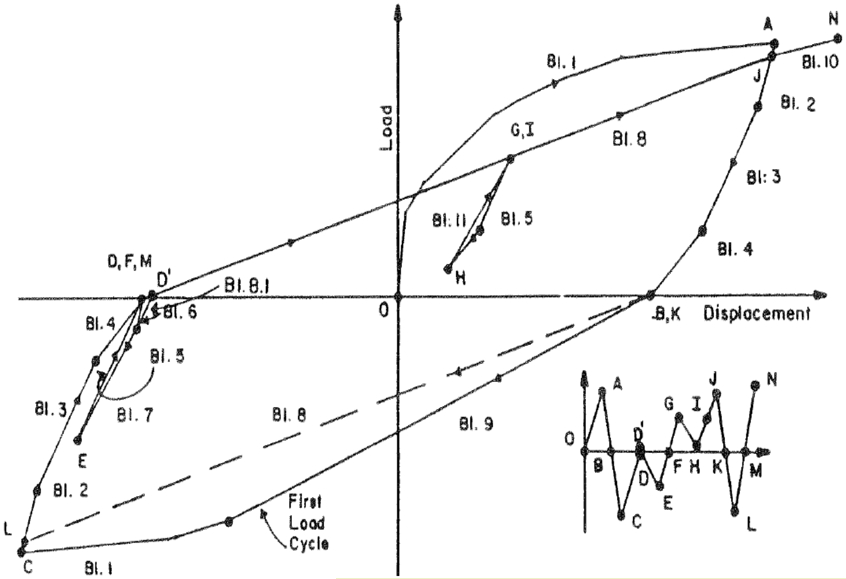
Nonlinear SSI Analysis computational steps:

- For the initial iteration, perform a linear SSI analysis using the elastic properties for the selected shearwall panels
- Compute concrete shearwall panel behavior in time domain that is used to calibrate the local panel hysteretic models associated to each nonlinear shearwall panel in complex frequency
- Perform a new SSI analysis iteration using a fast SSI reanalysis (restart analysis) in the complex frequency domain using the hysteretic models computed in Step 2 for all selected panels
- Check convergence of the nonlinear SSI response after new SSI iteration to stop; otherwise continue with a new iteration

Chen-Mertz Hysteretic Model for Low-Rise Shearwalls

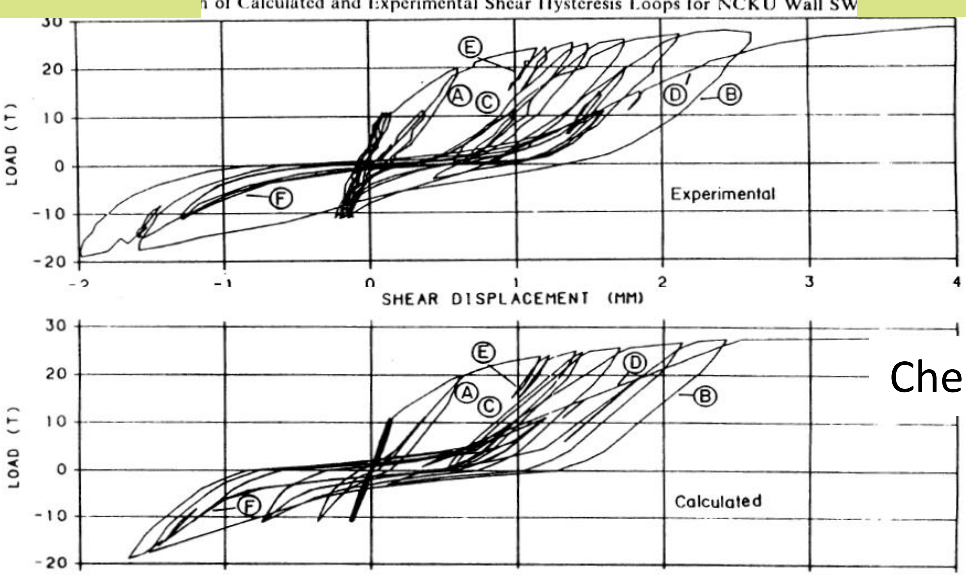


Wall In-Plane Shear



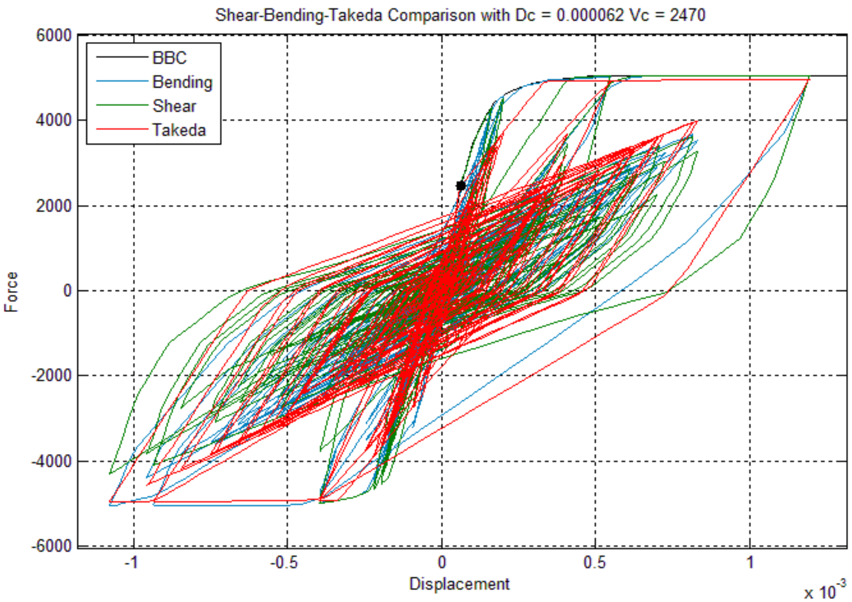
Wall In-Plane Bending

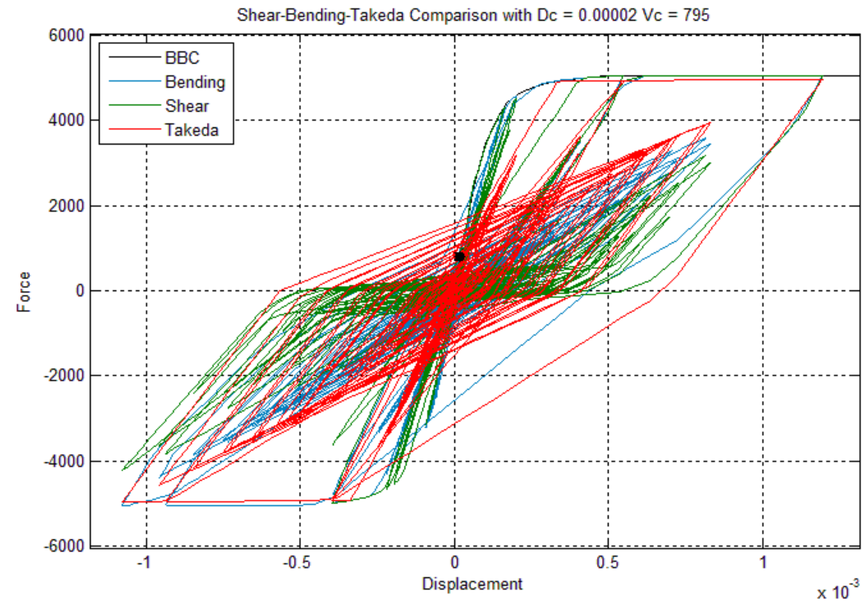
n of Calculated and Experimental Shear Hysteresis Loops for NCKU Wall SW

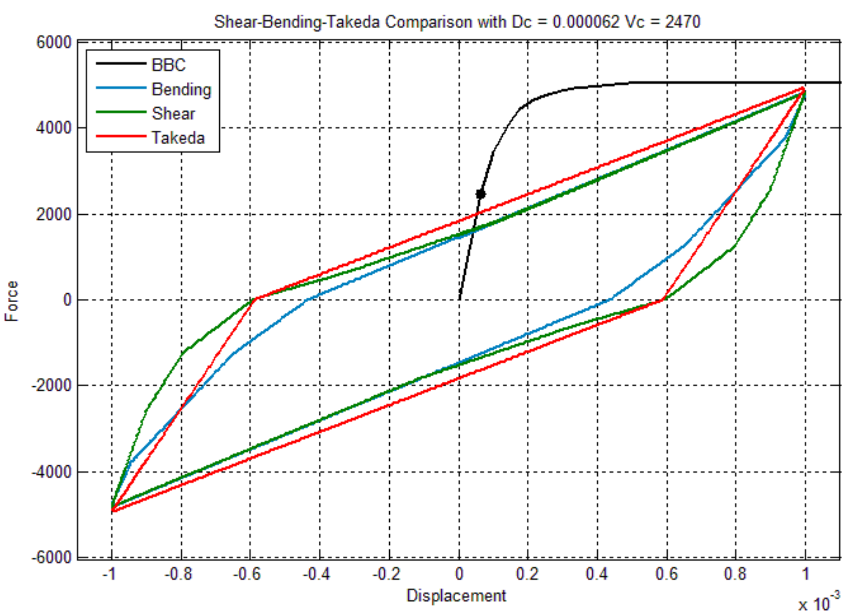


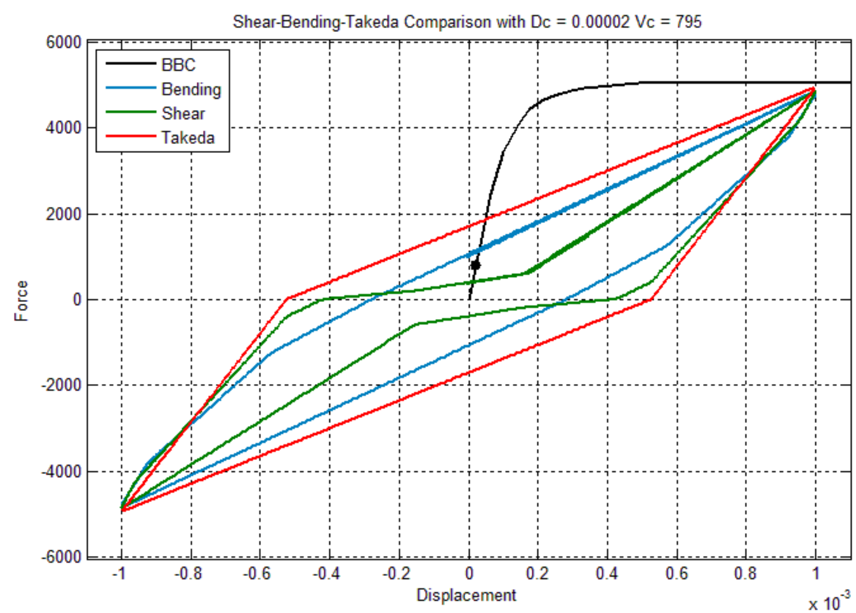
Cheng and Mertz, 1989

Reinforced Concrete Shearwall Bending and Shear Hysteretic Models: Chen-Mertz (CMB, CMS) and Takeda









Preliminary Preparation of Nonlinear Structure Model Using New SUBMODELER Commands

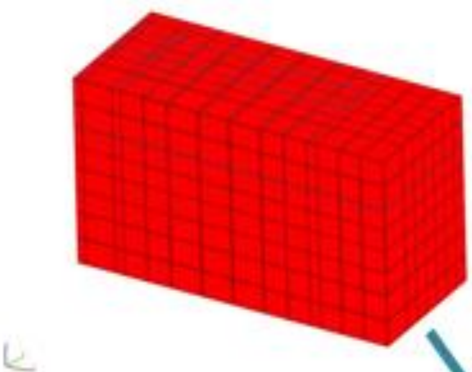
Step 1
Load elastic model into sub modeler

Step 2
Split model into shear walls using **Wallflr**, **Panelize**, and **Merge/Splitgroup** commands

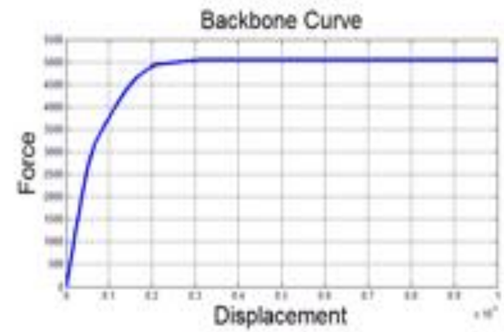
Step 3
Input shear wall options and material backbone curves

Step 4
Write new HOUSE and EQL module file using ***afwr*** command

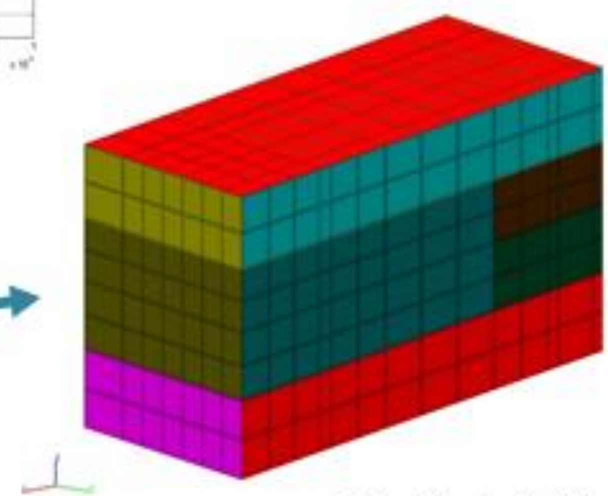
Step 5
Run equivalent linear analysis batch file



Original Elastic Model

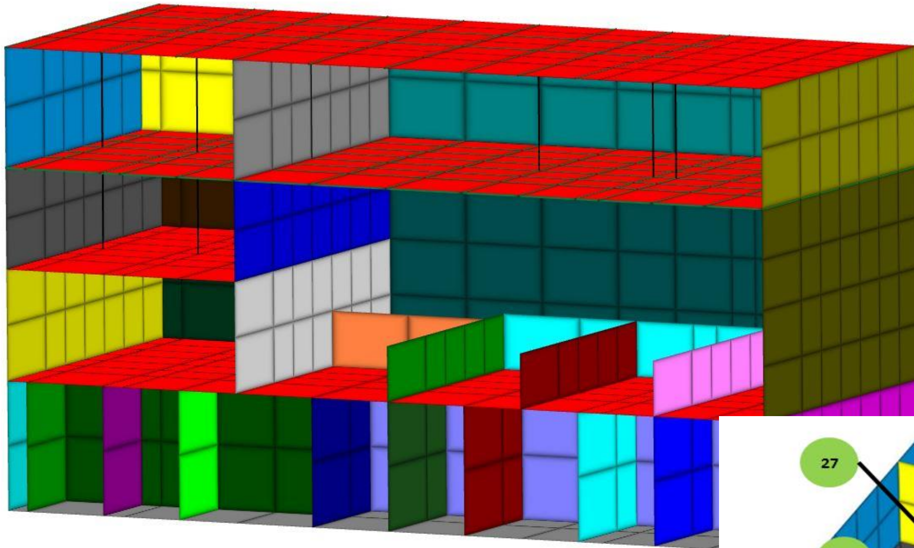


Submodeler
Panelize
Wallflr
Mergegroup
Splitgroup



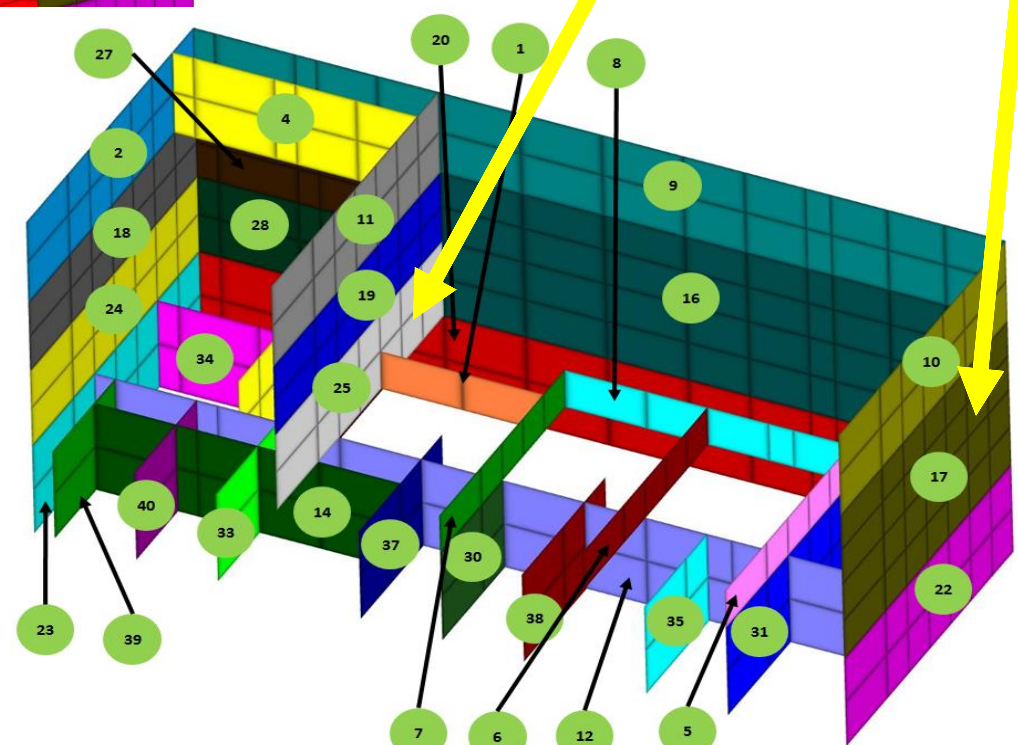
Original Elastic Model

Nonlinear Building Model Split in Wall Panels



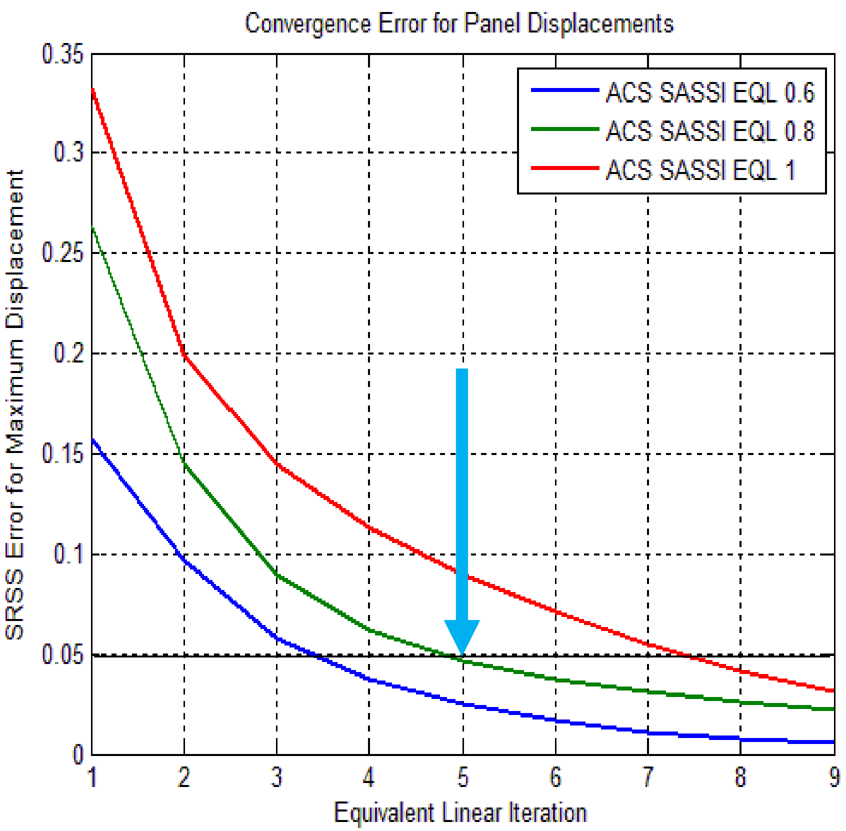
Nuclear Building Nonlinear Concrete Panels

Nuclear Building Split In Nonlinear Panels

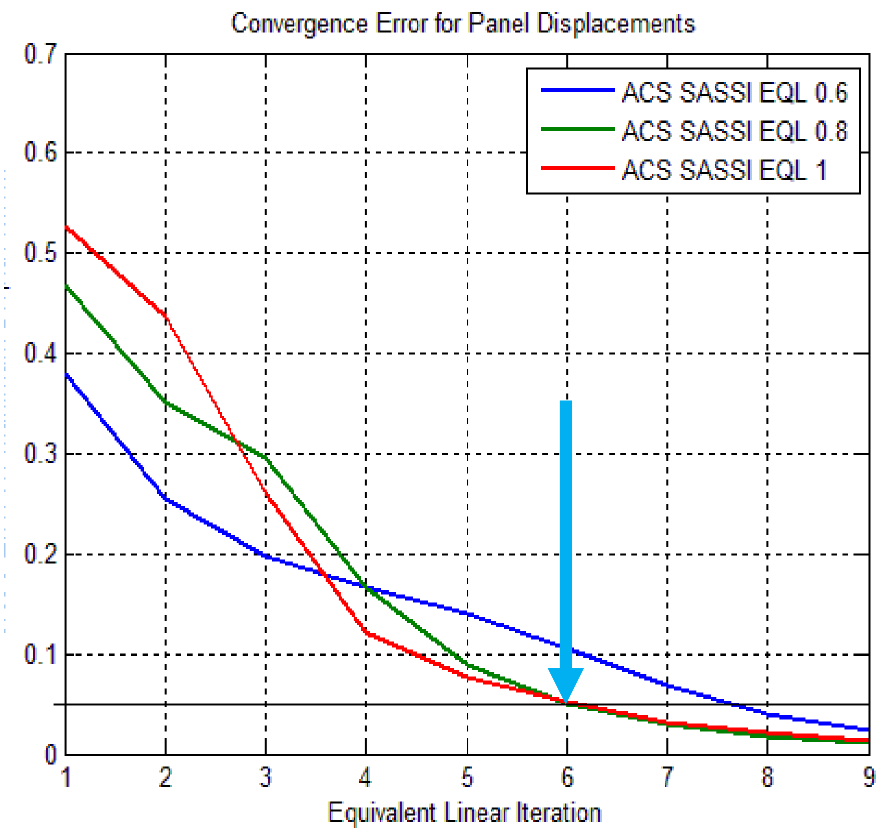


Nonlinear Solution Convergence is in 5-6 SSI Iterations Using "New Structure" Restart ANALYS Option

0.3g

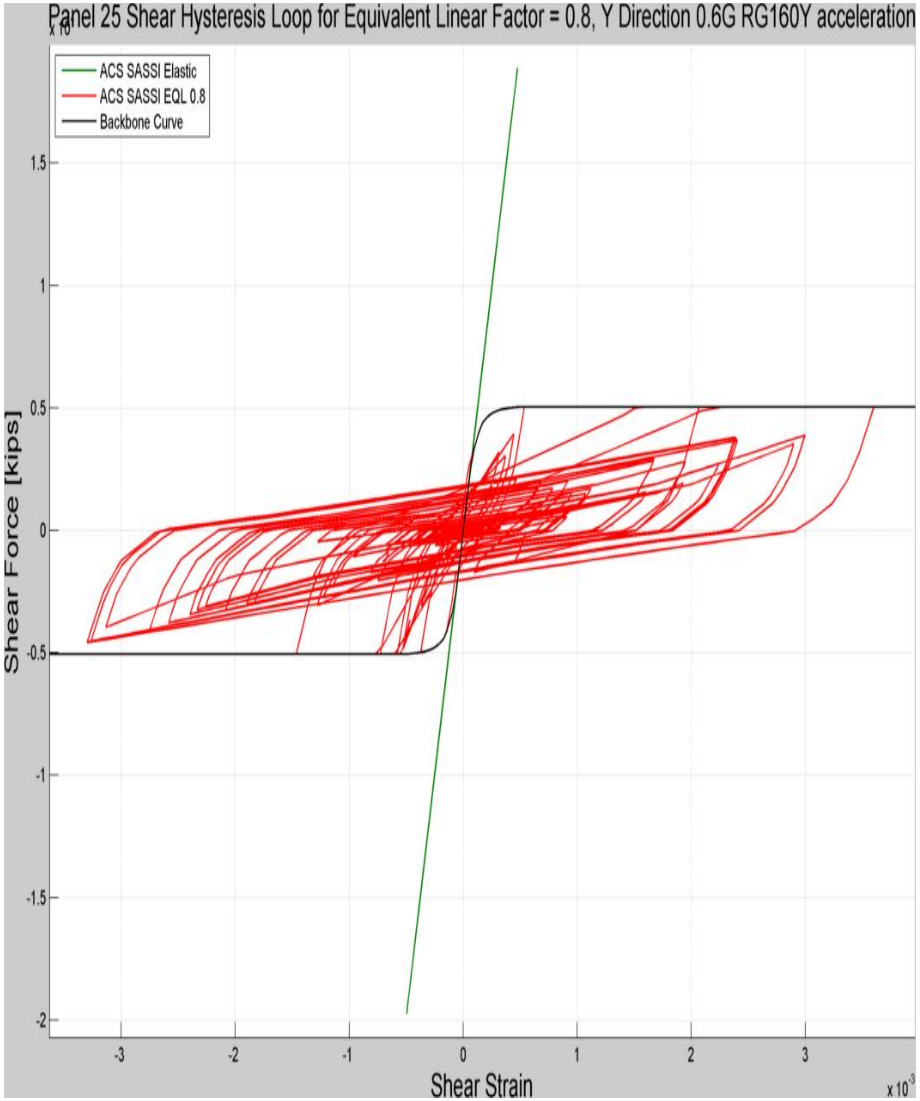


0.6g

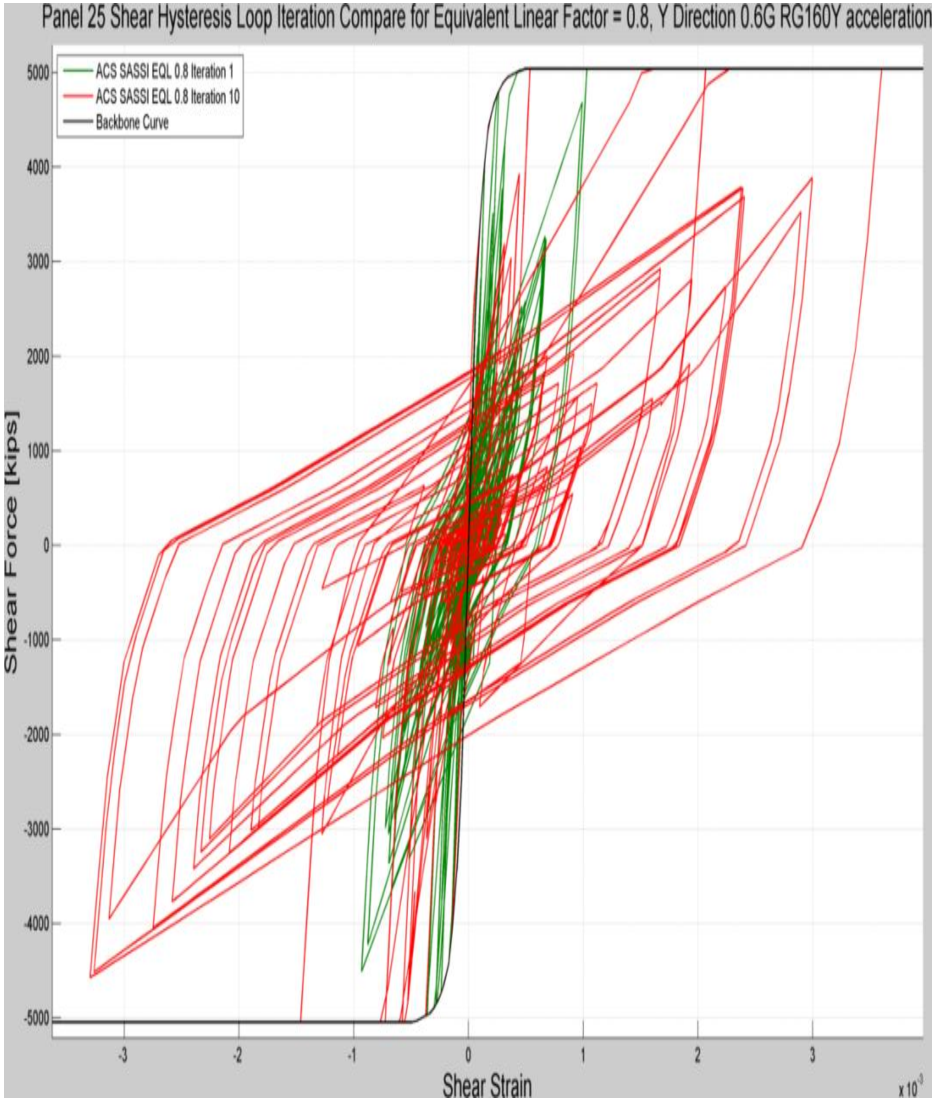


Nonlinear Bldg. SSI Analysis for 0.6g Earthquake

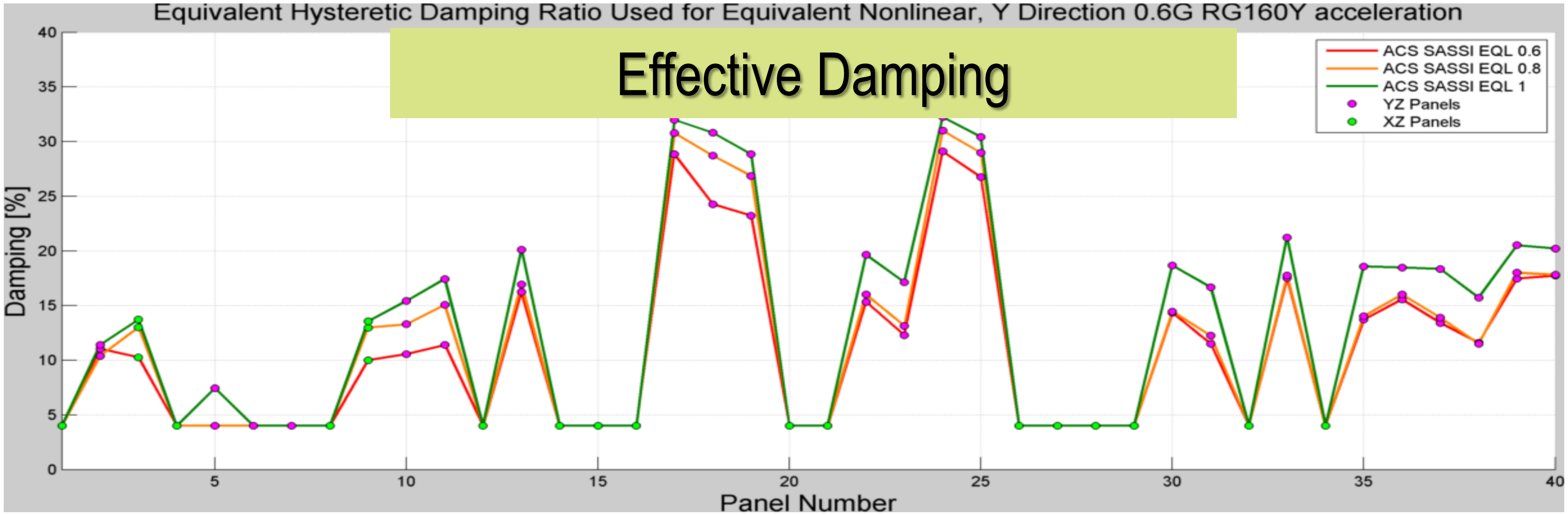
Elastic vs. Nonlinear

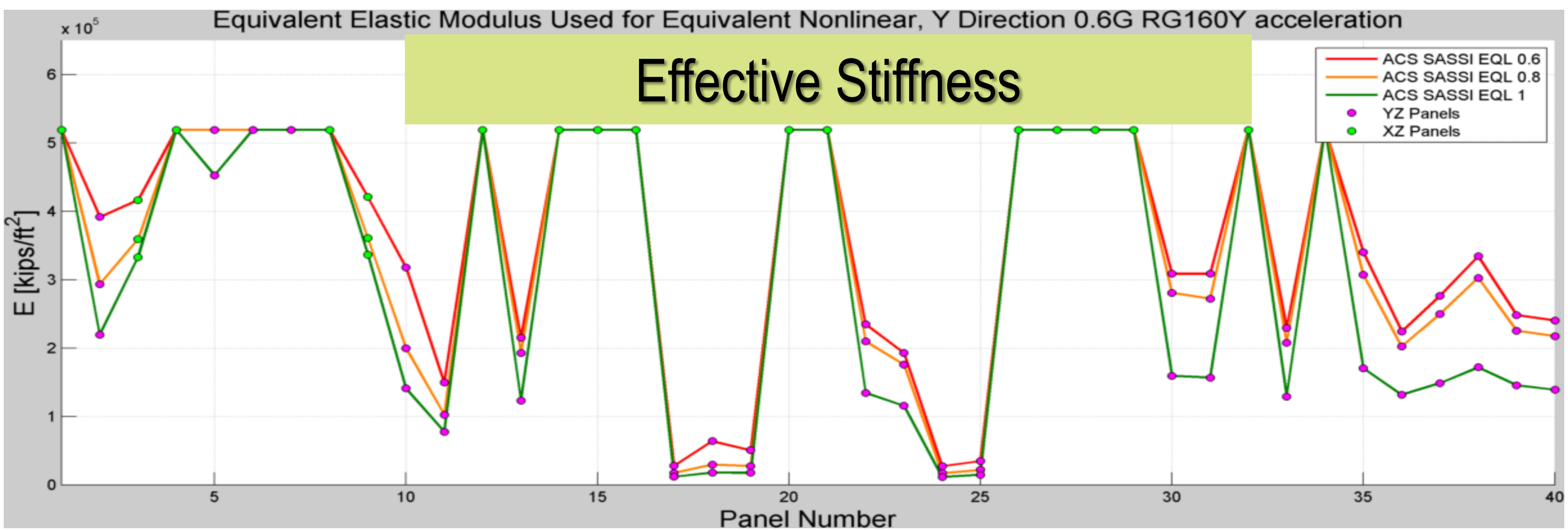


1st Iteration vs. Last Iteration

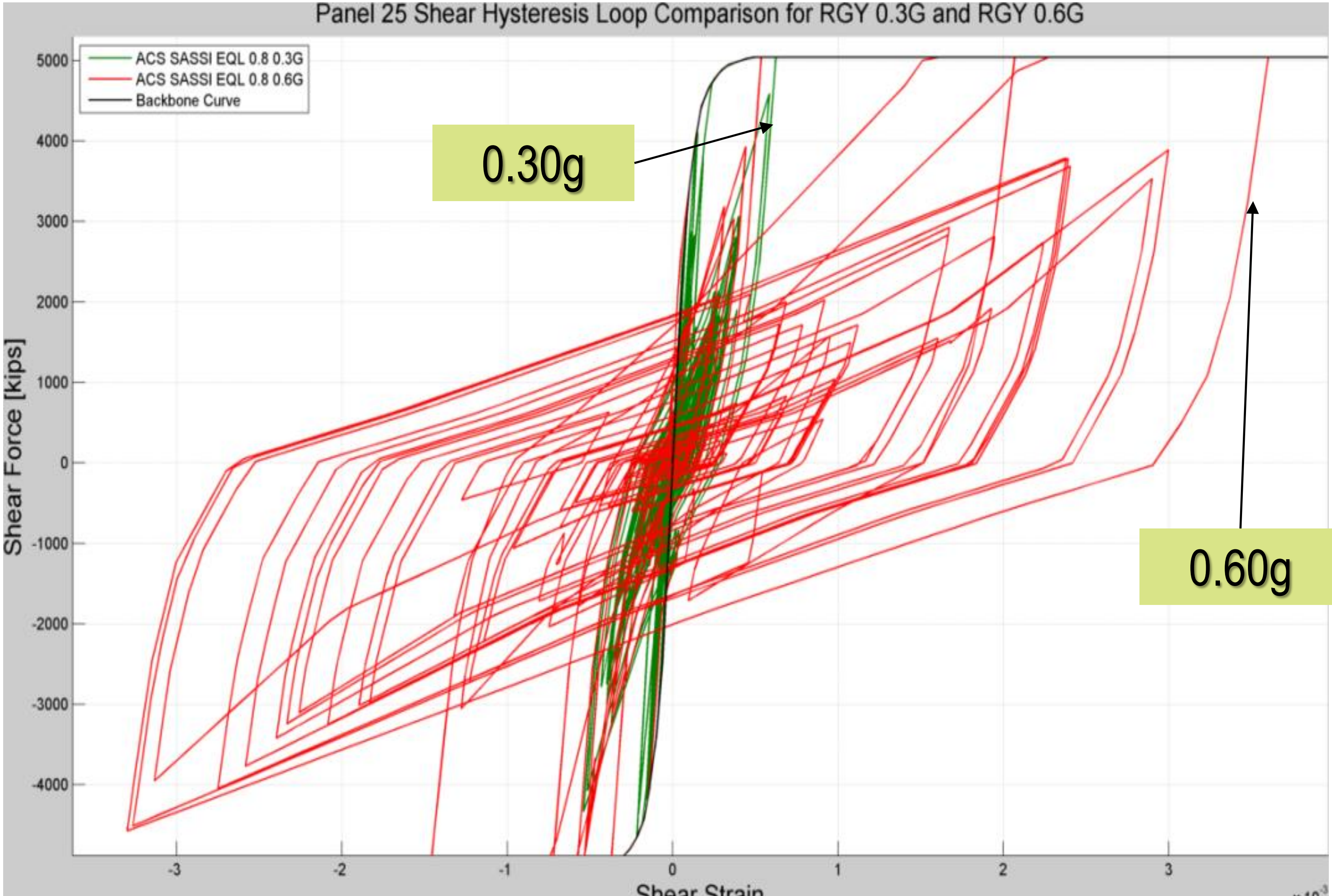


Nonlinear Bldg. SSI Analysis for 0.6g Earthquake. Final Equivalent Linear Panel Properties

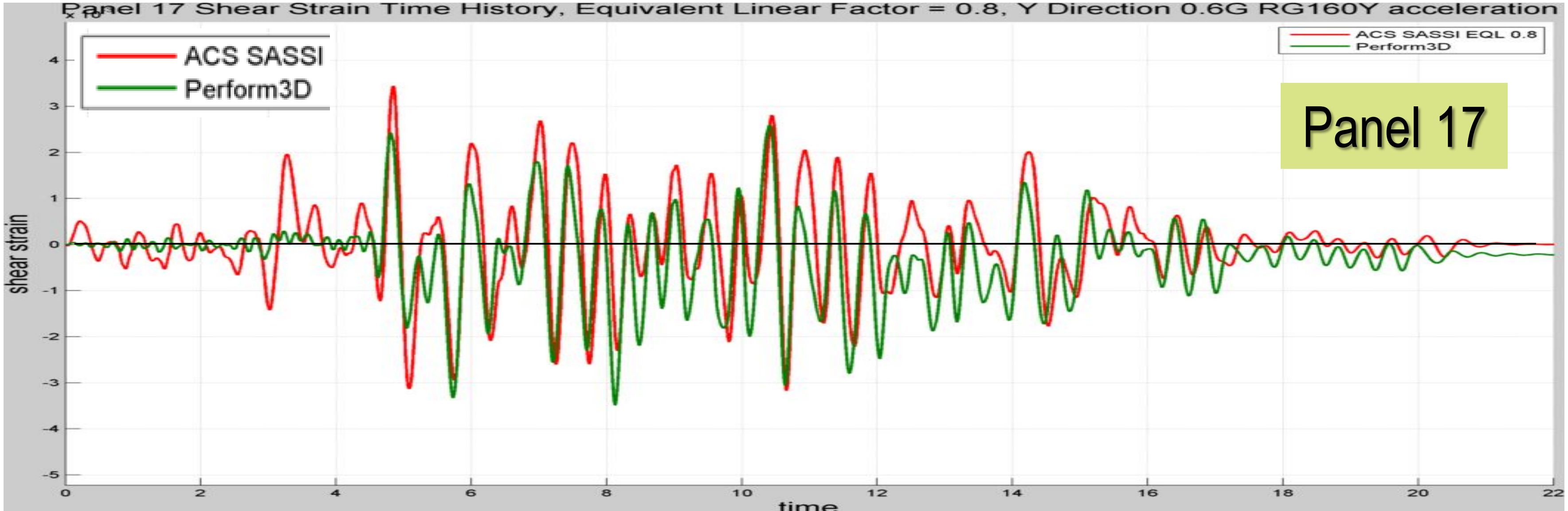


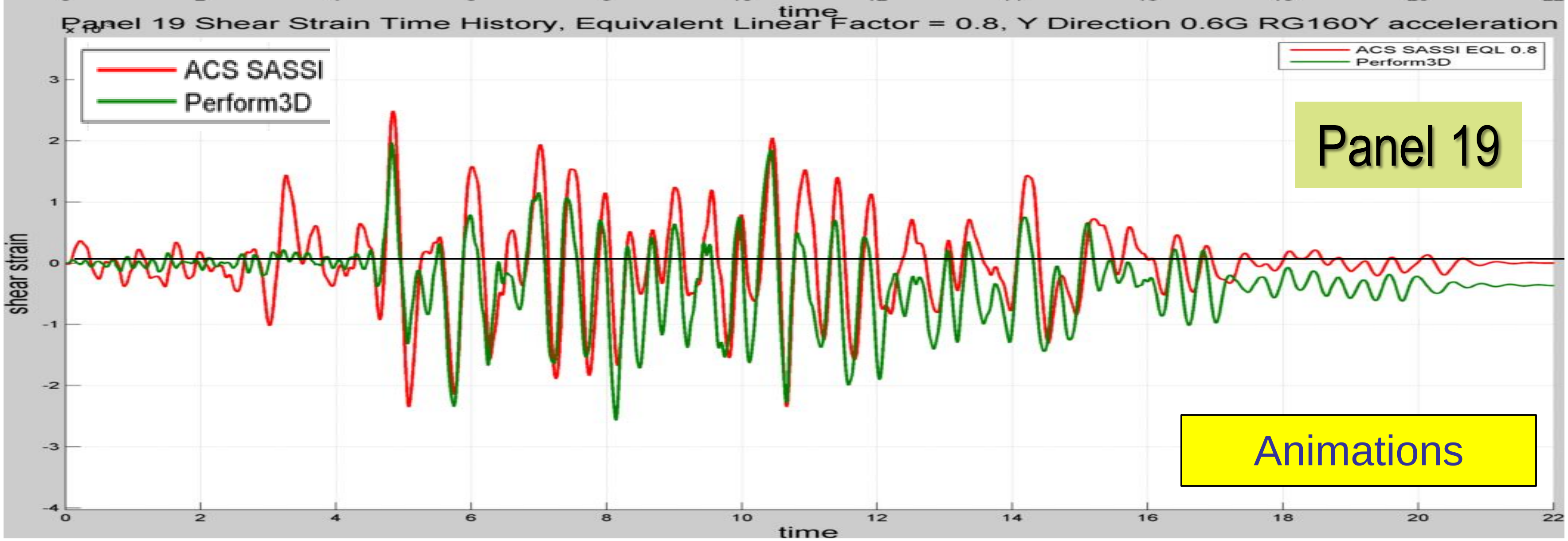


Nonlinear SSI Analysis for 0.3g (Green) and 0.6g (Red) Earthquake; Panel 17 Hysteretic Loops

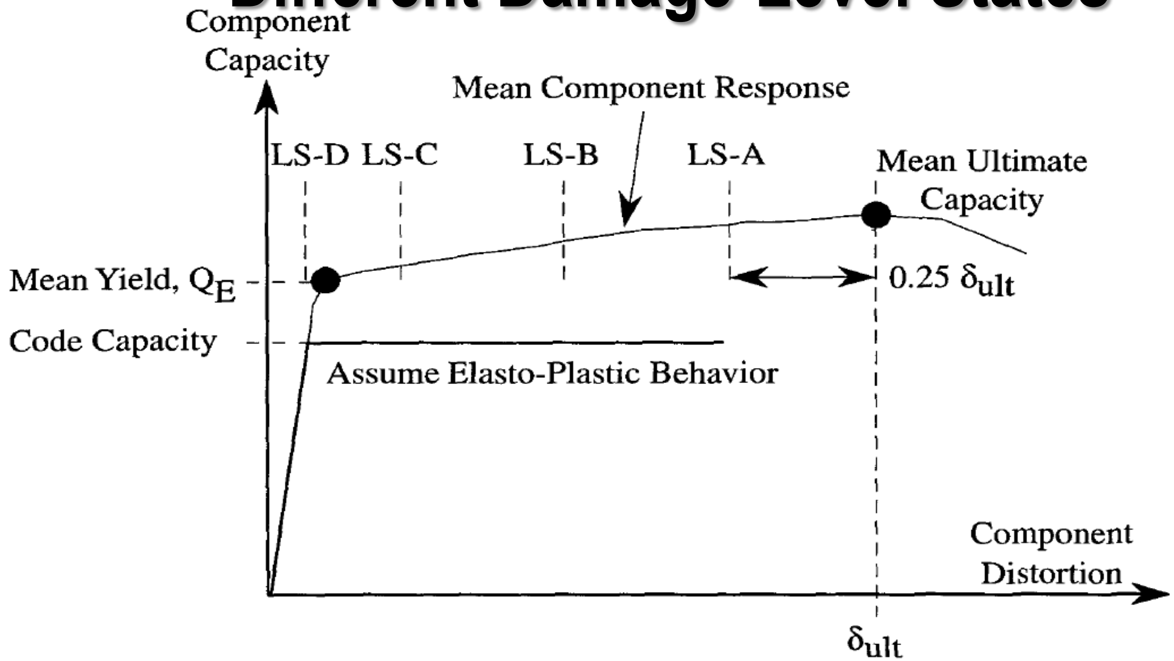


ACS SASSI vs. Perform3D Nonlinear Fixed-Base Analysis; Normalized Story Drifts, D/H





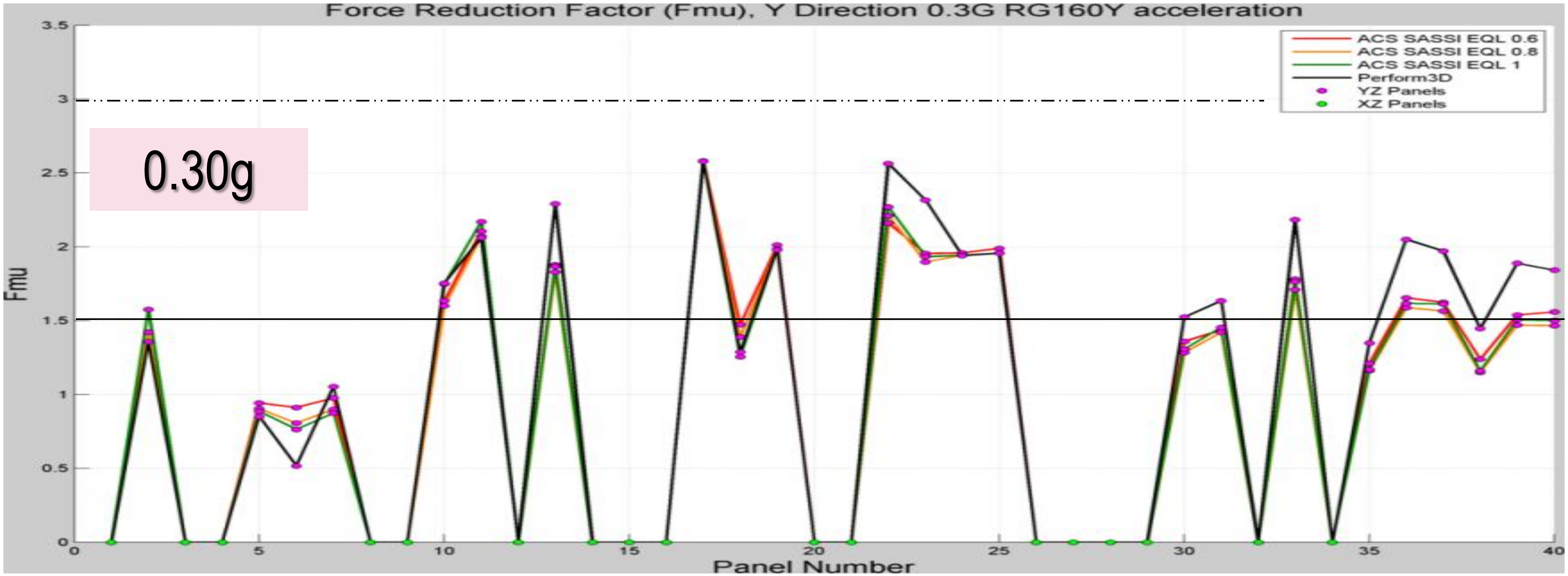
ASCE 43-05 Inelastic Reduction Factors with 95% NEP for Different Damage-Level States

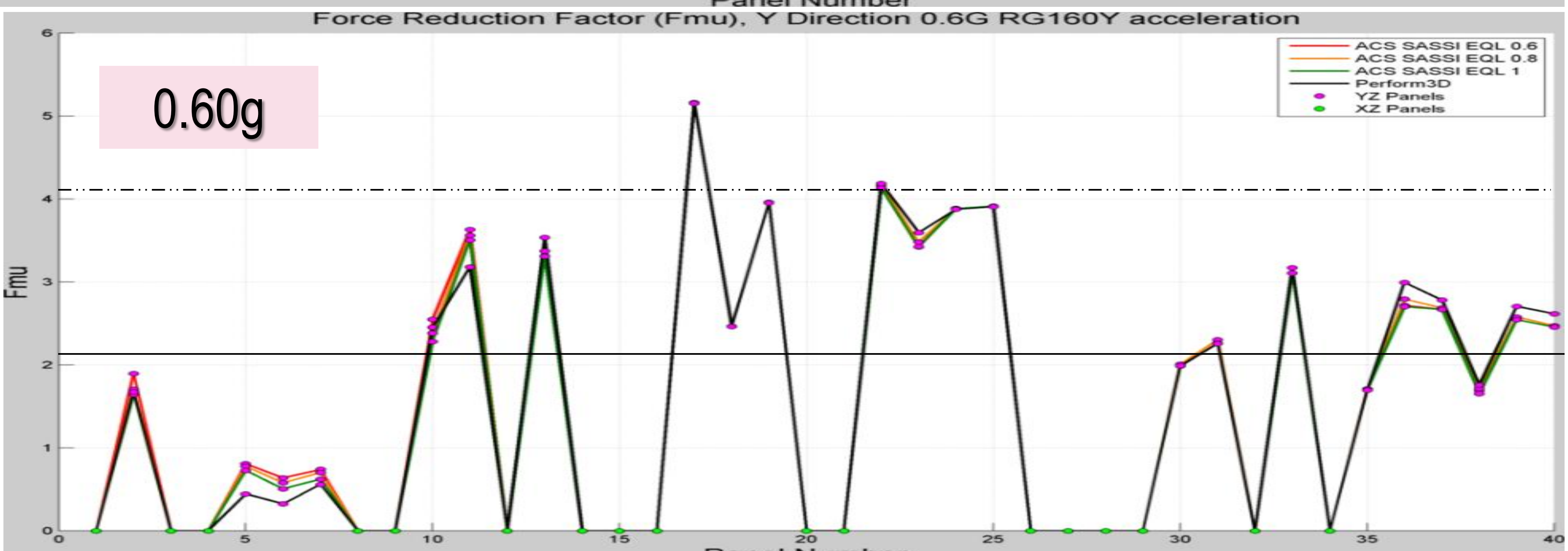


C5-4 Typical Load-Deformation Curve and Limit States

Limit State		LS-A	LS-B	LS-C
SMRF reinforced concrete moment frames				
Beams	$(15 \leq \ell/h)$	5.25	4.0	2.5
	$(\ell/h \leq 10)$	3.25	3.0	2.5
Columns**		2.0	1.75	1.5
Reinforced concrete shear wall, in plane:				
Bending controlled walls, $\frac{h_w}{\ell_w} \geq 2.0$				
	$6\sqrt{f'_c} < f_v$	2.25	2.0	1.75
	$f_v < 3\sqrt{f'_c}$	2.5	2.25	1.75
Shear controlled walls, $\frac{h_w}{\ell_w} < 2.0$		2.0	1.75	1.5

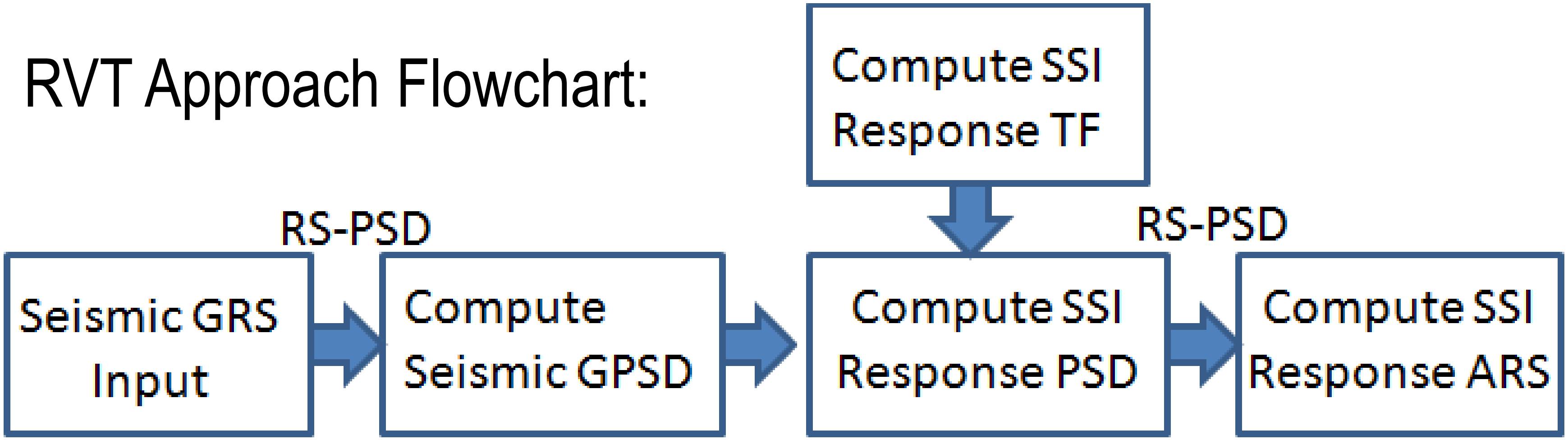
Inelastic Factors (Fe/Fn) for 0.30g and 0.60g Y-Dir Input





4) RVT Approach for Seismic SSI Analysis

RVT Approach Flowchart:



SDOF Transfer Functions:

$$H_0(\omega) = \frac{\omega_0^2 + 2i\omega_0\xi_0\omega}{\left(\omega_0^2 - \omega^2\right) + 2i\omega_0\xi_0}$$

Absolute Accelerations (ARS-APSD)

$$H_0(\omega) = \frac{\omega}{\left(\omega_0^2 - \omega^2\right) + 2i\omega_0\xi_0}$$

Relative Velocities (VRS-VPSD)

$$H_0(\omega) = \frac{1}{\left(\omega_0^2 - \omega^2\right) + 2i\omega_0\xi_0}$$

Relative Displacements (DRS-RPSD)

RVT Approach for SSI Analysis (Only Seismic Input)

The RVT based approach uses frequency domain convolution computations (no need to use time-histories) assuming a *linear system under a Gaussian seismic input:*

$$S_X(\omega) = |H(\omega)|^2 |H_0(\omega)|^2 S_u(\omega)$$

Response SSI SDOF Input

The RVT-based approaches include several options related to the *PSD-RS transformation*. These options are related to the stochastic approximation models used for computing the maximum SSI response overt a time period T, i.e. during the earthquake intense motion time interval.

The maximum SSI response can be expressed by using peak factors that are applied to the stochastic motion standard deviation (RMS). These quantities depend on the duration T, the mean crossing rate of the motion and probability level associated to the maximum response ("first passage problem").

Computation of Maximum Response in Time (RS)

$$\bar{X}_{\max} = p\sigma_X$$
$$\sigma_{X_{\max}} = q\sigma_X$$

1) M Kaul-Unruh-Kana stochastic model (MK-UK) (1978, 1981) :

$$p = \left[-2\ln\left(-\left(\frac{\pi}{T}\right)\left(\frac{\sigma_X}{\sigma_{\dot{X}}}\right)\ln(P) \right) \right]^{1/2}$$

Please note that this *p* is not the mean peak factor, since it provides maximum peak factor for any given NEP P

2) A Davenport (AD) (1964) for p and Der Kiureghian (1980) for q

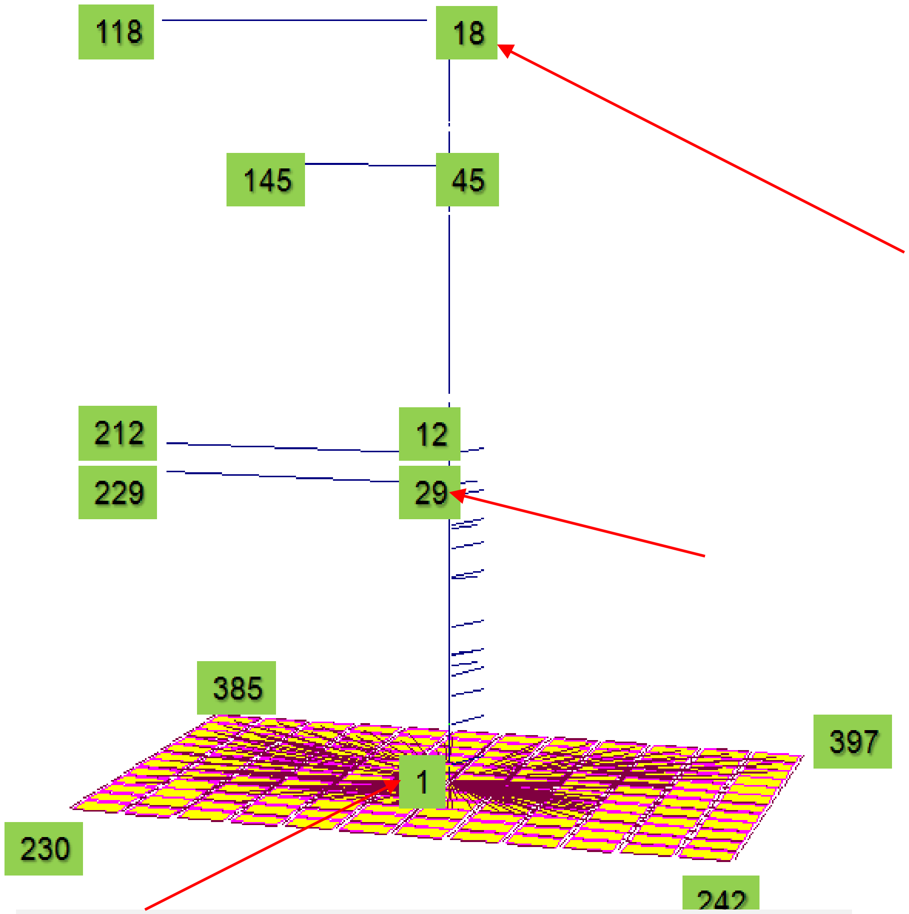
$$p = \sqrt{2\ln(\nu_0 T)} + \frac{0.5772}{\sqrt{2\ln(\nu_0 T)}} \qquad q = \frac{1.2}{\sqrt{2\ln(\nu_0 T)}} - \frac{5.4}{\left[13 + \left(2\ln(\nu_0 T)\right)^{3.2}\right]}$$

3) A Davenport Modified by Der Kiureghian (AD-DK) (1981,1983)

$$\nu_e T = \begin{cases} \max\left(2.1, 2\delta\nu_0 T\right) & ;0 < \delta \le 0.1 \\ \left(1.63\delta^{0.45} - 0.38\right)\nu_0 T & ;0.1 < \delta < 0.69 \\ \nu_0 T & ;0.69 \le \delta < 1 \end{cases} \qquad \delta = \sqrt{1 - \frac{\lambda_1^2}{\lambda_0 \lambda_2}}$$

EPRI AP1000 Stick RVT SSI Study

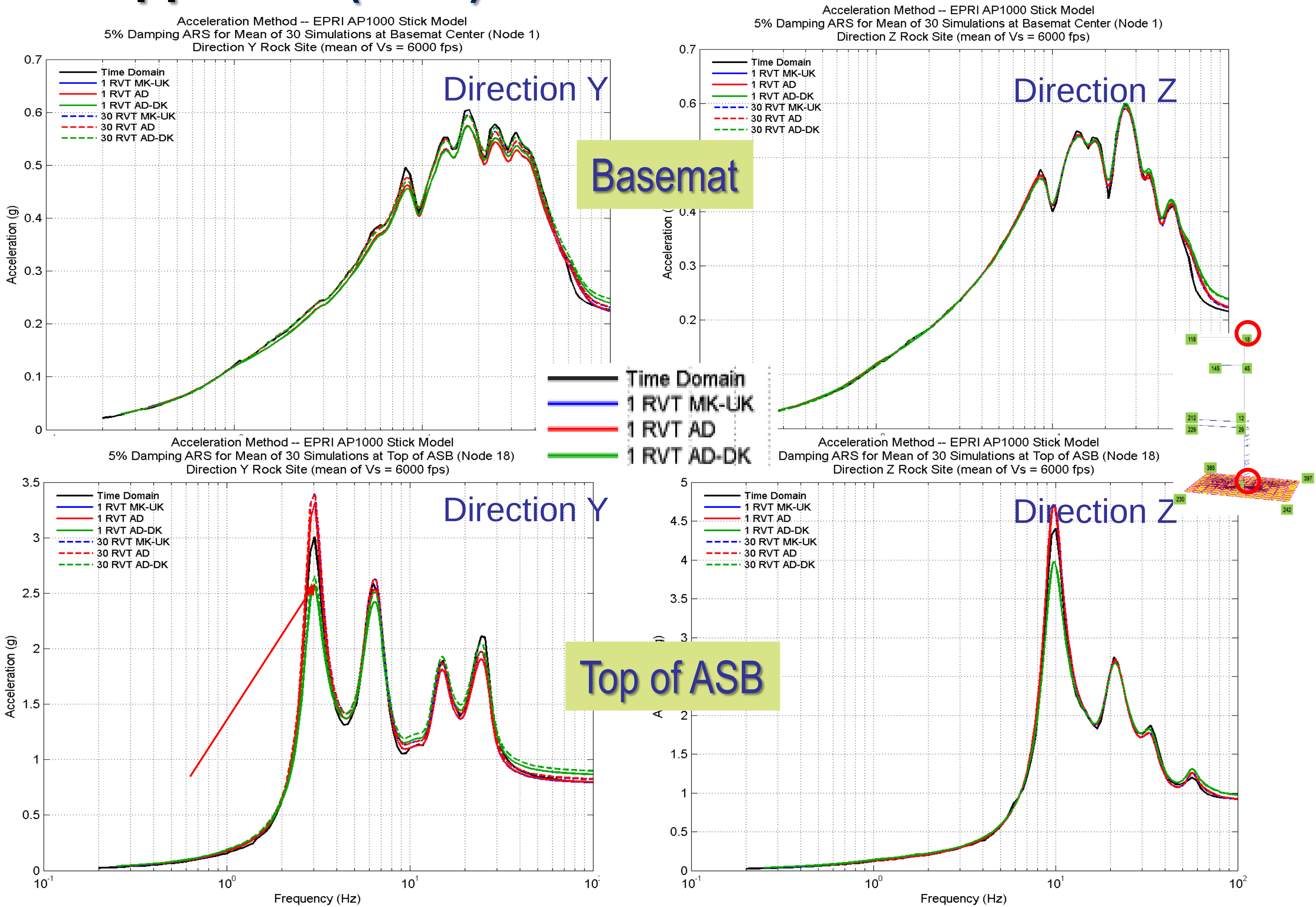
EPRI AP1000 NI Stick Model



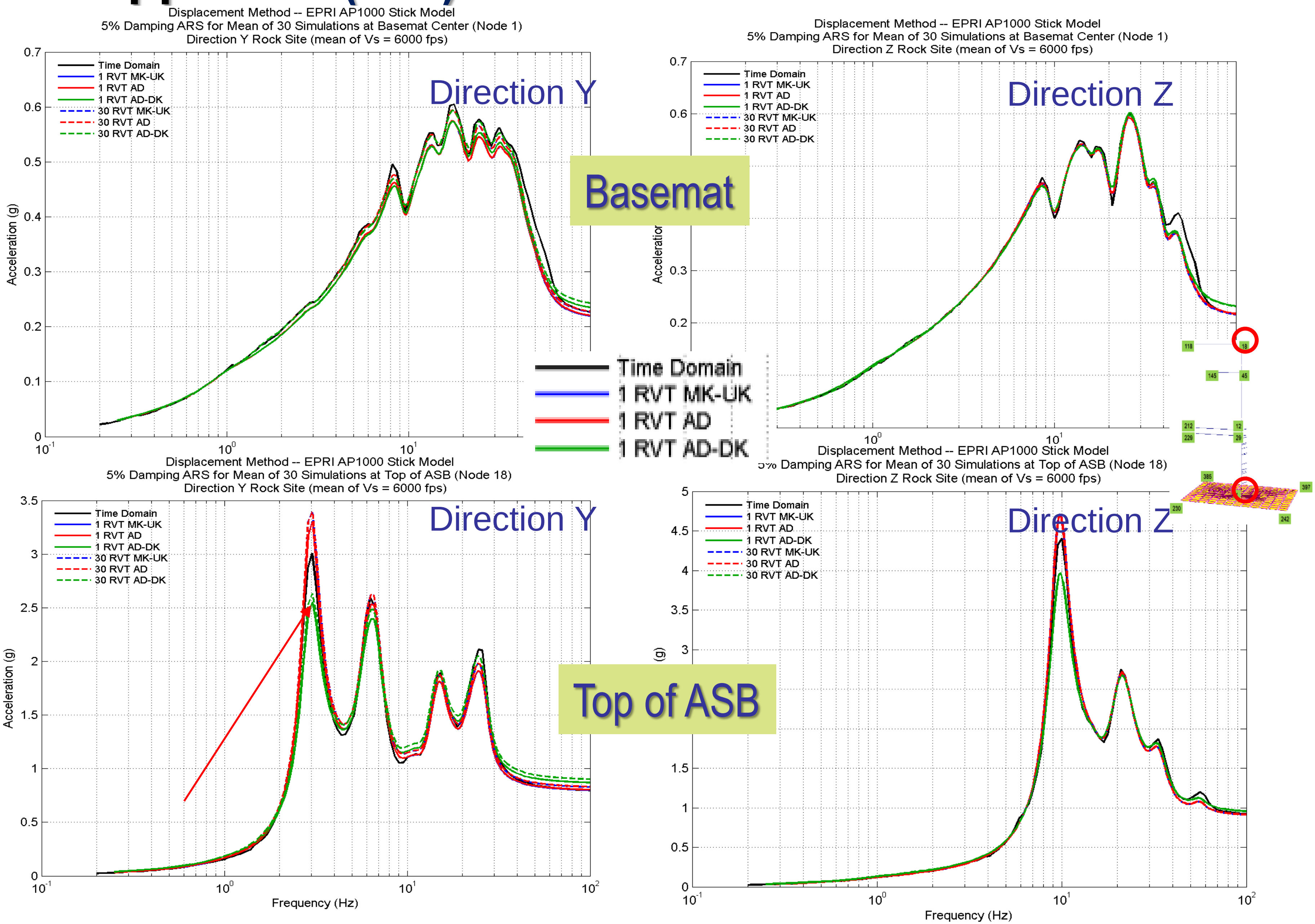
Case 1: Soil Site, Vs = 1,000 fps

Case 2: Rock Site, Vs = 6,000 fps

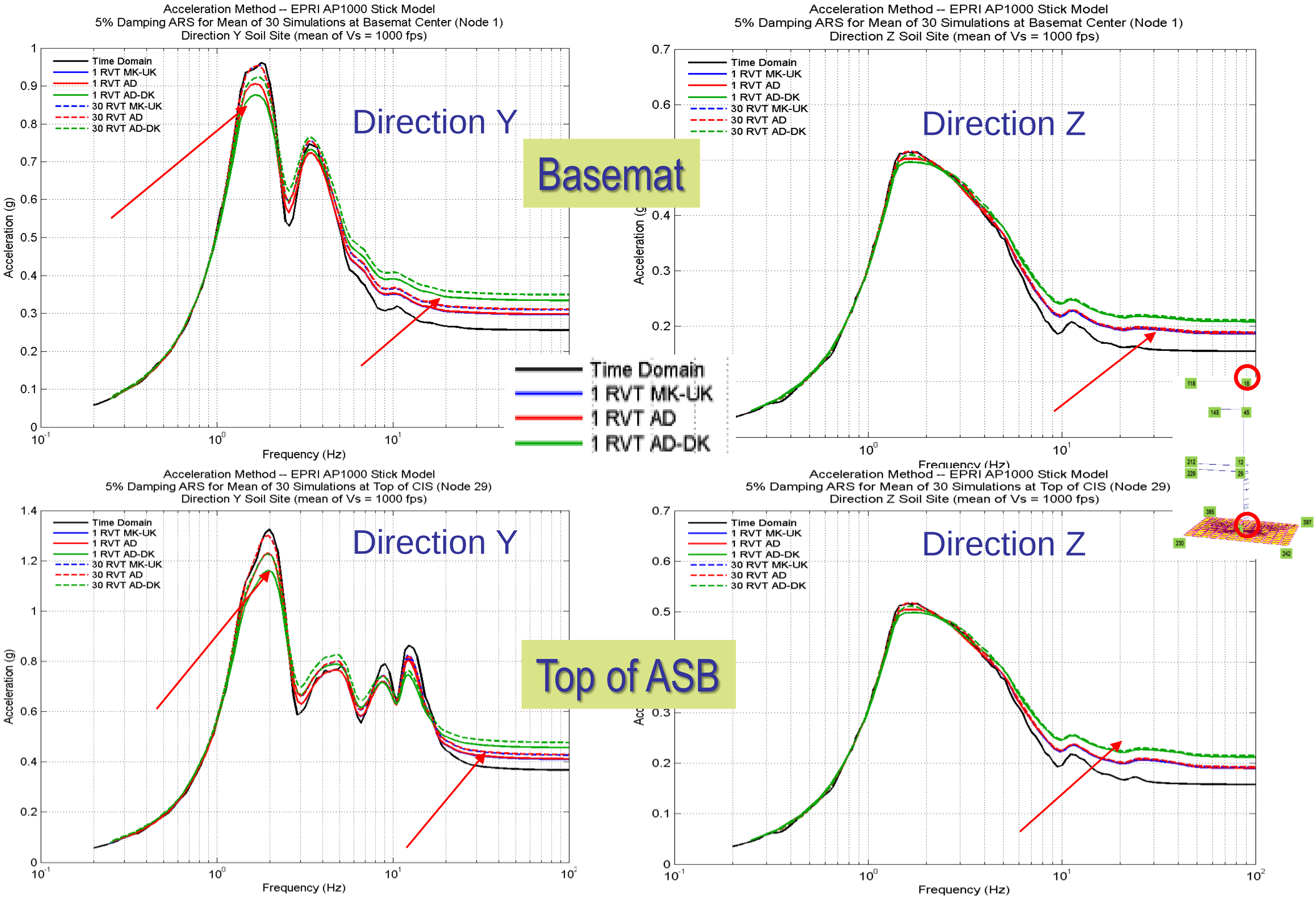
RVT Approach (ACC) vs. LHS for Rock – Mean ISRS



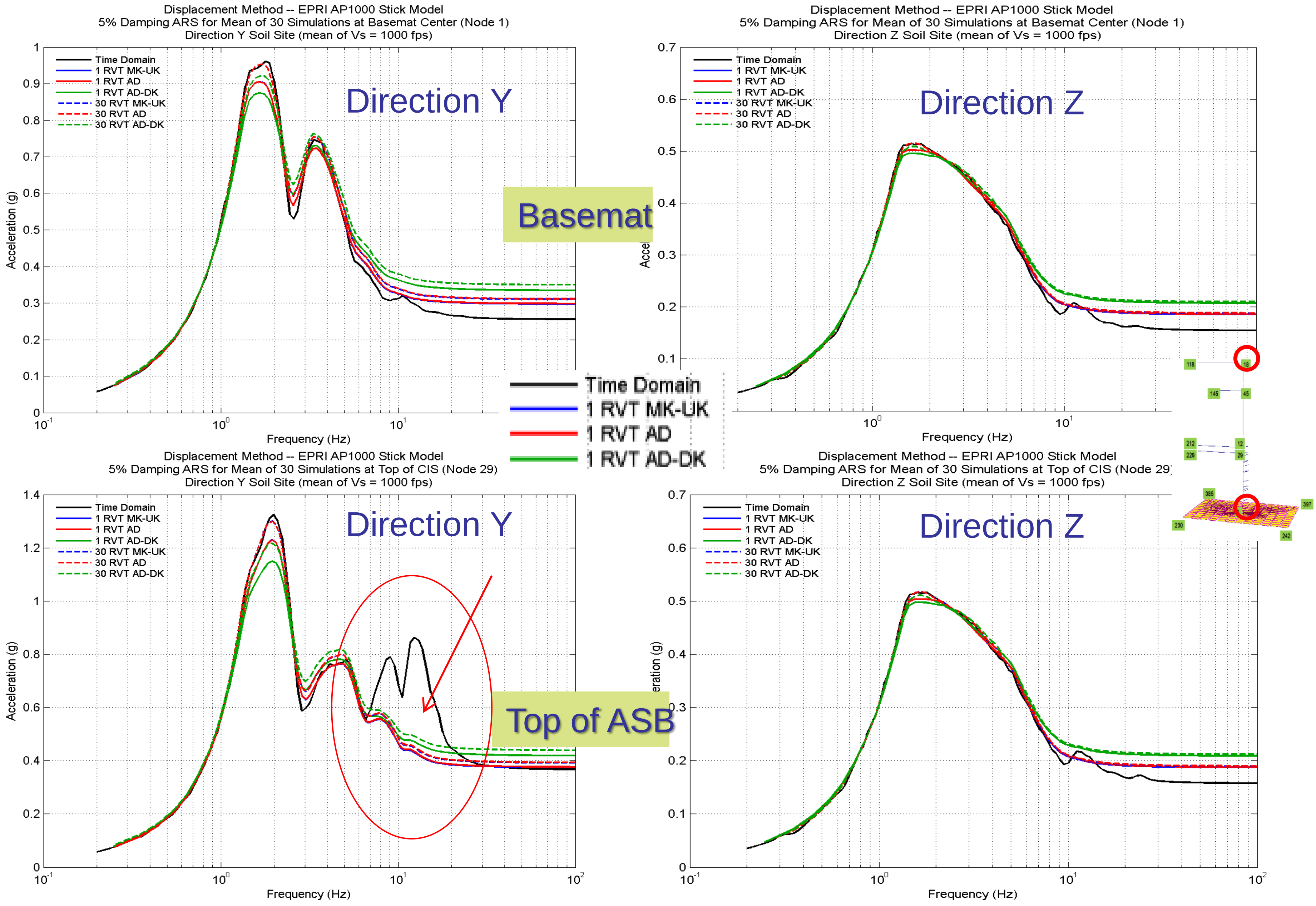
RVT Approach (DIS) vs. LHS for Rock – Mean ISRS



RVT Approach (ACC) vs. LHS for Soil – Mean ISRS



RVT Approach (DIS) vs. LHS for BE Soil – Mean ISRS



Conclusions for Investigated Cases

- For Probabilistic SSI analysis, the structure stiffness & damping uncertainties impact differently on ISRS for rock and soil sites
- For Probabilistic SSI analysis, the structure stiffness & damping uncertainties impact differently on ISRS depending on the floor elevation
- Probabilistic ISRS computed for 84% NEP show appear too low for *rock sites* due to the smoothing effect produced by statistical averaging on the sharp ISRS peaks - frequency shifts are an important parameter. *CAUTION! Guidelines needed; use higher NEP than 84%...?*
- RVT-based SSI approaches provide approximate solutions for the *mean* ISRS. However, the ISRS accuracy depend on the "*analytical equation*" used for computing maximum response (RS) of the Gaussian motion.

Probabilistic Site Response Analysis (PSRA) and Probabilistic SSI Analysis (PSSIA) *(Option PRO, May 15)*

Based on the new ASCE 04-2015 standard recommendations and guidelines for probabilistic seismic SSI analyses.

Probabilistic SSI responses should defined with the 80% non-exceedance probability level to be considered adequate for design.

ASCE 04-2015 probabilistic SR and SSI analysis models and methods are described in *Section 2.0 on Seismic Input and Section 5.5 on Probabilistic SSI*:

- GMRS/UHRS seismic input spectral content could be considered with randomly varying shape or not (Methods 1 and 2).
- Vs and D soil profiles should include spatial correlation with depth.
- Local structural effective stiffness K and damping D as functions of stress level, and highly dependent random variables.

ACS SASSI Option PRO Implementation

ACS SASSI probabilistic modeling should include:

SEISMIC INPUT:

- Both ASCE 04-2015 Methods 1 and 2 for spectral shape modeling for the seismic GMRS/UHRS input

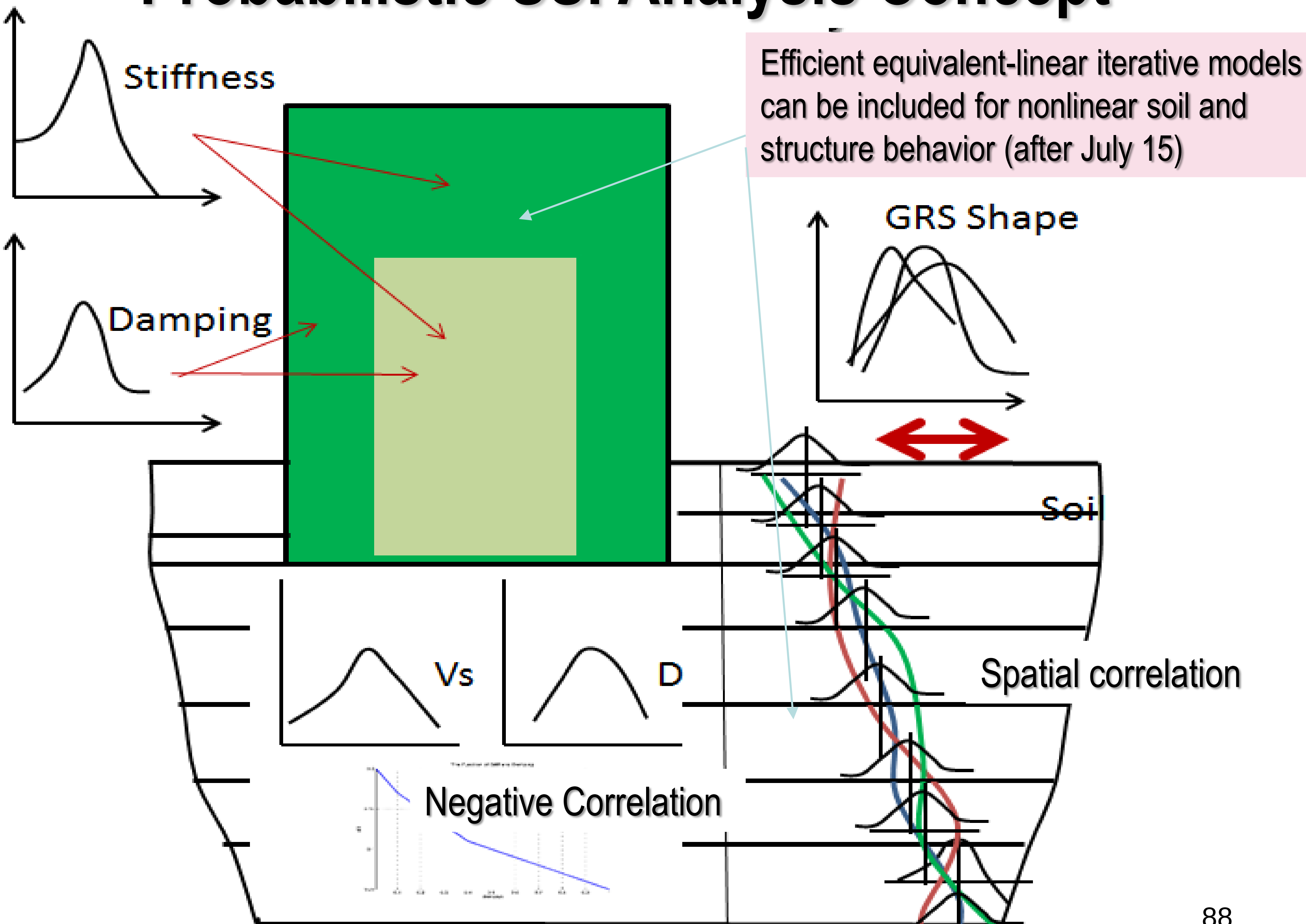
SOIL LAYER PROFILES:

- Low-strain soil shear wave velocity Vs and hysteretic damping D profiles modeled as 1D random fields, with dependent variables for each soil layer
- Effective soil shear modulus G and hysteretic damping D, as two random functions of soil shear strain for each soil layer

STRUCTURE :

- Stress-dependent equivalent-linear or effective stiffness K and structural damping D for each structural FE model group of elements (and materials within groups). The effective K and D depend on the stress levels in different parts of the structure. *Effective K and D can be accurately computed using Option NON.*

Probabilistic SSI Analysis Concept



Probabilistic SRA and SSIA Steps

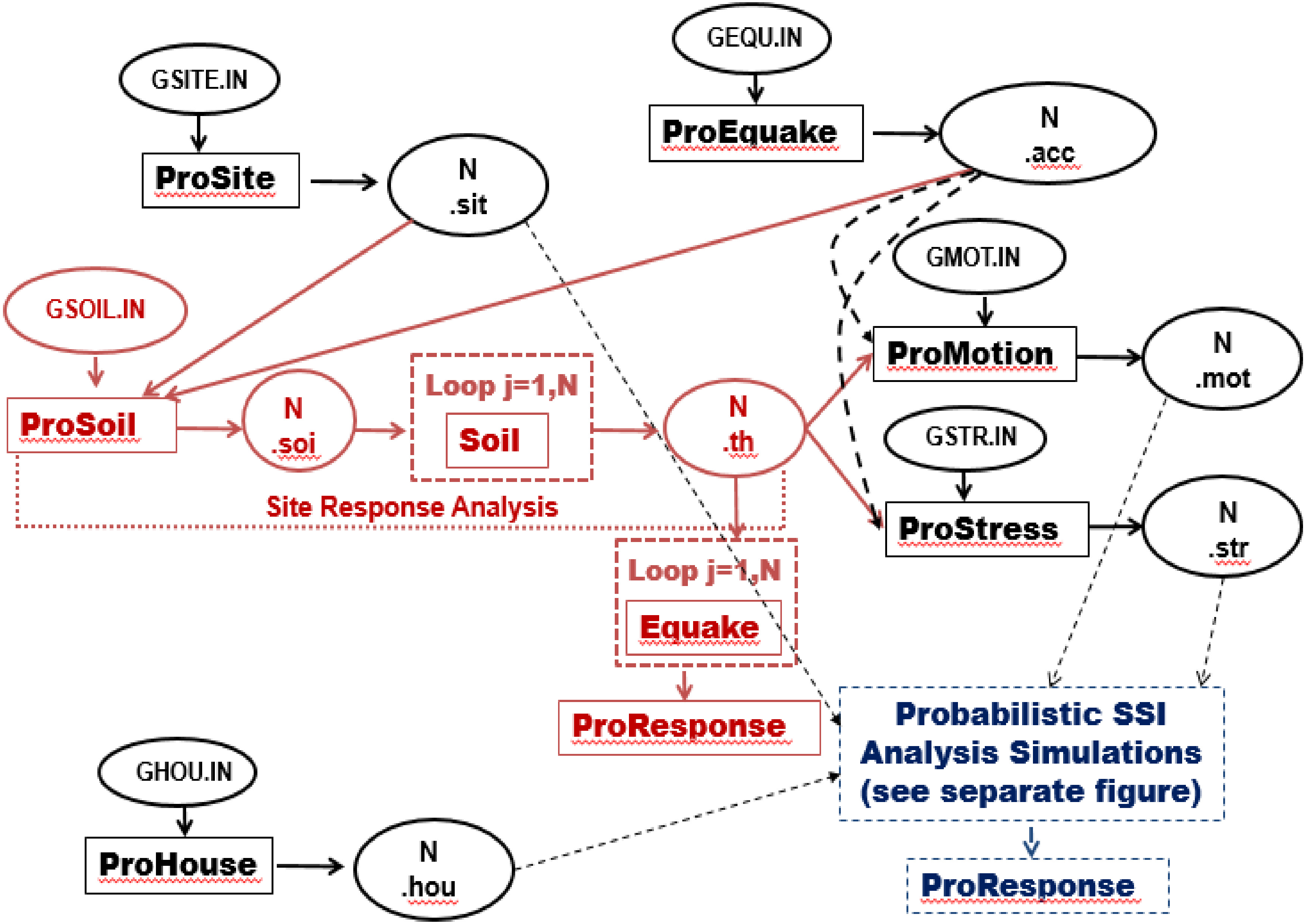
For a full probabilistic analysis three steps need to be completed:

1) **PREPROCESSING:** Using ACS SASSI PRO probabilistic SSI modules, generate statistical ensembles for Probabilistic SRA and/or Probabilistic SSI analysis input files using LHS simulations *(ProEQUAKE, ProSITE, ProSOIL and ProHOUSE)*

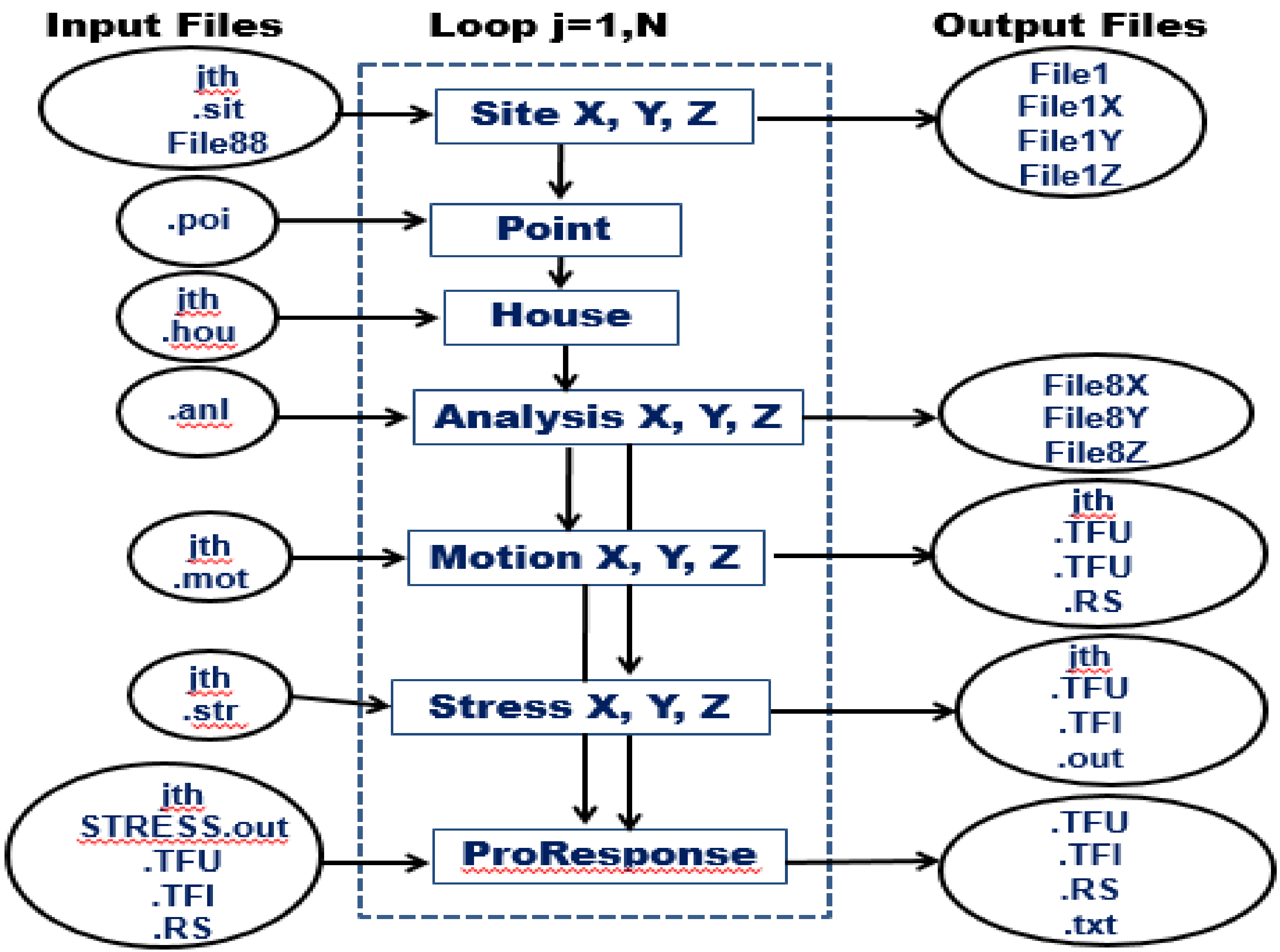
2) **ANALYSIS:** Using ACS SASSI deterministic SSI modules, run in batch the statistical ensembles of the LHS simulated input files to compute the LHS SSI response files (SITE, SITEP, SOIL, HOUSE, ANALYS, MOTION, RELDISP, STRESS).

3) **POSTPROCESSING:** Using ACS SASSI PRO post module, post-process statistically the ensembles of the LHS SSI responses *(ProRESPONSE)*

Probabilistic SSI Input Simulations
Using N LHS Samples

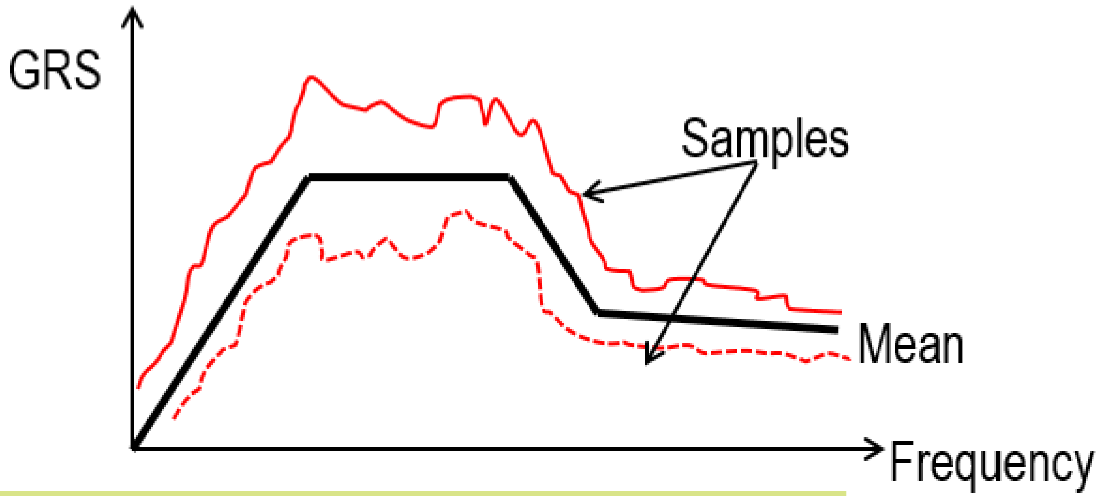


Probabilistic SSI Analysis Simulations
Using N LHS Samples

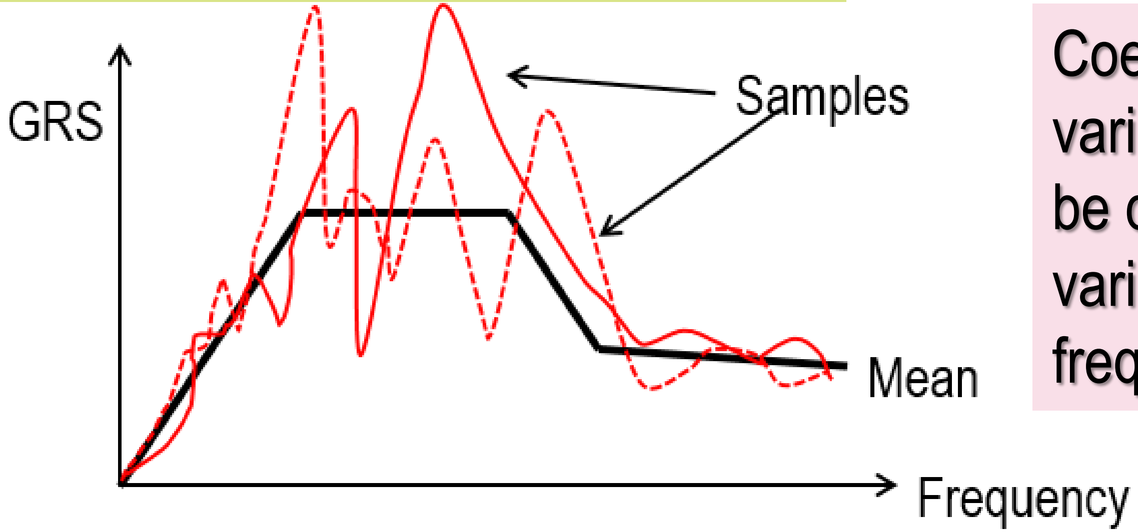


ProEQUAKE Spectral Shape Probabilistic Models

ASCE 04-2015 Method 1



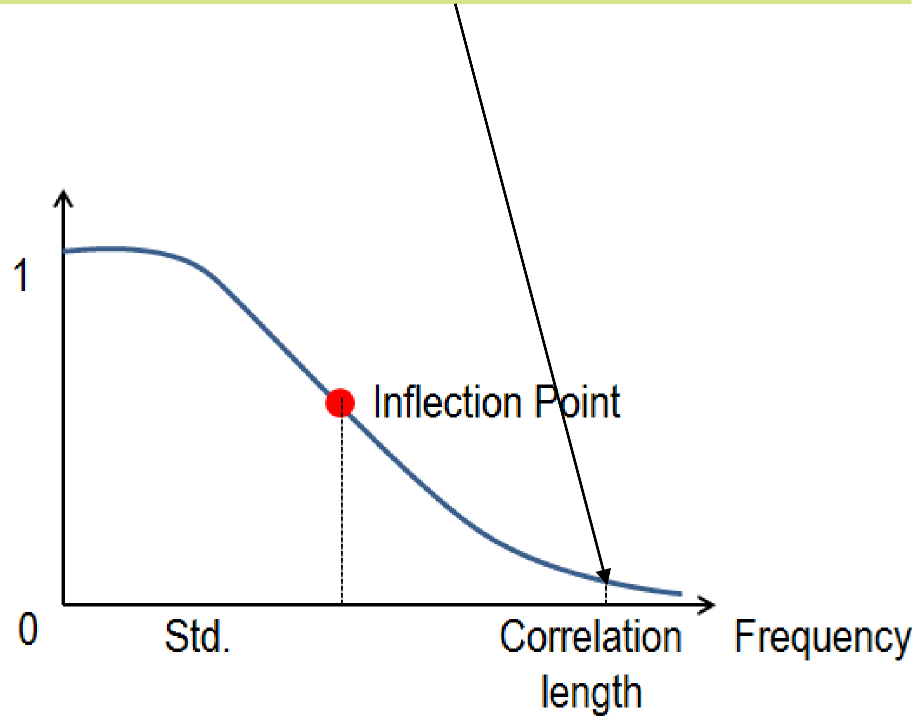
ASCE 04-2015 Method 2



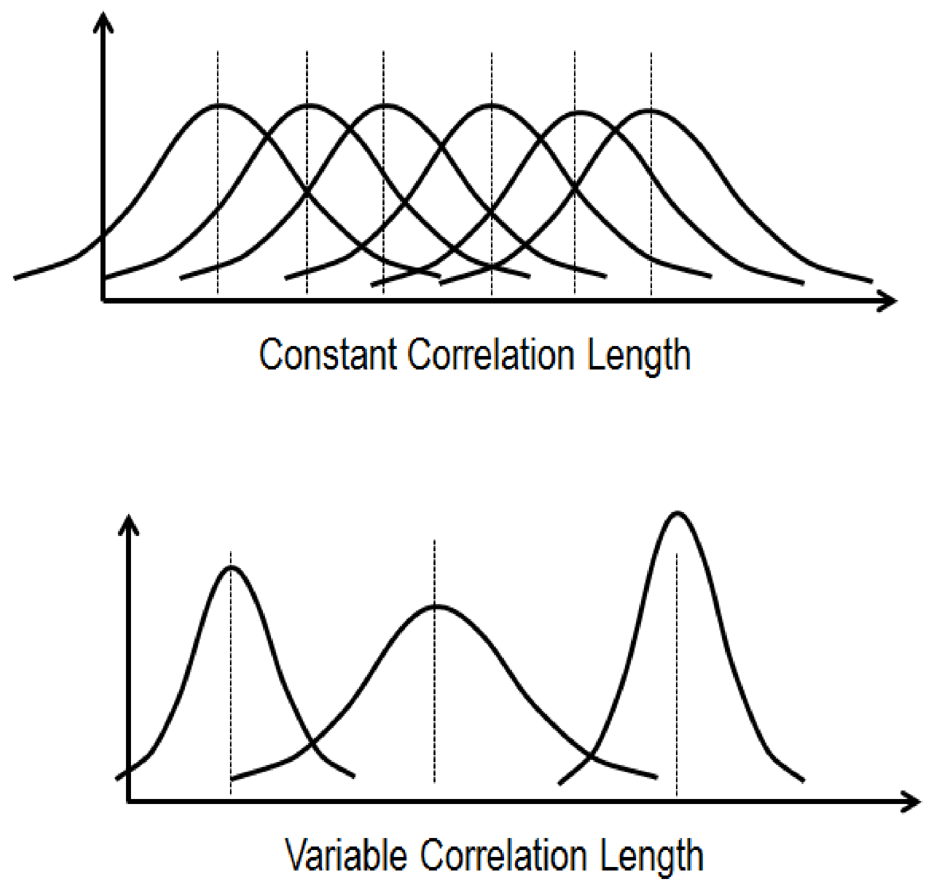
Coefficient of variation can be defined variable along frequency axis

GRS Spectral Shape Correlation Length Input

Correlation Length Definition

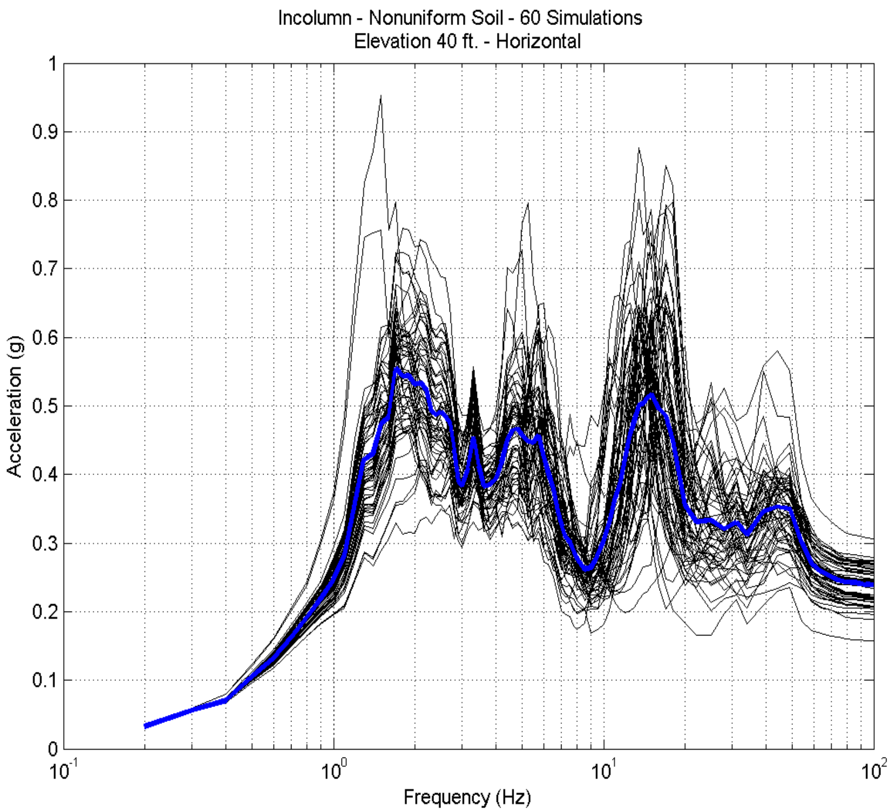


Constant vs. Variable Correlation Length

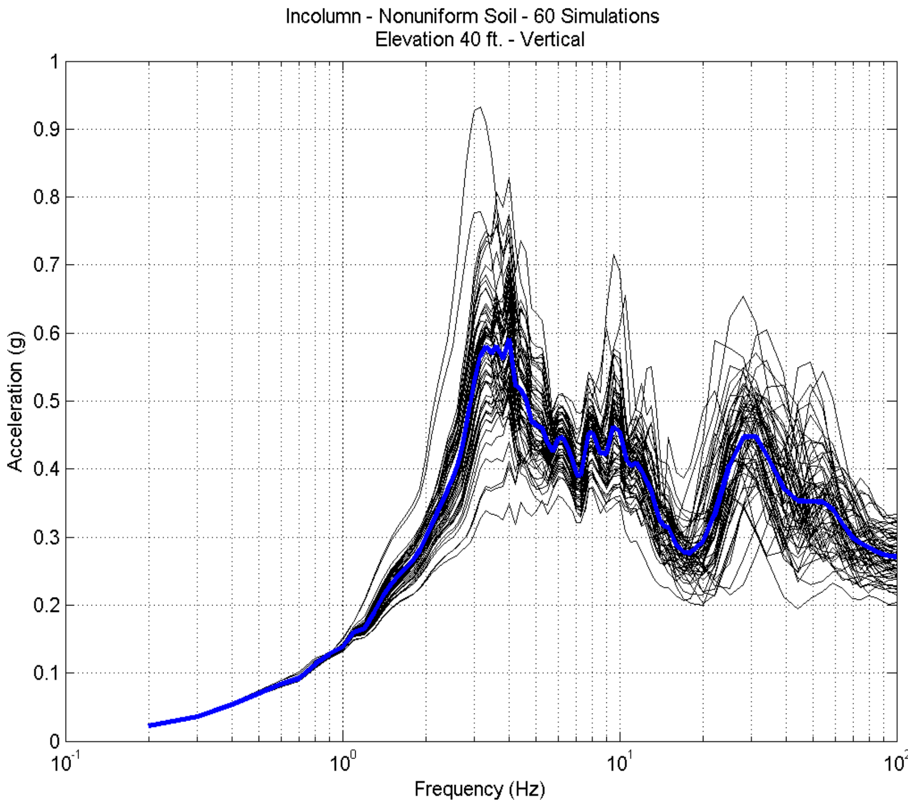


GRS Spectral Shape Correlation Matrix Input. Using Probabilistic Site Response Simulations

Horizontal



Vertical

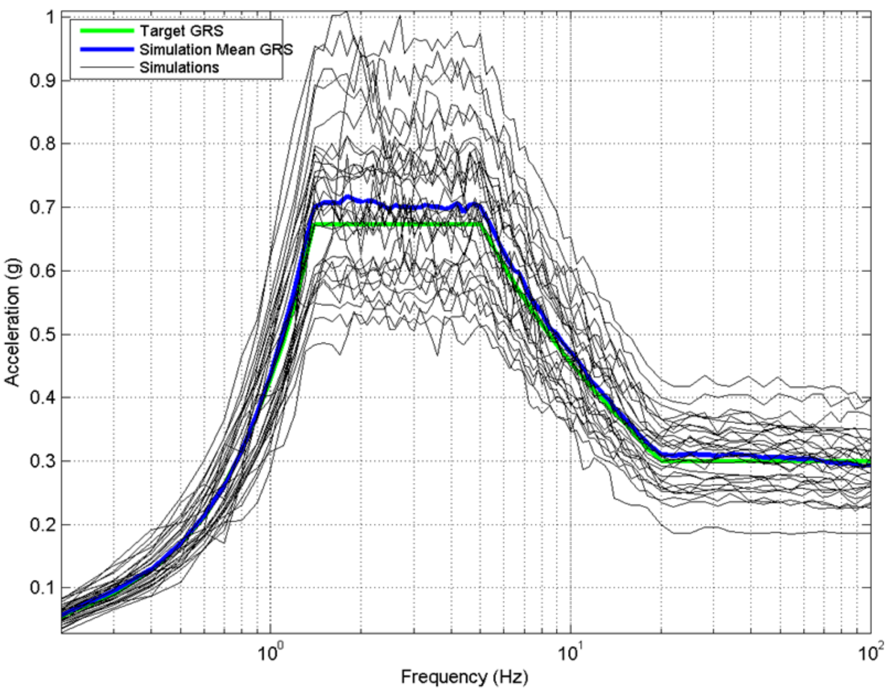


Seismic GRS Probabilistic Model Inputs

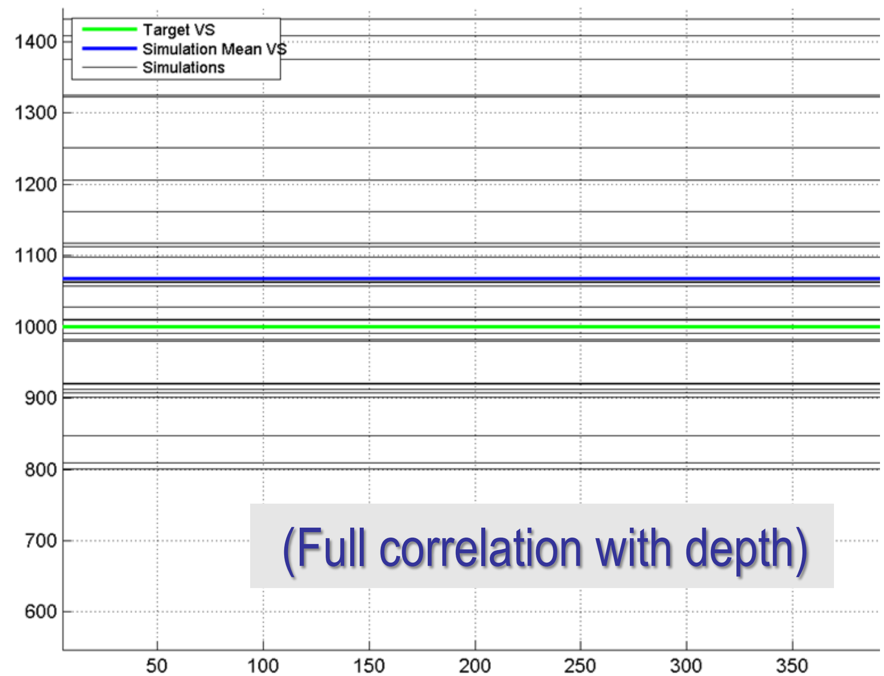
Input File Line Number	Variable Name (Input in free format)	Definition of Input Variables	Type
1	FRSI	Filenames for the simulated GRS inputs (ex. RSIxxx.RS)	Output
2	FRSO	Filenames for the computed GRS for simulated (ex. RS0xxx.RS	Output
3	FACC	Filenames for the computed acceleration histories (ex. ACCxxx.acc)	Output
4	FGEQU	Filenames for the simulated EQUAKE inputs (ex. GEQUxxx.equ)	Output
5	FBASEL	Filename for the seismic input mean GRS amplitude (ex. BASELINE.RSI)	Input
6	DAMPING	Damping ratio for the mean GRS input (in percent)	Input
6	GRAVACC	Acceleration of gravity for ground velocity and displacements	Input
7	DURATION	Duration of simulated acceleration histories (in seconds)	Input
7	TIMESTEP	Time step of simulated acceleration histories (in seconds)	Input
8	NSIMUL	Number of simulated seismic inputs for a single direction	Input
8	NFREQ	Number of frequencies in GMMEAN.RSI file	Input
8	INITRSI	Initial SEED Random Number for RSIxxx.RS simulation	Input
8	INITACC	Initial SEED Random Number for ACCxxx.ACC simulation	Input
9	OPTMETH	Option for the Method Used for GRS Simulation = 0 for Method 1 in ASCE 04-2015 = 1 for Method 2 in ASCE 04-2015	Input
9	DIR	Selected Input Direction: = 0 for X = 1 for Y = 2 for Z	Input
10	F1	1st Frequency for calculation of the c.o.v. factor (in Hz)	Input *
10	F2	2nd Frequency for calculation of the c.o.v. factor (in Hz)	Input *
11	OPTCOR	Option for GRS random shape correlation structure: = 0 for frequency-independent correlation length (scalar) = 1 for frequency-dependent correlation length (vector) = 2 for full correlation matrix for GRS frequencies (matrix)	Input
11	COV	Coefficient of variation of the GRS amplitude	Input
12	SIGMA	For OPTCOR = 0 is half of correlation length	Input
12	SIGMAV	For OPTCOR = 1 is half of correlation length vector	Input **
12	CORRMAT	For OPTCOR =2 is the correlation matrix file name	Input **

Simulated Probabilistic Seismic GRS (Method 1) and Soil Profile (Vs and D) Using Random Variables

Simulated GRS Inputs



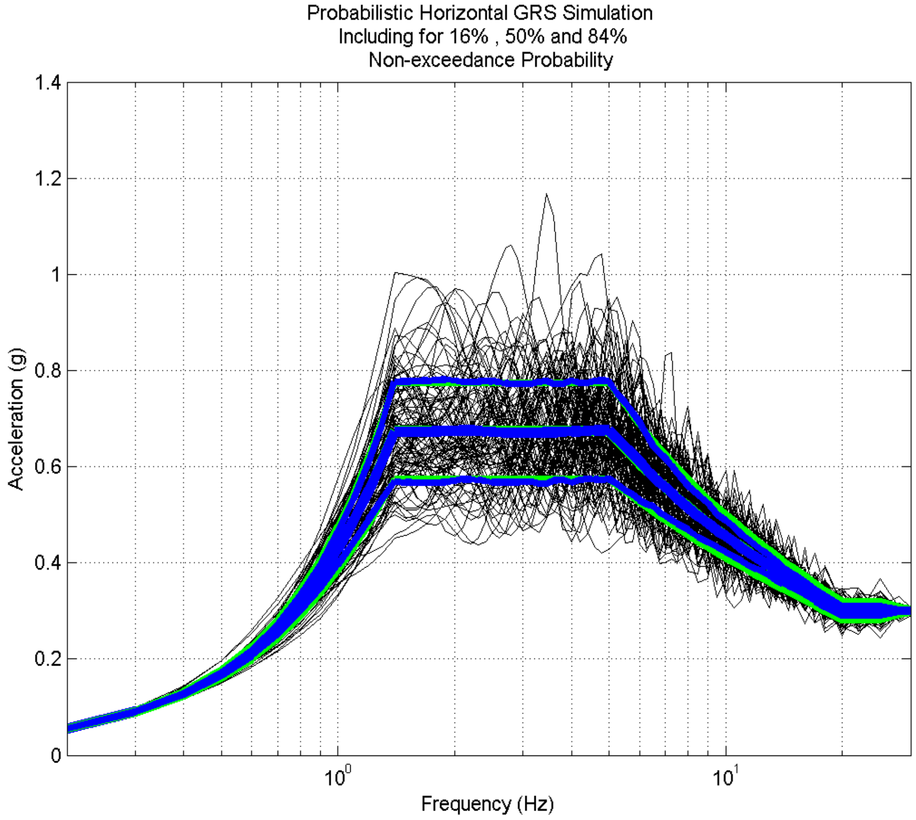
Simulated Soil Profiles



Note: Only 30 LSH simulations were used

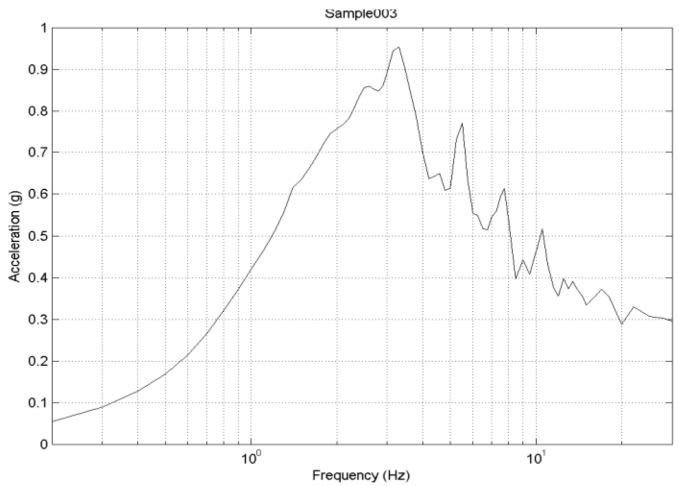
Simulated Probabilistic Seismic GRS (Method 2)

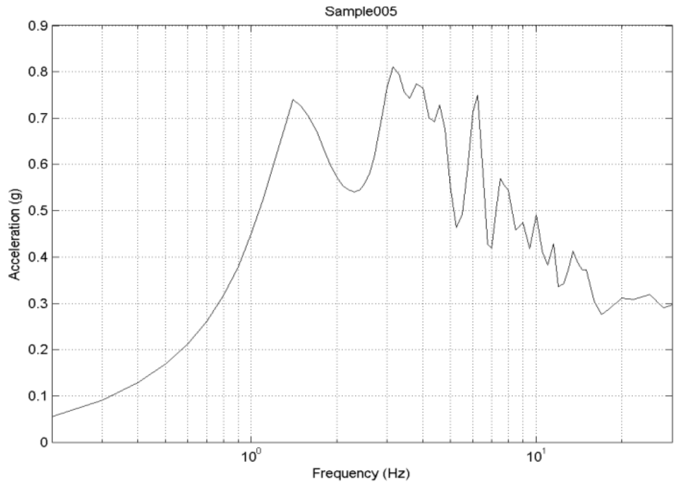
Probabilistic UHRS Input
0.30 ZPGA



c.o.v. = 15%; Correl. Length = 1 Hz

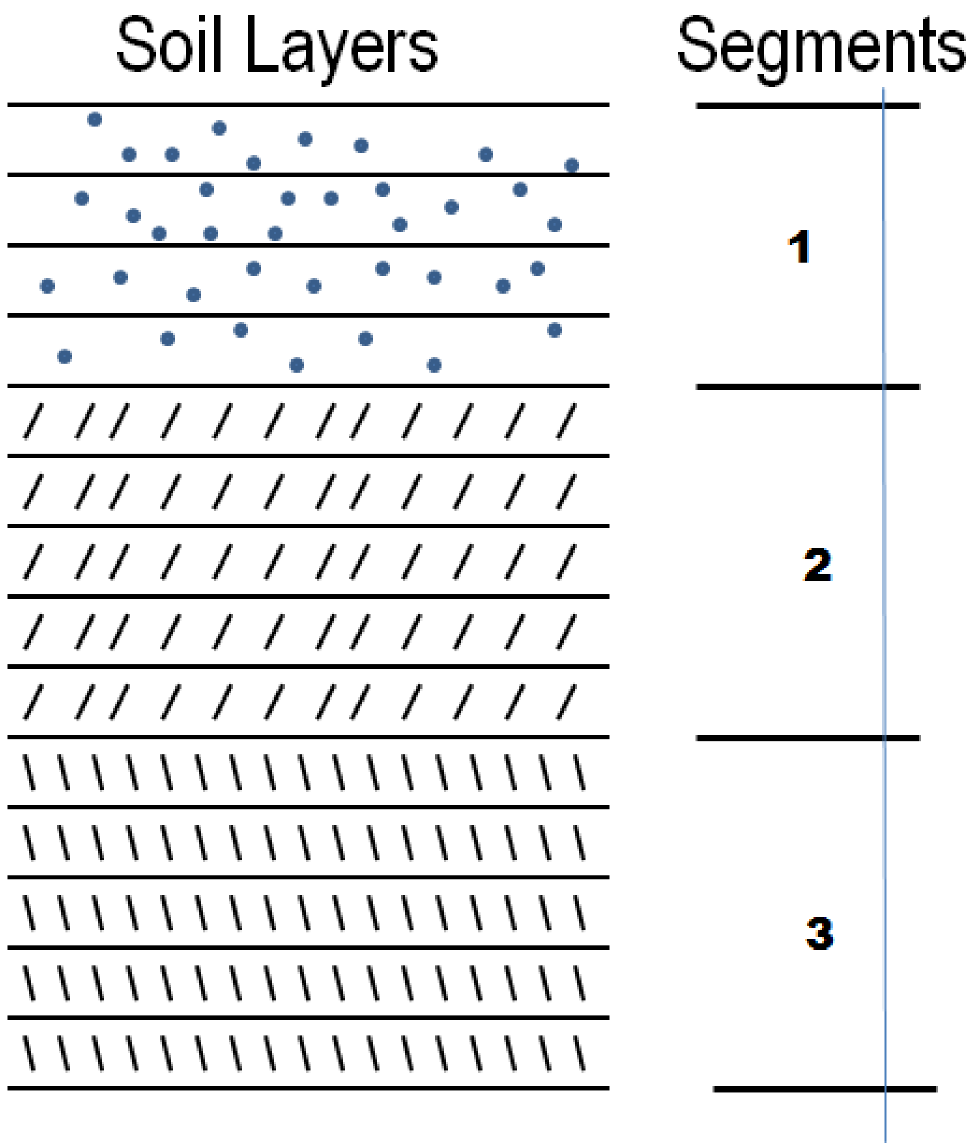
Simulated GRS





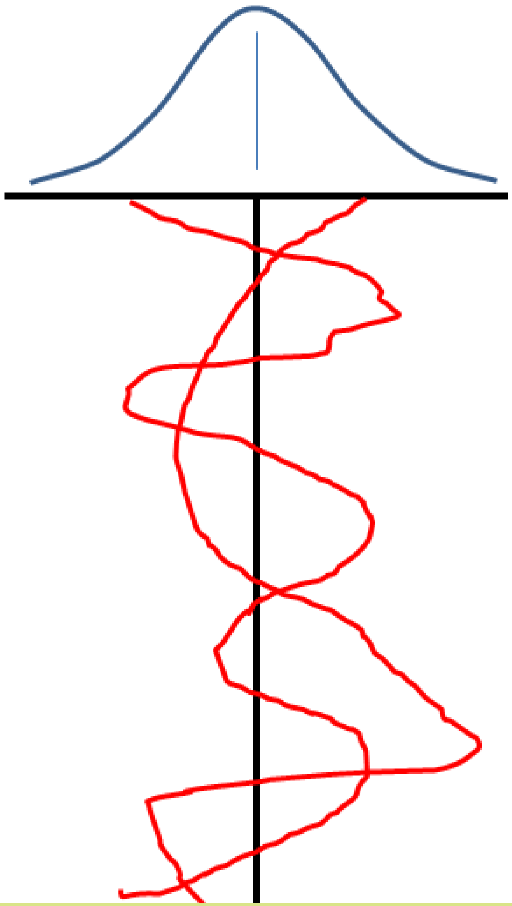
Random Samples

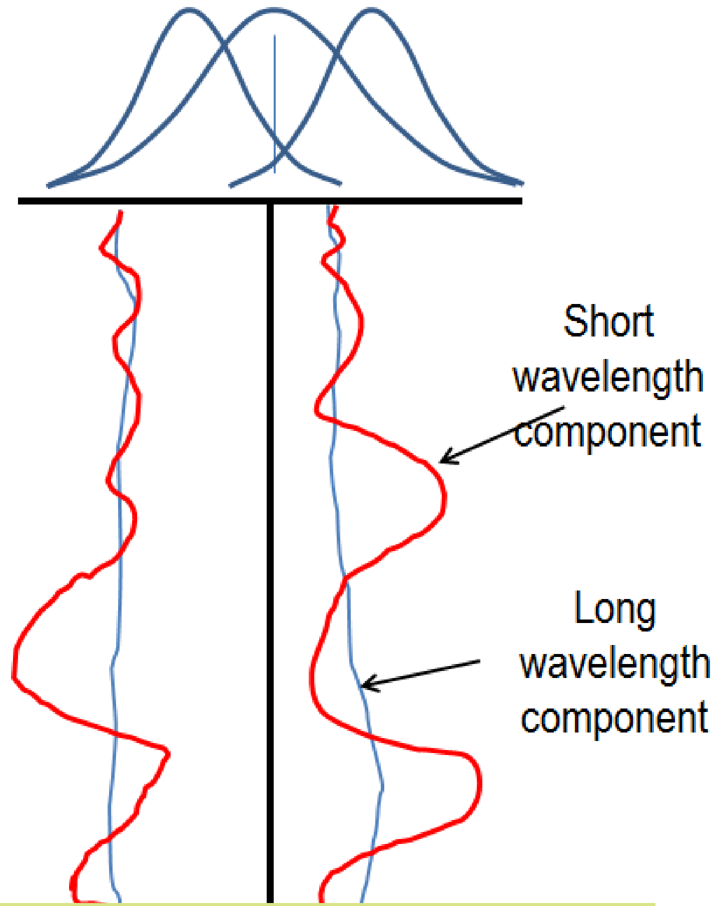
ProSITE Vs and D Soil Profile Probabilistic Models Using Multiple Segments Split



Different statistical properties for different soil profile segments in depth

Vs and D Soil Profile Probabilistic Models. Two Variation Scale Models Based on Field Data





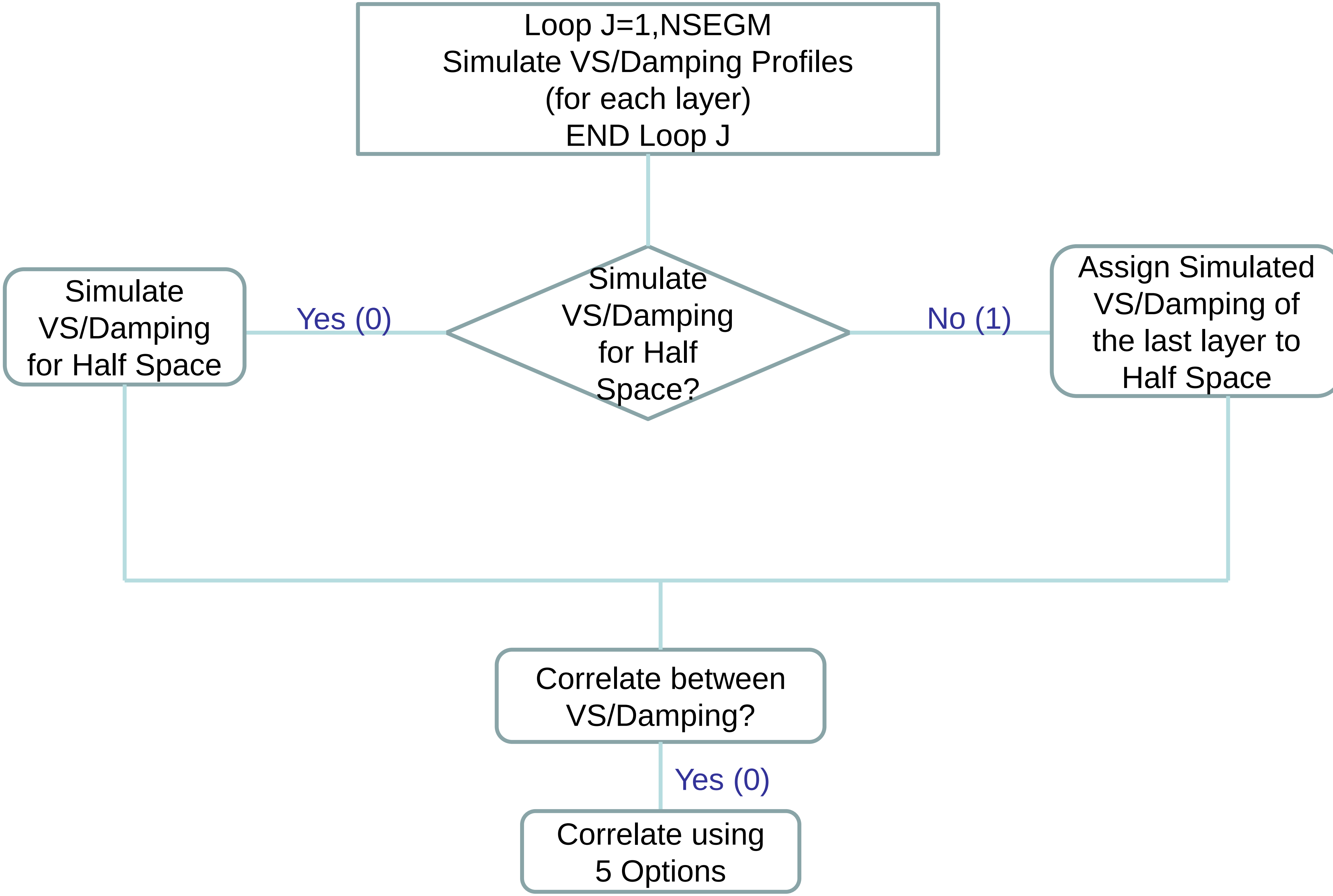
Model 1 (Simple)

1D Random Field Model

Model 2 (Composite)

Mixture of 1D Random Field Model: Two Wavelength Scale Variations

Vs and D Soil Profile Simulation Chart

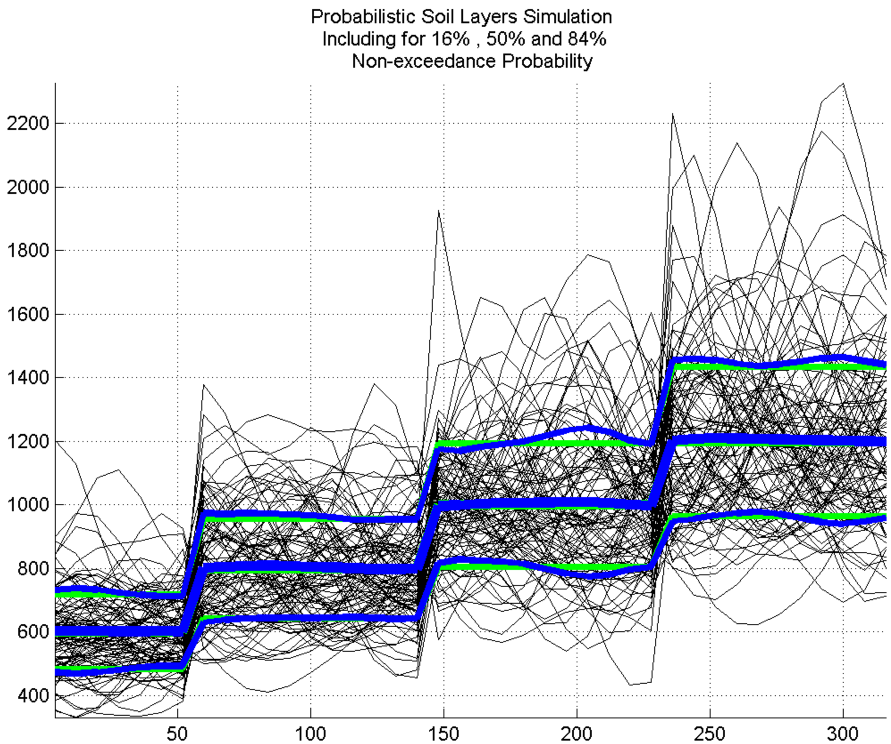


Vs and D Soil Profile Probabilistic Model Inputs

Input File Line Number	Variable Name (Input file in free format)	Definition of Input Variables	Type
1	NSIMUL	Number of simulated seismic input files	Input
1	OPTPDF	Option for probability distribution for Vs and D = 0 for Normal distribution = 1 for Lognormal distribution	Input
1	OPTVDCOR	Statistical dependence between soil layer Vs and D = 0 using a linear correlation coefficient = 1 assuming inverse variation dependence based an equal number of standard variations from mean values = 2 assuming statistical independence = 3 using a give an test-based response function Vs=f(D) = 4 using an inverse probability mapping between Vs and D that builds a statistical response function Vs=f(D)	Input
1	OPTSPCOR	Spatial correlation structure with depth for Vs and D = 0 for constant correlation length with depth (scalar) = 1 for variable correlation length with depth (vector) = 2 for full spatial correlation matrix (matrix)	Input *
1	OPTPROFIL	Soil profile random field models for Vs and D = 0 using 1D random fields = 1 using superposition of two random fields; a long-wavelength and a short-wavelength components	Input **
2	FBASEL	Filename for the mean soil profile (ex. BASELINE.SIT)	Input
3	FGSITE	Filenames for the simulated SITEP/SITE inputs (ex. GSITExxx.sit)	Output
4	NSEGM	Number of the soil profile segments (or number of sets of multiple soil layers above half-space)	Input
4	NTLAYER	Total number of soil layers without accounting for the half-space layer	Input
4	OPTHS	Option for the half-space layer random samples = 0 independent from soil above = 1 full correlated with the soil layer above	Input
4	NTLINE	Total number of lines in the BASELINE.SIT file	Input
Next Line (if OPTHS=0)			
		OPTVDCOR=0,1,2	
5	COVHSVS	If OPTHS=0 Coefficient of variation bedrock Vs	Input
5	COVHSD	If OPTHS=0 Coefficient of variation bedrock D	Input
		OPTVDCOR=3,4	Input
5	COVHSVS	If OPTHS=0 Coefficient of variation bedrock Vs	Input
	Loop J=1,NSEGM	Loop over the number of multi soil layer segments	
		OPTVDCOR=0,1,2	
6+(J-1)	NLAYSEG(J)	Number of layers in the segment (J)	Input
6+(J-1)	COVVS(J)	Coefficient of variation of Vs(J)	Input
6+(J-1)	INSEEDVS(J)	Initial SEED for Vs(J)	Input
6+(J-1)	COVD(J)	Coefficient of variation of D(J)	Input
6+(J-1)	INSEEDD(J)	Initial SEED for D(J)	Input
7 + NSEGM + J-1	CORLVS(J)	If OPTSPCOR=0,1,2,4 Correlation length of Vs(J)	Input
7 + NSEGM + J-1	CORLVS(J)	If OPTSPCOR=0,1,2,4 Correlation length of Vs(J)	Input
7 + NSEGM + J-1	CORMAT(J)	If OPTSPCOR=3 Filename of correlation matrix of Vs(J)	Input
8 + 2*NSEGM + J-1	CORVSD(J)	If OPTSPCOR=0 Filename of the correlation matrix	Input
9 + 2*NSEGM + J-1	COVMEANVS(J)	If OPTSPCOR=1 Coefficient of variation of long-wavelength component Vs profile	Input ***
9 + 2*NSEGM + J-1	COVMEAND(J)	If OPTSPCOR=1 Coefficient of variation of long-wavelength component D profile	Input ***
9 + 2*NSEGM + J-1	CORLLWVS(J)	If OPTSPCOR=1 Correlation length of the long-wavelength component Vs profile	Input ***
9 + 2*NSEGM + J-1	CORLLWD(J)	If OPTSPCOR=1 Correlation length of the long-wavelength component D profile	Input ***
Next 3 NSEGM+ NDATA*NSEGM Lines		OPTVDCOR=3	
6+(J-1)	NLAYSEG(J)	Number of layers in the segment (J)	Input
6+(J-1)	COVVS(J)	Coefficient of variation of Vs(J)	Input
6+(J-1)	INSEEDVS(J)	Initial SEED for Vs(J)	Input
7 + NSEGM + J-1	NDATAFCT(J)	Number of test response function data	Input
7 + NSEGM + J-1	INSEEDFCT(J)	Initial seed for response function noise	Input
	Loop I=1,NDATAFCT(J)	Loop over Vs=f(D)+noise response surface data	
8 + 2*NSEGM + (J-1)+I	VSDAT(J,I)	Response function Vs data points	Input
8 + 2*NSEGM + J-1+I	DDAT(J,I)	Response function D data pints	Input
8 + 2*NSEGM + J-1+I	VSNOISE(J,I)	Response function noise standard deviation	Input
9 + 2*NSEGM + J-1	CORLVS(J)	If OPTSPCOR=0,1,2,4 Half correlation length of Vs(J)	Input
9 + 2*NSEGM + J-1	CORLD(J)	If OPTSPCOR=0,1,2,4 Half correlation length of D(J)	Input
9 + 2*NSEGM + J-1	CORMAT(J)	If OPTSPCOR=3 Filename of correlation matrix of Vs(J)	Input
Next 3 NSEGM Lines		OPTVDCOR=4	Input
6+(J-1)	NLAYSEG(J)	Number of layers in the segment (J)	Input
6+(J-1)	COVVS(J)	Coefficient of variation of Vs(J)	Input
6+(J-1)	INSEEDVS(J)	Initial SEED for Vs(J)	Input
7 + NSEGM + J-1	RSMEANVS(J)	Mean Vs for building response function based on assuming rank correlation = -1	Input
7 + NSEGM + J-1	RSCOVD(J)	Coefficient of variation of Vs for building response function based on assuming rank correlation = -1	Input
8 + 2*NSEGM + J-1	RSMEAND(J)	Mean D for building response function based on assuming rank correlation = -1	Input
8 + 2*NSEGM + J-1	RSCOVD(J)	Coefficient of variation of D for building response function based on assuming rank correlation = -1	Input
9 + 2*NSEGM + J-1	CORLVS(J)	If OPTSPCOR=0,1,2,4 Correlation length of Vs(J)	Input
9 + 2*NSEGM + J-1	CORMAT(J)	If OPTSPCOR=3 Filename of correlation matrix of Vs(J)	Input

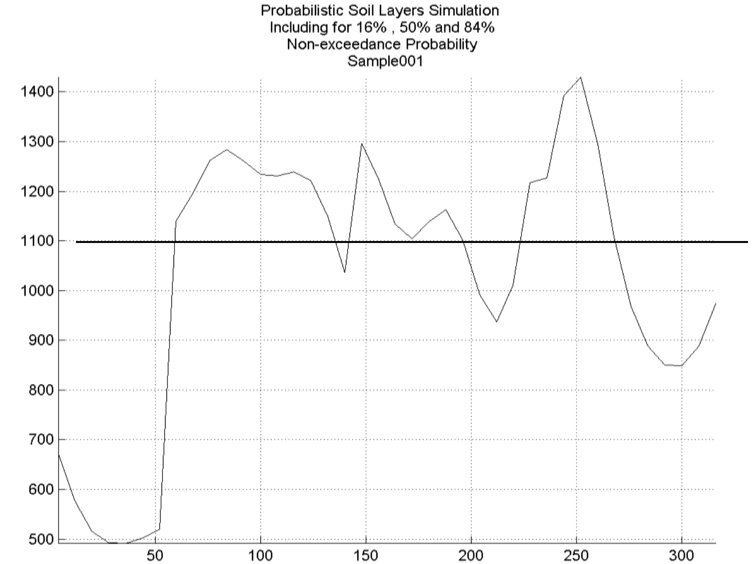
Simulated Probabilistic Soil Layer Profiles

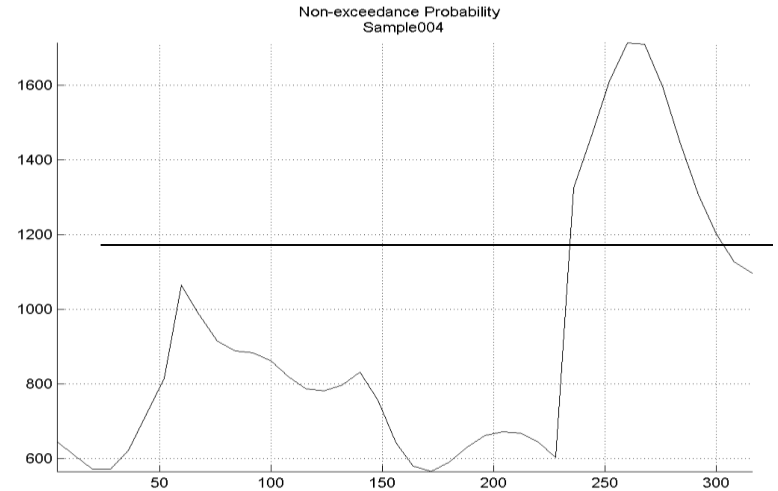
Probabilistic Soil Profile



c.o.v. = 20%; Correl. Length = 20 ft

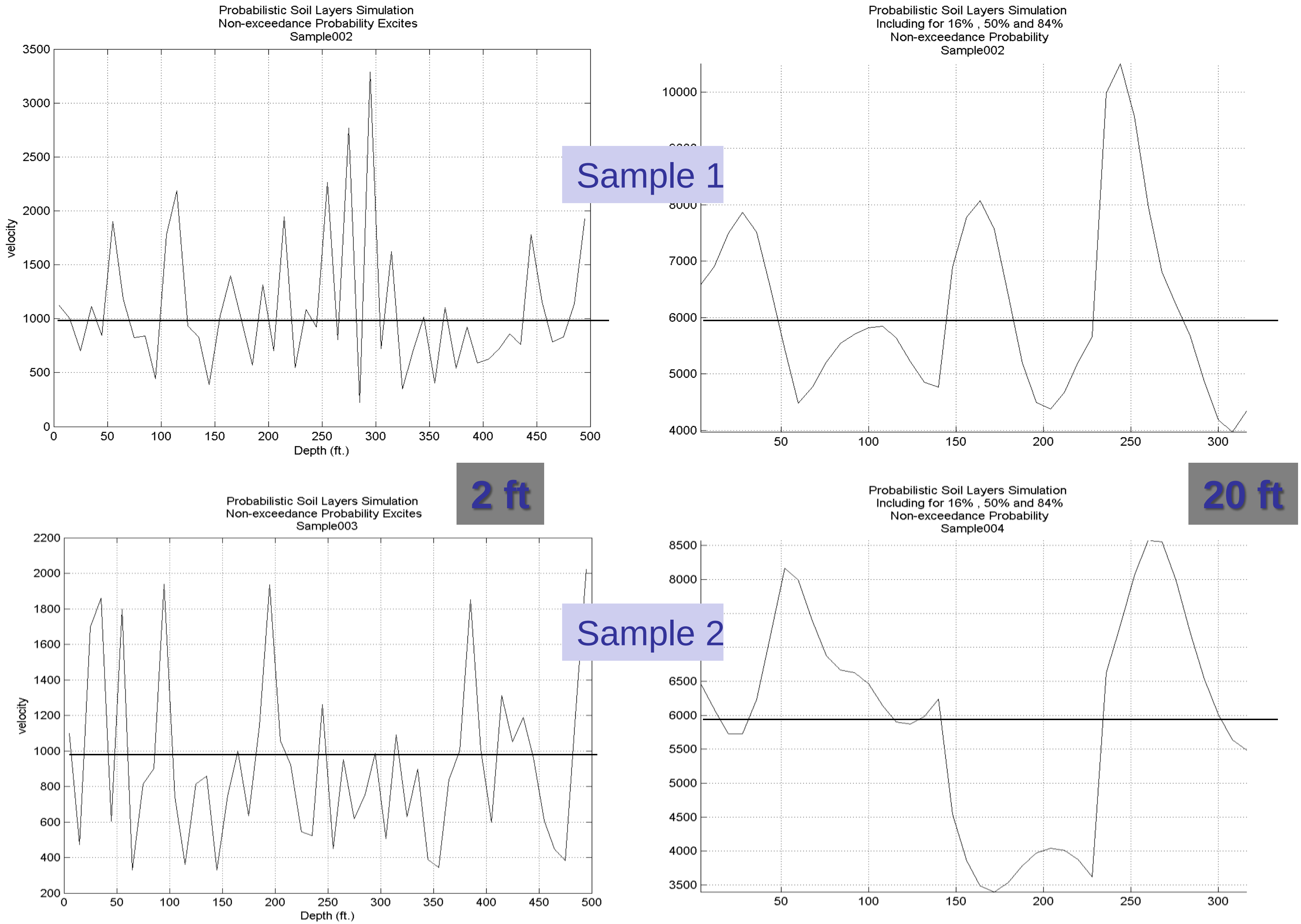
Simulated Soil Profiles





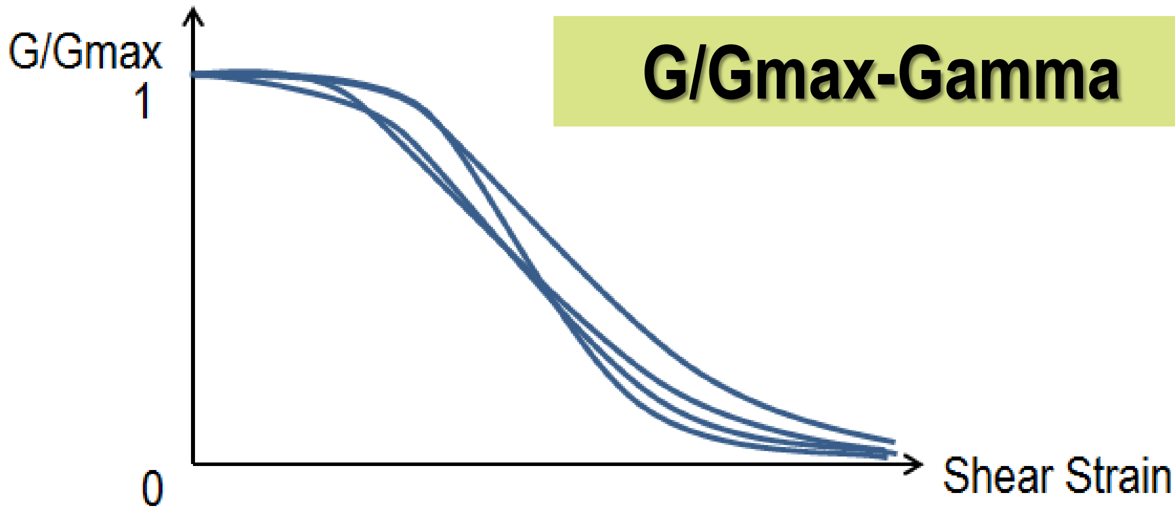
Random Samples

Effect of Spatial Correlation Length on Simulated Soil Profiles

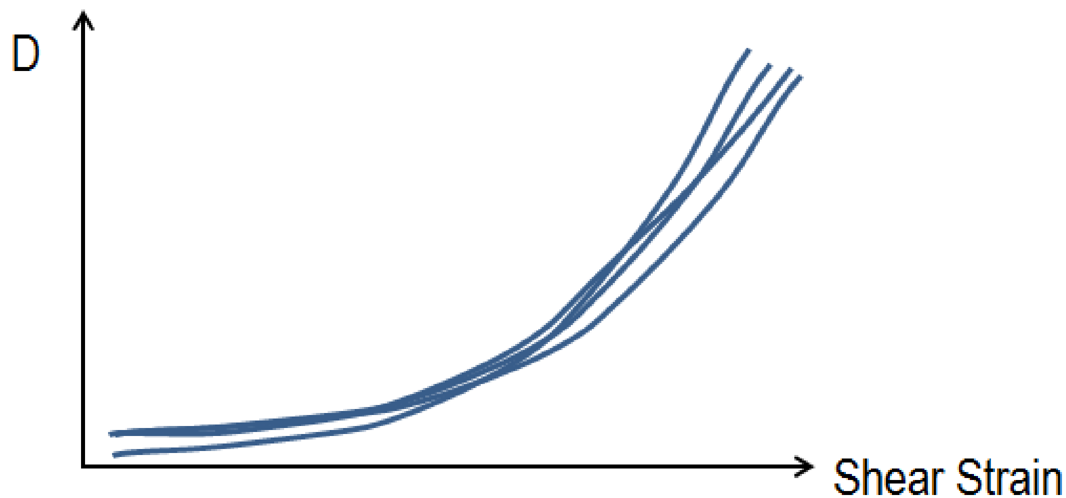


ProSOIL G/Gmax and D Probabilistic Curves.

G/Gmax-Gamma



D-Gamma



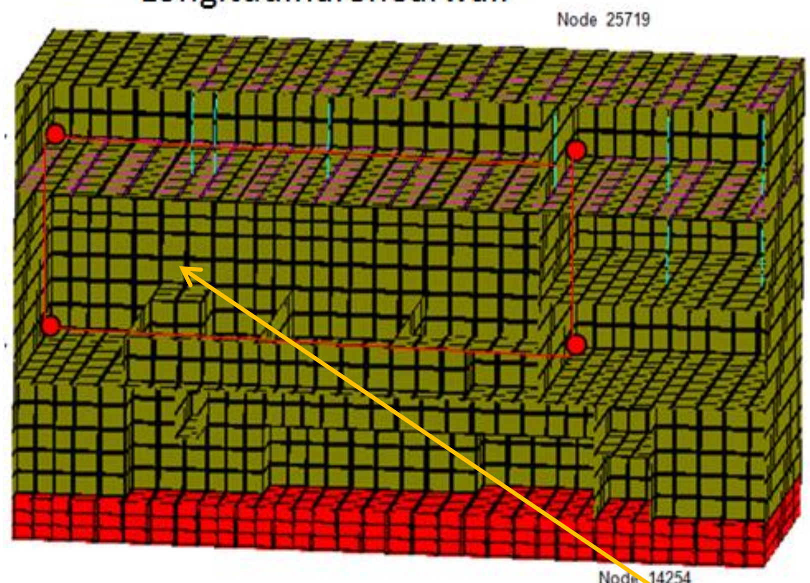
Soil curves assumed with large correlation lengths along the shear strain axis.

ProHOUSE Probabilistic Structural Modeling

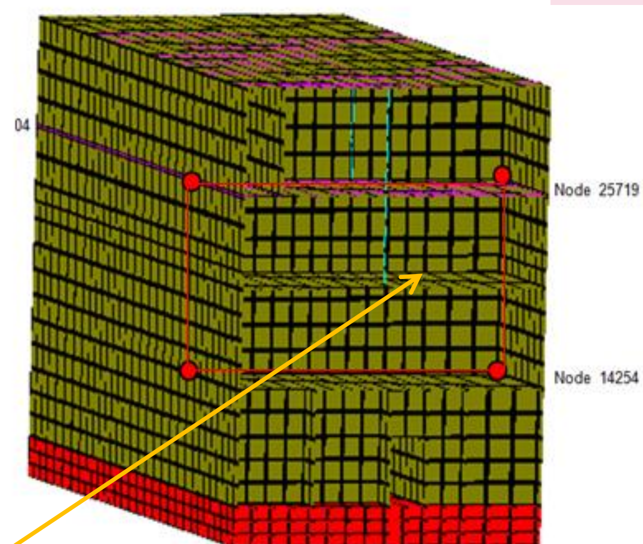
- Effective stiffness ratio Keff/Kelastic and damping ratio, Deff, are modeled as statistically dependent random variables.

- Keff/Kelastic and Deff can be considered negatively correlated, or having a complementary probability relationship, or Deff be a response function of Keff/Kelastic based on experiments

Longitudinal Shearwall



Transverse Shearwall

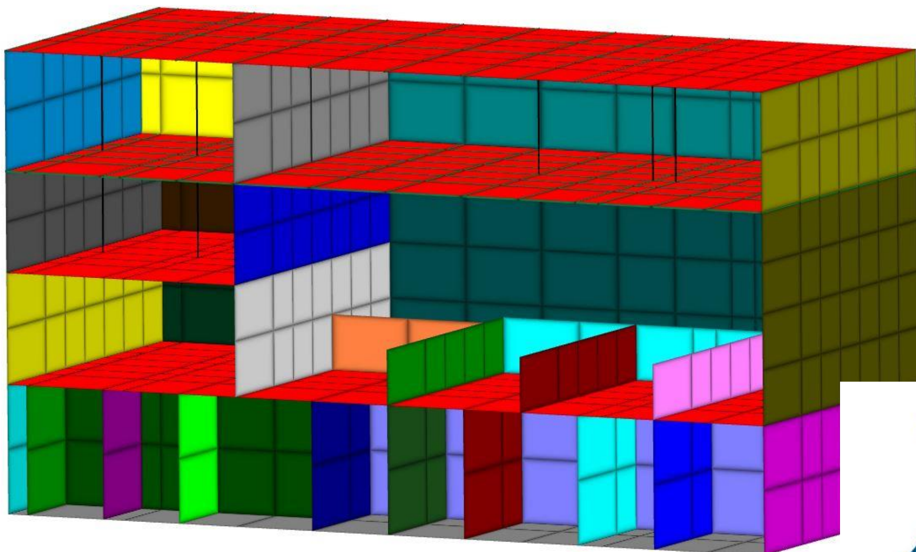


Deff = f (Keff/Elastic)

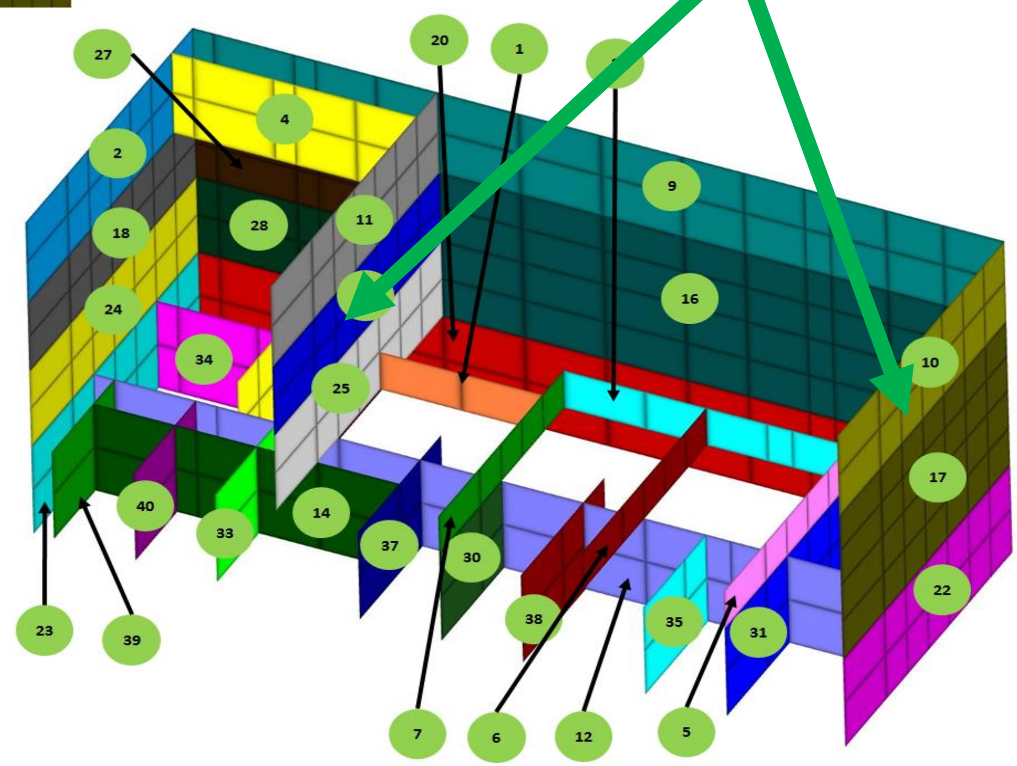
Stiff	Damp
0.01	0.20
0.10	0.17
0.20	0.15
0.30	0.13
0.40	0.11
0.50	0.10
0.60	0.09
0.70	0.08
0.80	0.07
0.90	0.06
1.00	0.05

- Keff and Deff are defined separately for each element group. Statistical correlation between different group Keff variables can be included.

Probabilistic Simulations of Local Structural Stiffness K and Damping D Using Option NON



Using Option NON is highly recommended (after July 15).
In next future Option PRO will include probabilistic BBC inputs.



Nuclear Building Split In Various Equivalent Linear Panels/Groups for PSSIA including Option NON capabilities

End of Part 1

Thank You!