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

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

2015 COPYRIGHT GP TECHNOLOGIES, INC. NOTES FOR ACS SASSI WORKSHOP, TOKYO, JAPAN
Number of SEEDs

Target PSD, if available (NUREG/CR 6728)
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

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

Applicable to generic, Abrahamson models and user-defined, site-specific coherency models.
Alfa factor for directional and radial coherency models

Optional optimal node renumbering for large-size models

Nonuniform motion can include frequency-dependent complex factors (amplitude and phase)

Using directly ANSYS FEA model for SSI in Option AA
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.
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 solution is always accurate.
- Use RELDISP always.
Spline Interpolation Good for Incoherent SSI; Eliminates Overshooting for Low-Frequency RS Peaks

ATF in Y-Direction

ATF in Z-Direction

Overshooting Option 0, 1 or 2
STRESS New GUI Inputs in SUBMODELER

Saves the element center stress, the .ess stress frames
Define Model Number. SUBMODELER can use multiple FE models.
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: MERGEOIL (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.*

```plaintext
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 substructuring approach. 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*

```plaintext
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
SUBMODELER Section-Cut Models

Transverse Wall Cut Using CUTVOL Command

Longitudinal Wall Cut Using CUTVOL Command
* 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\nreadstr,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\n```
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.
*Batch .pre input file of section cut for multiple frame data
actm,0
*Replace Directory Path
inp,demo8.pre,C:\DEMO_PROBLEMS\DEMO8\n * 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,-12.0317,0.0,0.0,1.0,0.0,0.0,1,.005,C:\DEMO_PROBLEMS\DEMO8\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

The Forces on the Cross Section, $Z = -12.0317$

<table>
<thead>
<tr>
<th>Time, second</th>
<th>$F_x$</th>
<th>$F_y$</th>
<th>$F_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.005</td>
<td>-343.777</td>
<td>12065.2</td>
<td>0.229168</td>
</tr>
<tr>
<td>0.01</td>
<td>-268.056</td>
<td>11539.5</td>
<td>0.30395</td>
</tr>
<tr>
<td>0.015</td>
<td>-189.851</td>
<td>10623.5</td>
<td>0.294853</td>
</tr>
<tr>
<td>0.02</td>
<td>-114.229</td>
<td>9320.15</td>
<td>0.227868</td>
</tr>
<tr>
<td>0.025</td>
<td>-44.3936</td>
<td>7645.78</td>
<td>0.144148</td>
</tr>
<tr>
<td>0.03</td>
<td>18.0423</td>
<td>5647.78</td>
<td>0.0908182</td>
</tr>
<tr>
<td>0.035</td>
<td>72.3351</td>
<td>3391.01</td>
<td>0.0894927</td>
</tr>
<tr>
<td>0.04</td>
<td>117.752</td>
<td>967.345</td>
<td>0.124164</td>
</tr>
<tr>
<td>...</td>
<td>...</td>
<td>...</td>
<td>...</td>
</tr>
<tr>
<td>1.97</td>
<td>-676.309</td>
<td>9226.0</td>
<td>0.149393</td>
</tr>
<tr>
<td>1.975</td>
<td>-631.161</td>
<td>8753.78</td>
<td>0.143451</td>
</tr>
<tr>
<td>1.98</td>
<td>-553.34</td>
<td>7862.11</td>
<td>0.158777</td>
</tr>
<tr>
<td>1.985</td>
<td>-450.432</td>
<td>6584.62</td>
<td>0.16545</td>
</tr>
<tr>
<td>1.99</td>
<td>-330.715</td>
<td>4981.56</td>
<td>0.133199</td>
</tr>
<tr>
<td>1.995</td>
<td>-202.2</td>
<td>3116.14</td>
<td>0.0512961</td>
</tr>
<tr>
<td>2</td>
<td>-72.0058</td>
<td>1058.89</td>
<td>0.0630017</td>
</tr>
<tr>
<td>2.005</td>
<td>53.7767</td>
<td>-1132.73</td>
<td>-0.172686</td>
</tr>
</tbody>
</table>

The Moments about Axes on the Cross Section, $Z = -12.0317$

<table>
<thead>
<tr>
<th>Time, second</th>
<th>$M_x$</th>
<th>$M_y$</th>
<th>$M_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.005</td>
<td>12605.2</td>
<td>0.229168</td>
<td>-800746</td>
</tr>
<tr>
<td>0.01</td>
<td>11539.5</td>
<td>0.30395</td>
<td>-5282.46</td>
</tr>
<tr>
<td>0.015</td>
<td>10623.5</td>
<td>0.294853</td>
<td>558887</td>
</tr>
<tr>
<td>0.02</td>
<td>9320.15</td>
<td>0.227868</td>
<td>779248</td>
</tr>
<tr>
<td>0.025</td>
<td>7645.78</td>
<td>0.144148</td>
<td>16700.6</td>
</tr>
<tr>
<td>0.03</td>
<td>5647.78</td>
<td>0.0908182</td>
<td>600032</td>
</tr>
<tr>
<td>0.035</td>
<td>3391.01</td>
<td>0.0894927</td>
<td>593052</td>
</tr>
<tr>
<td>0.04</td>
<td>967.345</td>
<td>0.124164</td>
<td>733816</td>
</tr>
<tr>
<td>...</td>
<td>...</td>
<td>...</td>
<td>...</td>
</tr>
<tr>
<td>1.97</td>
<td>9226.0</td>
<td>0.149393</td>
<td>698463</td>
</tr>
<tr>
<td>1.975</td>
<td>8753.78</td>
<td>0.143451</td>
<td>677218</td>
</tr>
<tr>
<td>1.98</td>
<td>7862.11</td>
<td>0.158777</td>
<td>5783.4</td>
</tr>
<tr>
<td>1.985</td>
<td>6584.62</td>
<td>0.16545</td>
<td>35783.4</td>
</tr>
<tr>
<td>1.99</td>
<td>4981.56</td>
<td>0.133199</td>
<td>21628.8</td>
</tr>
<tr>
<td>1.995</td>
<td>3116.14</td>
<td>0.0512961</td>
<td>184604</td>
</tr>
<tr>
<td>2</td>
<td>1058.89</td>
<td>0.0630017</td>
<td>305474</td>
</tr>
<tr>
<td>2.005</td>
<td>-1132.73</td>
<td>-0.172686</td>
<td>184558</td>
</tr>
</tbody>
</table>

**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 1\textsuperscript{st} step and ANSYS in 2\textsuperscript{nd} step for computing forces and stresses in structure using a more refined structural FEA modeling via ANSYS. The 1\textsuperscript{st} step is the overall SSI analysis that is identical with the analysis above mentioned at item i). The 2\textsuperscript{nd} step uses SSI responses as input BCs. The 2\textsuperscript{nd} step consists in an equivalent (quasi)static stress analysis using a much more refined FE mesh structural model (via ANSYS static analysis). The 2\textsuperscript{nd} 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)

ACS SASSI SSSI Model (1st SSI Analysis Step)

Non-Nuclear NB2 SSI Model

NB1 SSI Model

NB2 SSI Model

Refined ANSYS Model (2nd SSI Analysis Step)

NB2 Stress Analysis
Refined ANSYS Model

Seismic Forces
Foundation displacements

Demos 5, 6
OPTIONS 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

Structural Element Stress

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

Mesh Refinement

Variable Accuracy C-by-C

“Exact”

“Exact”

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

Exported SSI Model → 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

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)

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

[Image of ANSYS Static Load Converter window]

- **Data to Add From ACS SASSI to the ANSYS model**
  - Displacements
  - Acceleration
  - Displacement and Acceleration
  - Displacement for Soil Module

- **Use Multiple File Lists Inputs**

- **SASSI Model and Results Input**
  - Path: F:\ssi_results
  - HOUSE Module Input: solid_box.hou
  - Displacement Results: THD_04.105_00822
  - Trans. Acceleration Results: acc_04.105_00822

- **ANSYS Model and Data Input**
  - Path: F:\ansys_files

- **Mass Data for Intertial Load (Ignore for Displacement)**
  - Mass Type
    - Lumped Mass
    - Master Node Mass
  - Lumped Mass Data: lumped_mass.dat
  - Master Node Mass: master_mass.dat

- **ANSYS Output File**
  - ADPL File: mix_load_822.cmd

[Image of OK and Cancel buttons]
MAIN/LOADGEN Module GUI for Dynamics

Demo 5
OPTION AA: ACS SASSI-ANSYS Interface for SSI Analysis Using ANSYS Models

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.
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)

Surface (Soft Soil), Shell Box with Weighth Density=0, Node=400, X-FRF

- ACS Orig.
- ACS-ANS(ET63)

Surface (Soft Soil), Shell Box with Weighth Density=0, Node=400, X-FRF

- ACS Orig.
- ACS-ANS(ET181)
Using ANSYS Couple Nodes (CP Commands) and ANSYS Constrained Equations (CE Commands) (Pb 46)
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
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)

X Direction Acceleration at Node 1663

Time (Seconds)

Animation
Steps for Running SSI analysis Using ANSYS Model

**Step 1:** ANSYS is used to build FEA Models for Structure and Excavated Soil and produces K, M and C matrices for these.

**Step 2:** ACS SASSI SUBMODELER combines Structure and Excavated Soil after converting their ANSYS .cdb to ACS SASSI .pre format. Generate HOUSEFSA input file (.hou).

**Step 3:** Run HOUSEFSA run of SSI 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 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

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
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%_ana.out >> analys.inp
%bpath%\SITEB.exe < Site.inp
%bpath%\POINT3B.exe < Point.inp
%bpath%\HOUSEFSAB.exe < House.inp
%bpath%\ANALYSFSAB.exe < Analys.inp
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 are 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 absorption 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

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 Wallflt, 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
Wallflt
Mergegroup
Splitgroup

Backbone Curve

Displacement

Force
Nonlinear Building Model Split in Wall Panels

Nuclear Building Split In Nonlinear Panels

Nuclear Building Nonlinear Concrete Panels
Nonlinear Solution Convergence is in 5-6 SSI Iterations Using “New Structure” Restart ANALYS Option

Convergence Error for Panel Displacements

[Graph showing convergence error for panel displacements at 0.3g and 0.6g acceleration levels]
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

Effective Damping

Equivalent Hysteretic Damping Ratio Used for Equivalent Nonlinear, Y Direction 0.6G RG160Y acceleration

Effective Stiffness

Equivalent Elastic Modulus Used for Equivalent Nonlinear, Y Direction 0.6G RG160Y acceleration
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

Panel 17

Panel 19

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

![Graph showing component capacity and mean component response with various limit states (LS-D, LS-C, LS-B, LS-A) and corresponding reduction factors.]

C5-4 Typical Load-Deformation Curve and Limit States

<table>
<thead>
<tr>
<th>Limit State</th>
<th>LS-A</th>
<th>LS-B</th>
<th>LS-C</th>
</tr>
</thead>
<tbody>
<tr>
<td>SMRF reinforced concrete moment frames</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Beams</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$(15 \leq \ell/h)$</td>
<td>5.25</td>
<td>4.0</td>
<td>2.5</td>
</tr>
<tr>
<td>$(\ell/h \leq 10)$</td>
<td>3.25</td>
<td>3.0</td>
<td>2.5</td>
</tr>
<tr>
<td>Columns**</td>
<td>2.0</td>
<td>1.75</td>
<td>1.5</td>
</tr>
<tr>
<td>Reinforced concrete shear wall, in plane:</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bending controlled walls, $\frac{h_w}{\ell_w} \geq 2.0$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$6\sqrt{\frac{f_c}{f_y}} &lt; \frac{f_y}{f_y}$</td>
<td>2.25</td>
<td>2.0</td>
<td>1.75</td>
</tr>
<tr>
<td>$f_y &lt; 3\sqrt{\frac{f_c}{f_y}}$</td>
<td>2.5</td>
<td>2.25</td>
<td>1.75</td>
</tr>
<tr>
<td>Shear controlled walls, $\frac{h_w}{\ell_w} &lt; 2.0$</td>
<td>2.0</td>
<td>1.75</td>
<td>1.5</td>
</tr>
</tbody>
</table>
Inelastic Factors (Fe/Fn) for 0.30g and 0.60g Y-Dir Input

For 0.30g Y-Dir Input:

For 0.60g Y-Dir Input:
4) **RVT Approach for Seismic SSI Analysis**

**RVT Approach Flowchart:**

1. **Seismic GRS Input**
2. **Compute Seismic GPSD**
3. **Compute SSI Response PSD**
4. **Compute SSI Response ARS**

**SDOF Transfer Functions:**

\[ H_0(\omega) = \frac{\omega^2 + 2i\omega_0 \xi_0 \omega}{(\omega_0^2 - \omega^2) + 2i\omega_0 \xi_0} \]

Absolute Accelerations (ARS-APSD)

\[ H_0(\omega) = \frac{\omega}{(\omega_0^2 - \omega^2) + 2i\omega_0 \xi_0} \]

Relative Velocities (VRS-VPSPD)

\[ H_0(\omega) = \frac{1}{(\omega_0^2 - \omega^2) + 2i\omega_0 \xi_0} \]

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

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

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 over 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)

\[ \overline{X}_{\text{max}} = \rho \sigma_X \]
\[ \sigma_{X_{\text{max}}} = q \sigma_X \]


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

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 \( q \)

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

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

\[ \nu_e T = \begin{cases} 
\max(2.1, 2\delta \nu_0 T) & ; 0 < \delta \leq 0.1 \\
(1.63\delta^{0.45} - 0.38) \nu_0 T & ; 0.1 < \delta < 0.69 \\
\nu_0 T & ; 0.69 \leq \delta < 1 
\end{cases} \]

\[ \delta = \sqrt{1 - \frac{\lambda_1^2}{\lambda_0 \lambda_2}} \]
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

Acceleration Method – EPRI AP1000 Stick Model
5% Damping ARS for Mean of 30 Simulations at Basemat Center (Node 1)
Direction Y Rock Site (mean of Vs = 6000 fps)

Direction Y

Basemat

Direction Z

Acceleraion Method – EPRI AP1000 Stick Model
5% Damping ARS for Mean of 30 Simulations at Top of ASB (Node 16)
Direction Y Rock Site (mean of Vs = 6000 fps)

Direction Y

Top of ASB

Direction Z

2015 COPYRIGHT GP TECHNOLOGIES, INC. NOTES FOR ACS SASSI WORKSHOP, TOKYO, JAPAN
RVT Approach (DIS) vs. LHS for Rock – Mean ISRS

Displacement Method – EPRI AP1000 Stick Model
5% Damping ARS for Mean of 30 Simulations at Basemat Center (Node 1)
Direction Y Rock Site (mean of V_s = 6000 fps)

Displacement Method – EPRI AP1000 Stick Model
5% Damping ARS for Mean of 30 Simulations at Basemat Center (Node 1)
Direction Z Rock Site (mean of V_s = 6000 fps)

Displacement Method – EPRI AP1000 Stick Model
5% Damping ARS for Mean of 30 Simulations at Top of ASB (Node 18)
Direction Y Rock Site (mean of V_s = 6000 fps)

Displacement Method – EPRI AP1000 Stick Model
5% Damping ARS for Mean of 30 Simulations at Top of ASB (Node 18)
Direction Z Rock Site (mean of V_s = 6000 fps)

2015 COPYRIGHT GP TECHNOLOGIES, INC. NOTES FOR ACS SASSI WORKSHOP, TOKYO, JAPAN
RVT Approach (ACC) vs. LHS for Soil – Mean ISRS
RVT Approach (DIS) vs. LHS for BE Soil – Mean ISRS

Displacement Method – EPRI AP1000 Stick Model
5% Damping ARS for Mean of 30 Simulations at Basemat Center (Node 1)
Direction Y Soil Site (mean of Vs = 1000 fps)

- Time Domain
- 1 RVT MK-UK
- 1 RVT AD
- 1 RVT AD-DK
- 30 RVT MK-UK
- 30 RVT AD
- 30 RVT AD-DK

Displacement Method – EPRI AP1000 Stick Model
5% Damping ARS for Mean of 30 Simulations at Top of ASB (Node 29)
Direction Y Soil Site (mean of Vs = 1000 fps)

- Time Domain
- 1 RVT MK-UK
- 1 RVT AD
- 1 RVT AD-DK
- 30 RVT MK-UK
- 30 RVT AD
- 30 RVT AD-DK

Displacement Method – EPRI AP1000 Stick Model
5% Damping ARS for Mean of 30 Simulations at Basemat Center (Node 1)
Direction Z Soil Site (mean of Vs = 1000 fps)

- Time Domain
- 1 RVT MK-UK
- 1 RVT AD
- 1 RVT AD-DK
- 30 RVT MK-UK
- 30 RVT AD
- 30 RVT AD-DK

Displacement Method – EPRI AP1000 Stick Model
5% Damping ARS for Mean of 30 Simulations at Top of ASB (Node 29)
Direction Z Soil Site (mean of Vs = 1000 fps)

- Time Domain
- 1 RVT MK-UK
- 1 RVT AD
- 1 RVT AD-DK
- 30 RVT MK-UK
- 30 RVT AD
- 30 RVT AD-DK

2015 COPYRIGHT GP TECHNOLOGIES, INC. NOTES FOR ACS SASSI WORKSHOP, TOKYO, JAPAN
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 be 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 $V_s$ 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.*
Efficient equivalent-linear iterative models can be included for nonlinear soil and structure behavior (after July 15).

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 Analysis Simulations
Using N LHS Samples

Input Files
- .sit
- File88
- .poi
- .hou
- .anl
- .mot
- .str
- STRESS.out
- .TFU
- .TFI
- .RS

Loop j=1,N
- Site X, Y, Z
  - Point
  - House
  - Analysis X, Y, Z
  - Motion X, Y, Z
  - Stress X, Y, Z

Output Files
- File1
- File1X
- File1Y
- File1Z
- File8X
- File8Y
- File8Z
- jth
  - .TFU
  - .RS
  - jth
  - .TFI
  - .out
  - .TFU
  - .TFI
  - .RS
  - .txt

2015 COPYRIGHT GP TECHNOLOGIES, INC. NOTES FOR ACS SASSI WORKSHOP, TOKYO, JAPAN
**ProEQUIAKE Spectral Shape Probabilistic Models**

**ASCE 04-2015 Method 1**

- Coefficient of variation can be defined variable along frequency axis.

**ASCE 04-2015 Method 2**

![Graph showing spectral shape variations with mean and samples]
GRS Spectral Shape Correlation Length Input

**Correlation Length Definition**

- **Inflection Point**

**Constant vs. Variable Correlation Length**

- **Constant Correlation Length**
- **Variable Correlation Length**

Frequency vs. Correlation length

---

2015 COPYRIGHT GP TECHNOLOGIES, INC. NOTES FOR ACS SASSI WORKSHOP, TOKYO, JAPAN
GRS Spectral Shape Correlation Matrix Input. Using Probabilistic Site Response Simulations

**Horizontal**

Incolumn - Nonuniform Soil - 60 Simulations
Elevation 40 ft. - Horizontal

**Vertical**

Incolumn - Nonuniform Soil - 60 Simulations
Elevation 40 ft. - Vertical
# Seismic GRS Probabilistic Model Inputs

<table>
<thead>
<tr>
<th>Input File Line Number</th>
<th>Variable Name (Input in free format)</th>
<th>Definition of Input Variables</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>FRSI</td>
<td>Filenames for the simulated GRS inputs (ex. RS1xx.RS)</td>
<td>Output</td>
</tr>
<tr>
<td>2</td>
<td>FRSO</td>
<td>Filenames for the computed GRS for simulated (ex. RS0xxx.RS)</td>
<td>Output</td>
</tr>
<tr>
<td>3</td>
<td>FACC</td>
<td>Filenames for the computed acceleration histories (ex. ACxoxo.acc)</td>
<td>Output</td>
</tr>
<tr>
<td>4</td>
<td>FGEOQU</td>
<td>Filenames for the simulated EQUAKE inputs (ex. GEQUxxx.equ)</td>
<td>Output</td>
</tr>
<tr>
<td>5</td>
<td>FBASEL</td>
<td>Filename for the seismic input mean GRS amplitude (ex. BASELINE.RSI)</td>
<td>Input</td>
</tr>
<tr>
<td>6</td>
<td>DAMPING</td>
<td>Damping ratio for the mean GRS input (in percent)</td>
<td>Input</td>
</tr>
<tr>
<td>6</td>
<td>GRAVACC</td>
<td>Acceleration ratio of gravity for ground velocity and displacement</td>
<td>Input</td>
</tr>
<tr>
<td>7</td>
<td>DURATION</td>
<td>Duration of simulated acceleration histories (in seconds)</td>
<td>Input</td>
</tr>
<tr>
<td>7</td>
<td>TIMESJEST</td>
<td>Time step of simulated acceleration histories (in seconds)</td>
<td>Input</td>
</tr>
<tr>
<td>8</td>
<td>NSIMUL</td>
<td>Number of simulated seismic inputs for a single direction</td>
<td>Input</td>
</tr>
<tr>
<td>8</td>
<td>NFREQ</td>
<td>Number of frequencies in GMVMEAN.RSI file</td>
<td>Input</td>
</tr>
<tr>
<td>8</td>
<td>INITRA</td>
<td>Initial SEED Random Number for RS1xx.RSI simulation</td>
<td>Input</td>
</tr>
<tr>
<td>8</td>
<td>INITACC</td>
<td>Initial SEED Random Number for ACCxxx.ACC simulation</td>
<td>Input</td>
</tr>
<tr>
<td>9</td>
<td>OPTMETH</td>
<td>Option for the Method Used for GRS Simulation</td>
<td>Input</td>
</tr>
<tr>
<td>9</td>
<td>DIR</td>
<td>Selected Input Direction:</td>
<td>Input</td>
</tr>
<tr>
<td>10</td>
<td>F1</td>
<td>1st Frequency for calculation of the c.o.v. factor (in Hz)</td>
<td>Input *</td>
</tr>
<tr>
<td>10</td>
<td>F2</td>
<td>2nd Frequency for calculation of the c.o.v. factor (in Hz)</td>
<td>Input *</td>
</tr>
<tr>
<td>11</td>
<td>OPTCOR</td>
<td>Option for GRS random shape correlation structure:</td>
<td>Input</td>
</tr>
<tr>
<td></td>
<td></td>
<td>= 0 for frequency-independent correlation length (scalar)</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>= 1 for frequency-dependent correlation length (vector)</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>= 2 for full correlation matrix for GRS frequencies (matrix)</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>COV</td>
<td>Coefficient of variation of the GRS amplitude</td>
<td>Input</td>
</tr>
<tr>
<td>12</td>
<td>SIGMA</td>
<td>For OPTCOR = 0 is half of correlation length</td>
<td>Input</td>
</tr>
<tr>
<td>12</td>
<td>SIGMAV</td>
<td>For OPTCOR = 1 is half of correlation length vector</td>
<td>Input</td>
</tr>
<tr>
<td>12</td>
<td>CORRMAT</td>
<td>For OPTCOR =2 is the correlation matrix file name</td>
<td>Input</td>
</tr>
</tbody>
</table>
Simulated Probabilistic Seismic GRS (Method 1) and Soil Profile (Vs and D) Using Random Variables

Simulated GRS Inputs

Simulated Soil Profiles

(Full correlation with depth)

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

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
Simulate VS/Damping for Half Space?

Loop J=1,NSEGM
Simulate VS/Damping Profiles (for each layer)
END Loop J

Simulate VS/Damping for Half Space?

Yes (0)  Correlate between VS/Damping?

Yes (0)  Correlate using 5 Options

No (1)  Assign Simulated VS/Damping of the last layer to Half Space

Simulate VS/Damping for Half Space?
# Vs and D Soil Profile Probabilistic Model Inputs

<table>
<thead>
<tr>
<th>Input File Line Number</th>
<th>Variable Name</th>
<th>Definition of Input Variables</th>
<th>Type</th>
<th>Input/Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>NSIMUL</td>
<td>Number of simulated seismic input files</td>
<td>Input</td>
<td>Input</td>
</tr>
<tr>
<td>1</td>
<td>OPTPDF</td>
<td>Option for probability distribution for Vs and D</td>
<td>J-1</td>
<td>Input</td>
</tr>
<tr>
<td>1</td>
<td>OPTVDCOR</td>
<td>Statistical dependence between soil layer Vs and D</td>
<td>0 using a linear correlation coefficient</td>
<td>Input</td>
</tr>
<tr>
<td>1</td>
<td>OPTSPCOR</td>
<td>Spatial correlation structure with depth for Vs and D</td>
<td>J-1</td>
<td>Input</td>
</tr>
<tr>
<td>1</td>
<td>OPTPRFIL</td>
<td>Soil profile random field models for Vs and D</td>
<td>J-1</td>
<td>Input</td>
</tr>
<tr>
<td>2</td>
<td>FBASEL</td>
<td>File name for the mean soil profile (ex. BASELINE.SIT)</td>
<td>Input</td>
<td>Input</td>
</tr>
<tr>
<td>3</td>
<td>FGSITE</td>
<td>File name for the simulated SITEP/SITE inputs (ex. GSITExxx.SIT)</td>
<td>Input</td>
<td>Output</td>
</tr>
<tr>
<td>4</td>
<td>NSEGMD</td>
<td>Number of the soil profile segments (or number of sets of multiple soil layers above half-space)</td>
<td>Input</td>
<td>Input</td>
</tr>
<tr>
<td>4</td>
<td>NTLSR</td>
<td>Total number of soil layers without accounting for the half-space layer</td>
<td>Input</td>
<td>Input</td>
</tr>
<tr>
<td>4</td>
<td>OPThS</td>
<td>Option for the half-space layer random samples</td>
<td>J-1</td>
<td>Input</td>
</tr>
<tr>
<td>4</td>
<td>NTLINE</td>
<td>Total number of lines in the BASELINE.SIT file</td>
<td>Input</td>
<td>Input</td>
</tr>
</tbody>
</table>

# Input Lines

<table>
<thead>
<tr>
<th>Input Line</th>
<th>Variable Name</th>
<th>Definition of Input Variables</th>
<th>Type</th>
<th>Input/Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>7 + NSEGMD</td>
<td>7 + NSEGMD</td>
<td>Number of correlation length of Vs(J)</td>
<td>Input</td>
<td>Input</td>
</tr>
<tr>
<td>7 + NSEGMD</td>
<td>7 + NSEGMD</td>
<td>Number of correlation length of D(J)</td>
<td>Input</td>
<td>Input</td>
</tr>
<tr>
<td>8 + 2*NSEGMD</td>
<td>8 + 2*NSEGMD</td>
<td>Number of correlation length of Vs(J)</td>
<td>Input</td>
<td>Input</td>
</tr>
<tr>
<td>8 + 2*NSEGMD</td>
<td>8 + 2*NSEGMD</td>
<td>Number of correlation length of D(J)</td>
<td>Input</td>
<td>Input</td>
</tr>
</tbody>
</table>

# Output Lines

<table>
<thead>
<tr>
<th>Output Line</th>
<th>Variable Name</th>
<th>Definition of Input Variables</th>
<th>Type</th>
<th>Input/Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 + 2*NSEGMD</td>
<td>9 + 2*NSEGMD</td>
<td>Number of correlation length of Vs(J)</td>
<td>Input</td>
<td>Input</td>
</tr>
<tr>
<td>9 + 2*NSEGMD</td>
<td>9 + 2*NSEGMD</td>
<td>Number of correlation length of D(J)</td>
<td>Input</td>
<td>Input</td>
</tr>
</tbody>
</table>

# Additional Inputs

- **OPTVDCOR**: Spatial correlation structure with depth for Vs and D
- **OPTSPCOR**: Spatial correlation structure with depth for Vs and D
- **OPTPRFIL**: Soil profile random field models for Vs and D
- **FBASEL**: File name for the mean soil profile (ex. BASELINE.SIT)
- **FGSITE**: File name for the simulated SITEP/SITE inputs (ex. GSITExxx.SIT)
- **NSEGMD**: Number of the soil profile segments (or number of sets of multiple soil layers above half-space)
- **OPThS**: Option for the half-space layer random samples
- **NTLSR**: Total number of soil layers without accounting for the half-space layer
- **NTLINE**: Total number of lines in the BASELINE.SIT file
Effect of Spatial Correlation Length on Simulated Soil Profiles

![Graphs showing the effect of spatial correlation length on simulated soil profiles.](image)

Sample 1

Sample 2
ProSOIL G/Gmax and D Probabilistic Curves.

Soil curves assumed with large correlation lengths along the shear strain axis.
ProHOUSE Probabilistic Structural Modeling

- Effective stiffness ratio $K_{eff}/K_{elastic}$ and damping ratio, $D_{eff}$, are modeled as statistically dependent random variables.

- $K_{eff}/K_{elastic}$ and $D_{eff}$ can be considered negatively correlated, or having a complementary probability relationship, or $D_{eff}$ be a response function of $K_{eff}/K_{elastic}$ based on experiments.

- $K_{eff}$ and $D_{eff}$ are defined separately for each element group. Statistical correlation between different group $K_{eff}$ variables can be included.

$D_{eff} = f \left( \frac{K_{eff}}{K_{elastic}} \right)$

- $K_{eff}$ and $D_{eff}$ are defined separately for each element group. Statistical correlation between different group $K_{eff}$ 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!