

# **ACS SASSI**

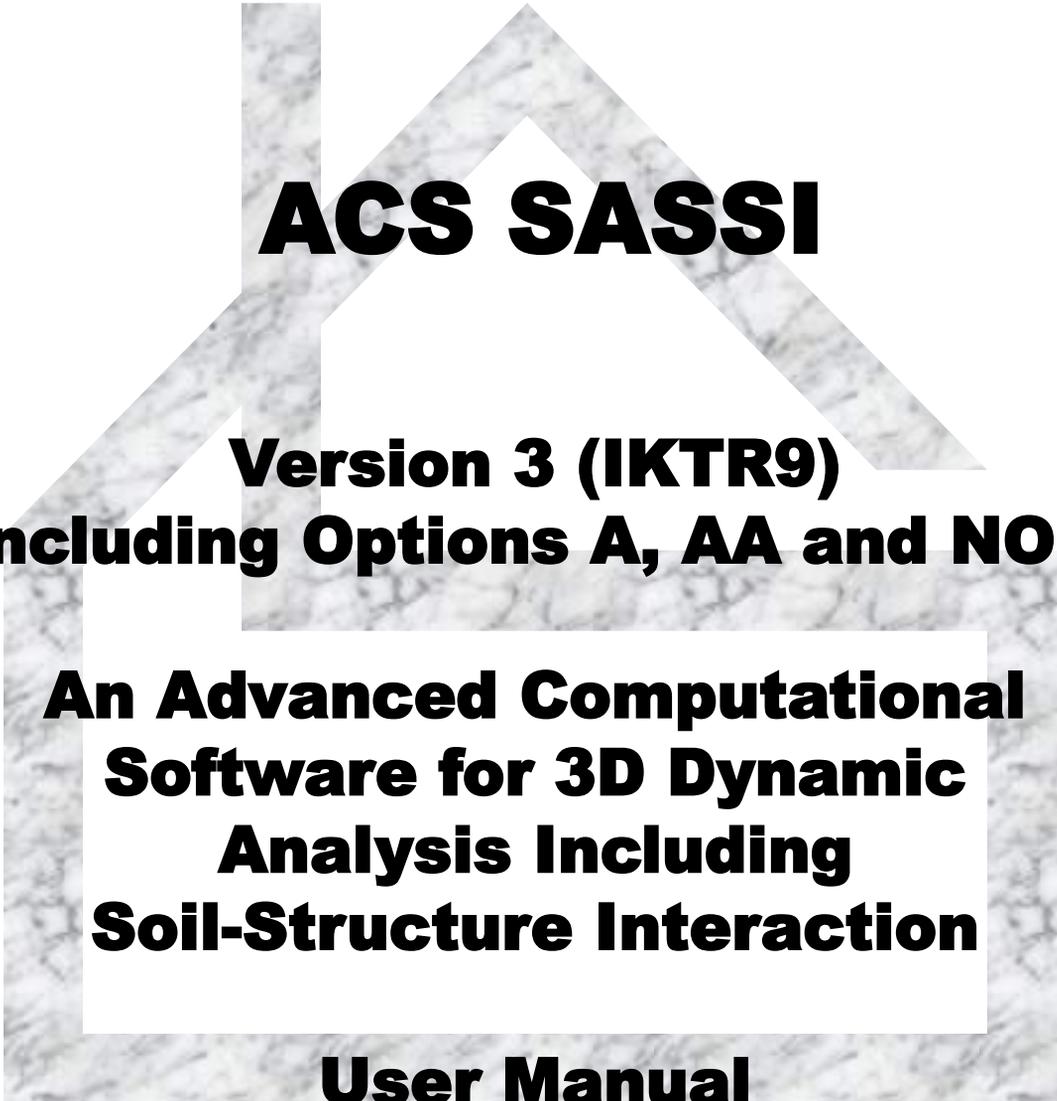
**Version 3 (IKTR9)  
Including Options A, AA and NON**

**An Advanced Computational  
Software for 3D Dynamic  
Analysis Including  
Soil-Structure Interaction**

**User Manual  
Revision 8**

**November 25, 2017**

**Ghiocel Predictive Technologies, Inc.  
4 South Main St., 3<sup>rd</sup> Floor, Pittsford, NY 14534, USA  
Phone: (585) 641-0379 / Fax: (585) 586-4672  
E-mail: [acs.sassi@ghiocel-tech.com](mailto:acs.sassi@ghiocel-tech.com)**



# **ACS SASSI**

**Version 3 (IKTR9)**

**Including Options A, AA and NON**

**An Advanced Computational  
Software for 3D Dynamic  
Analysis Including  
Soil-Structure Interaction**

**User Manual  
Revision 8**

# DISCLAIMER

GHIOCEL PREDICTIVE TECHNOLOGIES, INC. DOES NOT WARRANT THE OPERATION OF THE ACS SASSI VERSION 3 PROGRAM WILL BE UNINTERRUPTED OR ERROR-FREE. GHIOCEL PREDICTIVE TECHNOLOGIES, INC. MAKES NO REPRESENTATIONS OR WARRANTIES, EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. Ghiocel Predictive Technologies, Inc., in any case shall not be liable for any costs, damages, fees, or other liability, or for any direct, indirect, special, incidental, or consequential damages (including loss of profits) with respect to any claim by LICENSEE or any third party on account of or arising from this License Agreement or use the ACS SASSI Version 3 program.

The ACS SASSI Version 3 (Installation Kit Revision 9, IKTR9) includes two installation options for the same software:

i) IKTR9 for SSI models with up to 100,000 nodes. The ACS SASSI Version 3 IKTR7 was tested for 3D SSI models with up to 99,999 nodes and 33,000 interaction nodes on typical 192 GB RAM workstations running under MS Windows 7, 8 or 10 OS.

ii) IKTR9\_650K for SSI models with up to 650,000 nodes or 2,500,000 degrees of freedom. The ACS SASSI Version 3 IKTR9\_650K was tested for 3D SSI models with up to 640,000 nodes and up to 35,000 interaction nodes on typical 128 to 512 GB RAM workstations running under MS Windows 8 and 10 OS.

.

# Table of Contents

<b>1</b>	<b>INTRODUCTION .....</b>	<b>1</b>
1.1	DESCRIPTION OF ACS SASSI BASIC CAPABILITIES.....	5
1.2	SPECIFIC SSI MODEL SIZE RESTRICTIONS.....	10
1.3	MODULAR STRUCTURE CONFIGURATION .....	12
1.4	FINITE ELEMENT LIBRARY.....	13
1.5	MODELING CAPABILITIES AND LIMITATIONS.....	14
1.6	USER DATA INPUT .....	31
1.7	DATA CHECKING .....	32
<b>2</b>	<b>THEORETICAL BASIS FOR PERFORMING LINEARIZED SSI ANALYSIS .....</b>	<b>33</b>
2.1	FLEXIBLE VOLUME (FV) SUBSTRUCTURING .....	33
2.2	SITE RESPONSE ANALYSIS.....	36
2.3	IMPEDANCE ANALYSIS .....	36
2.4	STRUCTURAL ANALYSIS .....	37
2.5	SUMMARY OF COMPUTATIONAL STEPS .....	37
<b>3</b>	<b>DESCRIPTION OF SSI MODULES.....</b>	<b>39</b>
3.1	SEISMIC INPUT, SITE RESPONSE AND SSI ANALYSIS MODULES .....	39
3.2	PERFORMING SSI INITIATION AND RESTART ANALYSES .....	52
<b>4</b>	<b>APPLICATION GUIDELINES.....</b>	<b>55</b>
4.1	SSI ANALYSIS PROCEDURE .....	55
4.2	ACS SASSI RUNS.....	70
<b>5</b>	<b>MODEL CONSTRUCTION AND MANIPULATION.....</b>	<b>74</b>
5.1	Creating A New SSI Model .....	74
5.2	Loading and Saving Models.....	75
5.3	Using the Database.....	75
5.4	Without the Database .....	76
5.5	Transferring Files between Previous PREP and New UI.....	76
5.6	Macros.....	77
5.7	Merging Models.....	80
5.8	Submodeling and Calculations.....	81
5.9	Variables and Loops .....	83
<b>6</b>	<b>ACS SASSI USER INTERFACE (UI) MODULE MENUS AND OPERATIONS .....</b>	<b>85</b>
6.1	Model Menu.....	88
6.2	File Submenu.....	96
6.3	Plot Submenu .....	98
6.4	Modules Submenu .....	99
6.5	Options Submenu .....	113
6.6	View Submenu.....	187
6.7	Help Submenu .....	191
<b>7</b>	<b>PLOTTING OPERATIONS AND EXAMPLES.....</b>	<b>192</b>
7.1	Three-dimensional Plots .....	192
7.2	Two-dimensional Plots .....	199
7.3	Animated Plots .....	208
<b>8</b>	<b>TOOLBAR REFERENCE .....</b>	<b>215</b>
8.1	Main Toolbar .....	215
8.2	3D Plot Toolbar.....	216
<b>9</b>	<b>COMMAND REFERENCE .....</b>	<b>217</b>
9.1	Previous PREP Commands Included in New User Interface.....	217
9.2	General Commands Compatible with Previous PREP Versions.....	217
9.3	Node Commands.....	238
9.4	Element Commands.....	243

---

9.5	Load Commands .....	256
9.6	ACS SASSI User Interface Commands .....	260
9.7	ACS SASSI User Interface General Commands.....	260
9.8	Model Checking Commands .....	270
9.9	Model Generation & Combination Commands .....	271
9.10	Cut & Submodeling Commands .....	276
9.11	File Conversion Commands.....	280
9.12	Module Commands .....	283
9.13	Calculation Commands .....	286
9.14	Plot Commands .....	292
9.15	Programming Commands .....	308
9.16	Water Modeling Commands.....	313
9.17	Nonlinear & Panel Commands.....	314
9.18	Binary Commands .....	329
9.19	Thick Shell Commands.....	335
10	<b>CHECK ERRORS AND WARNINGS.....</b>	<b>336</b>
10.1	Overview .....	336
10.2	Errors.....	336
10.3	Warnings .....	351
11	<b>REFERENCES.....</b>	<b>353</b>

---

# 1 Introduction

The ACS SASSI software is a software in a continuous fast development. In addition to the computational speed improvements through a full parallelization of the code that makes it hundreds of times faster than the university SASSI code, ACS SASSI includes a totally new set SSI analysis capabilities that extends the linearized SASSI methodology to nonlinear soil and structure and probabilistic SSI problems including the effects of incoherent seismic wave fields. Please note that the ACS SASSI fast-solver is about hundreds times faster than the standard university SASSI2000 code.

*ACS SASSI Version 3* is a state-of-the-art highly specialized finite element computer code for performing 3D linear and non-linear soil-structure interaction (SSI) analysis for shallow, embedded, deeply embedded and buried structures under coherent and incoherent earthquake ground motions. The ACS SASSI software is an extremely user friendly, modern engineering software under MS Windows with a unique suite of SSI engineering capabilities. ACS SASSI is equipped with two translators for converting inputs of structural finite element models from ANSYS (CBD file) (ANSYS is a trademark of ANSYS Inc.) or university SASSI2000 (fixed format input files) to ACS SASSI, and also from ACS SASSI to ANSYS (APDL input file format). ACS SASSI uses an automatic management of all data resources, files, directories, and interconnections between different software modules. ACS SASSI can be run interactively for a single SSI model or batch for single and multiple SSI models.

In addition to software manuals, the most recent ACS SASSI Version 3 upgrades after IKTR4, have a supplementary “Help” support (that is not a part the NQA version documentation) that is available online if the user selects the “Help” option from the new ACS SASSI user interface (UI) menu bar. A number of twelve interactive demos are available to guide the users understand and use the main functionalities of the ACS SASSI code including Options A-AA and NON. Each demo has a detailed description and guidance on the inputs and on how to perform SSI analysis using specific functionalities. It is recommended these demos to be approached in ascending order from Demo 1 to Demo 12, also depending on the functionalities of interest for the analyst. In the demos, it is also shown how to use input Macros for post-processing, and some special commands to select new frequencies or select frames for animations of the contour plots of the nodal stresses in the structure.

The latest upgrade of the ACS SASSI Version 3 (Installation Kit Revision 9, IKTR9) described in this manual includes two installer kit options for the same software: i) IKTR9 for SSI models with up to 100,000 nodes and ii) IKTR8\_650K for SSI models with up to 650,000 nodes or 2,500,000 degrees of freedom. The ACS SASSI Version 3 IKTR8\_650K was tested for 3D SSI models with up to 625,000 nodes and up to 35,000 interaction nodes on 128 GB to 512 GB RAM workstations running under MS Windows 8 and 10. It is intentioned that the SSI model size limitation to 650,000

---

nodes to be relaxed from time to time in future, as the MS Windows workstation RAM and HDD resources continue increase.

The minimum hardware requirement is a 8-16 GB RAM workstations under the MS Windows 7, 8 or 10 OS that include a video card capable of supporting the OpenGL 4.0 libraries or higher for the 3D plotting and animations. For large size embedded SSI models with many interaction nodes as 15,000-20,000 interaction nodes, or even more, we suggest have the use of 192 GB to 512 GB RAM MS Windows 10 workstations.

It should be noted that for the large-size SSI models with more than 99,999 nodes, the integer fields were extended for the fixed format input files produced by the AFWRITE command for the FORCE, HOUSE, MOTION RELDISP and STRESS modules. However, the ACS SASSI UI .pre input file is not affected in anyway by the SSI model size.

It should be noted that the latest upgrades of the ACS SASSI Version 3 can run simultaneously up to tens or hundreds of dynamic load cases for a small runtime increase. Thus, for flexible foundation impedance problems that requires a large number of unit amplitude harmonic loads, ACS SASSI can run up to 500 separate load cases in a single run, depending on the size of the problem and the available RAM. Also, for seismic incoherent SSI analysis problems, ACS SASSI can run up to 50 stochastic simulations including X, Y and Z components for each in a single SSI analysis run, depending on the size of the problem and the available RAM. *These new capabilities of running many load cases in a single run makes the latest upgrade of ACS SASSI Version 3 to be at tens of time faster than the previous 2015 ACS SASSI Version 3 IKTR4 for running multiple load cases or incoherent SSI simulations.*

For seismic SSI coherent SSI analyses, ACS SASSI does not need any SSI restart analysis for the three-directional seismic input components, since it can solve the X, Y and Z input cases in the internal memory without the need to save the large restart files for each frequency. Restart files can be still useful for “New Structure” and “New Seismic Environment” restart analyses, such as nonlinear SSI analysis runs, or incoherent SSI analysis simulations.

The last upgrades of ACS SASSI Version 3, after IKTR4, include a totally new ACS SASSI User Interface (UI) that is much more capable than the previous version UI, including the MAIN, PREP and SUBMODELER modules available in the ACS SASSI Version 3 IKTR4. The new UI includes all the PREP and SUBMODELER commands of the previous versions, and also many new commands and powerful graphical capabilities for plotting SSI models and results. The new UI commands offer a much larger number of SSI modeling options than before.

In addition to the ACS SASSI main software SSI capabilities for performing deterministic linearized SSI analysis, there are number of companion, separate software modules that include additional advanced SSI capabilities. These advanced SSI capabilities are provided in the *Option A-AA* (ACS SASSI-ANSYS integration that provides a refined integrated two-step approach in

---

accordance with new ASCE 4-16 standard Section 4.6, *Option PRO* (probabilistic site response and probabilistic SSI analyses per new ASCE 4-16 standard recommendations in Sections 2 and 5.5 and in compliance with USNRC RG1.208, and *Option NON* (nonlinear structure to simulate the reinforced concrete cracking per new ASCE 4-16 standard Section 3.3 and USNRC SRP 3.7.2, and the post-cracking reinforced concrete behavior per ASCE 43-05 standard and new ASCE 43-16 draft, and USNRC SRP 3.7.2), or simulate the seismic base-isolators hysteretic behavior per new ASCE 4 standard Section 4.7 and 12.4). These advanced SSI analysis capability options are briefly described in the next paragraphs.

*The ACS SASSI-ANSYS integration capability included in Options A-AA covers an area of great interest for practical applications. This capability provides an advanced two-step SSI approach that can include more refined FEA structural models in the second step, including some local nonlinear material and/or nonlinear geometric effects in the structure or at foundation interface with the soil. There are two ACS SASSI-ANSYS interfacing options: i) Option "A" or "ANSYS" and ii) Option "AA" or "Advanced ANSYS". Demo problems are provided to help users understand how to best use the ACS SASSI-ANSYS interface using Options A and AA.*

i) *Option A is an ACS SASSI-ANSYS interfacing capability is based on an integrated two-step SSI approach, the 1<sup>st</sup> step is the overall SSI or SSSI analysis using the ACS SASSI SSI model and the 2<sup>nd</sup> step is the structural stress analysis using a refined ANSYS model with the input boundary conditions defined based on the SSI responses from ACS SASSI. Option A includes a fast automatic export of the seismic SSI boundary conditions for performing a detailed nonlinear or linear ANSYS stress SSI analysis using either quasi-static or dynamic models, or computing soil pressure on the foundation baseslabs and walls including soil separation effects. The 2<sup>nd</sup> step can have two distinct functionalities: i) perform structural stress analysis using refined ANSYS FEA structural models with detailed meshes, eventually including enhanced element types, non-linear material and plasticity effects, contact and gap elements, and ii) compute seismic soil pressure on the basement walls and slabs including the soil material plasticity, foundation soil separation and sliding using refined ANSYS soil models. Option A assumes that any nonlinearity included in the ANSYS model does not affect the SSI soil motion of the foundation-soil interface.*

ii) *Option AA in an Advanced ACS SASSI-ANSYS integration capability that permits to run SSI analysis using directly state-of-the-art ANSYS FE structural models. The Option AA ANSYS models could include advanced ANSYS FE types, pipes, shells including shear flexibility, coupled nodes, constraint equations, MPC elements, and even fluid elements (FLUID80) and the super-elements (MATRIX50). The Option AA uses directly the ANSYS FE model dynamic matrices with no need for the ANSYS FE model conversion into an ACS SASSI FE model. Only the model topology from the ANSYS FE model is transferred to ACS SASSI, but no material or constants, or other parameters. For computing structural stresses, the Option A should be used to transfer*

---

the SSI response motions at all time steps or selected critical steps as boundary conditions for ANSYS superstructure model.

The ANSYS MATRIX50 super-elements can be i) included in the ANSYS model within Option AA analysis, or ii) exported to the ACS SASSI model using an automatic conversion based on the general matrix (GM) elements. The Option A-AA has a separate manual that complements this user manual.

**WARNING:** For the Option AA analysis runs, the *HOUSEFS* and *ANALYSFS* modules (without the Option AA capability) should be replaced by the *HOUSEFSA* and *ANALYSFSA* modules. This can be accomplished with the ACS SASSI User Interface (UI) using the “Modules/Location” input selection for changing the default paths of the executable fields which were created during the software installation.

iii) *Option NON or nonlinear structure SSI analysis* uses a fast hybrid time-complex frequency approach based on a piece-wise equivalent-linearization for the nonlinear structure SSI analysis (Ghiocel et. Al, 2017). The ACS SASSI nonlinear SSI analysis is extremely fast, at least hundreds of times faster than the time-domain nonlinear SSI approaches based on numerical integration.

Option NON is applicable to the reinforced concrete structures for simulating the concrete cracking and post-cracking behavior in the low-rise shearwalls for the design-level and/or beyond-the-design-level seismic inputs. The Option NON was validated for the low-rise reinforced concrete shearwall buildings that fail primarily due to the in-plane shear deformation. Based on the time-domain hysteretic behavior, the elastic modulus and damping in each concrete wall are modified iteratively based on the local stress and deformation levels. No out-plane nonlinear concrete behavior is considered. However, Option NON can also consider the nonlinear concrete behavior due to the in-plane bending deformation effects. In the same nonlinear structure FE model, the analyst can include wall panels (parts of walls) that fail primarily either due to the shear deformation or the bending deformation, respectively. This has an useful practicality.

Option NON can be also used to include the effects of the hysteretic behavior of the seismic base-isolators using nonlinear spring elements. The nonlinear springs also might be used to simulate nonlinear pile-soil interface behavior, or even to identify the potential of the structure sliding on the ground including SSI effects. Option NON is validated for nonlinear translational springs in the three X, Y and Z global directions. Nonlinear rotational springs are not included at this time.

iv) *Option PRO for probabilistic site response analysis (PSRA) and SSI analysis (PSSIA)* using efficient LHS simulations. Option PRO is consistent with probabilistic site response and SSI procedures in the new ASCE 4-16 standard (Sections 2 and 5) and the USNRC guidance for computing the FIRS for the new site-specific licensing applications (Ghiocel, 2017).

---

The Option PRO probabilistic modeling includes: i) Response spectra shape model for the seismic motion input, ii) Soil shear wave velocity  $V_s$  and hysteretic damping  $D$  profiles, defined for each soil layer for low shear strain values, iii) Soil shear modulus  $G$  and hysteretic damping  $D$ , as random functions of the soil shear strain values for each soil layer, and iv) Equivalent linear values for the effective structural stiffness and damping for each group of elements depending on the stress levels in different parts of the structure. The Option PRO has a separate user manual. No information on Option PRO is provided in this manual.

The combined use of Options PRO and NON offers highly efficient, robust and accurate SSI solutions for the design-basis analysis and/or the seismic PRA/PSA fragility calculations. Using combined *Option PRO-Option NON*, the structural engineers will have the required SSI analysis tools to compute much more realistically the structural fragilities. Thus, the nonlinear reinforced concrete post-cracking behavior structures on ground could be efficiently evaluated in a business practical manner and not neglected, or crudely represented, as quite often was done in the past due to the lack of efficient nonlinear SSI analysis tools.

The ACS SASSI UI ensures the automatic administration of all resources, files, directories, and the interfaces between software modules. Comparing with previous releases in 2015, the ACS SASSI Version 3 upgrades in 2016 have a much more powerful User Interface (UI) that includes many new commands, parametric language macros, much faster post-processing for calculations, plotting and animations. Before discussing the new ACS SASSI UI module in detail, the key ACS SASSI SSI modeling and analysis capabilities are reviewed in next section.

## **1.1 DESCRIPTION OF ACS SASSI BASIC CAPABILITIES**

The latest ACS SASSI Version 3 SSI capabilities incorporate many advanced algorithms and specialized features. In comparison with the standard SASSI methodology, the ACS SASSI incorporates many additional SSI capabilities and specialized features, in addition to its much faster computational speed:

(i) *Generation of three-component input acceleration time histories compatible with a given design ground response spectrum* with or without time-varying correlation between the components. The user has also the option to generate acceleration histories using the complex Fourier phasing of selected acceleration records (called “seed records” in the new ASCE 4-16 standard). The software provides baseline correction and computes PSD and peak ground accelerations, velocities and displacements to be used by the analyst to check the US NRC SRP 3.7.1 requirements for the simulated ground accelerations.

(ii) *Evaluation of the seismic motion incoherency and wave passage effects.* ACS SASSI Version 3 includes state-of-the-art modeling including both isotropic (radial) and anisotropic (directional) incoherency models. Both stochastic and deterministic incoherent SSI approaches could be employed for simple stick models with rigid basemats. These incoherent SSI approaches

were validated by EPRI (Short et al., 2007) for stick models with rigid basemats, and accepted by US NRC (ISG-01, May 2008) for application to the new NPP seismic analysis. ACS SASSI includes six incoherent SSI approaches, namely, two simplified deterministic approaches that are the AS and SRSS approaches benchmarked by EPRI (Short et al., 2007), three other alternate deterministic approaches, and a rigorous stochastic simulation approach that is called “Simulation Mean” approach included in the 2006-2007 EPRI validation studies.

There are seven plane-wave incoherency models that can be used: the Luco-Wong model, 1986 (theoretical, not validated model), five Abrahamson models (empirical, isotropic or anisotropic, based on the statistical dense array records) and user-defined incoherency models. The Abrahamson models include the coherency models published in 1993, 2005 (all sites, surface foundations), 2006 (all sites, embedded foundations), 2007a (rock sites, all foundations), 2007b (soil sites, surface foundations) and user-defined coherency models. The recent ACS SASSI versions include directional or anisotropic Abrahamson coherency models in addition to the isotropic or radial Abrahamson coherency models included in earlier releases and also used in the 2007 EPRI studies.

The user-defined coherence functions are useful for particular sites for which more detailed seismological information is available, or for sensitivity studies including the effect of soil layering inclination or nonuniformity in horizontal plane. They are based on specific-site response data, eventually coming from the 2D nonlinear site response analysis using the equivalent-linear iterative approach for selected soil profile slices. The slice directions should in principle correspond to the two principal orthogonal directions of the soil layer slopes, namely for the maximum and minimum soil layer slopes. Thus, the user-defined coherence functions are usually different in the two selected orthogonal horizontal directions. For the incoherent SSI analysis using refined FE models with elastic foundations, we highly recommend the use of the stochastic simulation approach that includes no intrusion in the SSI system dynamics. The AS and SRSS deterministic approaches are simplified incoherent SSI approaches that should be applied only to rigid basemat stick models, as validated by the 2007 EPRI studies (Short et al., 2007). The use of AS or SRSS to elastic foundation FE models might not be necessarily appropriate since could produce considerably biased results, mostly overly conservative, but sometime unconservative, especially for the vertical direction for which the foundation basemats are much more flexible, as stated also in the new ASCE 4-16 standard.

The SRSS approach is more difficult to apply since it has no clear convergence criteria for the required number of the incoherent spatial modes to be considered for the incoherent SSI analysis. For flexible foundations, the number of required incoherent spatial modes required to reconstruct the free-field coherency matrix could be very large, in order of several tens or even hundreds of modes on a case-by-case basis. This makes SRSS impractical and even dangerous for FE elastic foundation problems.

**WARNING:** *The SRSS approach was implemented in ACS SASSI only for benchmarking purposes, since it was validated by EPRI for stick models with rigid basemats rather than for their*

---

*practicality. It should be also understood that the SRSS approach could provide incoherent responses that are overly conservative in the mid-frequency range, sometime higher than coherent responses, but potentially non-conservative in the high-frequency range. We do not recommend the use of the SRSS approaches for the design-basis nuclear projects which use FE SSI models with elastic foundations.*

(iii) *Nonlinear hysteretic soil behavior capability is included in the ACS SASSI main software for seismic SSI analysis using the Seed-Idriss iterative equivalent linear procedure for both the global (due to wave propagation in free-field) and the local soil nonlinearity (due to SSI effects). The nonlinear soil site response is included in SOIL module based on SHAKE methodology. For 2D and 3D SSI models, the local soil nonlinear behavior can be included using near-field soil PLANE or SOLID elements (defined in the HOUSE model input). The ACS SASSI code uses a fast SSI reanalysis (or restart) solutions for the equivalent-linear soil SSI iterations that takes advantage of the already computed soil impedance matrix available from the SSI initiation run. This feature reduces the run time per SSI iteration by a factor of 2 to 5 times depending of the foundation embedment size. For nonlinear soil SSI analyses performed in batch mode, the simultaneous X-Y-Z input effects can be considered at each SSI iteration using the COMB\_XYZ\_STRAIN auxiliary program.*

(iv) *Nonlinear hysteretic structure behavior capability is included in Option NON. The iterative equivalent-linearization can be applied for modeling the cracking and post-cracking reinforced concrete wall behavior using shell elements, or for modeling the rubber-bearing hysteretic materials for the seismic base-isolation using nonlinear spring elements. The nonlinear springs can be also applied for modeling the local pile-soil interface nonlinear effects due to pile slipping in the vertical direction that can be an important aspect during intense seismic motions.*

(v) *The nonuniform soil motion, or multiple seismic input motion option, includes the capability to consider variable amplitude seismic input motions. The nonuniform motion input is applicable to continuous foundations assuming that the free-field motion complex amplitude varies in the horizontal plane after specific frequency dependent spectral patterns. These patterns are described by the user using complex amplification factors (or relative transfer functions) computed with respect to the reference amplitude motion. The nonuniform motion assumption could be combined with motion incoherency and wave passage to create more realistic seismic environments.*

(vi) *Seven interpolation schemes for the complex responses are implemented for complex responses. The newer interpolation scheme that uses bi-cubic splines is recommended for complex FE models under incoherent seismic inputs (for which the number of SSI frequencies should be usually larger than 200). For such cases, when number of frequencies is sufficiently large, the bi-cubic spline interpolation provides most accurate results for incoherent analysis since it does not create any spurious peaks or valleys (option = 6). The bi-cubic spline interpolation should be applied only if the number of SSI frequencies is sufficiently large, so that spectral peaks are not clipped by the smooth spline interpolation. Different interpolation techniques could perform*

---

differently on a case-by-case basis, especially for highly complex FE models with coupled responses, especially for incoherent motions. The various interpolation options that are available in the code provide the structural analyst a set of powerful tools for identifying and avoiding the occurrences of spurious spectral peaks in the computed transfer functions of structural motions and stresses. The first six options (options 0-5) were implemented in the original SASSI 1982 scheme that uses a non-overlapping moving window, the university SASSI2000 scheme that uses a weighted average moving window, and four new interpolation schemes including two non-overlapping window schemes with different shifts and two average overlapping moving window schemes with different numbers of sliding windows. To check the interpolation accuracy, convenient comparative plots of the computed TFs versus the interpolated TFs can be easily obtained using the GUI graphics.

(vii) *The new Fast Flexible Volume (FFV) method* provides accurate and numerically efficient SSI analysis solutions for deeply embedded structures (DES) such as small modular reactors (SMRs). The FFV method, in addition to the interaction nodes defined at the outer surface of excavation volume, includes interaction nodes defined by internal node layers within excavation volume. Using the INTGEN command, the user can automatically generate the interaction nodes for the FFV method or other methods, as FV and FI-FSIN (SM) or FI-EVBN (MSM). The FFV method speeds the SSI analysis of deeply embedded structures by up to tens of times faster than the traditional, reference FV method.

(vii) *Automatic selection of additional SSI calculation frequencies* that are required to improve the accuracy of the interpolated TF that is applicable to both the node acceleration TFs (ATF) and the element stress TFs (STF). This feature that can be implemented using UI commands and macros, is an important practical capability, especially for larger size FE model applications, because it saves a lot of labor effort and ensures high quality of the SSI analysis.

(viii) *Visualization of complex TF variation patterns* within the entire structural model for selected, SSI calculation frequencies. The complex TF patterns are visualized on the structure using colored *vector plot* animations including all three-directional components (red for X, green for Y and blue for Z). The TF amplitude is given by vector length, and the TF phase is given by vector orientation. This capability is extremely useful for checking the correctness of the FE modeling and understanding the structural dynamic behavior.

(ix) *Computation and visualization of the amplitude TF or spectral accelerations* for a selected damping value at a given SSI calculation frequency for the entire SSI model using either structural deformed shape or *bubble plots*. The deformed shape plots are animated structural plots with a controlled movie frame speed, so that they can be also viewed as static plots. For selected resonant frequencies, the spectral amplitudes or the ZPA values can be plotted as a deformed shape plot.

---

(x) *Computation and visualization of structural acceleration and relative displacement time histories using structural deformed shape plots.* The deformed shape plots can be static structural plots for selected times, or maximum values, or structural animations of the SSI response variation in time during the earthquake action.

(xi) *Computation and contour plotting of the average nodal seismic stresses* (for all six components in global coordinates) in the entire structure, or for selected parts of the structure based on the element center stresses for the SHELL and SOLID elements. Both maximum and time-varying values of nodal stresses are computed and available for plotting. The approximation is based on the assumption that element center and node stresses are equal (no shape function extrapolation is included). For sufficiently refined finite element models this approximation appears reasonable. Contour stress plotting can be either static maximum values or animated time-varying values at selected time frames (automatic frame selection is included). Maximum element center stresses values are also available in a convenient text file format.

(xii) *Computation and contour plotting of seismic soil pressure on foundation walls using near-field SOLID elements.* The nodal pressure is computed based on averaging of adjacent element center pressures. Both maximum and time-varying values of nodal seismic pressures are computed and available for plotting. The analyst can also automatically combine the seismic soil pressures with the static soil bearing pressures and then, plot the resultant soil pressure of foundation walls and mat. Contour seismic soil pressure plotting can be either static contour plots of maximum values or animated contour plots of time-varying values at selected time frames (an automatic frame selection capability is included).

(xiii) *Post SSI analysis calculations for superposition of the co-directional SSI effects in terms of acceleration, displacement of stress time-histories and in-structure response spectra* using UI commands and macros. For time histories both the algebraic summation available. For in-structure response spectra (ISRS) post-processing i) the weighted linear combination and ii) the square-root of sum of square (SRSS) for the superposition of the X, Y and Z co-directional effects are implemented. The analyst can also compute the envelope, broaden and average ISRS from multiple spectral curves. Post-processing calculations can be also used for computing the maximum structural stresses, forces and moments, and/or the maximum seismic soil pressure on walls and mat with or without including the soil static bearing pressure component.

The ACS SASSI NQA Version 3 has been tested, verified, documented and released under the Ghiocel Predictive Technologies Nuclear Quality Assurance Program which is in compliance with the requirements of 10 CFR50 Appendix B, 10 CFR21, ASME NQA-1 2008 and Addenda Subpart 2.7 2009. The ACS SASSI NQA version comes with a complete set of software documentations that were developed under the quality assurance requirements of the GP Technologies NQA-1 Level Program. The ACS SASSI NQA version documentation includes the user and verification manuals and the V&V computer files for a large set of various seismic V&V problems, including shallow, embedded and buried foundations, rigid and flexible foundations, piles, subjected to

various different seismic environments, different surface and body seismic waves, motion incoherency and directional wave passage along an arbitrary horizontal direction, multiple support excitations for isolated foundations, linear or nonlinear SSI analysis.

The current ACS SASSI NQA Version 3 IKTR9 includes a set of 56 SSI verification and validation (V&V) problems, many of these including several subproblems. In these V&V problems, the computed SSI results using ACS SASSI are compared against benchmark results based on published analytical solutions or computed using other validated with computer programs, including SHAKE91, SASSI2000 and ANSYS. Each SSI verification problem tests a different capability of the ACS SASSI NQA code. The total number of the V&V computer input files and output files for all the SSI verification problems of the ACS SASSI NQA version is several thousands of files that require about few hundreds of MB of the hard drive space.

## **1.2 SPECIFIC SSI MODEL SIZE RESTRICTIONS**

In current ACS SASSI Version 3 code, the SSI model size limitation is either 100,000 nodes for IKTR9 or 650,000 nodes or 2,500,000 equations for IKTR9\_650K. On the MS Windows PCs with 16GB RAM, SSI problems with sizes up to 100,000 nodes including up to 8,000 interaction nodes can be run efficiently. For the SSI problems including larger-size models with more than 10,000 interaction nodes, up to 20,000 interaction nodes, MS Windows PCs with RAM from 32 GB up to 192 GB are recommended. For very large-size SSI problems with more than 15,000-20,000 interaction nodes, the MS Windows 10 workstations with 512 GB RAM are required.

As a general practical recommendation for the industry applications, we suggest keeping the number of interaction nodes to less than 20,000 interaction nodes, if possible, so that SSI runs can be done relatively quickly without putting too much strain on the project schedule. A straight forward option is to reduce the number of interaction nodes by selecting faster approximate SSI substructuring methods than the reference Flexible Volume (FV) method, such as Flexible Interface (MSM or ESM) or Fast-FV, as described in next sections.

Other maximum SSI model size limitations applicable are:

EQUAKE Module:

- Number of time steps for simulated acceleration histories = 32,768

SOIL Module:

- Number of time steps for simulated acceleration histories = 32,768
- Number of soil material curves = 100
- Number of data for soil curves = 11
- Number of soil layers = 200

SITE Module:

- Number of soil layers = 200

- 
- Number of half-space layers = 20
  - Number of analysis frequencies = 500

## POINT Module:

- Number of soil layers = 200
- Number of half-space layers = 20
- Number of analysis frequencies = 500
- Number of embedment layers = 50

## FORCE Module:

- Number of analysis frequencies = 500

## HOUSE Module:

- Number of nodes = 99,999 nodes for IKTR9  
= 650,000 nodes or 2,500,000 dofs for IKTR9\_650K
- Number of interaction nodes = 100,000 (impractical since it needs about 4-5 TB RAM)
- Number of materials or cross-section geometries = 9,999
- Number of analysis frequencies = 500
- Number of embedment layers = 50
- Number of multiple support (foundation zones) = 5,000
- Number of interaction nodes per embedment layer to run incoherent analysis = 20,0000

## ANALYS Module:

- Number of analysis frequencies = 500

## MOTION Module:

- Number of analysis frequencies = 1500
- Number of time steps or Fourier frequencies = 65,536
- Number of damping values for response spectra calculations = 5

## RELDISP Module:

- Number of analysis frequencies = 1500
- Number of time steps or Fourier frequencies = 65,536

## STRESS Module:

- Number of analysis frequencies = 1500
- Number of time steps or Fourier frequencies = 65,536
- Number of elements per group = 100,000
- Number of element groups per model = 10,000

## NONLINEAR Module:

- Number of nodes = 300,000 nodes
- Number of data points for backbone curves = 100

---

### **1.3 MODULAR STRUCTURE CONFIGURATION**

The configuration of the ACS SASSI Version 3 main program includes a number of eleven distinct SSI analysis interrelated modules, as shown in Figure 1.1. Most of these SSI modules are programmed in advanced FORTRAN languages, except the C++ routines for performing a fast matrix assembly using advanced GPU numerical libraries.

For a linearized SSI analysis the run sequence of the SSI modules is SITE-POINT-HOUSE-ANALYS-MOTION-RELDISP-STRESS. If the user desires to simulate a spectrum compatible accelerations for the SSI analysis input, then, the EQUAKE module should be used. If user desires to perform a nonlinear site response analysis using either the SHAKE methodology based on the equivalent-linearization of the soil hysteretic behavior, or using nonlinear time-domain nonlinear soil model based on the DEEPSOIL theory, he needs to use the SOIL module.

In addition to the basic eleven SSI modules, there are two other modules required for Option A-AA (LOADGEN and SS12ANSYS), and two other modules required for Option NON (NONLINEAR and COMBIN\_XYZ\_THD). All these modules are shown in Figure 1.1. In Figure 1.1, the names of the SSI modules that belong to the Options A-AA and Option NON are in blue letters. The text files produced by different SSI modules are circled with red line, while the binary files are circled with green line. The text files produced by the NONLINEAR modules are not shown in Figure 1.1.

In addition to the modules shown in Figure 1.1 there is another complex software module programmed in VC++ language called ACS SASSI UI that covers completely all the graphical user interface functionalities for building and checking SSI models, performing SSI analysis, review results and post-process results. The post-processing includes many capabilities for handling results and post-processing results either computationally and graphically, including animations. The ACS SASSI UI has also an advanced parametric language that permits creating a variety of macros for efficient pre- or post-processing.

If nonlinear soil is considered, then, the equivalent-linear SSI iterations are based on using repeatedly the run sequence HOUSE-ANALYS (restart)-STRESS (eventually including COMB\_XYZ\_STRAIN run). For nonlinear structure SSI analysis the NONLINEAR module is also required. The nonlinear SSI analysis module run sequence is SITE-POINT-HOUSE-ANALYS-MOTION-RELDISP-COMBINE\_XYZ\_THD-NONLINEAR for the first SSI iteration, and then, HOUSE-ANALYS (restart)-MOTION-RELDISP-COMB\_XYZ\_THD-NONLINEAR for the following SSI iterations.

The Option PRO has a separate user manual.



- 
- |   |                                    |
|---|------------------------------------|
| • 3D thin plate/shell elements (Kirckhoff theory) | type SHELL (4 nodes)               |
| • 3D thick plate/shell elements (Mindlin theory)  | type TSHELL (4 nodes)              |
| • 2D plane strain elements                        | type PLANE (4 nodes)               |
| • 3D spring elements                              | type SPRING (2 nodes)              |
| • 3D general matrix elements (stiffness and mass) | type General Matrix (2 or 3 nodes) |

The excavated soil volume could be modeled using the following finite element types:

- SOLID
- PLANE

The external force loads can be:

- nodal forces
- nodal moments
- nodal translational masses
- nodal rotational masses

## 1.5 MODELING CAPABILITIES AND LIMITATIONS

The ACS SASSI code has the following modeling capabilities and limitations:

### 1.5.1 Soil Deposit, Excavation and Structural Modeling Basic Aspects

1. The site soil layering for SSI analysis consists of semi-infinite viscoelastic horizontal soil layers on a rigid base or a semi-infinite elastic or viscoelastic half-space. The half-space is simulated by additional soil layers added automatically by the SITE module, plus a viscous boundary at the bottom. The non-linear behavior is approximated using the Seed-Idriss iterative equivalent linear procedure.

**WARNING:** *The number of soil layers should be larger than 20 layers. This requirement is important when uniform soil deposits need to be modeled. A small number of soil layers can affect the accuracy of the Rayleigh and Love wave modes computed by SITE. The number of half-space layers that is defined by the user should be 10 to 20 computational layers for accurate results.*

**WARNING:** *The soil layer properties should include the effects of soil hysteretic behavior in the free-field. The soil layer dynamic properties, Vs and Damping, for the seismic SSI analyses should be defined by the linearized, effective, or equivalent soil properties computed using the Seed-Idriss iterative equivalent linear model within the SOIL module, as functions of the soil layer shear strains. The SOIL module uses the SHAKE methodology for simulating the vertically propagating*

---

*wave within horizontal soil layering. Then, based on the user SITE input selection, if ACS SASSI can automatically transmit the SOIL equivalent soil layer properties to the SITE module and the HOUSE module for the excavated soil layers.*

It should be noted that in the recent ACS SASSI upgrades, the SOIL model includes in addition to SHAKE methodology a new special option for the soft soil situations for a time-domain numerical integration solution. The nonlinear time-domain option uses a hyperbolic hysteretic model for soil behavior, similar to the DEEPSOIL software soil model (at this time without including water table effects).

2. The building structures are discretized using standard 2D and 3D finite elements connected at nodal points. Each nodal point on the structure may have up to six displacement degrees of freedom. The user has the freedom to delete one or more of the degrees of freedom thereby reducing the size of the problem accordingly. The FE library includes basic elements such as BEAM, SPRING, PLANE, SOLID, SHELL/TSHELL and General Matrix (GM) elements.

The GM elements can be used to include super-element matrices in the FE structural model. An automatic conversion capability is provided in Option AA for converting the ANSYS MATRIX50 super-elements into the ACS SASSI super-elements using the GM elements.

For the thin shell/plate FE structural models using SHELL elements, the use of the FIXROT or FIXSHLROT command improves the numerical conditioning for the thin SHELL models by avoiding the occurrences of numerical singularities due to the zero in-plane rotational stiffness in the Kirchhoff plate element formulation. For oblique SHELL elements, the FIXSHLROT and FIXROT commands automatically add a small-stiffness torsional spring for the in-plane shell rotations. The torsional spring stiffness default value is equal to 10. The user can modify the torsional spring stiffness value, so that this torsional spring stiffness value is not larger than 10% of the shell element bending stiffness.

The FIXSLDROT and FIXSPROT commands take care of fixing the unused free rotational DOFs for the solid and spring elements, respectively. The use of these commands save both disk storage and run time.

The FIXROT command has a similar effect with combining the FIXSHLROT, FIXSLDROT and FIXSPROT commands.

***WARNING:*** For thin shell SSI models using SHELL elements, especially including oblique shell elements with respect of global coordinate system, we highly recommend the use of the FIXROT (or FIXSHLROT) command that may improve considerably the numerical conditioning of the FE structural model for the dynamic SSI analysis. For thick shell models using the newer TSHELL

---

*elements, a small rotational stiffness is automatically added in HOUSE, so that there is no need to the use the FIXROT command.*

3. For nodes that are common to the BEAM or SHELL/TSHELL elements and SOLID elements, the nodal rotational displacements of the BEAM or SHELL/TSHELL elements are not transmitted to the SOLID elements. This is because the SOLID elements have only three nodal translations and no rotation. To transmit the bending moments from the BEAM or SHELL/TSHELL element to the SOLID elements, the user has to include in the FE model additional massless BEAM or SHELL/TSHELL elements along the edges or the faces of the SOLID elements. By doing this the nodal rotations are transmitted to the SOLID elements through the node rotations of the additional BEAM or SHELL/TSHELL elements. The added BEAM and SHELL/TSHELL elements transmit the bending moments from the original BEAM and SHELL/TSHELL element nodes by force couples at the SOLID element nodes.

Another situation to pay attention is the connection between BEAMs and SHELLs/TSHELLs. Since there is no plate stiffness for the in-plane SHELL rotation (drilling), the BEAM rotation in shell plane cannot be transmitted to the SHELL nodes. For such situations, the user should build an additional local tripod BEAM systems to transmit the node rotation to the neighbor nodes. This is typical FEA modeling.

Additional SSI model checking commands that *warn* the user about potential modeling mistakes are FIXEDINT and HINGED. The two commands identify the mistakenly fixed interaction nodes, and, respectively, any potentially undesired hinges between the SOLID, SHELL/TSHELL and BEAM elements.

***WARNING:*** *The use of the FIXEDINT and HINGED commands is strongly recommended to check the SSI model correctness before the production SSI analyses are performed.*

4. The excavated soil volume FE model is idealized by either standard 2D PLANE strain in 2D SSI analysis or 3D SOLID elements in 3D SSI analysis. If no near-field soil elements are included in the FE model, then, the excavated soil elements should be only connected with the structure elements only at the nodal points of the foundation-soil interface. If the near-field soil elements are included in the FE model to simulate the back-fill soil or irregular soil layering, then, the excavated soil elements should be connected only with the near-field soil elements (not with the structural elements) at the node points situated at the boundary between the near-field soil and the far-field soil layering that simulates the surrounding soil media.

***WARNING:*** *A very important SSI model checking is achieved using the EXCSTRCHK command that identifies if any excavation volume internal nodes are connected incorrectly to the structure basement nodes.*

5. All the interaction nodes lie on the soil layer interfaces and have only translational degrees of freedom. Rotational structural motions are transferred by the nodal translations.

6. The SSI interaction nodes are defined differently depending on the SSI substructuring approach to be used. In ACS SASSI there are four main SSI substructuring approaches: i) The Flexible Volume (FV) method (also called Direct method) with SSI interaction nodes defined for all the excavated soil volume nodes, ii) the Flexible Interface method (FI) with SSI interaction nodes defined only at the Foundation-Soil-Interface-Nodes (FI-FSIN) (also called Subtraction) and iii) the Flexible Interface method (FI) with SSI interaction nodes defined at all the Excavated-Volume-Boundary-Nodes (FI-EVBN) (also called Modified Subtraction Method, MSM, or sometime Extended Subtraction Method, ESM), and iv) the Fast-Flexible Volume (FFV) method with interaction nodes including the FI-EVBN interaction nodes, plus additional internal node layers of the excavated soil. In other words, the FFV uses as interaction nodes all the excavated outer-surface nodes plus a reduced number of internal horizontal node layers within the excavated soil (Ghiocel, 2015a).

The ACS SASSI UI module includes the INTGEN command that can be used to automatically define the interaction nodes for different SSI substructuring methods, such as FV, FI-EVBN, FI-FSIN and FFV. The ACS SASSI implementation for the FV (Direct), FI-FSIN (Subtraction Method, SM) and FI-EVBN (or MSM) methods are described in Section 2.1.

The FFV method is recently introduced in ACS SASSI to deal with deeply embedded SSI models such as small-modular reactors (SMR). The FFV method is much faster than the reference FV method and is highly accurate for deeply embedded SSI models (Ghiocel, 2013a, 2014b, 2015a). The FFV method is extremely useful for deeply embedded larger-size SSI models that include a very large number of FV interaction nodes, say 30,000 to 50,000 interaction nodes, which slows down the SSI runs and increase the HDD storage way too much for fast-project schedule. Using FFV, the total number of interaction nodes can be largely reduced, and by this the entire SSI analysis runtime and hard-disk space can be reduced possibly by tens of times. For SMRs, the FFV method is an extremely useful method since the FI-EVBN and FI-FSIN methods may fail to produce reasonable accurate results.

**WARNING:** *The new ASCE 4-16 standard require a preliminary validation study if any other method than FV is used, such as SM, MSM or FFV. The validation against FV has to be done before SM, MSM or FFV can be used for the SSI production runs. As recommended in the ASCE 04-2016 standard, the validation analysis can be done for simple “excavated soil models” including no structure or foundation part. The validation study shall compare the computed acceleration transfer functions (ATF) at the common nodes between the structure and excavated soil. Rather than considering the solely “excavated soil model” (“the swimming pool model”), the use of a simplified massless foundation model to determine the kinematic SSI effects using SM, MSM or FFV against FV is suggested. For the deeply embedded SSI models, such as SMRs, in*

---

*deep soft soil deposits, the “excavation soil model” could become numerically poorly conditioned, and therefore, should be avoided. The most complete validation study is to consider the full SSI model and check ATF at critical locations within the nuclear building (Ghiocel, 2015a).*

**WARNING:** *The new ASCE 4-2016 standard recommends the use of quarter FE models for performing comparative “excavated soil” analysis using SM or MSM against FV. This saves a lot of computational analysis time. However, we noticed that for some SM models that behaves highly unstable for the full models, they behave less unstable for the quarter models. Therefore, we do not recommend the use of quarter models to qualify the SM or MSM models. It seems that for the SM or MSM quarter models the symmetric and antisymmetric kinematic boundary conditions imposed to the quarter models help the numerical instability and, therefore, they could behave more stable than the full models (Ghiocel, 2015a)*

7. When selecting the substructuring method for SSI analysis, the user should make a trade-off between the required accuracy of results and the computational effort involved. The FV method is accurate, but is also very computational intensive, and by this is limiting the size of the FE structural model. The soil impedance calculations are proportional with the power 2-3 of the number of interaction nodes. It is not uncommon that the FV method runtime could take tens of times of the FI-FSIN and FI-EVBN method runtimes. The FI-EVBN method is several times faster than the FV method and only few times slower than the FI-FSIN method.

The FI-FSIN could become numerically unstable in the higher frequency range depending on the surrounding soil stiffness and excavation configuration. For stiffer soil sites or rock sites, the FI-FSIN method is expected to provide accurate results coincident with the FV and FI-EVBN method results. The larger the SSI model excavation is, the more effective the FI methods are in terms of speed. The problem is to make sure that the FI methods maintain the accuracy of SSI results. Because of the need to check FI accuracy, preliminary sensitivity studies using the FI and FV method are always recommended when dealing with embedded structures. Typically FI-EVBN provides both numerical accurate and reasonable computational speed when compared with the reference FV method. The FI-EVBN (or MSM) appears to be appropriate for shallowly embedded nuclear island models. The FI-EVBN method may fail for deeply embedded models, such as SMR, for which FFV or FV are appropriate (Ghiocel et. Al, 2013a, Ghiocel, 2014b, Ghiocel, 2015a).

8. The interaction nodes shall always include all the nodes at the excavation volume-far field soil interface nodes. Skipping any nodes at the excavation-far field soil interface with the intention to reduce the number of the interaction nodes could affect largely the SSI solution accuracy. Sensitivity studies shall be performed to see the level of inaccuracy introduced in the SSI solution by skipping nodes at the excavation FE model-far field soil interface.

9. An important SSI modeling aspect is that the excavation volume mesh should be as uniform as possible. Nonuniform meshes create nonuniform local soil impedances at the interaction nodes

---

that could affect the accuracy of SSI analysis. Therefore, regular uniform foundation meshes are recommended, as also recently recommended by the NRC consultants and the Brookhaven National Lab (BNL) SSI experts (Nie et al., 2013). This can be accomplished using transition meshes between the excavation volume mesh and the structure basement. As a note, we plan to include an user friendly automatic transition mesh capability in a next upgrade of the software. This will be accomplished by using few new commands in the ACS SASSI UI.

For nonuniform meshes, there is no precise rule on how to select the radius value for the soil axisymmetric model used to compute the soil layering flexibility matrix. A practical rule that is to consider the average radius value computed for all excavation volume solid elements. However, preliminary sensitivity studies using maximum, average and minimum radius values are always recommended to confirm that using average radius value is conservative. If the radius sensitivity studies indicate significant differences in the SSI responses of interest as a result of the radius value variation, then, the SSI responses from the sensitivity analyses should be enveloped. It should be noted that the torsional and rocking effects introduced by nonuniform mesh could be amplified for incoherent inputs.

10. Sometimes, some interaction nodes are defined outside of the excavation volume. This could happen when there are buried BEAM or SHELL/TSHELL elements (that have no volume) that extend outside of the basement space that encloses the excavation volume. The interaction nodes list should include these nodes of buried BEAM or SHELL/TSHELL elements. This can be done automatically for SHELL/TSHELL elements by using the ETYPE command (= 2) combined with the INTGEN command.

11. For an accurate SSI modeling consistent with the SASSI Flexible Volume theory, the excavated soil nodes should be different than the structural basement nodes, except at the foundation-soil interface where the structural basement and excavated soil share the common nodes or are connected by rigid/very stiff springs.

More generally, for the SASSI FE models that include surrounding backfill, the near field soil-far field soil interface, which is the outer surface of the excavated soil should be considered for connection with FE model. All the other internal excavated soil nodes should be independent and not connected with the basement nodes, even they might have the same nodal coordinates. This is a basic SSI modeling rule that ensures that the two coupled FE subsystems, structure and excavated soil, vibrate independently with the exception of the common nodes placed at the interface between them.

Unfortunately, sometimes in the past practice, this rule of using separated meshes in the embedment for the basement and the excavated soil was ignored. This modeling violation could produce very poor results, especially if FI methods are used. It should be noted that the FV method provides often close results for SSI models with separated meshes and unique mesh in

---

the basement. This situation is limited to applications with structural basements having stiff walls and floors. For flexible structural systems placed in the basement, such piping systems or equipment, the lack of not using separate meshes can affect significantly the accuracy of their SSI responses.

**WARNING:** A very important SSI model checking is achieved using the `EXCSTRCHK` command that identifies if any excavation volume internal nodes are connected incorrectly to the structure basement nodes.

**WARNING:** For embedded structures problems, it is always possible that numerical instabilities could occur at isolated frequencies. This instability could occur for all SSI methods including the FV method, and depends on the particularity of the problem. We recommend that the user should carefully revise and understand SSI results by inspecting the computed acceleration transfer functions (ATF) at different node locations. By inspecting the computed ATF at several nodal locations, the frequencies that manifest solution instabilities should be identified. The unstable computed results at those frequencies should be not be considered for interpolating the transfer functions in the MOTION and STRESS modules. Sensitivity studies by considering adjacent frequencies to those suspected for spurious results, are recommended for clarification. The user can remove any frequency that appears to produce a spurious solution using the auxiliary interactive DOS code called "Remove\_Frequencies\_from\_FILE8.exe" that is installed by default in a zip file included in the C:\ACSV300\ folder.

12. The structural mass matrix is assumed to be 50% lumped and 50% consistent except for the structural beam elements and plate elements where consistent mass matrix and lumped mass matrix are used, respectively.

13. Material damping is introduced by the use of complex moduli, which leads to effective damping ratios which are frequency independent and could vary from element to element.

### 1.5.2 Dynamic Loading for Seismic and Vibration Analysis

1. The seismic environment may consist of an arbitrary three-dimensional superposition of inclined body and surface seismic waves.
2. Earthquake excitation is defined by a time history of acceleration that is called control motion. The control motion is assigned to one of the three global directions at the control point which lies on a soil layer interface.
3. Both coherent and incoherent input motions with or without directional wave passage effects can be considered. For incoherent motions, different incoherency models should be used for rock and soil sites, respectively. Up to 50 stochastic incoherent SSI simulations including the

---

X, Y and Z components can be run in a single SSI run, depending on the SSI model size and the available workstation RAM.

4. In addition to seismic loads, it is possible to introduce external forces or moments such as impact loads, wave forces, or loads from rotation machinery acting directly on the structure. The external forces are applied at the nodal points and are assumed to have similar time histories. However, it is possible to assign different maximum amplitudes and arrival times for each dynamic load applied at a nodal point. This feature enables the program user to define moving dynamic loads on the structure. Please note that up to 500 dynamic load cases can be run in a single SSI analysis run, depending on the SSI model size and the MS Windows workstation available RAM.
5. Transient input time histories such as earthquake record or impact loads are handled by the Fast Fourier Transform technique. Therefore, the time histories must be specified at equal time intervals. The total duration of the Fourier period should be significantly larger than the excitation duration. After the input excitation is ended, the rest of the Fourier period points should have a zero amplitude excitation. This last time interval part of the Fourier period in time is called “quite zone” or “training zero part” and covers the duration during which the SSI system has free-vibration. This quite zone duration also depends on the SSI system component lowest-damping value. Besides, the total number of points in the time histories must be a power of 2. Maximum number of Fourier or time history points is 32,768.
6. The *seismic analysis* option and the *external force/vibration* analysis option cannot be applied in the same single run. Separate linearized SSI analyses are required, and final results can be obtained by linear superposition.

### 1.5.3 Finite Element Library Description

The current finite element library includes the basic element types including, solid, shell/plate, beam, plane, spring and general matrix elements, as follows:

1. Three-dimensional SOLID element (eight-node solid, including degenerated pyramids) with three translational degrees of freedom per node. This element may also include nine incompatible displacement modes in this element when it is used to model the structure. Nonlinear SOLID elements are possible using the iterative equivalent-linear procedure implemented in HOUSE and STRESS modules.
2. Three-dimensional BEAM elements (three nodes) with three translational and three rotational degrees of freedom per node.
3. Three-dimensional SHELL (Kirckhoff theory) and TSHELL (Mindlin-Reissner theory) shell element (four nodes plate, including also triangles) with three translational and three

---

rotational degrees of freedom per node. Nonlinear shell wall models are possible only in Option NON based on the iterative equivalent-linear procedure. The nonlinear wall hysteretic behavior is limited to in-plane shear or bending deformation effects.

4. Two-dimensional four-node PLANE strain finite element (four nodes, or triangles) with two translational degrees of freedom per node. Nonlinear PLANE elements are possible using the iterative equivalent-linear procedure implemented in HOUSE and STRESS modules.
5. The three-dimensional SPRING finite element (two nodes) with three translational and three rotational degrees of freedom per node. Nonlinear springs are possible only in Option NON using the iterative equivalent-linear procedure. For nonlinear analysis, the 3D spring elements, which can have up to three translational stiffnesses should be separated in groups of 1D nonlinear springs, to be able to handle the differential damping for each DOF based on the nonlinear spring hysteretic behavior.
6. Three-dimensional General Matrix (GM) stiffness and mass element (2 nodes for global coordinates and 3 nodes for local coordinates). These elements can be used to define super-element matrices as complex stiffness/mass matrices. The GM elements have three translational and three rotational degrees of freedom per node. The GM elements have more generally 3 nodes. If only the 2 nodes, I and J, are defined that then the stiffness and mass inputs are provided in global coordinate systems. If 3 nodes, I, J and K, are defined then, the inputs are provided in local coordinate system. See the MXI command for details.

The GM element can be used to define super-elements and/or external subsystem matrices, such as equipment or fluid effects. Option AA includes a new automatic conversion capability from the ANSYS MATRIX50 super-element to the ACS SASSI GM elements.

#### **1.5.4 Nonlinear Soil and Structure Hysteretic Behavior**

1. Using SSI substructuring methods, the ACS SASSI software solution in complex frequency is restricted to linearized SSI analysis solutions. However, nonlinear SSI analysis including soil and structure hysteretic behavior can be performed using an iterative equivalent linearization algorithm. For each iteration, a fast SSI reanalysis (restart analysis) using the “New Structure” option should be used.

2. For nonlinear 3D solid or 2D plane elements, typically used to model soil local nonlinear behavior, only the HOUSE and STRESS modules should be run differently while ANALYS should be run using the restart for “New Structure”. The local nonlinear soil hysteretic behavior modeled by 3D solid or 2D plane elements is handled automatically by running iterative SSI iterations by looping the HOUSE-ANALYS(restart)-STRESS modules. For the 2D soil layer models, PLANE elements should be used, while for the 3D soil models (typically in the adjacent zone to

foundation), SOLID elements should be used. It should be noted that the user can select of using either the effective soil i) shear strain component, or ii) maximum shear strain in 2D soil elements, or iii) the octahedral shear strain in 3D soil elements.

To combine the effective shear strain computed using STRESS from directional inputs and provided in FILE74 for X, Y and Z inputs, the COMB\_XYZ\_STRAIN aux program should be used for each SSI iteration. The constitutive soil material curves for nonlinear site response or SSI analysis, i.e. the shear modulus and damping as functions of soil shear strain, should be input by the user for the SOIL module that creates the FILE73 text files from where they are read by the STRESS module during SSI iterations.

For the 1D soil models that implies horizontal soil layers, the SOIL module can be used for modeling the soil hysteretic behavior using either the Seed-Idriss iterative equivalent-linear model based on the SHAKE methodology, or a true nonlinear soil model based on the DEEPSOIL theory.

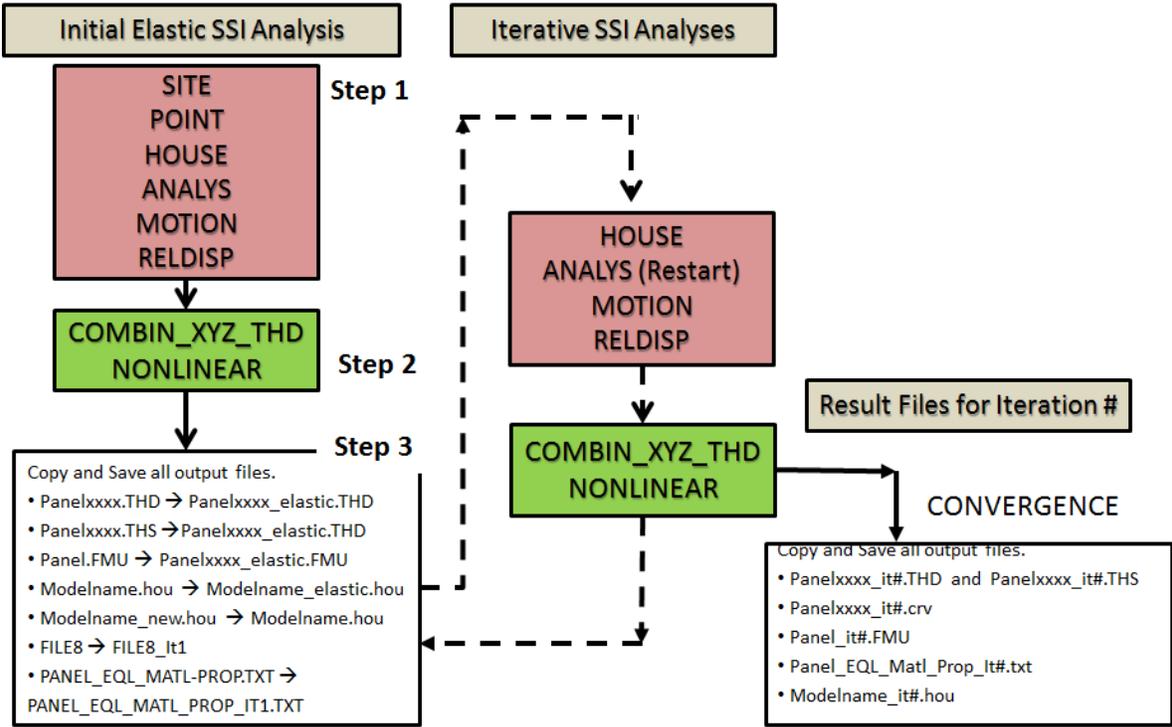


Figure 1.2 Nonlinear SSI Analysis and Generated Files Produced by NONLINEAR Module

The Option NON includes the two modeling choices: i) nonlinear RC wall panel elements (macroshell elements) and ii) nonlinear spring elements with translational dofs. Demo 9 shows an example of a nonlinear SSI analysis for low-rise reinforced concrete shearwall nuclear building subjected to large earthquake level of 0.60g. Demo 10 shows an example of a seismically base-isolated nuclear building model.

---

For the nonlinear structural behavior in 3D SSI analyses, the Option NON (NONLINEAR module) is required. The Option NON can address the reinforced concrete cracking and post-cracking behavior for low-rise shearwalls, nonlinear rubber base-isolators behavior, nonlinear pile-soil interface behavior, and even limited foundation sliding effects.

The panels are subdivisions of the walls that are assumed to be subjected to the uniform shear or bending deformation. The wall panels are described as a set of shell elements in vertical plane. Panels should have rectangular shapes, but they can include shell elements with triangular shape. Also, panels can have any orientation in horizontal plane, not necessarily being parallel to the global coordinates.

The NONLINEAR module run chart is shown in Figure 1.2. The produced filenames as shown in Figure 1.2 are for a typical nonlinear SSI analysis of a concrete shearwall structure that includes a set of nonlinear wall panels. If nonlinear springs are used the “PANEL” string in the names of the files will be replaced by the “SPRING” string.

The computational steps involved by NONLINEAR module are:

- Perform the initial linear elastic (uncracked) SSI analysis using the initial elastic properties for the nonlinear elements
- Compute the local behavior of nonlinear elements in time domain based on the local relative displacements, that is then used to calibrate the local linearized hysteretic models associated to each nonlinear element in complex frequency
- Perform a new SSI analysis iteration using a fast SSI restart analysis in the complex frequency domain using the linearized hysteretic models computed in Step 2 for nonlinear elements
- Check convergence after new SSI iteration to stop or continue.

**WARNING:** Auxiliary DOS batch programs are available to combine the co-directional effects after each SSI iteration, so that that nonlinear element behavior is computed based on the three-directional seismic input, not separate for each unidirectional input. These aux programs are called COMB\_XYZ\_STRAIN for handling nonlinear 3D solid and 2D plane elements using the STRESS shear-strain results, and COMB\_XYZ\_THD for handling nonlinear shell wall and spring elements in Option NON using the RELDISP relative displacement results.

The NONLINEAR result files include nonlinear structure displacements, accelerations, forces in elements, ductility ratios with respect to the initial element stiffness (cracking for concrete hysteretic models), and inelastic absorption reduction factors for the forces in the nonlinear elements. These inelastic absorption factors are defined by the ratio between the initial elastic

forces and the effective nonlinear forces. The inelastic absorption factors depend on the shear strain level. For concrete shearwall buildings these computed inelastic factors correspond to the inelastic absorption reduction factors defined in the ASCE 43-05 standard for different state limits that are used to evaluate the structure wall functional or ultimate capacities.

The nonlinear structure SSI or SSSI analysis also provide better predictions of the inter-building displacements for establishing the appropriate gap sizes. The motion incoherency is an additional important influential factor that can reduce substantially the gaps between the neighboring buildings during the earthquake motion.

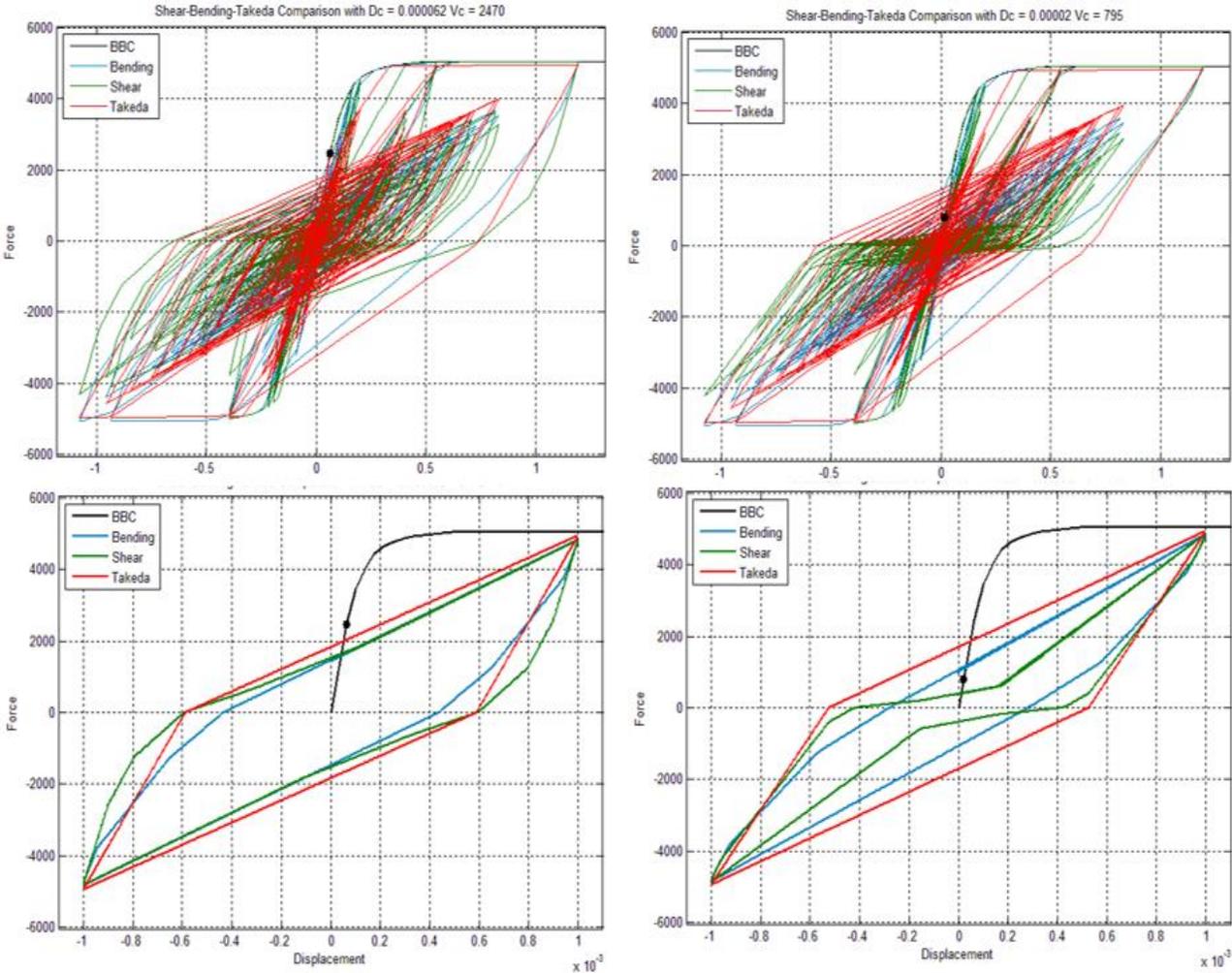


Figure 1.3 Comparison of Cheng-Mertz Shear (CMS), Cheng-Mertz Bending (CMB) and Takeda (TAK) Hysteretic Model Force-Strain Loops for the Same Shear Strain History and Single Cycle.

*Right:* Low cracking point back-bone curve producing pinched loops for CMS model;

*Left:* Higher cracking point back-bone curve producing no pinched loops for CMS model.

For the low-rise reinforced concrete shearwall buildings, the following hysteretic models are currently implemented: 1) Cheng-Mertz Shear (CMS) model, 2) Cheng-Mertz Bending (CMB)

model (not included in this version) and 3) Takeda (TAK) model. These hysteretic models are compared in Figure 1.3 for the same shear strain time history input.

The Verification Manual validates the ACS SASSI Option NON results for a low-rise reinforced concrete typical nuclear building against the nonlinear time-domain integration results obtained using the specialized reinforced concrete dynamic FE program called PERFORM3D (Trademark of Computer and Structures, Inc., USA).

For typical low-rise concrete shearwall nuclear buildings with the height less than the width that are affected primarily by the in-plane shear deformation, the CMS hysteretic model is most appropriate. For wall panels that are most affected by the in-plane bending deformation, the TAK hysteretic model is most appropriate. ‘

To compute the wall panel shear and bending deformation, the displacements computed at the four corner nodes of the panel are used. Figures 1.4 shows how the panel shear strain and bending rotational strain (curvature) are computed in local wall plane axes after the rigid body motion of the panel is extracted. The panel edge differential strains are considered. The shear strain is defined by the average vertical edge rotation (left), while the bending rotational strain (or curvature) is defined by the horizontal edge relative rotation (right).

The uniform axial strain in vertical direction are also computed, so the user can compute the vertical seismic axial forces. For low-rise shearwall buildings the axial force effects are usually quite small. However, the can be included for computing the panel shear or bending capacities as discussed in Section 6.

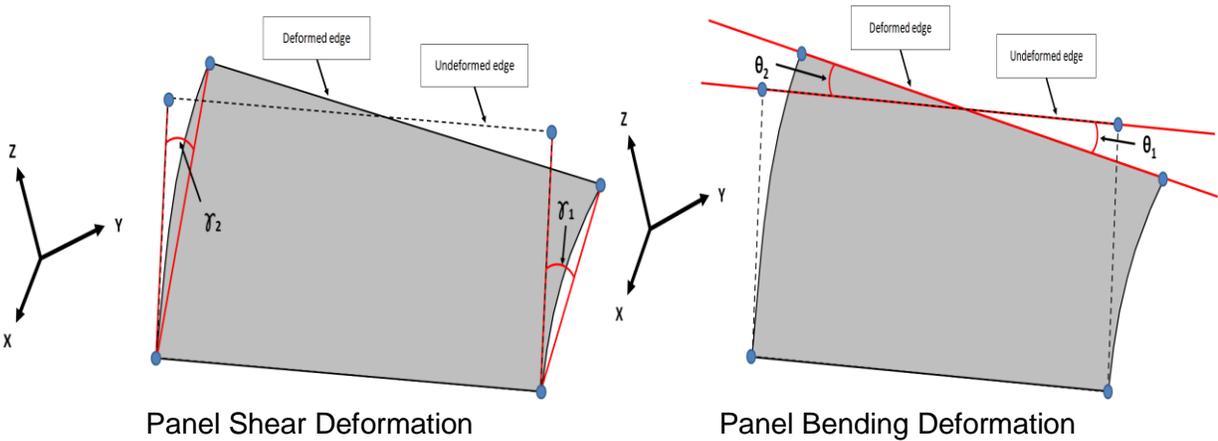


Figure 1.4 Panel Deformation for Shear and Bending Loading (No Axial Force Effects Included)

Based on the hysteretic behavior of each wall panel, the local equivalent-linear properties are computed after each SSI iteration. The stiffness reduction is applied directly to the elastic modulus for each panel. This implies, under the isotropy material assumption, that the shear, axial and

bending stiffnesses suffer the same level of degradation. Poisson ratio is considered to remain constant. Thus, in the current implementation of Option NON, the wall panel shear stiffness modification as a result on nonlinear behaviour is fully coupled with the bending stiffness. This is a reasonable assumption *only* for low-rise shearwalls for which the nonlinear behaviour is governed by the shear deformation, while bending effects play an insignificant role. Based on various experimental tests done at Cornell University, Gergely points out in NUREG/CR 4123 (Gergely, 1984) that in low-rise walls such as those that occur in the modern nuclear power plants, the flexural distortions and associated vertical yielding play a negligible role. This was also recognized by many other research studies, including the EPRI report on “Methodology for Developing Seismic Fragilities” (Reed and Kennedy, 1994).

If the analyst wants to consider the axial effects in the concrete walls, he will need to consider the combined seismic vertical forces with the vertical gravity forces in the panel when defining wall panel shear and bending capacities. The vertical axial forces can be computed by the analyst in a static separate or combined SSI analysis under vertical loads, in order to adjust the BBC input. This can be done by converting the SSI model to ANSYS and do a separate static analysis for gravity loads in ANSYS or it can be done in ACS SASSI for static or combined SSI analysis. To perform a static gravity analysis in ACS SASSI, the user needs to simulate a very low frequency vertical acceleration excitation with an amplitude of 1g, i.e. a one-period harmonic excitation that is exactly equal with the Fourier period duration. For example, using 32,768 Fourier points with a time step of 0.005 seconds, this one-period duration is about 163.84 seconds. For combined SSI analysis under vertical loads, the vertical seismic excitation can be added on the top of the one-period variation, the best over the middle of the first half-period variation defined with a negative sign to simulate gravity force direction.

**WARNING:** *The CMS and TAK hysteretic models included in the current Option NON are typical for nonlinear concrete wall behavior under the horizontal seismic inputs. In the current Option NON there is no hysteretic model for handling axial deformation in wall panels under vertical uniform forces. It should be noted that new ASCE 4-17 recommends to reduce only the shear and bending wall stiffnesses due to the concrete cracking, while the axial stiffness remains unchanged. Therefore, we suggest consider the structure behave nonlinear under the two horizontal components and linear elastic under the vertical seismic component. However, the horizontal and vertical displacements computed at the corners of each wall panel should include for each SSI iteration, the combined effects of the three seismic input components. This is achieved by using the COMB\_XYZ\_THD auxiliary program that is automatically included in the batch run file generated by the NONLINBAT, 1. However, the COMB\_XYZ\_THD.inp text file that is the input of the COMB\_XYZ\_THD auxiliary should be defined by the user (see example file for the COMB\_XYZ\_THD auxiliary program included on the installation DVD).*

For the nonlinear SSI analysis, the nonlinear spring elements, which can have up to three translational stiffnesses, should be separated in spring groups of 1D nonlinear springs, to be able to handle the differential stiffness and damping for each dof based on the local spring hysteretic

---

behavior.

The Problem 51 of the Verification Manual validates the ACS SASSI Option NON results for a seismically nuclear concrete building against the nonlinear time-domain integration results obtained using the ANSYS V15 program (Trademark of ANSYS, Inc., USA).

For nonlinear structure analysis in Option NON the user is required to define the material constitutive back-bone curves (BBC) and the hysteretic model type he wants to use. The BBC definition is the structural analyst responsibility. The BBC input should be defined in terms of the nonlinear force-relative displacement relationship for springs, and the in-plane nonlinear shear force-shear strain relationship, or the in-plane bending moment-curvature relationship for wall panels under primarily shear deformation or bending deformation. The BBC could be input using either an external data file using the BBC commands or the directly by inputting the data point coordinates X (strain) and Y (force/moment) vectors using the BBCX and BBCY commands.

To build the BBC for each wall panel two options are available to the analyst: 1) use directly the existing technical recommendations (see *WARNING* below) for computing the in-plane peak/ultimate wall panel capacities, and based on these and the elastic stiffnesses build the BBC for each panel, or 2) perform a separate nonlinear static pushover analysis (outside of ACS SASSI) to compute the BBC for each panel.

The BBC should include the effect of vertical axial forces, if significant. Thus, the effect of vertical axial force should be evaluated, if negligible or significant, and if significant to include the axial force effect in the wall panel capacities. Typically, for the low-rise shearwall buildings the axial force effects are small and can be neglected, as mentioned in the previous page.

It should be noted that if the user is not certain about the type of the governing deformation mechanism, shear or bending, in certain panels, then, he can run few preliminary comparative analyses assuming that those particular panels fail either in shear or bending. Based on the preliminary analyses, user can identify which type of deformation governs each of these panels.

***WARNING:*** *The wall panel BBC curves should have a smooth shape and variation that describes the nonlinear behavior of the wall panels under the lateral seismic loading. The BBC could be built based on the existing pertinent technical recommendations, or computed using static nonlinear push-over FE analysis. For estimating the low-rise shearwall panel capacities there are a significant number of sources in the literature that provide empirical equations for computing the wall panel shear capacities (Gulec and Whittaker, 2009, Wood, 1990, ACI 349-08, Barda et al., 1977). Using the SHEAR command the user can check the computed shear capacity values based on different shear capacity equations. Using the BBCGEN command smooth BBC curves can be automatically generated for many wall panels.*

We recommend to avoid using the popular Barda equation that is applicable only to barbell walls with heavy flanges, for predicting wall panel shear capacities for typical nuclear shearwall buildings, since it could overly estimate the panel shear capacities. Recently, the Barda equation was taken out of the final ASCE 43-17 standard draft. The ATC 72-1 Option 3, 2010, provides additional guidance for reducing the shear peak capacities to account for many cyclic degradation effects.

The ACS SASSI Option NON can handle shearwalls with large openings as described below. If the structure has walls with significant openings the use of the EDGE command is highly effective to split the wall panel into subpanels. For example, for a wall with openings like the wall shown below in Figure 1.5. On the right side is defined a single wall panel similar to solid wall without openings. We assume that the single panel is included in the element group 1. If the panel has openings, it is shown that the use of the EDGE 1,0,0,1 command creates the new groups 2 and 3 for the subpanels shown in the red color. The panel is split in three subpanels. Then, the second application of the EDGE, 2 command for the group 2, that includes the openings, results in the split of this group 2 into other three groups, the groups 2, 4 and 5 that include the subpanels around the openings in the red color. The panel is now split into five subpanels, as shown in the lower left-side plot. The BBC should be defined for each subpanel based on the panel geometry and reinforcement.

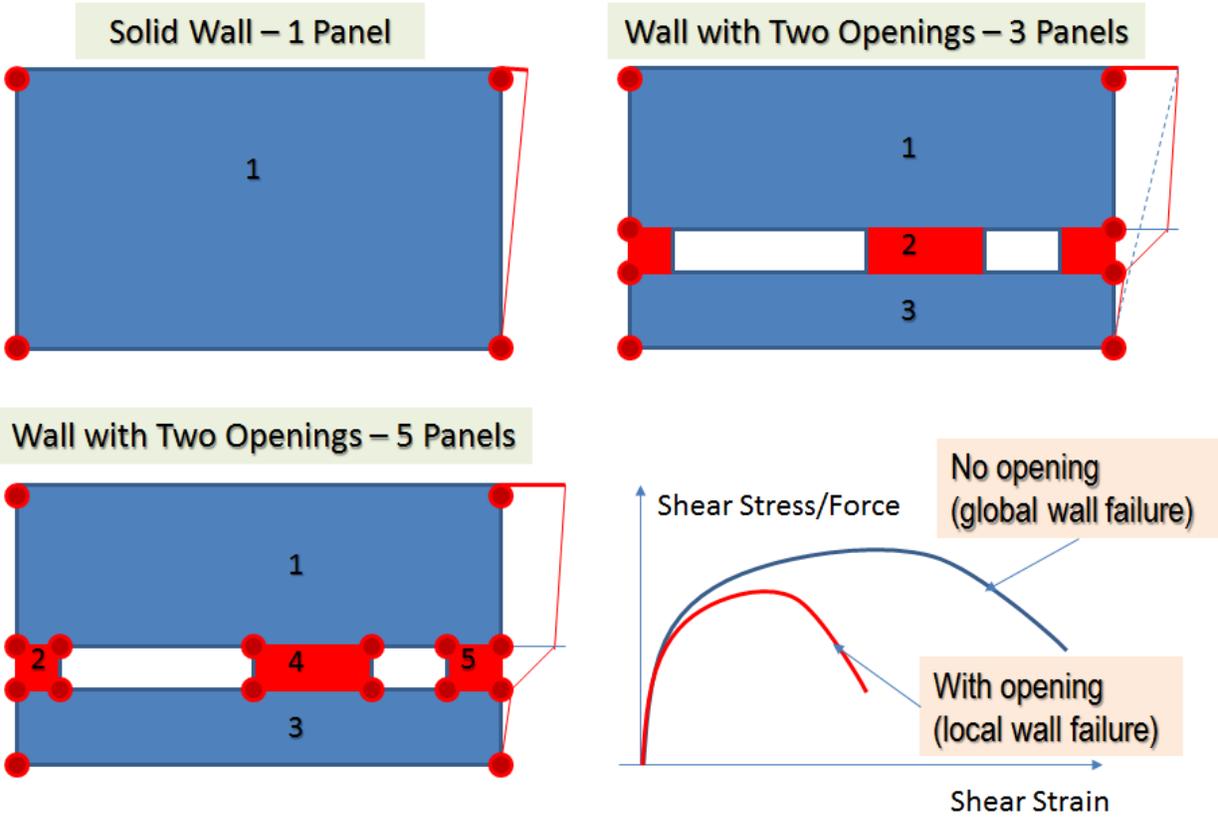


Figure 1.5 Subpanel Modeling for Concrete Wall with Large Openings

The effect of openings have large impact of the overall wall behavior as illustrated in Figure 1.5, in the lower right-side plot that shows the BBC for the solid wall vs. the BBC for the same wall with the openings. If large openings are present, then, the wall failure is basically a failure under the local large strains in the subpanels/piers that border the openings, as shown in Figure 1.5.

Based on the user's choice, the damping can include only the hysteretic damping, or the sum of the viscous-elastic damping and the hysteretic damping. The user also can use a damping cut-off value, or a damping reduction scale factor. The damping cut-off value should be used to ensure that the computed effective damping does not go above the maximum damping value accepted in the ASCE 4 and 43 standards and the USNRC guidelines. For example, for the seismic SSI design-basis analysis, the use of a damping cut-off value of 7% for considering concrete cracking will ensure that the effective damping in the walls will not be larger than the maximum acceptable damping value of 7% in any wall panel.

**WARNING:** *Simplified shapes, such bilinear or trilinear BBC with sharp corners may severely affect the convergence of the iterative equivalent-linear procedure and may produce oscillatory convergence or even divergence.*

**WARNING:** *For nonlinear wall panels and springs, the slope of the first segment of the BBC curves should correspond to the elastic stiffness included in the initial HOUSE input file before any iteration. In the NONLINEAR module, the BBC curves are normalized to the initial stiffness values defined by the first segment slope of the BBCs, and then, these normalized BBC will be applied to the initial elastic modulus value defined in the HOUSE input file (.hou file). In the HOUSE input, the elastic modulus is used for shells, and the elastic spring stiffness is used for springs. For concrete wall panels, the BBC elastic shear stiffness values defined by the cracking point should be equal to  $GAshear$  for shear type BBCs, where  $G$  is the elastic shear modulus and  $Ashear$  is the cross-sectional shear area, and equal to  $EI$  for bending type BBCs, where  $E$  is the elastic longitudinal modulus and  $I$  is the moment of inertia of the cross-sectional area. The effects of vertical axial forces, if significant, should be included in the BBCs since affect the panel capacities.*

## 7. SSI Solution Interpolation Scheme in Frequency

The last ACS SASSI version include seven efficient interpolation schemes for accurate approximation of the complex response transfer functions (TF), for both nodal acceleration TF (ATF) and structural stress/force TF (STF). For complex FE model structures of nuclear complexes, the existence of several structure models on common mats and the high density vibration mode frequencies in mid and high frequency ranges could produce significant deviations in the interpolated ATF or STF obtained using the interpolation options 0 to 5, especially, if the number of selected SSI frequencies is not sufficient. These deviations are produced by spurious

---

narrow-band spectral peaks and valleys in the interpolated TF due to insufficient frequency points. In these cases additional frequencies are needed. A different interpolation scheme using bi-cubic complex splines is implemented for the interpolation option 6. This spline interpolation is the best choice for incoherent SSI analyses for which the number of SSI frequencies should be larger than for coherent SSI analysis, say at least 200 SSI frequencies.

Typically, 40 to 80 SSI frequencies could be sufficient for simple, stick structural models that captures major global modes of the structure vibration, and 100 to 200 frequencies for complex flexible structural models that include many local modes. For incoherent SSI analysis we suggest the use of a larger number of frequencies in the 200-300 range (Ghiocel, 2013b, Ghiocel, 2014a). Please also see the 2007 EPRI validation study for ACS SASSI application to incoherent SSI analysis (Short et al., 2007).

Additional details on the SSI frequencies selection is provided in Section 4.1.2.

***WARNING:*** *The number of SSI frequencies should larger for nonlinear SSI analysis than for linear SSI analysis. It is suggested use a denser frequency grid for frequency ranges for which significant spectral peaks are expected to occur due to the nonlinear behavior, i.e. spectral peaks shifts to lower frequencies.*

### 1.5.5 System of Parameter Units

For SSI analysis any system of units may be chosen to be used as long as these units of the input data are consistent in all the program modules. Acceleration of gravity can be either 32.2 ft/sec<sup>2</sup> for the British Unit (BU) system or 9.81m/s<sup>2</sup> for the International Unit (IU) system.

## 1.6 USER DATA INPUT

ACS SASSI offers several ways of entering input data:

- through the instruction line
- through the menu commands
- through the import of input text files (*INP,<fname>*- instruction) (see Section 9.2.18)

Using the ACS SASSI PREP instructions, the input data can be entered in free format, with the comma “,” as a delimiter between the instruction parameters. The general form of an instruction is an alphanumeric “keyword”, which defines the instruction, followed by parameters separated by “,”: “Keyword, p1, p2,..., pn”. Instruction names are not case-sensitive.

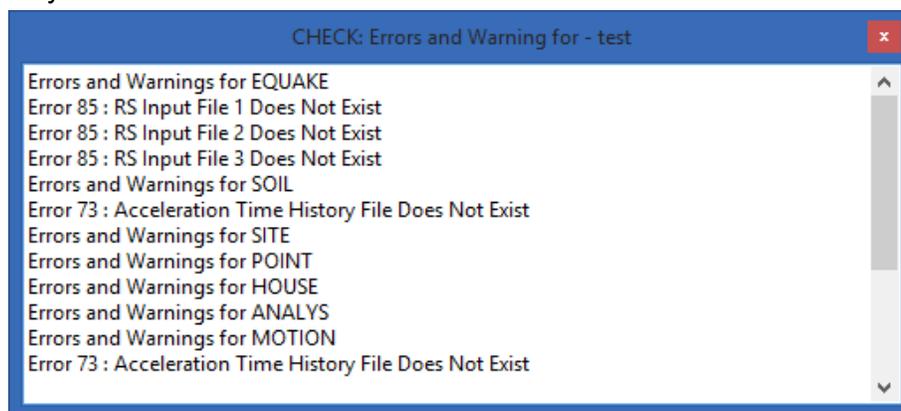
Input data file may be entered in any order, by alternating the instruction line, dialog boxes, menus, and data files.

## 1.7 DATA CHECKING

Data checking can be done in several ways:

- during entering data, by generating data lists
- by interactive plotting of the model or of the selected model parts, soil layers, time history files, response spectra, and shear modulus - shear strain / damping ratio - shear strain curves
- graphically, by different colors associated with the elements in function of element groups, properties, materials,....
- the *Plot Info* window can be activated by pressing the right mouse button, in which case it will show all properties of the selected element.

Once the SSI analysis model is generated, the *CHECK* instruction (see Section 9.2.7) can be activated to verify data correctness and to show in the check window the error and/or warning messages, if any.



*CHECK* is automatically activated when selecting the instruction for writing the analysis files, *AFWRITE* (see Section 9.2.3). If errors are detected, the affected analysis file(s) will not be written. The maximum number of error and warning messages can be controlled by the **Options/Check** menu command (see Section 6.6.1).

## 2 Theoretical Basis for Performing Linearized SSI Analysis

In this section some basic assumptions and SSI analysis procedures are reviewed.

### 2.1 FLEXIBLE VOLUME (FV) SUBSTRUCTURING

In the FV substructuring approach, the dynamic solution is computed for the coupled structure-excavated soil system defined by the differential complex dynamic stiffness obtained by subtracting the excavated soil from the basement. The difference between the dynamic complex stiffnesses of the two coupled subsystems, structure and excavated soil, characterizes SSI interaction effects for an embedded foundation problem. As a simple example, if the complex dynamic stiffness of the structure is equal to the complex dynamic stiffness of the excavated soil at all excavated soil nodes, then, the difference is zero and there is no SSI interaction. The larger is the difference between these two coupled subsystems, the larger are the SSI effects on the response of the embedded structure.

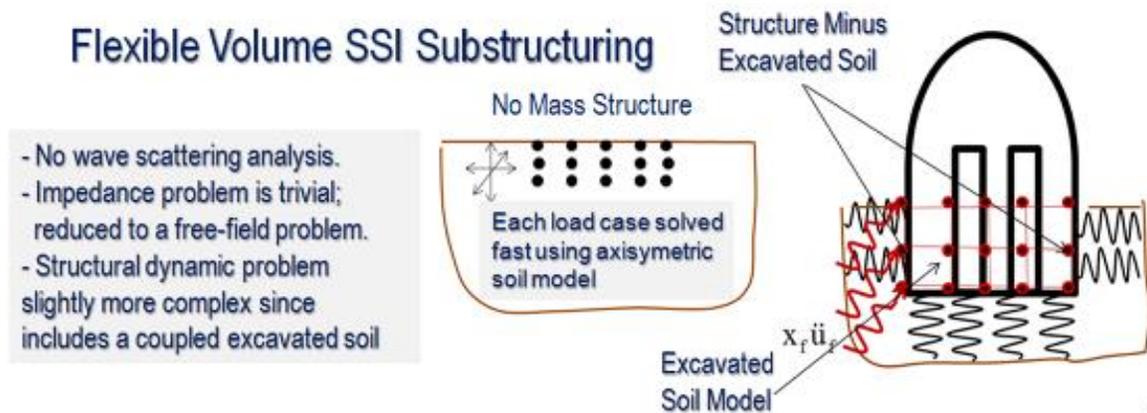


Figure 2.1: Flexible Volume Method Concept

In the FV substructuring the SSI system is partitioned into three coupled substructures, namely, the free-field system, the structural system and the excavated soil system as shown in Figure 2.2. The FE model part consists of two FE models: i) the structural system and ii) the excavated soil system. The difference between the reference FV method and FI methods is that in the reference FV method (or Direct method) all the excavated soil dynamic degrees of freedom are considered to be interaction nodes for computing the SSI solution, while for the FI methods (or Subtraction Method or Modified Subtraction Method) only a subset of the excavated soil dynamic degrees of freedom are used as interaction nodes for computing the SSI solution in complex frequency. The

FI methods are basically approximation of the FV method with the intention to reduce the computational effort associated with the soil impedance calculations for embedded structures.

As shown in Figure 2.2, in the reference FV method, the interaction between structure and foundation occurs at all excavated soil nodes (Figure 2.2a), while in the FI methods, in either FSIN or EVBN implementation, the interaction nodes are defined only on the excavation volume lateral surface (Figure 2.2b). Thus, the FI methods are significantly faster than the FV method.

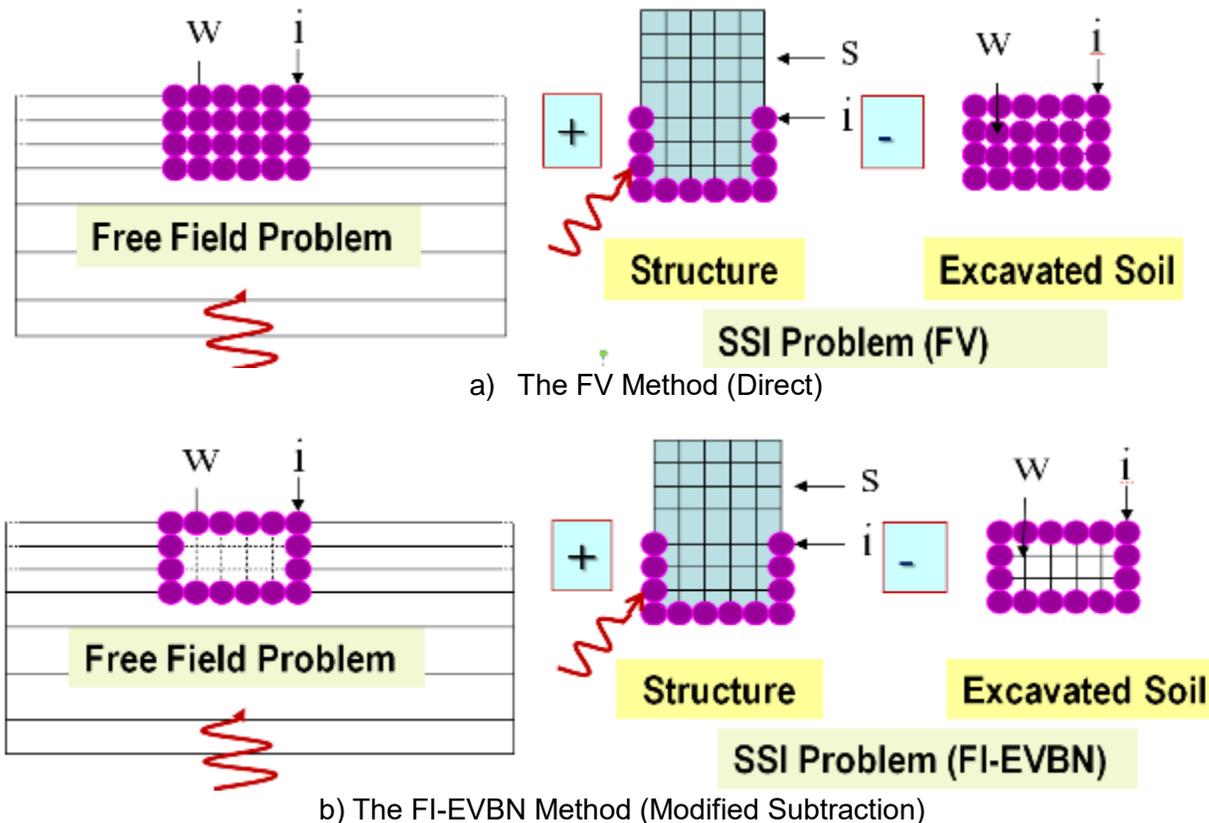


Figure 2.2: The Flexible Volume SSI Substructuring Approaches

For the FI-FSIN method (or Subtraction in the ASCE 4-17 standard), the excavated soil dynamic degrees of freedom are reduced to the SSI nodes that are located only at the foundation-soil interface. For the FI-EVBN method (or Modified Subtraction in ASCE 4-17 standard), in addition to the SSI interaction nodes defined at the foundation-soil interface, includes SSI interaction nodes defined at the ground surface of excavated soil as shown in Figure 2.2b. The additional SSI interaction nodes placed at the ground surface of excavated soil improve significantly the SSI response accuracy. These ground surface SSI interaction nodes are of key importance to capture accurately the seismic wave scattering effects that are produced by the scattered surface waves.

In addition to the FI methods, the Fast FV method was recently introduced. The FFV method improves the FI-EVBN method accuracy by increasing the number of interaction nodes by adding

new layers of interaction nodes that are internal nodes of excavated soil volume (Ghiocel, 2013a). The FFV method is a more accurate method than FI methods for deeply embedded structures such as SMRs. However, the way in which the FFV interaction nodes are selected from the internal excavated soil nodes should be made on a case-by-case basis based on the expert judgment and/or preliminary sensitivity analyses using alternate selections for the internal excavation interaction nodes.

It should be noted that the SSI analysis matrix formulation is the same for the FV and FI methods, except that the equations of motion of the excavated soil system which are different depending on the selection of the SSI interaction nodes. For the FI methods the equations of motion of the internal nodes of the excavated soil are only approximate since they miss the soil impedance and the free-field seismic load terms.

The SSI problem matrix equation is:

$$\begin{bmatrix} \mathbf{C}_{ii}^s - \mathbf{C}_{ii}^e + \mathbf{X}_{ii} & -\mathbf{C}_{iw}^e + \mathbf{X}_{iw} & \mathbf{C}_{is}^s \\ -\mathbf{C}_{wi}^e + \mathbf{X}_{wi} & -\mathbf{C}_{ww}^e + \mathbf{X}_{ww} & \mathbf{0} \\ \mathbf{C}_{si}^s & \mathbf{0} & \mathbf{C}_{ss}^s \end{bmatrix} \begin{Bmatrix} \mathbf{U}_i \\ \mathbf{U}_w \\ \mathbf{U}_s \end{Bmatrix} = \begin{Bmatrix} \mathbf{X}_{ii} \mathbf{U}'_i + \mathbf{X}_{iw} \mathbf{U}'_w \\ \mathbf{X}_{wi} \mathbf{U}'_i + \mathbf{X}_{ww} \mathbf{U}'_w \\ \mathbf{0} \end{Bmatrix} \quad (2.1)$$

from which the final total motions of the structure can be determined. In these equations, the subscripts s, i, and w refer to degrees of freedom associated with the nodes on superstructure, basement and excavated soil, respectively.  $\mathbf{C}$  is the dynamic stiffness matrix defined by:

$$\mathbf{C}(\omega) = \mathbf{K} - \omega^2 \mathbf{M} \quad (2.2)$$

where  $\mathbf{M}$  and  $\mathbf{K}$  are the total mass and complex stiffness matrices, respectively,  $\mathbf{u}$  is the vector of complex nodal point displacement;  $\mathbf{X}_{ff}$ , is a frequency-dependent matrix representing the dynamic boundary of the foundation at the interaction nodes. The  $\mathbf{X}_{ff}$  is referred to as the impedance matrix.

According to the complex frequency domain formulation, the solution of the linearized SSI problem reduces to three main computational steps (for each frequency):

1. Solve the site response problem to determine the free field motion  $\mathbf{u}'_f$  within the embedded part of the structure.
2. Solve the impedance problem to determine the free-field impedance matrix  $\mathbf{X}_{ff}$ .
3. Solve the SSI problem by forming the entire SSI system complex stiffness and load vectors and solving the above equations for the final SSI response displacements.

## 2.2 SITE RESPONSE ANALYSIS

The original site is assumed to consist of horizontal soil layers overlying a uniform half space. All material properties are assumed to be viscoelastic. Usually, the values of the soil shear stiffness and hysteretic damping of each layer correspond to the iterated equivalent-linear values computed from the nonlinear free-field analysis. The linearized site response analysis for the SSI analysis is handled in the SITE module. The nonlinear site response analysis to compute the iterated equivalent-linear soil layer properties is performed using the SOIL module via the SHAKE equivalent-linearization methodology.

Only the free-field displacements of the layer interfaces where the structure is connected are of interest. The SITE linearized displacement amplitude solution is expressed in the form:

$$u_f'(x) = U_f' \exp[i(\omega t - kx)] \quad (2.3)$$

where  $U_f'$  is a mode shape vector which contains the interface amplitudes at and below the control point ( $x=0$ ) and  $k$  is a complex wave number which expresses how fast the wave propagates and decays in the horizontal  $x$ -direction. Effective discrete and iterative eigen solution methods are used for determining the complex wave mode shapes and wave numbers corresponding to control motion at any soil layer interface for the inclined P-, SV-, and SH- waves, and Rayleigh waves and Love surface waves.

Two types of seismic wave combinations are possible: i) SV-, P- and Rayleigh waves (2 DOFs per node) and ii) SH and Love waves (1 DOF per node). A complete field of recorded seismic waves will include all the five types of component waves. However, for typical seismic SSI analysis, as required by ASCE and USNRC, only vertically propagating SV, SH and P wave component types should be considered. To best reconstruct a complete 3D seismic wave field with vertically propagating waves, we suggest considering SV waves for X direction, SH waves for Y direction and P waves for Z direction.

ACS SASSI includes also the incoherent motion option for describing seismic random field input motion environment. The wave passage effects can be also included. Technical details on the ACS SASSI incoherent SSI analysis methods could find in 2006-2007 EPRI reports (Short et. al., 2006, 2007, Ghiocel, 2007a) and in other papers (Ghiocel, 1998, Ghiocel et. al., 2007b, 2009a and 2009b, 2013b, 2014a, 2015b).

## 2.3 IMPEDANCE ANALYSIS

As previously stated, the soil impedance matrix represents the dynamic stiffness of the free-field soil layering at the interaction nodes. Thus the soil impedance matrix is determined as the inverse

---

of the free-field dynamic soil flexibility (compliance) matrix. Depending on the selected FV substructuring method, FV, FI or FFV, the size of the soil flexibility and impedance matrices will be quite different.

## 2.4 STRUCTURAL ANALYSIS

The FV substructuring approach for seismic SSI analysis is shown in Figure 2.2. The FE model part can include the structure, plus eventually some surrounding backfill soil, and the excavated soil. The backfill soil included in the FE part is called the near-field field soil, while the soil layering outside the FE model part that is connected with the excavated soil FE part at its outer surface interaction nodes is called far-field soil. This backfill soil zone can be modeled either in 2D space or 3D space using either PLANE or SOLID finite elements, respectively.

## 2.5 SUMMARY OF COMPUTATIONAL STEPS

The SSI system steady-state equations of motion for the near-field zone are expressed in matrix form as shown in Section 2.1. As indicated by the matrix equation in Section 2.1, for each frequency,  $\omega$ , the impedance matrix, the load vector, and the dynamic stiffness matrices ( $\mathbf{C} = \mathbf{K} - \omega^2 \mathbf{M}$ ) for the structure (including the basement and any soil irregular zone) and the excavated soil are formed.

After forming the complex system of equations of motion, this must be efficiently solved. The matrices involved have often large sizes, especially for the 3D SSI problems with significant embedment that need to be optimally handled depending on the available RAM and the HDD storage space.

The SSI solution algorithm includes the following step.

Below are given details on the operations which must be performed for each SSI frequency step of the SSI analysis:

1. *Form complex dynamic stiffness of structure:* The total frequency-dependent complex stiffness of the structure is computed using the total stiffness and mass matrices.
2. *Form complex dynamic stiffness of excavated soil:* The total frequency-dependent complex stiffness of the excavated soil is computed using the total stiffness and mass matrices.
3. *Form impedance matrix:* If the flexible volume or interface methods are used, then, the entire flexibility matrix for the interaction nodes is constructed using the flexible volume method. The flexibility matrix is then inverted in place, using an efficient subroutine for symmetrical matrices, to obtain the impedance matrix, which is stored in the same form.

4. *Form total stiffness of soil-structure system.* The total stiffness is obtained by adding the impedance matrix to and subtracting the excavated soil volume dynamic stiffness from the total stiffness of the soil-structure system.
5. *Form load vector:* For seismic analysis, the load vector is computed by multiplying the impedance matrix by the free-field motion vector. For forced vibration analysis, the load vector is formed directly from the given nodal external forces.
6. *Solution of linear complex equation system:* The transfer function matrix is determined by solving linear system. This matrix corresponds to the absolute acceleration transfer functions (ATF) for seismic analysis and displacement transfer functions (DTF) for forced vibration analysis.

---

## 3 Description of SSI Modules

In this section we describe the main functionalities of the basic SSI modules shown in Figure 1.1 used for the SSI analysis calculations. In addition to the basic SSI modules, we also included the Option NON modules. The Options A and AA modules are described in a separate documentation called “ACS SASSI-ANSYS Integration Capability” User Manual. The Option PRO on probabilistic site response and SSI analysis has also a separate user manual.

### 3.1 SEISMIC INPUT, SITE RESPONSE AND SSI ANALYSIS MODULES

Short descriptions of the SSI module functionalities are provided in the following:

#### 1. Module EQUAKE

The EQUAKE module generates earthquake acceleration time histories that are compatible with given seismic design ground response spectra. The EQUAKE module combines both frequency domain and time domain algorithms to generate the spectrum compatible accelerograms. The frequency domain matching algorithm is based on the Levy-Wilkinson (LW) algorithm, while the time domain matching algorithm is based on an Abrahamson (AB) algorithm also implemented in the RspMatch. The LW algorithm is used first to get a RS approximation, and then, the AB algorithm is used to improve accuracy of the computed RS to the target RS.

The EQUAKE module can be used to simulate acceleration histories with random phases, or based on the so-called “seed records” as described in the new ASCE 04-2016 standard. In the latter case, the simulated accelerograms preserve the Fourier phasing of the “seed record” components for X, Y and Z directions.

The EQUAKE module uses a refined baseline correction algorithm that includes the complex frequency algorithm used in the FLUSH code, plus additional polynomial corrections in time-domain. The EQUAKE generated acceleration, velocity and displacement time histories are saved in the text files with the extension .acc .vel and .dis. The user defines the file name for the acceleration time history that is also used for velocity and displacement histories.

The generated spectrum-compatible input accelerations are in compliance with the US NRC requirements included in SRP 3.7.1 for single time-history input, Option 1, Approach 2. The applied SRP criteria include the following aspects:

- Total motion duration is at least 20 seconds; if input duration is less than 20 seconds, a warning message will show up on screen and in the output file;

- 
- The Nyquist frequency is not higher than 100 Hz. The maximum frequency is given by the time step size. If the time step size is larger than 0.005 seconds, a warning will show up on screen and in the output file;
  - Minimum 100 points per frequency decade are used between the lowest and the highest frequencies as defined in the input text file that contains the response spectrum amplitudes (with the extension .rsi in Demo 1). The input file format is two columns, frequency and amplitude for a given damping ratio.
  - The computed 5% damping response spectrum will have no point with more than 10% below the target spectra and no more than 30% larger than the target response spectrum at any frequency;
  - No more than 9 adjacent frequency points falling below the target response spectrum are permitted.

The EQUAKE module also computes a few feature parameters of the computed acceleration time histories; such as strong motion duration (between Arias intensities 5% and 75%),  $V/Av$  and  $AD/V^2$  (A: peak ground acceleration; V: peak ground velocity; D: peak ground displacement). If more than one-component acceleration time history is generated, then, the stationary and non-stationary cross-correlation coefficients are computed. If the user gives the target power spectral density (PSD), EQUAKE checks if the generated acceleration time history meets power spectral density requirements.

The EQUAKE input file has extension .equ and it is created by the AFWRITE command.

The generated accelerograms are then used for site response analysis and SSI analysis through the SOIL, MOTION and STRESS modules. The EQUAKE module computes the response spectra (in .rso file), power spectral density (PSD) (in .psd file) and the positive frequencies portion of the complex Fourier Transform (FFT, in .fft file) of the simulated acceleration histories, or external acceleration histories input by the user. The PSD is computed using a plus/minus 20% frequency averaging intervals in compliance with the ASCE 4 standard and the USNRC requirements. The strong motion duration is defined by the time interval between 5% and 75% Arias intensities. To compute the FFT and PSD only the strong motion duration part is considered.

The .psd file contains a two column data set. The 1st column is frequency and the 2nd column is the computed PSD amplitude. The units depend on the user selection for the gravity acceleration units. If gravity acceleration unit is  $\text{ft/s}^2$ , then, the computed PSD unit is in  $\text{inch}^2/\text{sec}^3$ . If the gravity acceleration units is  $\text{m/s}^2$ , then, the computed PSD unit is  $\text{cm}^2/\text{sec}^3$ .

The .fft file contains a three column data set. The 1st column is frequency, the 2nd column is Fourier transform real part and the 3rd column is the Fourier transform imaginary part. It should be noted that are only positive frequencies are included.

The EQUAKE output file includes the input data information, the generated acceleration time history input parameters, the statistical pair correlations between components for the entire motion duration (stationary correlation) and for a 2-second moving window (nonstationary correlation). The nonstationary correlation values for recorded motion components (NS, EW and Vertical) could be used to generate simulated acceleration histories with the same nonstationary correlation patterns. This is an alternative to the use of recorded motion phasing for simulating acceleration histories. Also, the nonstationary correlation information provides useful insights on the incoming wave patterns for the recorded motions. Nonstationary correlation could be used for computing the principal axes of motion.

**WARNING:** *The nonstationary correlation option is not mentioned in the ASCE standards and the USNRC guides. If the nonstationary correlation option is used, then, the response spectrum compatibility for the correlated components should be regained by inputting the generated correlated acceleration histories as “seed” records and, by running the EQUAKE module again.*

## 2. Module SOIL

The SOIL module performs a nonlinear site response analysis under vertically propagating S waves using the Seed-Idriss iterative equivalent-linear model for the soil hysteretic behavior (included in SOIL-EQL). The input file has extension .soi and it is created by the AFWRITE command. The SOIL module is based on the SHAKE code methodology with some additional programming improvements done over years. The computed equivalent soil properties can be sequentially used in the SSI analysis. In addition to the output file, SOIL produces also other text files with extension .TH that are response time histories for plotting purposes. The TH files include time histories for accelerations (ACCxxx), soil layer strains (prefix SNxxx) and stresses (prefix SSxxx). The xxx notations refers to free-field soil layer number (numbering is done from the ground surface to the depth).

SOIL also produces the text file FILE73 that contains the material soil curves that are used for the non-linear SSI analysis by the STRESS module, and FILE88 with the iterated, equivalent-linear or effective soil properties that are used by SITE is non-linear SSI option is selected by the user.

In the last revision of the SOIL module, we also included a nonlinear time domain capability using a hyperbolic hysteretic soil model (included in the SOIL-NON submodule) based on the DEEPSOIL software theory developed by the Urbana-Champaign university researchers. To make this new capability convenient to the user, the input necessary to run this option is kept to minimal as shown in the new SOIL input window lower part. The output files have the same names as for the equivalent-linear analysis option. The V&V Problem 49 shows the validation of the SOILNON module against the DEEPSOIL program.

**WARNING:** *This nonlinear time domain site response analysis option in SOIL is provided not for direct application to the nuclear projects. This option should not be used for the site response analysis for nuclear licensing projects, except when is fully justified by the analyst. This nonlinear time-integration option is not a part of the standard site response procedures and practice in US for defining the seismic inputs for SSI analysis of nuclear facilities. This nonlinear time domain capability is provided in SOIL only as a benchmark tool useful for soft soil deposits that may have significant nonlinear behavior under severe earthquakes. This option is viewed as an additional, useful engineering tool to get additional insights on the local soil condition effects and compare with the standard procedure based on the equivalent-linear hysteretic soil models.*

**WARNING:** *This nonlinear time domain option is valid only for nonlinear convolution analysis when the input motion is defined by the compatible motion at the bedrock.*

### 3. Module SITE

The SITE module solves the site response problem. The input file has extension .sit and it is created by the AFWRITE command. The control point and wave composition of the control motion has to be defined in the input files. The information needed to compute the free-field displacement vector used is computed and saved on disk in FILE1.

The SITE program also stores information required for the transmitting boundary calculations in FILE2. The actual time history of the control motion is not required in this program module, but later in the MOTION module. In addition to the output and binary files FILE1 and FILE2.

### 4. Module POINT

The POINT module consists of two subprograms, namely POINT2 and POINT3 for 2D and 3D SSI problems, respectively. The input file has extension .poi and it is created by the AFWRITE command. The POINT module computes information required to form the frequency dependent flexibility matrix. The results are saved on FILE3. FILE2 created by program module SITE is required as input. Thus, the SITE module must be run before the POINT2 or POINT3 module.

### 5. Module HOUSE

The HOUSE module forms the element mass and stiffness matrices of all elements. The name of HOUSE module is HOUSEFS for fast-solver code option and HOUSEFSA for Option AA. The input file has extension .hou and it is created by the AFWRITE command. The FEA model may include only the structure, and also the near-field soil, especially of irregular soil zones are present. For the Option AA, the HOUSEFSA input includes uses directly the ANSYS model matrix files, as described in the “ACS SASSI-ANSYS Integration Capability” User Manual.

For the baseline code with the standard solver, the structure and basement stiffness and mass matrices are stored in FILE4. The structure and basement stiffness and mass matrices are stored in the COOSK and COOSM files.

The random field decomposition for incoherent motions is performed in this module. The HOUSE results for incoherent SSI or nonuniform/multiple support excitation option are stored in FILE77 that is later used by ANALYS. If the user desires to check the accuracy of the coherence kernel decomposition, HOUSE can output the incoherent mode contributions. If many incoherent stochastic simulations are executed, then, a set of FILE77s will be produced. Each FILE77 file includes the incoherent motion information for all three directions in space, X, Y and Z.

HOUSE requires FILE74 as input from STRESS and produces FILE78 that is a non-empty file only if the nonlinear SSI analysis option is used. FILE78 is used by STRESS during the SSI nonlinear iterations.

The HOUSE module can be executed independent of SITE and POINT modules, except for embedded models. Thus, HOUSE requires that the SITE input file, the .sit file, to be available in the working directory.

The HOUSE module incorporates an optimizer for the node numbering. This node numbering optimizer is highly effective for large-size SSI models with significant embedment. The Option AA HOUSEFSA module uses automatically the optimizer. We always recommend use of the node numbering optimizer for larger size embedded SSI models. To select the node renumbering optimizer, the analyst can use the UI Options/Analysis/HOUSE input window. After running HOUSE, a new HOUSE input text file with the .hounew extension and a new .map extension text file will be saved in the same working directory. This .hounew input file contains the new optimized SSI model, while the .map file contains the original nodes and the corresponding new node numbers after optimization is applied. The user should use the .hounew node numbers for post-processing.

**WARNING:** *If the node numbering optimizer is used, the new node numbers should be used to select the MOTION and RELDISP node outputs. The optimized model input is described in the .hounew extension file that is automatically used for creating the FILE4 (.n4 extension), COOSK and COOSM files that are further used as input binary files for the ANALYS module run. The FILE8 that includes the complex SSI response solution and is used by MOTION, RELDISP and STRESS is based on the renumbered model.*

**WARNING:** *Option AA uses automatically the HOUSE optimizer and generates a modelname.map file for the node mapping, including the pairs of the new nodes and the old nodes for the renumbered and the original SSI models.*

## 6. Module FORCE

The FORCE module forms the load force vector for external load cases. The input file has extension .frc and it is created by the AFWRITE command. This module is not used for seismic problems, except for computing foundation impedances. The external loads may correspond to impact forces, rotating machinery, or simple unit forces to be used to determine the impedance of a flexible foundation. The results are stored in FILE9. If multiple load cases, up to 500 if the available RAM permits, are used in a single SSI batch run for ANALYS that will save substantial run time, then, the FILE9 file obtained after each load case run using the FORCE module should be copy to a new file that has the same name FILE9 with the number of the load case appended to it, such the FILE9001, FILE9002, FILE9003....FILE9500. Only one digit load case number can be appended to FILE9.

## 7. Module ANALYS

The ANALYS module computes the linearized SSI problem solution for the required frequency steps. The input file has extension .anl and it is created by the AFWRITE command. FILE1, FILE3 and FILE4 are always required as input files. For the external load cases FILE9, and for incoherence analysis FILE77 are also required as input.

ANALYS performs the following computational steps:

1. Forms the soil flexibility matrix
2. Computes the soil impedance matrix at interaction nodes.
3. Determines seismic load vector, including incoherency effects in considered
4. Solves the equation system for each frequency step and obtains complex response transfer functions for each degree of freedom.

The solution output computed by the ANALYS module contains the complex transfer functions which depending on the option required are from the control motion to the final motions or from external loads to total displacements. In either case, the SSI TF results are stored in FILE8 that is used by MOTION, RELDISP and STRESS for computing SSI responses.

To perform the restart analysis, the SSI initiation run should be done with the ANALYS input option checked for "Saving Restart Files". The restart database files are called COOXxxx and COOTKxxx when the xxx denotes the three digits used to define the frequency order number; for example, COOX001 and COOTK001 files are required for performing restart of the 1<sup>st</sup> frequency. The associated index files, COOXI and COOTKI should be also present in the restart analysis working directory. Also, the DOFSMAP, FILE90 and FILE91 files should exist in the working directory for the ANALYS restart. Due to these name formats only the full set of all sequential SSI frequencies can be used for the SSI restart analysis using ANALYS module. If user desires to use a frequency subset he needs to rename the COOXxxx and COOTKxxx files accordingly.

If the global “unconstrained” soil impedance analysis option is selected ANALYS also produces FILE11 that is a quite large size.

**WARNING:** *The global soil impedances should be only applied for surface stick SSI models with rigid basemats. Its accuracy can be crude for elastic foundations, and its interpretation difficult. Please also note that for embedded SSI models the “unconstrained” global impedances are computed correctly only if the FI-FSIN method is used.*

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

It should be noted that the ANALYS module can run simultaneously multiple load cases in a single run without need for any restart. For coherent seismic SSI analysis, ANALYS can run simultaneously the all three X, Y and Z direction inputs. For incoherent seismic SSI analysis, ANALYS can run up to 50 simulations in a single run. For vibration analysis or external forces, ANALYS can run simultaneously up to 500 external forcing load cases. The number of simultaneous cases depends on the SSI model size and available RAM. If the number of simultaneous cases is too high for the available RAM, then, ANALYS will give an error message of “access violation”.

## 8. Module MOTION

The MOTION module reads the transfer functions from FILE8 and performs an efficient frequency domain interpolation using a complex domain scheme. The input file has extension .mot and it is created by the AFWRITE command. The interpolated transfer functions are then, used to compute the SSI response motions at a set of nodes selected by the user.

The MOTION module requires only FILE8 as input. If the baseline correction option is used the computed acceleration, velocity, or displacement response spectra may be requested at different location points and degrees of freedom (this produces a significantly more approximate solution for relative displacements in a structure than using the RELDISP module), the nodal point motions including acceleration, velocity and displacements are saved in the FILE13 text file. The FILE13 format includes 4 columns which provides as each time step, the accumulated time, the absolute acceleration, absolute velocity and absolute displacements.

The computed acceleration histories and response spectra can be optionally saved on FILE12.

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

---

the in-structure response spectra (RS) for selected damping values. These text file names are xxxxxTR\_y.ext, where xxxxx is the node number, y is the DOF that can be X, Y or Z, and .ext is the extension that can be TFU, TFI or ACC. The node number is 5 digits for SSI models with less than 100,000 nodes and 6 digits for SSI models with more than 100,000 nodes. For response spectra files, the file names are xxxxxTR\_yzz.RS, where zz is the order number of the damping ratio value (for example, 01 and 02 for two selected values of the damping ratio of 0.02 and 0.05). See Table 3.1 for more details on the SSI response text files.

If the MOTION post-processing restart option is used, then additional text files for post-processing are generated in the \TFU, \RS and \ACC subdirectories (ACCR is included for rotational accelerations). These frame text files contain the SSI response values computed for all active nodal DOF at each frequency step or time step. These frame files are used by the ACS SASSI PREP module to create structural bubble plots, TF vector plots, contour plots, or deformed shape animations. See Table 3.2 for more details on frame text files. In addition to the ACC frames, MOTION also generates the ACC\_max.txt files that contain the maximum acceleration or ZPA frame.

MOTION can generate also compressed binary databases for the computed SSI response acceleration histories (Modulename\_ACC.bin file). Please see the BINOUT command for this. The COMBACCCDB command can be used to combine the X, Y and Z input direction nodal acceleration histories from the three directional binary databases.

The MOTION module can be also used to generate response spectra for external acceleration histories that are listed in the CONTTRS file. These external files need to have the same format with the .ACC files that contain the acceleration histories computed by MOTION based on SSI analysis solution. The external acceleration file names need to have the same extension .ACC. The computed response spectra will be saved in files with the same name and the .RSO extension.

## 9. Module STRESS

The STRESS module computes requested stress, strain, and force time histories and peak values in the structural elements. The input file has extension .str and it is created by the ACS SASSI UI AFWRITE command. The module STRESS requires FILE4 and FILE8 as inputs. STRESS produces FILE74, if the nonlinear SSI analysis option is employed. For nonlinear SSI, STRESS also uses FILE78 produced by HOUSE as an input.

In addition to the output file STRESS produces also some specific text files useful for post-processing. These text files include the extension .TFU,.TFI and .THS that contain structural element stress responses in each selected element, respectively, the computed TF (TFU),

interpolated TF (TFI) and stress time histories (THS). These text file names have the format `etype_gnum_enum_comp` plus extension; for example, `BEAMS_003_00045_MXJ` that contains the MX moment at node J for the BEAM element number 45 that belongs to Group 3. See Table 3.1 for more details on SSI response text files.

STRESS can also generate also compressed binary databases for the computed SSI response element output histories (Modulename\_STRESS.bin file). Please see the BINOUT command for this. The COMBTHSDB command can be used to combine the X, Y and Z input direction element component output histories from the three directional binary databases. See also Demo 9 for the post-processing of the element stresses saved in separate directional binary databases.

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

Table 3.1: Useful Text Files for the ACS SASSI Result Verification and Post-Processing; Node numbers requires 6 digits for SSI models with more than 99,999 nodes.

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

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

Table 3.2: Post-Processing Frame Files Produced by MOTION, RELDISP and STRESS

The STRESS module in addition to the above files also generates an important text file named ELEMENT\_CENTER\_ABS\_MAX\_STRESSES.TXT that contains the maximum element stress components in the center of the elements (calculated by STRESS) for all elements.

**WARNING:** This ELEMENT\_CENTER\_ABS\_MAX\_STRESSES.TXT file is generated only if all the element stresses are selected in all elements using the post-processing check box.

**WARNING:** This ELEMENT\_CENTER\_ABS\_MAX\_STRESSES.TXT file with the computed maximum element center stress results are not usable for the beam elements, for which the nodal forces and moments are needed, i.e. not in the center of the elements, but at the nodes. The center stress results are useful only for shell, solid and spring elements.

The format of this file is as follows for a model with n groups:

```
[ # of groups]
[1st group element type] [group #] [ordered group #] [# of elements in group]
[2nd group element type] [group #] [ordered group #] [# of elements in group]
...
[nth group element type] [group n] [ordered group #] [# of elements in group]
[1st group element type] [group #] [ordered group #]
[element #] [stress comp 1] [stress comp 2] [stress comp 3] [stress comp 4] [stress comp 5] [stress comp 6]
```

Example of ELEMENT\_CENTER\_ABS\_MAX\_STRESSES.TXT

```
3
SOLID      1      1      3
SHELL      2      1      1
SOLID      3      1      5
SOLID      1      1
1 0.69504 0.61290 0.93326 0.21454 1.36011 0.45008
2 0.82394 0.70086 0.68225 0.20217 0.65360 0.34301
3 1.63296 1.09535 1.41437 0.49395 1.40079 0.39915
SHELL      2      1
1 6.98477 12.51727 9.28106 0.87311 0.58065 0.44423
SOLID      3      1
1 1.07909 1.12969 1.91468 0.10359 1.03872 0.25992
2 0.33023 0.38077 0.49889 0.08123 0.41770 0.15688
3 0.53114 0.43295 0.65837 0.08003 0.33819 0.07529
4 0.54051 0.49153 0.96988 0.14669 1.12478 0.26478
5 0.22013 0.31832 0.54288 0.15972 1.58830 0.29268
```

---

If the STRESS post-processing restart option is used, then additional text frame files for post-processing are generated in the \NSTRESS subdirectory. These text frame files are used by the ACS SASSI UI module to create structural node stress contour plots, static (for a selected time or for maximum stress values) or animated. The STRESS post-processing handles only SOLID and SHELL elements for 3D SSI models. If the SSI model contains both SOLID and SHELL elements, the frames include only average node stresses for the membrane stresses. For the SHELL elements only, separate frames are generated for the average node bending stresses (the file extension include letters bd from bending). See Table 3.2 for more details on frame text files.

If the SSI model includes near-field soil elements that are adjacent to the foundation walls, then the soil pressure frames can be generated. The soil pressure frames are saved in \SOILPRES subdirectory. In addition to the seismic soil pressures frames at each time step, a single frame with maximum soil pressures is also generated. The user can also create total soil pressure frames including the static bearing pressures plus the computed seismic pressures. The static pressure text file is named STATIC\_SOIL\_PRESSES.TXT and is generated when the soil pressure frames are requested. The format of this file is the same as for the ELEMENT\_CENTER\_ABS\_MAX\_STRESSES.TXT file, except that there is only one column for the element soil pressures. When it is generated the first time by the STRESS restart analysis for soil pressure option, the static pressure file has only zero values. Then, if the user inputs the non-zero static pressure values and runs again the STRESS post-processing restart for soil pressure option, these non-zero static pressures are added to the seismic pressures values using algebraic summation and the total soil pressures are saved in the soil pressure frames stored in the \SOILPRES subdirectory.

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

**WARNING:** *There is no text frame files generation for the TSHELL element.*

**WARNING:** *The STRESS frame files contain average nodal stresses and average nodal pressures to be used only for plotting purposes. The nodal stresses and soil pressures were computed directly from the SOLID element center stresses or pressures (normal stress to the solid element face). The nodal stress was assumed to be equal to element center stress that introduce a certain level of approximation of the nodal stresses (no shape functions are used). In addition, the nodal averaging process could produce stresses and pressures could produce values that are difficult to interpret and use. The accurate stress and soil pressure values to be used by the analyst for the SSI calculations and seismic design are the computed values in the element centers (that are provided in the STRESS outputs, or the text files called*

---

*ELEMENT\_CENTER\_ABS\_MAX\_STRESSES.TXT* and *pres\_max\_ele* if post-processing option is activated), not the nodal average values. However, the average nodal stress and soil pressure add invaluable information for understanding the SSI model seismic behavior and for identifying the critical stress zones, or critical pressure areas on the foundation walls and mat.

For the nonlinear SSI analysis option, STRESS generates the FILE74 after each SSI iteration. FILE74 is then used by HOUSE for the next SSI iteration. The COMB\_XYZ\_STRAIN module is companion to STRESS module and combines the directional shear strains using SRSS for the three seismic input directions.

The element output frames at all time steps which are named ESTRESS\_framenumber.ess are also available in the UI Options/Analysis/STRESS input window. These files are required for the Section-Cut calculations as illustrated in the Demo 8 and the Problem 47 of Verification Manual.

**WARNING:** *There is no text frame ESTRESS files generation for the TShell element.*

## 10. Module RELDISP

The RELDISP module uses the interpolated complex ATF produced by MOTION (.TFI files) to compute the relative displacements at different selected nodes. The input file has extension .rdi and it is created by the AFWRITE command. RELDISP produces an output file with the computed maximum nodal relative displacements. This module also produces extension .TFD and .THD files that contain the nodal relative displacement complex TF and the relative displacement time histories. See Table 3.1 for more details on the SSI response text files.

If the RELDISP post-processing restart option is used, then additional text files for post-processing are generated in the \THD subdirectory (includes \THDR for rotational displacements). These frame text files contain the SSI response values computed for all active nodal DOF at each time step. These node frame files are used by the UI module to create structural deformed shape animations. See Table 3.2 for more details on frame text files.

RELDISP can also generate also compressed binary databases for the computed SSI response relative displacement histories (Modulename\_STRESS.bin file). Please see the BINOUT command for this. The COMBTHDDB command can be used to combine the X, Y and Z input direction element component output histories from the three directional binary databases. However, for the relative displacement combination of X, Y and Z input direction, it is needed to run RELDISP three times for each direction to compute the principal responses and coupled responses that are saved in separate binary databases; for example, for the X-input direction, the three binary databases will be Modelname\_TR\_X\_THD.bin, TR\_Y\_THD.bin and TR\_Z\_THD.bin.

To combine these three binary database in a single database (Modelname\_THD.bin), the COMBDISPDIR command shall be used. This command should be used for each input direction.

### **11. Module COMBIN**

The COMBIN module combines results computed for different frequencies from two ANALYS runs. This module is useful when after the solution was obtained it is found that some additional frequencies are needed to be included. The COMBIN module requires two solution files of FILE8 type as inputs. These input files are renamed FILE81 and FILE82. The output file of this module is a new FILE8 obtained by combining the two solution files.

### **12. Module NONLINEAR**

The NONLINEAR module functionalities and options are discussed in detail in Section 1.5.4 and Demos 9 and 10. The computational steps involved by the NONLINEAR module are:

- Perform the initial linear SSI analysis using the initial elastic properties for the nonlinear elements
- Compute the local behavior of nonlinear elements in time domain based on the local relative displacements, that is then used to calibrate the local linearized hysteretic models associated to each nonlinear element in complex frequency
- Perform a new SSI analysis iteration using a fast SSI restart analysis in the complex frequency domain using the linearized hysteretic models computed in Step 2 for nonlinear elements
- Check convergence after new SSI iteration to stop or continue.

The COMB\_XYZ\_THD module is companion of the NONLINEAR module and is used to combine co-directional displacement histories for X, Y and Z inputs as described in Section 1.5.4 and illustrated in Demos 9 and 10.

## **3.2 PERFORMING SSI INITIATION AND RESTART ANALYSES**

To be able to run successfully ACS SASSI SSI analysis the user should maintain all the text input files produced by the AFWRITE command in the same working directory. The sequence for a linear seismic SSI analysis is EARTHQUAKE (if needed), SOIL (if needed) and then, SITE, POINT, HOUSE, ANALYS, MOTION (if needed) and STRESS (if needed). The sequence for the linear external force vibration analysis SSI analysis is SITE, POINT, FORCE, HOUSE, ANALYS, MOTION (if needed) and STRESS (if needed).

---

For nonlinear soil SSI analysis, in addition to the regular SSI module input files, two additional input text files are needed. These are the files with extensions .liq and .pin (see HOUSE input definition for nonlinear SSI analysis option). The .liq extension file is generated by HOUSE during the initial run. If .liq file is not empty and contains number 1, then, HOUSE will use FILE74 for the model input materials for the next SSI iteration. A similar file with the .liq file, called PANEL.NON or SPRING.NON exists also for the NONLINEAR module run for nonlinear panels and nonlinear springs. If file the .non extension file exists and contains the number 1, then, NONLINEAR will use the previous iteration SSI results and run a nonlinear analysis for a new SSI iteration. If .non is not existent, then NONLINEAR will run a linear analysis for the 1<sup>st</sup> iteration.

The ACS SASSI SSI analysis runs can be done interactively or in the batch mode. If the SSI runs are done interactively, then, each SSI module is executed at a time under user control using the UI “Modules” menu selections, or a set of SSI modules is executed automatically using Options\Write “Running SSI Modules” menu.

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

```
SSI_module_name.exe < SSI_module_name.inp
```

where SSI\_module\_name could be SITE, or POINT or ANALYS. The SSI module executables are installed by default in the C:\ACSV300\EXEB directory and are also provided with the ACS SASSI installation DVD for the Batch run mode. Each input file with the SSI\_module\_name and the extension .inp contains only three input lines:

```
modelname  
modelname.ext_input  
modelname_SSI_module_name.out
```

where ext\_input is the extension provided by the AFWRITE command.

For the NQA version, the V&V runs are prepared to be run by the V&V analyst in the batch mode. The V&V batch run file also provides a set of detailed examples on how to build efficiently SSI analysis batch run files.

The following changes of problem parameters are required for different types of dynamic SSI analysis restart or reanalysis:

- a. Change in the control motion: “New Time History” (MOTION restart)

Suppose results are required for a different time history or response spectrum of the control motion. Then, as long as the nature of seismic environment, i.e., the type of wave field, is not

changed, only the module MOTION has to be re-executed.

b. Change in seismic environment: "New Seismic Environment" (ANALYS restart)

Suppose that structure was originally analyzed for the effects of vertically propagating SV or SH waves and after that the user wants to input other types of waves for example, vertically propagating P-waves, or Rayleigh waves, causing the same motion at the control point as in the free field. In these cases the SITE and ANALYS modules have to be re-executed. If the incoherency of seismic motion is considered, then, since incoherency modify the seismic loads on structure, the HOUSE and ANALYS modules have to be re-executed.

c. Change in dynamic loading: "New Dynamic Loading" (ANALYS restart).

If changes are made in dynamic loads applied directly on the structure, only program modules, FORCE, ANALYS and MOTION have to be re-executed. However, if only the time history of dynamic loads is changed while the loading pattern is not changed, only the program module MOTION has to be re-executed.

d. Change in structure or near-field soil: "New Structure" (ANALYS restart)

If changes are made in the structure or near field soil without changing interaction nodes, the HOUSE, ANALYS and MOTION and STRESS modules have to be re-executed. For non-linear SSI analysis iterations, the STRESS module computes the effective soil shear modulus and damping in the near field soil elements and transmits to HOUSE the new iterated values through FILE74.

---

## 4 Application Guidelines

A set of guidelines describing the step-by-step procedure for a SSI analysis is given in this section. This section includes two sub sections 4.1 and 4.2.

Section 4.1 describes the steps for typical linearized SSI analysis as implemented in ACS SASSI. In the last part of this section, these steps are further elaborated and the important parameters of the analysis are discussed in more detail.

Section 4.2 describes the ACS SASSI modules which are to be executed to perform the initiation, post-process and restart analysis.

### 4.1 SSI ANALYSIS PROCEDURE

#### 4.1.1 Computational Steps for Linearized SSI Analysis

The seismic SSI analysis involves the following preliminary steps:

- Step 1. Generate the spectrum-compatible acceleration input time history set of the control motion using the EQUAKE module in compliance with the ASCE 4 standard and USNRC SRP 3.7.1 guidance.
- Step 2. Determine the important frequency ranges for the SSI response by examining the dominant frequencies of the structure. These frequencies can be obtained for fixed-base FE model using a standard finite element program via modal analysis. The ACS SASSI UI has automatic converters to import or export FE models from ANSYS to ACS SASSI and vice versa. Based on the initial SSI analysis, additional frequencies should be considered to capture correctly all structural vibration modes. The SSI solution for the new added frequencies should be computed independently and then the result database for original and added frequencies should be integrated using the COMBINED module.
- Step 3. Based on the control motion frequency content, plus the 90% cumulative modal mass criterion given in Section 3 of ASCE 4 standard, determine the cut-off frequency of the SSI analysis. The cut-off frequency should be equal or smaller than the maximum wave passing frequency thru the mesh of the soil layering and the FE model.
- Step 4. Based on the vertically propagating seismic wave assumption, specify the location of the control motion, and compute the free-field iterated strain-compatible soil properties using the SOIL module.
- Step 5. Consider the linearized soil profile, based on the free-field iterated soil properties obtained from the SOIL module, for the site response analysis using the SITE module. This can be done automatically by the user, by selecting the “Nonlinear” option in the SITE module input dialog. For embedded SSI models, the ACS SASSI

---

UI also automatically transfers the iterated soil profiles to the HOUSE model input for excavated soil layers.

- Step 6. Build the structure FE model, or models for multiple structures, including eventually backfill soil as the near-field soil zone.
- Step 7. Build the excavated soil FE model for each embedded structure considered in the model. For adjacent structures, the multiple excavated soil volumes can be merged in one larger soil excavation volume.
- Step 8. Select the computational method for computing the impedance matrix and performing SSI analysis, such as FV, FI or FFV.
- Step 9. Select the frequencies for which the site response and point load problems are to be solved. It is recommended to choose at least 40 to 80 frequencies for simple stick models and 100 to 300 frequencies for complex FE models. These frequencies are selected according to the information obtained from Steps 1 and 2 and later on, can be increased, if necessary, to improve the accuracy of the interpolated transfer functions.
- Step 10. Perform initiation SSI analysis to compute the acceleration complex transfer functions at all the nodes of the SSI system.
  - a. Execute the SITE module in Mode 1 and 2 based on the information of Steps 5 and 9 and the specified location of the control motion. This analysis yields the information needed to form the transmitting boundary in the program module POINT. The Mode 1 information is saved in FILE2. Mode 2 performs the site response analysis by using FILE2 as input and the .sit input file that describes the nature of the seismic wave field. This analysis yields a set of free-field motions which are saved in FILE1.
  - b. Execute the POINT module using also FILE2 as input and specifying the maximum embedment of the structure and also the radius of the point load in the .poi input file. This analysis yields the information on point load solution, saved in FILE3.
  - c. Execute the HOUSE(FS) fast-solver module based on the information of Steps 6, 7, and 8. The HOUSE run yields to the complex stiffness and mass matrices of the structure and excavated soil that are saved in the FILE4 that (is renamed as modelname.N4) and the COOSK and COOSM matrix files. If Option AA is used, then, HOUSE(FSA) run will need as inputs additional ANSYS FE model matrix files, as described in the “ACS SASSI-ANSYS Integration Capability” User Manual. In Option AA is used for defining the FE model topology (nodes and element connections) needed also for equation mapping and post-processing using the ACS SASSI UI.
  - d. Execute the ANALYS(FS) fast-solver module using the FILE1 (for seismic option), FILE9 (for external force option), and FILE3 and FILE4 (modelname.n4) as inputs. The ANALYS run yields to the soil impedance matrices, the triangularized stiffness of the total system, and the acceleration

---

complex transfer functions (ATF) for all the user requested SSI frequencies and all the nodal points in FILE8. The frequencies selected at this stage for interaction analysis are based on the information obtained from Steps 1 and 2.

For large-size embedded SSI models, to shorten the execution time of the ANALYS module, as well as limit the size of the scratch files generated by this program, it is recommended to break the frequency range into several frequency subranges and perform separately the SSI analysis for each subrange.

- Step 11. Based on the SSI solution obtained in Step 10, i.e. using the computed transfer functions, compute the response of the system in time domain. This step, usually consists of the following computer runs:
- a. Execute the COMBIN module, if necessary, to combine the transfer function databases (FILE8s) obtained for different frequency subranges in Step 10.
  - b. Execute the MOTION, RELDISP and STRESS modules, based on the results of Step 10 and item a. of this step, to compute the complex response of the system. Please check the computed and interpolated ATFs from MOTION and STFs from STRESS to clarify if additional SSI frequencies are needed to improve the accuracy of the SSI analysis results.
- Step 12. At this stage additional new SSI frequencies might be considered based on the review of the ATF and STF results in Step 11.
- Step 13. Perform a restart SSI analysis if changes in either the superstructure (“New Structure” restart), or the seismic environment (“New Seismic Environment” restart) occur. In the case of forced foundation vibration problems:
- a. The time history of control motion in Step 1 is replaced by the reference time history of the external dynamic forces.
  - b. Step 4 is of no use.
  - c. The iterated soil properties in Step 5 are replaced by the initial soil properties.
  - d. The site response problem is eliminated from Step 9 and second part of the item a of Step 10.
  - e. Item d. of Step 10 is replaced by a different analysis performed by the FORCE module to obtain the load vector on FILE9, which replaces FILE1 in item e. of Step 10.
  - f. The dynamic environment in Step 13 is replaced by the external dynamic forces.

---

### 4.1.2 Engineering Considerations

In order to make effective use of the ACS SASSI program, in addition to the considerations in Section 1.5, the user should also consider the following important SSI modeling aspects:

#### 1. Rigid vs. Flexible Basement

The basement is always assumed to be flexible in the FE modeling. As a result, no saving is obtained if a less realistic assumption such as the rigid basement assumption is used. It is recommended to carry out the SSI analysis with actual visco-elastic properties of the basement. If it is necessary to evaluate the effect of the rigid basement assumption, a restart analysis can be performed by selecting a value for the elastic modulus for the basement that is  $10^4$  and  $10^5$  times larger.

#### 2. Surface vs. Embedded Structure

It is possible in practice to treat an embedded structure as surface structure if the embedment effects are not significant on the SSI responses of interest. By neglecting the embedment effects, a large run time saving can be achieved for solving the SSI problem, since the number of interaction nodes is largely reduced for surface foundation. However, the neglect of embedment is not necessarily conservative for all frequencies. In narrow frequency ranges, it could be that sometimes, there will be response amplification for the embedded model at some locations. The neglect of embedment is in principle globally conservative in terms of structural forces, but this does not guarantee that some in-structure response spectra (ISRS) might exhibit some amplifications and new spectral peaks for narrow frequency bands.

#### 3. 2D SSI vs. 3D SSI

The 2D SSI analyses are not recommended, especially for the soil sites due to their inability to model correctly the complex dynamic stiffness of the foundations. The 2D SSI models have usually a larger radiation damping, and therefore, they are expected to be unconservative. The use of 2D SSI models are *justified* for sensitivity studies on the 2D soil layering configuration effects of SSI responses. The 2D SSI sensitivity studies could include the effects of nonuniform, sloped soil layering, inclined baserock or topographic features. The ACS SASSI has also the capability to perform the nonlinear site response analysis for 2D soil profiles using equivalent linearization. For particular problems, this capability is very important. The 2D nonlinear site response analysis is also valuable for building site-specific directional incoherency models that include the effects of specific features of the soil layering at the NPP site.

#### 4. Coherent vs. Incoherent Motion

The ACS SASSI code has the capability to consider incoherent motions for seismic analysis more realistically than any other SSI analysis program. The incoherent motion effects are induced by the fact that the free-field soil motion has not identical motions in all points in the horizontal plane. Until last decade, the incoherent motion effects were generally ignored, although they are a part of the seismic wave propagation physics, as already proved by the real dense arrays earthquake records in the recent past.

It has been shown that the incoherent motion effects may affect the seismic response of large foundation size buildings, such as those of NPP facilities. Please note that incoherent motion effects can also amplify the structure-soil-structure interaction (SSSI) effects, especially for soil sites (Ghiocel 2015b).

The incoherent seismic motion option is limited to 3D models. Also, quarter-models and half-models and 2D SSI models are not permitted.

The ACS SASSI code can consider both the seismic motion incoherency and the directional wave passage effects. Both stochastic and deterministic incoherent SSI approaches could be employed. A number of the incoherent SSI approaches implemented in ACS SASSI were validated during the 2007 EPRI studies (Short et. al., 2007) and then, further endorsed by US NRC (ISG-01, 2008) for application to the new advanced nuclear reactor projects. Additional information on these incoherent SSI approaches could be find in other publications (Ghiocel, 2007, 2009a, 2009b, 2013b, 2014b, 2015b).

The ACS SASSI software includes seven incoherent SSI approaches, namely, five simplified deterministic approaches based on the so-called AS and SRSS approaches used in the 2007 EPRI studies. However, only two of the implemented deterministic approaches were benchmarked in the 2007 EPRI studies. The stochastic simulation approach, called Simulation Mean, was considered the *reference* incoherent SSI approach in the 2007 EPRI report (Short et. al., 2007).

ACS SASSI includes a total of seven plane-wave incoherency models incorporated in the code: the Luco-Wong model, 1986 (Luco and Wong, 1986, Model #1, theoretical, but not validated in practice), and five Abrahamson models (empirical, based on seismic dense array records, Models #2 through #6), 1993 and 2005 models for all sites and surface foundations (Abrahamson, 1993 and 2005, Models #2 and #3, respectively), 2006 model for all sites and embedded foundations (Abrahamson, 2006, Model #4) 2007 model for hard-rock sites and all foundations and 2007 model for soil sites and surface foundations (Abrahamson, 2007, Models #5 and #6, respectively), and the user-defined coherency model, Model #7.

---

For all incoherent SSI analysis applications, but especially for those with larger size, elastic foundations, we strongly recommend the use of the stochastic simulation approach that is both accurate and numerically efficient. It also includes additional useful information for the analyst' engineering judgment on the incoherent SSI response statistical variation (motion incoherency is a random phenomena). The AS and SRSS deterministic approaches are both approximate and with limited application to rigid foundation SSI models, as validated by the 2007 EPRI studies. The AS and SRSS deterministic approaches could be applied to simple stick models with rigid mats. AS and SRSS were evaluated by EPRI only for stick models with rigid mats. Their use for complex FE models with flexible foundation have to be demonstrated by preliminary sensitivity studies that should indicate that at the corner locations of the structural floors the ISRS estimates obtained using AS or SRSS are reasonably accurate or conservative with respect to stochastic simulation.

The SRSS approach is more difficult to use, and is more time-consuming than the stochastic simulation approach, since it needs separate SSI analyses for the incoherent spatial modes. The SRSS approaches has been made available in ACS SASSI per nuclear industry request for benchmarking purposes, rather than for their practicality, accuracy or numerical efficiency merits.

The SRSS results could be improved at a certain level by adding more incoherent spatial modes in the SSI analysis. For stick models with rigid mats the use of 10 incoherent modes is usually sufficient as shown in the 2007 EPRI studies, but for large-size elastic foundation mat SSI models, the use of tens or even hundreds of incoherent modes might be needed to get a reasonable accuracy for the SSI responses (Ghiocel, 2013b, 2014b, 2015b). SRSS has also a conceptual limitation since it neglects the couplings between the closely-spaced incoherent modal responses.

The ACS SASSI can be also used to apply the SRSS approach used in the 1997 EPRI report (Tseng and Lilahanand, 1997), but not validated by the 2007 EPRI studies. This SRSS approach is called herein SRSS FRS. The SRSS FRS uses the SRSS rule to combine the end results computed for different incoherent modes that is can be applicable to ATF amplitudes, ISRS, maximum displacements or ZPAs. The SRSS FRS is different than the SRSS TF approach.

The SRSS TF which was validated by the 2007 EPRI studies uses the SRSS rule to combine the ATF amplitudes for different incoherent modes assuming that their phases are zero (as for static responses). The resulting ATF with the SRSS combined amplitudes are then used to compute the ISRS results.

***WARNING:*** For flexible foundation SSI models, the number of the required incoherent modes in the SRSS approaches could be very large, tens, or even hundreds, that makes the SRSS approaches highly impractical for complex SSI model problems. This aspect is critical for the vertical direction for which the basemat is much more flexible, and tens of incoherent modes might not be sufficient to capture accurately the high frequency ISRS spectral peaks at some locations. It should be also noted that deterministic SRSS approaches have no way to uniquely and correctly

---

*select the incoherent eigen-mode signs, plus or minus, that are sensitive to the interaction node numbering, and affect the modal complex ATF responses.*

**WARNING:** *For computing structural forces, the SRSS TF approach should be applied in conjunction with the STF modal responses. However, such an approach was not validated in the 2007 EPRI studies and is not currently implemented in ACS SASSI.*

We do not recommend using the AS and the SRSS approaches for licensing projects.

Sometime in practice, for the embedded models, the mesh for each embedment level could be different. Since ACS SASSI uses the projection of all interaction nodes on the horizontal plane for computing the free-field coherency matrix, it could happen that for such cases some node locations are very close, and this might create an ill-conditioned coherency matrix. For example, if the surrounding backfill soil mesh included in the FE model has a slightly inclined node plane on the perimeter surface from the vertical plane, then, the projection of some nodes is almost at the same locations which creates an ill-coherency matrix. In this particular case, the “per level” approach could be used.

**WARNING:** *For the multilevel embedment SSI applications, the user should investigate if there is any problem with the coherency matrix eigen-solution. This can be done by extracting additional HOUSE output information on the incoherent mode contributions. A final checking should be reviewing the ANALYS ATF output for the zero-frequency solution (or frequency number 1) that in the input direction should have a value close to 1.00. Large deviations of the zero-frequency ATF amplitude from 1 indicates that the coherency matrix is ill-conditioned due to the space distributions of interaction nodes with the embedment. For such unusual situations, the incoherent modes can be determined separately for each interaction node embedment level, and then, the coherency matrix eigenvectors can be assembled for the entire set of interaction nodes using the DOS auxiliary program called Build\_FILE77.exe that is provided at the installation time in the default subfolder C:\ACSV300\Build\_FILE77. However, BuildFILE77 will provide accurate results ONLY if the incoherent modes at different levels maintain the same node numbering pattern, so that the incoherent eigen-modes are not flipping their signs at different levels. This may imply a significant checking effort for each mode at each level.*

The “per level” approach is also required for deeply embedded foundations in soil sites for which the coherence functions at the surface and the foundation levels are different. *Additional consulting tech support is recommended for such complex situations for which additional checking efforts are required.*

#### 5. Isolated vs. Multiple Structures

Using ACS SASSI, the structure-soil-structure interaction (SSSI) problems can be analyzed in 3D space. As shown by recent investigations, these SSSI effects can be significant on a case-by-case situation, especially for soil sites. The SSSI effects can be significant, depending on a

particularity of the problem. The SSSI affects mostly the local responses, such as ISRS, and seismic soil pressures on the foundation walls and mats (Ghiocel, 2014, 2015b).

The existence of different foundation levels for neighboring buildings usually amplify significantly the SSSI effects and increase the soil pressures on the embedded walls.

The motion incoherency effects could also affect the SSSI effects, especially for softer soil-sites where both SSSI and incoherency effects are both significant. The motion incoherency could produce significant reduction of the SSSI inter-building spacing, eventually produce impacts of the buildings. For soil sites, the minimum gap sizes between buildings computed for incoherent motion are about 2-3 times larger than for coherent motion. Incoherent SSSI analyses should be based on the stochastic simulation approach with no phase adjustment that provides the “theoretically exact” approaches based on Monte Carlo simulation.

#### 6. Non-uniform Seismic Motion Input

Results have shown that for highly nonuniform soils in horizontal plane, the seismic input is not the same over the foundation area, especially for the large-size foundations, or for multiple structure foundations in SSSI models. The in-situ soil columns measured at different boreholes could be different. Often, the soil columns from different in-situ boreholes could indicate slightly different soil motion amplifications to ground surface. The ACS SASSI code can consider a variable amplitude seismic motion inputs for a single continuous foundation, or for separate foundations. In this case, the foundation should be divided into multiple zones that for each one the user input a slightly different seismic excitation motion using frequency-dependent amplitude amplification factors. Motion incoherency and wave passage effects can be included in addition to nonuniform or multiple input excitations to create a more realistic seismic environment. Up to 5,000 zones can be considered to define nonuniform soil motions under a foundation.

#### 7. Symmetry of the System

The ACS SASSI code has the capability to take advantage of the symmetry of structure subjected to either symmetric or anti-symmetric loading patterns. The SSI analysis runtime can be drastically reduced by utilizing this capability of the program. The parameter in the SYMM command is used to specify the number of symmetrical planes of the system.

**WARNING:** *The incoherent motion and impedance evaluation options are not applicable for half or quarter models.*

#### 8. Rigid Base Rock vs. Halfspace Condition

The program has the capability to simulate the existence of a visco-elastic halfspace below the bottom of the user-defined soil layers. In case of the visco-elastic halfspace simulation, the program automatically adds a set of additional halfspace soil layers below the user-defined soil

---

layers with the overall variable depth of  $1.5 * V_s / f$  where  $V_s$  is the shear wave velocity of the halfspace and  $f$  is the frequency of analysis. This added “buffer” of computational layers are automatically defined in the SITE module input based on the user input for the selected number of generated halfspace layers. In addition to these computational halfspace layers, viscous dashpot elements are added at the bottom of the layered soil deposit. The input parameter for the number of halfspace layers is recommended to be in the range of 10-20 sublayers. If number of computational sublayers is zero, there is no visco-elastic halfspace simulation.

#### 9. SSI Cut-off Frequency

The SSI cut-off frequency is an important SSI analysis parameter since it not only sets an upper limit on the number of frequencies to be analyzed, but also controls the SSI model size by constraining the maximum allowable element sizes, and thus, the sizes of the stiffness and mass matrices of the SSI problem. The SSI cut-off frequency depends on the seismic input frequency content and the structure dynamic behavior.

The factors governing the selection of the cut-off frequency are:

- a. The frequency content of the input motion.
- b. The dominant frequencies of the entire dynamic system
- c. The time increment of the input time history.

The cut-off frequency varies typically between 30Hz-40Hz for soil sites, and 60Hz–70Hz for rock sites.

#### 10. Selection of SSI Frequencies

The frequencies to be selected for the SSI analysis depend on the frequency components of the input motion and system response, i.e. the narrowness and the amplitude of spectral peaks, and how close the spectral peaks are located relative to each other. This information can be preliminary evaluated based on the modal fixed-base analysis of the structure and engineering judgment from previous experiences. The fixed-base natural frequencies will give some hints on the approximate location of the peaks in the structure. The importance of each peak can be seen from either the mode participation factors or the fixed-base transfer function shapes. However, the user is warned that the SSI effects may drastically affect both the effective stiffness and damping of the structure-soil system and produce significant frequency shifts. Therefore, the analyst should be careful interpreting the fixed-base results for extrapolation to the SSI results.

The frequency interpolation scheme of complex transfer functions incorporated into the program, since the effect of the soil-structure interaction is to flatten the sharp peaks, sometimes eliminates some of the structural peaks. Usually using no more than 40-80 frequencies is sufficient for the SSI solution for simple structures or stick models. However, for complex FE models the number

of frequencies should be between 100 and 200 for coherent analysis, and between 200 to 300 frequencies for incoherent analysis.

The complex transfer functions for nodal acceleration and element stresses/forces are then computed for all the Fourier frequencies up to cut-off frequency by interpolation in the complex frequency domain. ACS SASSI uses seven smart interpolation schemes to compute transfer functions at the Fourier frequencies.

If an interpolated transfer function peak is obtained close to the mid-point of an interval defined by two consecutive frequencies and its interpolated amplitude is significantly higher than the neighboring computed amplitudes, it is recommended to include an additional frequency. Therefore starting the analysis with fewer frequencies (e.g. 80) is possible for coherent SSI analyses. By then examining the ATF and adding new frequencies, the significant SSI peaks can correctly be captured. The ACS SASSI UI module commands and macros can be used to create an automatic algorithm to compute the new required frequencies needed based on the identification of the ATF spectral peaks that are obtained by interpolation. Demo 3 shows an example on this. Also, the TFU-TFI plot capability that can be selected from the UI menu to compare the computed ATF (TFU extension files) with the interpolated ATF (TFI extension files). For incoherent SSI analysis we recommend use at least 200 SSI frequencies from the start.

#### 11. Computation of SSI Frequency Points (NFREQ) and Steps

The transfer functions are computed at discrete frequency points which are integer multipliers of the frequency step. The total number of SSI analysis frequencies is 500. For general analysis, the frequency step, DF, is calculated by  $DF = 1/DT/NFFT$ , where the parameters DT and NFFT are the time step and number of points to be used in the Fourier transform of the time history, respectively. For single harmonic forced vibration analysis, the time history input is not required, therefore, the user can directly specify DF.

Once the frequency step is defined, the number of frequency points,  $f_i$  are determined through the use of integer frequency numbers, NFREQ<sub>i</sub>, defined as follows:

$$NFREQ_i = f_i / DF \quad i = 1, 2, \dots, NF \quad (4.1)$$

where NF is the total number of frequency points selected for the analysis according to item 6 of this section. The maximum frequency number to be specified is controlled by the cut-off frequency and can be obtained as follows:

$$NFREQ_{NF} = f_{NF} / DF \quad (4.2)$$

where  $f_{NF}$  is the cut-off frequency.

---

## 12. Soil Deposit Modeling

The foundation soil is defined by semi-infinite elastic/viscoelastic layers resting on a rigid base rock or viscoelastic halfspace. The allowable layer thickness for the SSI analysis is determined using the simple rule that the layer thickness must not exceed one fifth of the wavelength at the highest frequency component of the SSI analysis. This criterion is based on the thin layer method assumption that displacements within each layer varies linearly. Based on this 1/5 wavelength rule, the geotechnical soil profile, usually, is subdivided into a number of computational sublayers that satisfy the criterion.

**WARNING:** *For deep soft soil deposits having non-uniform variations with depth, a large number of soil layers could be required to transmit a sufficiently high frequency content for vertically propagating waves. In these situations, the user should carefully revise and understand SSI results by inspecting the computed transfer functions at different node locations. In some situations, when soil properties are not uniform with depth, or when Poisson ratios are large, above 0.47, numerical instabilities could occur at isolated frequencies in the free-field solution. This result can affect SSI response at few isolated frequencies. By inspecting the computed acceleration transfer functions (ATF) at several nodal locations, the frequencies that manifest instabilities can be identified. The computed results for those frequencies need to be dropped out from SSI analysis. The SSI solution at these frequencies should not be considered for interpolating the complex transfer functions in the MOTION and STRESS modules. Sensitivity studies by considering adjacent frequencies to those that are suspected to produce spurious results, are highly recommended.*

Irregular backfill soil zone adjacent to the structure can be incorporated into the FE model. Also, if the analyst wants to get the seismic soil pressures on the foundation walls and mat and include the local nonlinear soil hysteretic behavior, then he needs to include in the SSI model one or few adjacent soil elements surrounding the foundation walls and under basemat. These adjacent soil elements could be solid elements for 3D problems and plane elements for 2D problems. However, it should be noted that these adjacent soil elements might slowdown significantly the SSI analysis runtime, typically by a factor up to 2. An alternate is to use springs between the excavation volume perimetral interaction nodes and basement perimetral nodes (duplicate nodes with same coordinates). Please see the INTGEN command for automatic generation of these springs. In this case, the computed axial forces in the springs distributed by the afferent area provide estimates of the seismic soil pressures on the embedded walls.

In addition, Option A can be used for computing the seismic pressure evaluation including the nonlinear foundation-soil separation and sliding effects.

If the local soil hysteretic behavior (secondary nonlinearities) is anticipated to affect significantly the SSI seismic pressures, which often is the case for deeply embedded foundations, then, the

---

use of adjacent nonlinear near-field soil elements is required for performing a nonlinear SSI analysis using equivalent-linear iterations.

For pile foundations, nonlinear springs can be used in Option NON for modeling the pile-soil interface slipping. The SOLIDPILE command can be used to generate automatically the springs surrounding the piles, between the piles and the backfill soil.

### 13. Structural Modeling

The structure should be modeled by the 2D or 3D finite elements to closely describe the structure geometry. The available FE library in ACS SASSI includes the basic 3D beam, shell, solid, spring elements, and the 2D plane elements. The 3D general matrix elements could be used to define substructure matrices and super-elements, or include the fluid inertial effects. Option AA has an automatic converter of the ANSYS MATRIX50 super-elements (.sub files) to the ACS SASSI general matrix elements (.pre files).

The selection of the elements and nodal points follows the general rules of the FE analysis practice and theory.

The specific rules for defining correctly the interaction nodes are:

- a. The interaction nodes are defined for the excavated soil model, not on the structure model. If the excavated soil volume outer surface has common nodes with the structure, i.e. as should be if no surrounding backfill soil is included with the structure FE model, then, these common nodes which also belong to the structure will be interaction nodes.

Except these common nodes, all the other FE structure model nodes including the basement nodes, shall not be defined as interaction nodes. Also, none of the internal excavation volume nodes shall be connected to a structural node.

- b. All the interaction nodes of the excavation should be below the ground surface and must lie at the soil layer interfaces.
- c. For embedded models, interaction nodes should be always numbered in the ascending order.

**WARNING:** A very important SSI model checking is achieved using the EXCSTRCHK command that identifies if any excavation volume internal nodes are connected incorrectly to the structure basement nodes.

### 14. Excavated Soil Volume Modeling

In the SSI analysis of the embedded structures, the excavated soil volume must be modeled by either solid elements for 3D SSI or plane elements for 2D SSI. The excavation volume is typically connected at the perimetral interaction nodes with the structure if no backfill soil zone is included.

---

The element sizes of the excavated soil elements are controlled by the distance between the interaction nodes. Internal nodes in the basement that are a part of an internal structure vibrating independently from soil media, need to be defined separately from the SSI interaction nodes.

The SSI results for embedded models are quite sensitive to the excavated soil mesh size. The vertical mesh size is selected based the above 1/5 wavelength criterion for ensuring an accurate transmission of the high-frequency vertically propagating wave components. However, for the horizontal mesh size the use of the same 1/5 wavelength criterion, could be too restrictive and could create an unnecessary computational analysis burden. Please also see in this section item 19 for additional details.

#### 15. SSI Substructuring Methods

In ACS SASSI there are three main SSI flexible volume substructuring methods available for the SSI analysis calculations, FV, FI and FFV. Key details on the application of these three methods and guidance are provided in Section 1.5.1, Section 2.1 and in this section at the item 19, shown below.

**WARNING:** *The new ASCE 4-2016 standard and the USNRC SRP 3.7.2 require a preliminary validation study if any other method than FV is used, such as SM, MSM or FFV. The validation against FV has to be done before SM, MSM or FFV can be used for the SSI production runs. As recommended in the ASCE 4-2016 standard, the validation analysis can be done for simple “excavated soil models” including no structure or foundation part. The validation study shall compare the computed acceleration transfer functions (ATF) at the common nodes between the structure and excavated soil. Rather than considering the solely “excavated soil model” (“the swimming pool model”), the use of a simplified massless foundation model to determine the kinematic SSI effects using SM, MSM or FFV against FV is suggested. For deeply embedded SSI models, such as SMRs, in deep soft soil deposits, the “excavation soil model” could become numerically poorly conditioned, and therefore, should be avoided. The most complete validation study is be to consider the full SSI model and check ATF at critical locations within the nuclear building (Ghiocel, 2014a).*

**WARNING:** *The ASCE 04-2016 standard and the USNRC SRP 3.7.2 requires a preliminary validation study using quarter SSI models for the excavation volume if any other method than FV is used, such as SM, MSM or FFV. It should be noted that more recent investigations have indicated that for unstable SM or MSM embedded SSI models, the quarter models are much more stable than the full models. The kinematic conditions imposed due to the symmetry and antisymmetry planes to the quarter models make the quarter models more stable numerically than the full models (Ghiocel, 2015a).*

#### 16. Addition of SSI Frequencies

The combination and addition of frequencies can be accomplished using the COMBIN module, as long as the specified frequencies reside on the computed FILE8 data files. For example, suppose the program modules SITE and POINT were executed for 10 frequencies (0.98, 2.93, 4.88, 6.84, 8.79, 10.74, 12.70, 14.65, 15.62 and 17.58Hz) and FILE9 and FILE3 were created. Furthermore, it is assumed that the program module ANALYS was executed only for 5 frequencies which reside on the above mentioned FILE9 and FILE3. The SSI analysis is to be repeated for two new frequencies (2.93 and 15.66 Hz) and the results are to be combined with those of the old frequencies. Since the new frequencies reside on the above mentioned FILE9 and FILE3 (and FILE4 or modelname.N4 results are frequency independent), the ANALYS module is executed to solve for the new 2 frequencies. Subsequently, the COMBIN is used to add the computed system response for the two new frequencies to the old ones and create a new solution FILE8 data file.

#### 17. Flexibility and Impedance Matrices of the Foundation

The computation of the free-field flexibility matrix follows the point load solution of a soil column. The soil column is modeled by a FE model with plane strain elements for 2D models, or axisymmetrical elements for 3D models.

The soil impedance matrix,  $\mathbf{C}$  (including real part  $\mathbf{K}$  and imaginary part  $\mathbf{D}$ ) is computed by inversion from the free-field soil compliance matrices,  $\mathbf{f}$  and  $\mathbf{g}$ ,

$$\mathbf{K} + i\omega\mathbf{D} = (\mathbf{f} + i\mathbf{g})^{-1} \quad (4.3)$$

where  $\omega$  is the frequency of analysis and  $i = \sqrt{-1}$ .

The columns of the soil compliance matrix are obtained by applying separate unit harmonic amplitude forces or moments for all the translational dofs of the interaction nodes. By inverting the compliance matrix for each frequency the corresponding soil impedance matrices is computed. Efficient parallel algorithms are implemented for inverting the flexibility matrix that is a fully populated dense matrix. The size of the soil flexibility and impedance matrix is about 14.4 GB for a number of 10,000 interaction nodes, and scales further by power 2 for higher number of interaction nodes.

#### 18. Nonlinear Hysteretic Soil Behavior

Since the complex frequency domain is used for computing SSI responses, the ACS SASSI program is restricted to only linearized FE systems. However, approximate nonlinear soil behavior free-field and SSI analyses can be performed using an equivalent-linear iterative procedure (SHAKE methodology).

---

In the application of the equivalent-linear procedure to seismic SSI problems, it is useful to consider the nonlinear soil effects associated with two types of non-linearity: (i) Global nonlinearity includes the nonlinear soil hysteretic behavior effects due to the seismic wave propagation in free-field, often also called the primary nonlinearity, and (ii) Local nonlinearity, includes the nonlinear soil hysteretic behavior effects due to SSI, called also the secondary nonlinearity. In practice, typically only the primary soil nonlinearity is considered. This is not true for the FEA models that include a soft backfill soil zone surrounding foundation, or include multiple FEA structural models for computing the SSSI effects between heavy neighboring buildings. The secondary effects may have influence on the distribution of the seismic soil pressures on embedded foundation walls.

The secondary soil nonlinearity effects imply an extended near-field soil zone in the SSI model. The effective soil shear strains within the near-field soil zone are computed iteratively according to equivalent linear procedure. Each SSI iteration is treated as a restart SSI analysis with “*New Structure*” that is about 2 to 4 times faster than the initiation SSI analysis. After each directional SSI iteration run is done for the X, Y and Z inputs, the effective strain for the three-directional input is computed using the COMBIN\_XYZ\_STRAIN.exe auxiliary program. The aux program uses the SRSS rule to compute the effective shear strain based on the superposition of the effective shear strain for each input direction written in FILE74 by the STRESS module.

#### 19. Excavated Soil and Structure FE Model Discretization

In order to accurately transmit the seismic waves through the discrete FE model, the vertical mesh size should be so that the largest size of each element does not exceed 1/5 of the shortest wavelength that corresponds to the highest frequency component of interest in the SSI analysis. This shortest wavelength corresponds to the cut-off frequency of the SSI analysis. Since the mass matrix computation in the code consists of 50% lumped mass and 50% consistent mass, the 1/5 wavelength criterion is reasonable for constructing the SSI models.

The 1/5 wavelength criterion is important for sizing the far-field and near-field soil layers, and the excavation soil volume elements in the vertical direction. This criterion is appropriate for the excavation volume vertical element sizes, if seismic input composition is predominantly constituted of vertically propagating waves. However, for the excavated soil horizontal mesh size the 1/5 wavelength criterion could be too restrictive for many practical situations, on a case-by-case basis, depending on the wave scattering effects.

Inside the excavated soil volume due to wave scattering effects, there is a mixture of incident body waves and scattered body waves and surface waves. Since the surface waves in a viscoelastic half-space have a horizontal propagation with a traveling speed that is relatively close to shear wave speed, the excavation horizontal mesh size should be close or equal to the vertical mesh size. However, in practice, for realistic layered soil deposits, the horizontal mesh size can be slightly relaxed. Often, reasonably accurate SSI results could be obtained for horizontal mesh

sizes that are about 1.2-1.5 times, or sometime even twice, larger than the vertical mesh size on case-by-case situation.

The required horizontal mesh size depends on the mixture of incident and scattered wave composition. If the seismic wave pattern in the excavated soil volume consists of 1D vertically propagating waves, then the horizontal element sizes in excavation volume are not restricted at all. Thus, the excavation volume horizontal element sizes could be several times larger than vertical sizes. The larger the discrepancy between the complex dynamic stiffness of the structural basement and the excavated soil volume is, the more refined horizontal mesh is required, since larger the wave scattering effects occur.

**WARNING:** *Sensitivity mesh studies are always recommended to validate the excavated soil horizontal mesh size. This is desirable since a larger horizontal mesh size impacts largely favorably on the SSI analysis runtime. Sensitivity studies can be done using quarter FE models of the excavation soil volume with no structure included. Without any preliminary mesh sensitivity study, the excavation horizontal mesh size should be taken close to vertical size.*

## 20. Half-Space Simulation

In order to simulate the halfspace condition at the bottom boundary, two techniques, variable depth method and viscous boundary method at the base are included. In the variable depth method, up to 20 extra half-space computational layers with total thickness of  $1.5\lambda$  and with the properties of halfspace are added to the soil profile. The wave length,  $\lambda$ , is the shear wave length in halfspace and is a function of frequency. Thus, the added soil layer thickness varies with frequency. The choice of  $1.5\lambda$  arise from the observation that fundamental modes of Rayleigh wave in halfspace decay with depth and essentially vanish at a depth corresponding to  $1.5\lambda$ . Furthermore, the  $1.5\lambda$  layer thickness is subdivided into  $n$  layers with increasing thickness with depth. The use of 20 half-space layers is suggested to provide best numerical accuracy to half-space simulation. With this technique, the layer thickness will increase with depth and decrease with frequency.

The soil model with added extra layers is further improved by replacing the rigid boundary at the base of the extended layer system with viscous boundary by placing dashpots in horizontal and vertical directions. The halfspace simulation is achieved in the SITE module.

## 4.2 ACS SASSI RUNS

The first step in running the SSI analysis code is to determine to which of the following three analysis run types the problem belongs:

- a. Initiation SSI Solution Runs
- b. Post SSI Post-Processing Runs

---

c. Restart or Reanalysis SSI Solution Runs

#### 4.2.1 Initiation Run for Seismic Linearized SSI Solution

The initiation SSI analysis consists of several consecutive SSI module runs:

1. EQUAKE Run > produces the spectrum compatible seismic input accelerations for SOIL, MOTION, RELDISP and STRESS
2. SOIL Run > produces the iterated, effective soil properties for SITE based on nonlinear free-field analysis (SHAKE methodology)
3. SITE Run (Mode 1 and 2) > produces the free-filed soil wave modes and soil layering information needed by POINT, HOUSE and ANALYS
4. POINT (POINT2 or POINT3) Run > produces the soil layering flexibility/compliance results for the ANALYS input
5. HOUSE Run > produces structural complex K and M matrices and the eigen-solution incoherency motion information to be used directly by ANALYS and STRESS
6. ANALYS run (Mode 1) > produces SSI complex acceleration transfer function (ATF) solution for seismic analysis or SSI displacement transfer function (DTF) solution for vibration or external force analysis.

The ANALYS runs for seismic analysis can be executed for a single direction X, or for three direction X, Y and Z. See next sections for details. ANALYS initiation and restart are done in same way.

The ANALYS runs for vibration analysis can be executed for a single load case, or for up to 500 load cases, depending on the SSI model size and available RAM. See next sections for details. ANALYS initiation and restart are done in same way.

#### 4.2.2 Post-Processing Runs

Once FILE8 is obtained, this file is used to compute the required response of the system through one or more of the following runs:

1. COMBIN run (to merge different set of SSI frequencies in old FILE8s in a new FILE8)
2. MOTION run (accelerations, ATF, ARS)
3. RELDISP run (relative displacements)
4. STRESS run (stresses/forces, STF)

The COMBIN run is necessary only if new frequencies are to be added to the old FILE8s (they must be renamed FILE81 and FILE82). The MOTION and STRESS runs are independent to each other and depend on the scope of the analysis. RELDISP needs the MOTION run to be done

---

before, to compute the complex ATF files (.TFI extension) for all the nodes of interest for computing relative displacements.

#### 4.2.3 Restart SSI Solution Runs (ANALYS Restart)

The ANALYS restart modes include:

- a. New Structure (Mode 2)
- b. New Seismic Environment (Mode 3 for seismic)
- c. New Dynamic Loading (Mode 3 for external loads)

Each of the above modes involves only two computer runs as described below.

#### 4.2.4 New Structure or Near-Field Strain-Dependent Soil Properties Restart

This restart mode performed for both the seismic and foundation vibration problems when the FE model is changed without modifying embedment nodes, consists of the following runs:

1. HOUSE run (new FE model)
2. ANALYS run (Restart for “*New Structure*”, Mode 2)
3. STRESS (stresses in new FE model)

The new FILE4 (modelname.n4) obtained from the HOUSE run is used with the old FILE1 (seismic analysis) or FILE9 (vibration analysis) as input to the ANALYS restart run that creates FILE8. This restart is also used for nonlinear SSI analyses for the equivalent-linear SSI iterations. For nonlinear soil SSI analysis, HOUSE, ANALYS and STRESS have to be run for each SSI iteration. The FILE74 and FILE78 text files that are generated at each SSI iteration by STRESS and HOUSE are useful for checking the solution convergence. The COMB\_XYZ\_STRAIN auxiliary program combines using the SRSS rule the effective soil strains computed for each direction in the nonlinear soil element groups and saved in FILE74 by the STRESS module. A new FILE74 will be generated after the combination of the results for X, Y and Z inputs.

#### 4.2.5 New Seismic Environment or Incoherent Simulation

This restart mode performed only for seismic problems when the type of seismic input or its location is changed. It consists of the following two runs:

1. SITE run (for the new seismic environment, or for X, Y and Z directions)
2. HOUSE run (new incoherent stochastic simulation run for X, Y and Z directions)
3. ANALYS run (restart for “*New Seismic Environment*”, Mode 3)

The FILE1 obtained from SITE is used as input for the ANALYS run. The ANALYS run then creates a new FILE8 or a set of FILE8s. For coherent analysis ANALYS can run three input directions X, Y and Z at the same time and produce three FILE8s, FILE8X, FILE8Y and FILE8Z. In this case, FILE1X, FILE1Y and FILE1Z needs to be created with SITE before ANALYS is run.

If the seismic motion incoherency is considered, then, HOUSE can run up to 50 incoherent motion simulations and create up to 50 FILE77s that will be used by ANALYS for creating 150 FILE8s for all 50 incoherent motion simulations for X, Y and Z directions. These 50 FILE77s are named FILE77001,...,FILE77050.

For incoherent simulations, ANALYS can run up to 50 simulations each for the X, Y and Z inputs in a single run, producing up to 150 FILE8s, depending on the SSI model size and available RAM. These FILE8 files are named FILE8001, FILE8002 and FILE8003 for 1<sup>st</sup> simulation for X, Y and Z inputs...and so on.

The SSI restart files required by ANALYS are the COOXxxx and COOTKxxx files, where xxxxx denotes the frequency order number. Additional restart files which are needed are COOXI, COOTKI, DOFSMAP, FILE90 and FILE91.

#### **4.2.6 New Dynamic Loading**

This restart mode, which can be performed for the vibration problems, consists of the following two runs:

1. FORCE run
2. ANALYS run (restart for "*New Dynamic Load*", Mode 3)

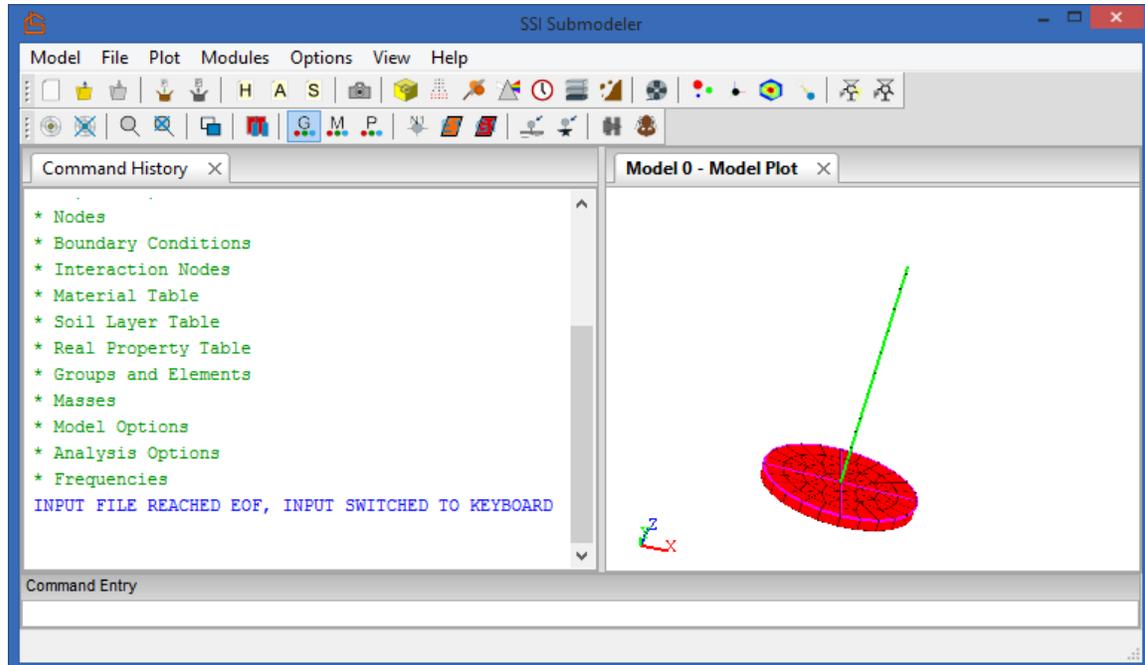
The new FILE9 obtained from the FORCE run, is used as input for ANALYS run. ANALYS run then creates a new FILE8. ANALYS can also run in restart mode many load cases, up to 500 cases. The FILE9s should be created before ANALYS is run. These files are named FILE9001, FILE9002,...,FILE9500.

## 5 Model Construction and Manipulation

In this section we describe the operation required to build a new SSI model, and how to create submodels or merge models, and how to build parametric language macros for post-processing the SSI analysis results.

### 5.1 Creating A New SSI Model

- Define the nodes for the structure, starting from the bottom. See the N command.
- Set the interaction nodes using INTGEN or INT commands, and fixed boundary conditions, if needed, using the D command
- Define the groups and elements using the GROUP and E commands
- Check the elements by opening a Model Plot window by using the Plot > Model > Element menu or the Model
- Define the materials, soil layers, and beam properties using the M, L, and R command
- Assign materials and properties to the elements using the MSET and RSET command
- Check the assigned materials and properties in the Model Plot window by showing the elements in their material / property color and comparing their color with the material / property color shown when selecting the Options > Colors menu or the Color command.



- Define the nodal masses with the MT and MR commands and set the mass units option using the MUNITS command. If the General Element are used to define nodal masses using the MXM command, then the mass units are controlled by the MOPT command.

- 
- Check the masses by displaying them in the Plot>Model>Nodes which opens the node plot window
  - Set the model options by selecting the Options > Model Menu
  - Define the analysis frequency set using the FREQ command
  - Define the analysis options by selecting the Options / Analysis menu command
  - Run the AFWRITE command to write the analysis files for the selected modules
  - Run the desired SSI modules

## **5.2 Loading and Saving Models**

The way models are loaded and save has changed in UI from the previous releases. In earlier versions ACS SASSI the user must create a database, and then create a new empty model in and add it to the existing database before any work on a model can begin. The new UI allows the user to work on a model without require the user to specify a place to save a model until a later time. The UI has a database that will be described later. Please note that this database and model data is not fully compatible with the database in previous releases up to IKTR4.

## **5.3 Using the Database**

The UI has a single two-level database structure. The single database structure replaces the multiple user-defined databases in previous releases. The single database is in a location cached by the system, so that the user will never have to search the directory structure to find old databases that have not been used in some time. The two-level database structure allows the user to organize all of the SSI models on a single computer account by project, much similar to how the multiple database models in the previous GUI was intended. Finally, the database is now considered to be a convenience feature and is no longer required to use any functionality.

When the user attempts to open a model for the first time in the UI Load Model window tree structure will be empty. At this time the user must add a group. All models must reside within a group. If the user attempts to define a model without a group, the model will not be saved in the database structure. After the user defines a group the user must select the group he wants to add the model to in the tree window. Once the group is selected, the user can press the add model button and provide the model information as they would in the previous GUI releases. Once the user clicks OK the tree will update with the added model. Then, the user can select the model and open it if desired. If the user has a model selected and attempts to add a new model that model will be added to the group in which the selected model resides.

When the user opens the model there may be errors about files not existing, this is to be expected when creating a new model. The database expects the model to exist and attempt to load the model when the open button is pressed. The database is setup to allow user to reorganize the models on their system and to add models to the database that have been created without the database in previous UI execution. For this reason, the database no longer creates the files that

---

the model loading code is trying to load before the loading sequence which causes the error messages.

#### **5.4 Without the Database**

As stated earlier, the database is no longer required in the use of the new UI. If the user is only going to use a model for a single analysis, the database will only add few additional steps to the process. For that reason the user can directly use the .pre file input using the INP command or the File>Input menu, or one of the converters to load a model into the UI session. If the user uses this method to input the model, the SAVE, RESUME and AFWRITE commands will not work because the .pre input file does not specify any model name and path. These commands may actually run, but there is no way to tell where the models will be written.

If the user wishes to use any of this functionality there is a new command MDL which specifies the model name and path. Once the MDL command is used on a model the SAVE, RESUME and AFWRITE commands will work properly. The MDL command can be issued before or after the model input without problems.

In the previous releases, the UI echoes all the .pre input file commands entered to the screen. This took a large amount of time and the output overwhelmed the screen and eliminated any chance of the user to find an error in the .pre file. Due to this fact it was decided for the new UI that only comments, warnings, errors and the model information text will be echoed to the screen. This changed the model input speed from a .pre file from 20 minutes to under 1 minute.

If the user has create and saved the model in a previous GUI run, the user can use the MDL to specify the model and then, use the RESUME command to load the model. The SAVE, RESUME and AFWRITE commands work in the same way as they work in the previous ACS SASSI PREP module. These commands do not update the database. If the user wants to add a saved model to the database the user can use the Model Open popup window to modify the database and add the location of the saved model.

#### **5.5 Transferring Files between Previous PREP and New UI**

Both original PREP available in the releases up to IKTR4 and the new UI starting after IKTR4, uses the .pre input text file as inputs. The parametric language of PREP is covered by the new UI, plus many other new commands. Thus, older .pre file from previous versions are expected to be readable by the new UI with no changes.

The original PREP had a window to open ASCII formatted files for view. This functionality was limited because a user could not edit a file that was being viewed inside of this window or copy this file with a different name or to a different location. This window has been replaced in new UI

---

with a text editor window that allow the user to view, edit and save files inside of the ACS SASSI User Interface. This window behaves much like a simple text editor application but a user can connect the window to the command entry box Using the Input > Connect to Command Entry menu option. When the user selects this option all commands that do not cause an entry error will be recorded to this window as long as the text window remains open and no other windows are connected to the Command Entry. The Command Entry bar is similar to the original PREP instruction line however not the user can select previously entered commands by pressing the up and down arrows while the text cursor is blinking in the Command Entry bar.

The user can now copy data from the Command History window and paste that data into another location. Information and warning text types have been added to the output each one of these type has its own distinct user selectable color. Finally, users can filter the output of the Command History if they desire by using the options in the View Menu.

## **5.6 Macros**

ACS SASSI generates a large amount of data that tends to be processed using repetitive tasks. Due to the rigid command structure and the way data is stored in original PREP, it was impossible to automate the execution of these tasks. The new UI commands do not have the same constraints that limited the ability of the PREP to access and modify the data that required the addition of the batch functionality in PREP. For this reason, many new UI commands have been introduced that replace the batch functionality.

While these new commands improve upon the efficiency of some of the batch options, it does not reduce the burden of the user to create and verify a file to do these operations. Some batch files need to perform hundreds or thousands of operations on data generated by the SSI modules. The macro functionality introduced in the ACS SASSI UI has been implemented to allow these operations to be performed more efficiently.

A macro in the ACS SASSI UI is a specially formatted .pre file that contains variables to allow for the procedure in that .pre file to be looped multiple times, replacing that variable each time. For example, a macro used to compute the SRSS of the acceleration response spectra would contain a single set of the commands needed to perform that operation once, in this case replacing a node number with a variable, allowing this macro to be called for a list of nodes. An example of this is shown in the next section. The Macro variables are defined by a positive integer number that is encapsulated with dollar signs (example \$1\$, \$2\$ ..., \$10293\$). Macros are loaded into the UI with the use of the LOADMACRO command. The user can then call the Macro by using the MACRO command. When the user calls the MACRO command, the variables are replaced by the arguments given to the instruction. The integer in the variable is directly related to the argument number in the command, so the first argument in the MACRO command will replace \$1\$ in the macro, the second replaces \$2\$, and so on for all the macro arguments.

Macros can be as simple or as complex as the user wishes. There is no MACRO command limitations on what a Macro contains including calls to other Macros. It is suggested that a Macro not use the LOADMACRO command because the repeated call to LOADMACRO will hurt performance, if the same data is being loaded repeatedly. All Macro names will be translated to all upper case letters, therefore Node and NODE will be the same Macro. There is a 50 character limit on the Macro name and a 3000 character limit on what each variable can contain. Each command line in the Macro is also limited to 3000 characters. There is no limit on the number of variables that one Macro can have. A Macro can have the same name as a command, however this does not overwrite the UI command.

Demo 3 contains a number of useful examples of using the UI macros for post-processing results.

### 5.6.1 Basic Macro Example

The basic example uses the macro to replace the N Command

```
* Node-Macro.pre A Simple example Macro
N,1,$1$,$2$,$3$

* MacroTest Pre will be equivalent to N,1,13.52,15,100.25
LoadMacro,Node,.\Node-Macro.pre
Macro,Node,13.52,15,100.25
```

### 5.6.2 Graphing Macro Example

The graphing example creates a Macro to get the SRSS of 3 lines, save the result and captures the plot of all of the lines in an image. The usage of the Macro will then create the SRSS line and graph for all points in the model.

```

* SRSS-macro.pre
* Load 3 lines SRSS data and output result to a file
* and capture image of all the lines on the graph
READSPEC,$1$,1,1
READSPEC,$2$,1,2
READSPEC,$3$,1,3
SRSS,4,1,2,3
WRITESPEC,$4$,4
* Capture images.
SPECPLOT,1,2,3,4
CAPTUREPLOT,$4$.png
CLOSEPLOT

* SRSS_folder.pre
* Create SRSS files for all nodes in the model
LOADMACRO,SRSS,SRSS-macro.pre
MACRO,SRSS,Node1x.rs,Node1y.rs,Node1z.rs,Node1srss.rs
MACRO,SRSS,Node2x.rs,Node2y.rs,Node2z.rs,Node2srss.rs
MACRO,SRSS,Node3x.rs,Node3y.rs,Node3z.rs,Node3srss.rs
MACRO,SRSS,Node4x.rs,Node4y.rs,Node4z.rs,Node4srss.rs
      ...
MACRO,SRSS,Node5214x.rs,Node5214y.rs,Node5214z.rs,Node5214srss.rs

```

### 5.6.3 Nested Macro Example

This is a modification of the graphing example that uses the same SRSS-macro.pre to show how the user should nest macros. Instead of the first macro SRSS-macro.pre being called directly the new macro will simplify the input of SRSS\_folder.pre.

```

* Nested-macro.pre
* Use SRSS-macro.pre to create a graph
MACRO,SRSS,Node$1$x.rs,Node$1$y.rs,Node$1$z.rs,Node$1$srss.rs

* SRSS_folder_Nested.pre
* Create SRSS files for all nodes in the model
* Does the same thing as SRSS_folder.pre
*
* Load the Macros
*
LOADMACRO,SRSS,SRSS-macro.pre
LOADMACRO,NESTED,Nested-macro.pre
*
* Use the Macros
*
MACRO,NESTED,1
MACRO,NESTED,2
MACRO,NESTED,3
MACRO,NESTED,4
      ...
MACRO,NESTED,5214

```

## 5.7 Merging Models

The ACS SASSI UI introduces some new functionality to merge multiple models together automatically. These commands are useful for merging a structural model with a model containing the excavation elements for that model or creating an SSSI model with multiple structures. During the merging process, the user can control how coincident nodes are merged, or can apply a translation in the X, Y, and/or Z direction. The MERGE command is more useful for creating SSSI models, as it allows for an X, Y, or Z direction translation during the merging process. The MERGESOIL command is useful for merging a structural model with an excavation model, as it provides control over how coincident nodes are merged. These two commands are described in the following sections, and also in the command reference in Section 9.9.

### 5.7.1 Merge Models to Build SSSI Models

The general MERGE command will take two SSI models already in memory and combine them together to make a third, new combined model. The combined SSI model will not change any of the data from the first model, while the second model data will be offset (node numbers, group numbers, material numbers, etc.), so it does not conflict with the first model. The MERGE command also translates the second model nodes by a constant supplied by the user in the command parameters. If the user needs to rotate one model in relation to another model the user should use the ROTATE command one of the SSI model in memory before using the MERGE command.

Below is an example of the MERGE command. This example assumes both models are in the proper location to in reference to each other. User should adjust the model geometry, before the MERGE command is used. For details, please see the MERGE command.

```
* Code to merge 2 models.  
* The merged model will be in model 3  
Actm,1  
Inp,Model1.pre  
Actm,2  
Inp,Model2.pre  
Actm,3  
Merge,1,2,0,0,0
```

### 5.7.2 Merge Excavation Volume with Structure

The MERGESOIL command is used to integrate the structure FE model with the excavation volume model. The excavation volume model nodes should be in the same coordinate system as the structure model. Also, the structure model should be made of structural elements (ETYPEGEN,1), while the excavation model should be made by excavation elements (ETYPEGEN,2). The excavation model can be created by the user using the EXCAV command if the basemat geometry includes an identical node layer mesh for all underground elevation.

For Option AA, the MERGESOIL command is required to integrate the structure and excavation FE models created in ANSYS (.cdb files) and then converted to ACS SASSI (.pre files) using the UI ANSYS to ACS SASSI converters. When using ANSYS to generate the model matrices for SSI analysis, the user must have separate structural and excavation volume models. The user needs an ACS SASSI model for its topology information for performing SSI runs and to post-process and visualize results such as acceleration and displacement time histories, ISRS, based on the ANSYS model. This MERGESOIL command will connect the interface of the two FE models based on the user selected option in the third argument of this command.

Below is an example of using the MERGESOIL command. This example uses INP to disambiguate which model is structure and which is excavation. It is important to set ground elevation before MERGESOIL, so that the command can weld the two models interface nodes properly based on the user selected option for the connection type. For details, please see MERGESOIL command.

If the MERGESOIL command is used in conjunction with Option AA, then, the mapping file for the excavation which is the last argument, should have a specific name “modelname\_Excav.map”. Please see Demo 7 on Option AA for more information on this.

```
* Code to merge an excavation to the soil.
Actm,1
Inp,struct.pre
etypegen,1
* Adjust Ground Elev. before merge
Actm,2
Inp,Soil.pre
* Adjust Ground Elev. before merge
etypegen,2
Actm,3
MergeSoil,1,2,1,modelname_Excav.map
```

## 5.8 Submodeling and Calculations

The ACS SASSI UI allows the user to isolate sections of a larger model to perform engineering calculations on that section. The commands relating to this functionality are described in the following sections, as well as in the Command Reference Section 9.10.

### 5.8.1 Section Cut Calculations and Submodels

A section cut in the ACS SASSI UI is a user-defined subset of elements to be used for engineering calculations. These elements are selected using the commands CUTADD and CUTVOL, which are described in Section 9.10. The elements selected for a section cut can be viewed selecting Plot > Cuts in the menu bar of the ACS SASSI UI.

### 5.8.2 Section Cut Calculations

The ACS SASSI UI has the functionality to perform cross-section design calculations on models stored in memory. To use these commands, the user must first create a section cut with the elements to be included in the calculation. Then, the user should create a cross sectional model using the CSECT command. The cross-sectional model is a 3D model base on where the plane defined in the CSECT command intersects with the elements in the section cut. The section cut/cross-section plane intersection is then extruded out to a unit thickness to create the 3D model, which is used instead of the 2D for plotting purposes. The user may also use the calculation commands on a non cross-section model, for example, computing the global base forces and moments for the entire building.

The cross-sectional model will have the same group/element numbers as the original model but the node numbers and locations will be changed to create the 3D unit thickness cross-section. The UI module finds the intersection of an element with the plane and will adjust the extents of the element based on this intersection. The UI calculates the intersection of a shells with the cross sectional plane and shell thickness will be adjusted in the cross section when the shell face is at an oblique angle to the section cut plane.

For the section cut calculations, the user should first load the element stress data (.ess files) for the model, before the cross-section is created.

Below is a section-cut example that includes building the section-cut submodel and then, computing the section cut forces and moments. First, the section-cut submodel is created. Then, the section-cut calculations are executed using CALCPAR command for a single user selected time. If user desires to compute section-cut force and moment time histories, then, the CALCSECTHIS command should be used.

#### 1<sup>st</sup> step: create section-cut submodel

```
* Create a submodel of elements above ground surface and save it to a file
* Ground surface at 2.53
inp,model.pre
*<blank> can be lack of any characters or just white-space
cutvol,3,<blank>,<blank>,<blank>,<blank>,2.53
cut2sub,3,1
actm,1
write,model above ground.pre,C:/users/user/.../
```

The <blank> in the CUTVOL command represent lack of arguments provided to the command. <blank> is used to illustrate the argument fields that will be left as default values. See the command reference for a description of what the default values are for this command.

#### 2<sup>nd</sup> step: section-cut calculations at the user selected time/given stressfile.txt

```
* Create a submodel of a wall and do calculations on it
inp,model.pre
readstr,stressfile.txt
cutvol,3,52.5,52.8,-320,320,2.53,45.22
* Create the crossection for calculation
csect,1,3,55.6,0,4,1,0,0
actm,1
calcpar,1,0,0,0,1,0
calcm
```

Please see the Demo 8 example for more details on the section-cut calculations.

### 5.8.3 Transfer Commands

The Transfer commands act similarly to the section-cut commands, however these commands directly transfer elements to the new model instead of to a section cut, overwriting any elements already defined with that element number in the destination model. For this reason it is suggested that the user applies the section-cut commands instead for creating a submodel, as this allows for the review of the elements to be added to the new model before the transfer is executed.

## 5.9 Variables and Loops

A variable in the ACS SASSI UI module is a list of strings that the user can access elements of at a later time and an integer counter. A variable is declared using the VAR Command, the first argument is the variable name the rest of the arguments are the element of the variable list. Variable names in the ACS SASSI UI are not case sensitive, therefore defining two variables whose name's differ only in capitalization will results in the first variable defined being overwritten. A simple example of the VAR command usage is VAR, X, 1.23, 2.83, 3 (Note: In later examples in this section, it will be assumed that this command was declared when describing how the user can access this variable later in this section). The user can access the variable in any command (with the exception of the FOREACH command) by putting a prefix @ in front of the variable name and by using the postfix operators that will be described.

If the user only uses the prefix and declares @X a command argument the ACS SASSI User Interface will replace the instance of @X with the variable counter which is set to 0 by default. The + and - post-fix operators will add or subtract an integer from the counter and replace the variable with the updated counter @X+5 will add 5 to the X counter and that value will replace the entire statement. ++ and -- are the same as if the user added or subtracted 1 ( @X++ and @X+1 are equivalent statements). The = post-fix operator will set the counter equal to the integer following the number and that number will be substituted for the variable @X=-15 will set the counter to -15 and -15 will be substituted into the argument. To access the variable list the [] operator must be used. Inside of the brackets the user enters an integer position in the variable list that is desired. The list numbering starts at 1 so using the example variable 1.23 will replace the @X[1] statement, while @X[2] will be replaced by 2.83.

A variable list cannot be modified once declared the user can only change what is in a variable list by overwriting the Variable using the VAR command. The Variable counter get set/reset to 0 every time the VAR command redefines a variable. The counter can only be modified while being referenced in a valid command, so that the SETVAR command has been introduced. This SETVAR command is a placeholder to allow the user to set or check variable counters without affecting the model.

Once a variable has been declared the user can execute a single command loop by using the FOREACH command. The FOREACH command take a variable name without the @ prefix and the command the user wishes to loop through. The number of times the loop will execute is dependent on the number of elements in the variable list (FOREACH,X,... would execute 3 times). While in the loop a separate loop counter can be accessed by using the pound sign operator (#) with the post-fix operators. Loops can be nested but nested loops cannot use the same variable, this situation will cause an infinite loop (this situation should cause an error and loop execution will not occur). The loop can only take one command but the user can pair FOREACH with a macro to get a multiple command loop.

Below is an example of simple nested loop using three variables for declaring a block of 5x5x5 nodes. In this example, the N command is used to define nodes. The node counter variable "NNUM" is incremented with each loop execution by the "@NNUM++", which is defining the node numbers 1 through 125 as it loops through the values of the X, Y, and Z variables.

```
* Loop.pre A simple nested loop example
* Declare a 5x5x5 block of nodes
* NNUM = Node Number Counter
*   X = X coordinate
*   Y = Y coordinate
*   Z = Z coordinate
Var,NNUM
Var,X,1,2,3,4,5
Var,Y,1,2,3,4,5
Var,Z,1,2,3,4,5
ForEach,Z,ForEach,Y,ForEach,X,N,@NNUM++,@X[#],@Y[#],@Z[#]
```

## 6 ACS SASSI User Interface (UI) Module Menus and Operations

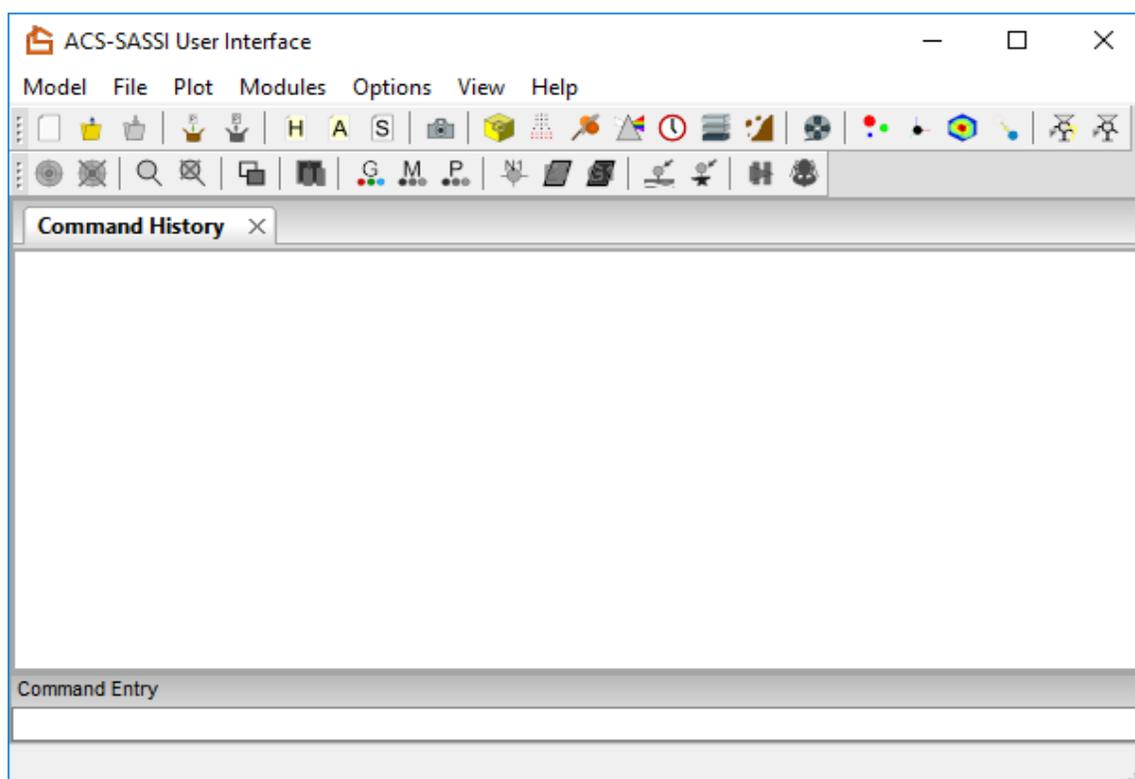
In this section we present in details the ACS SASSI UI operations, including how the user needs to input the SSI analysis option data for all the SSI modules. The table shown below details all the menu options available in this version.

Menu Options		Description	
Model	New	Create an empty Model	
	Open	Open a model from the database	
	Save	Save a SSI model (Binary Database format)	
	Input	Load a .pre file	
	Converters	SASSI .hou	Load a Model from SASSI .hou file
		ANSYS .cdb	Load a Model from ANSYS® .cdb file
		GT-STRUDL Database	Load a Model from GT-STRUDL® Database (Not included)
	Output	Write a model to a .pre file	
	Export to ANSYS	Export model to ANSYS® input	
	Export to STRUDL (not included)	Export model to GTSTRUDL® Database (Not included)	
Exit	Close ACS SASSI User Interface		
File	Open	Open a ACSII text format file	
	Export Image	Export a plot image	
	Export Table	Export a Data table from a 2d plot	
Plot	Model	Elements	Element Plot
		Nodes	Node Plot
	Cuts		Plot the data in a section-cut of the active model
	Spectrum TFU-TFI		Load files with the spectrum file format and plot a 2d line plot
	Time History		Load files with the time history file format and plot a 2d line plot
	Soil Layers		Show the soil layer plot
	Soil Properties		Show the soil properties plot
	Non Uniform Soil Field		Show the non-uniform soil layer (not active)
	Process Animation		Process frames for animation

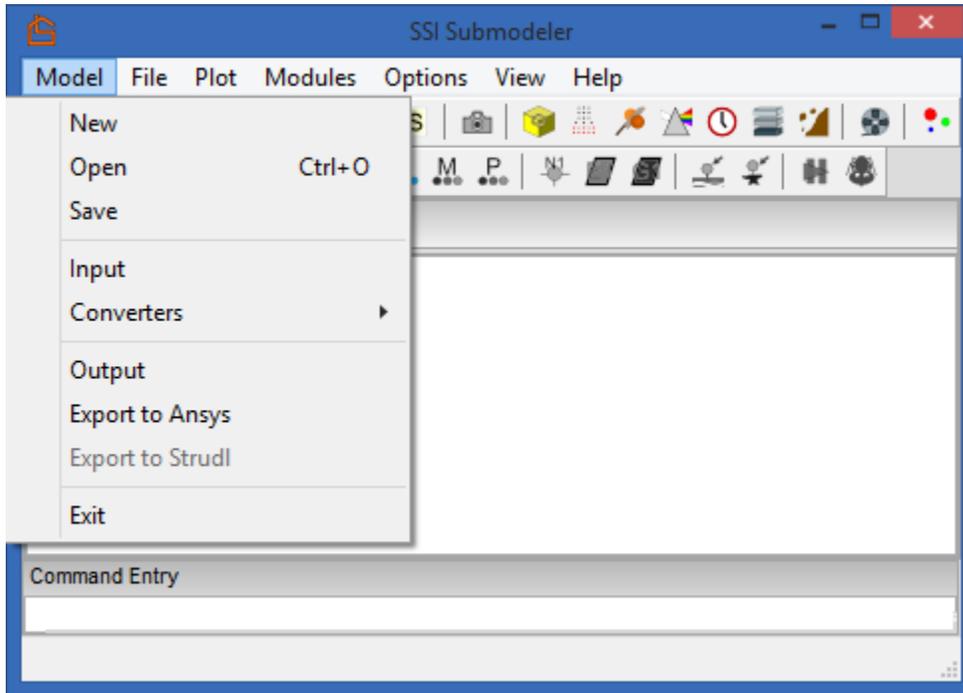
Menu Options		Description
	Frame List	
	Bubble	Plot the bubble animation
	Vector	Plot the vector animation
	Contour	Plot the contour animation
	Deformed Shape	Plot the deformed shape animation
Modules	Location	Location listing of the modules
	Extension	Extensions associated with each module
	EQUAKE	Run the EQUAKE module
	SOIL	Run the SOIL module
	LIQUEF (not included)	Run the LIQUEF module (Not Active)
	SITE	Run the SITE module
	POINT	Run the POINT module
	HOUSE	Run the HOUSE module
	PINT (not included)	Run the PINT module (Not Active)
	FORCE	Run the FORCE module
	ANALYS	Run the ANALYS module
	COMBIN	Run the COMBIN module
	MOTION	Run the MOTION module
	STRESS	Run the STRESS module
	RELDISP	Run the RELDISP module
	NONLINEAR	Run the PANEL module
	ANSYS Eq. Static Load	Run the LOADGEN module for static load
ANSYS Dynamic Load	Run the LOADGEN module for dynamic load	
ANSYS Super Element Utilities	Run the SSI2ANSYS module for ANSYS MATRIX50 super-element use in Option AA, or for conversion to ACS SASSI model using GM element	
Options	Model	Sets the model options.
	Write	Modify file writing options for AFWRITE/WRITE commands
	Check	Sets the check options
	Analysis	Sets the SSI analysis options for the ACS SASSI modules.
	Windows Settings	Sets the options for the active window.

Menu Options		Description	
	Colors	Sets the colors for UI module.	
	Font	Changes the fonts for UI module	
	Shader Options	Set more plot options for active 3D plots	
	Reset Plot	Reset plot to default settings	
View	Check Errors	Show the error checking window	
	Command Window	Show or open command history window	
	Command Display	Command Echo	Show echoes in the command history
		Output Confirmation	Show output confirmation in command history
		Comments	Show Comments in command history
		Warning & Errors	Show Warning and errors in command history
	Toolbars	Main Toolbar	Show/hide the main toolbar
Plot Toolbar		Show/hide the plot toolbar	
Help	Help	Load online Help for ACS SASSI use	
	About	Show About box	

A screen shot of the ACS SASSI UI opening screen under Windows 10 defaults is shown below.



## 6.1 Model Menu

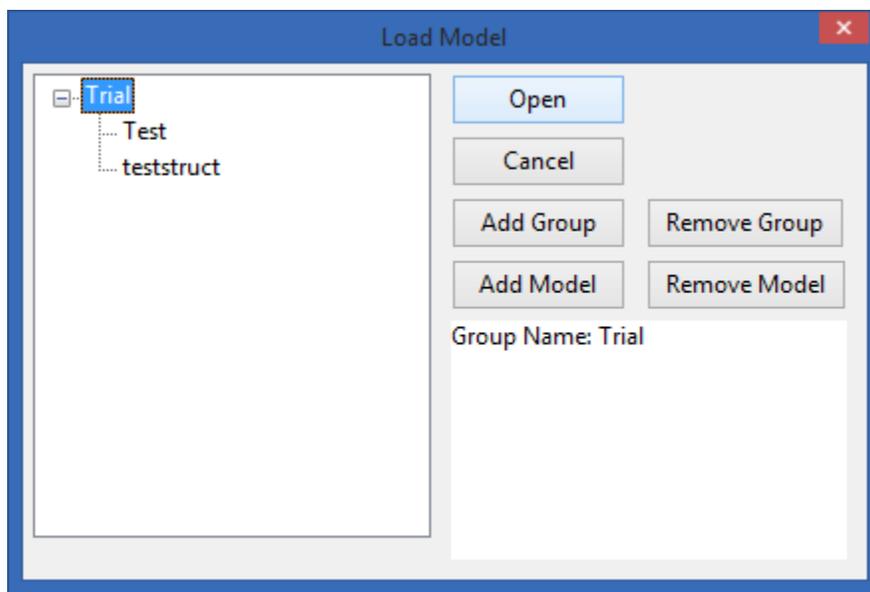


### 6.1.1 New

This menu option creates an empty model with a model number that is not currently in use by the UI module. Then, UI changes the active model to the new model number. This menu option does not delete or modify any models that are in the UI memory before this menu option is selected.

### 6.1.2 Open An Existing Model

The Model>Open menu option allows the user to open previously used models and create destinations for new models in a two-level database. When the user clicks on the menu a window will appear with a tree structure on the left and buttons and an information window on the right. If the user has not used this functionality before on this computer, the tree window will be blank.



To begin from a blank tree the user must add a group. A group is like a folder where similar models can be stored. The user can organize groups in any way desired however all models must be in a group. Any attempt to add a model outside of a group will not add a model to the tree. When the user clicks the add group button a window will pop-up and ask for a name for the group. Once the user clicks ok the group will be added to the bottom of the tree.

To remove the group from the tree the user can select the group by left clicking on it with the mouse and then hitting the remove group button when the group is highlighted. When this happens the user will be prompted for a confirmation if the group is to be removed. If the user clicks yes the group will be removed. A second popup will appear and ask the user if he wants to delete all of the data from the models in that group. If the user selects yes the files and folders for the models in the group will be removed from the hard drive. If the user selects no, the model files and folders will still exist.

Once the user has a group in the tree structure, a model can be added to the group by clicking on the desired group to highlight it. Next the user can hit the add model button which will pop-up a window that asks the user to enter model information. Once the user clicks ok a model will be added to the group. The user can add a new model which has not yet been defined, a model that already exist in another group, transferred from another computer, or created in another way. If the user wishes to remove a model the user can press the remove model button when the desired model is highlighted in the tree. The program will prompt the user with a similar set of pop-up that were described when the remove group button is pressed.

The user can open the model by highlighting it in the tree and clicking open button. Once the open button is clicked the window will close and the model will be loaded. If the user has just defined

the model, there may be errors that appear stating that files are not found. This is normal because the program is attempting to load files that have not yet been created. These files will be created when the user issues a SAVE command.

The model database file itself is stored in a file called SASSIdb.xml on the user hard drive in a default location that varies by the operating system. This file contains the model/group tree structure found in the window's left hand side. Other than model name and location of the models this file contains no other information about any models on the hard drive.

### **6.1.3 Save**

The Save Menu option performs the SAVE command from the menu. The model name and path should be defined for the active model before using the Save option.

### **6.1.4 Input**

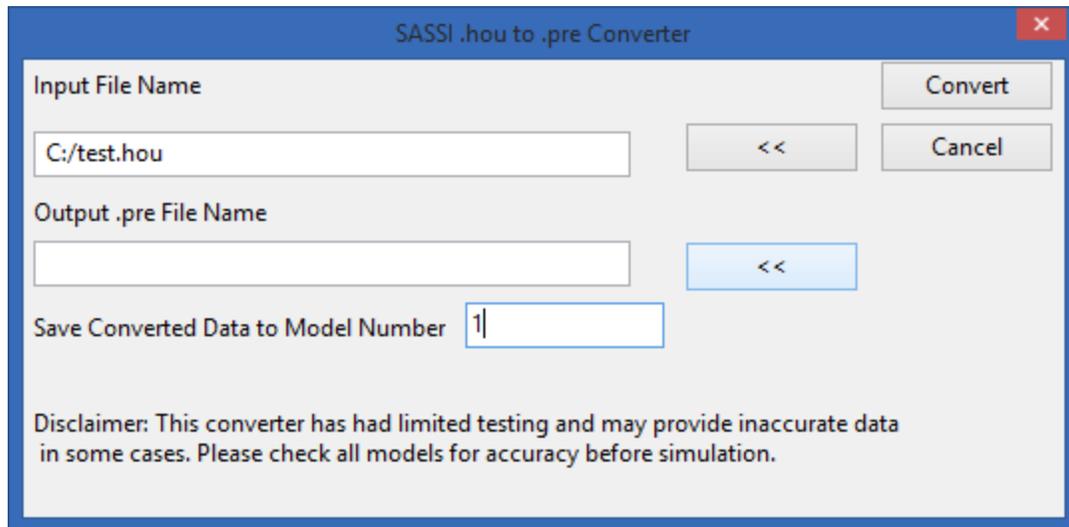
The Model>Input menu option allows the user to select/browse for a .pre format input file and load that into the UI module. Once the user selects the proper file, a progress bar should appear in the lower right-hand corner and give the user an idea of how much of the .pre file has yet to execute. (Note: The progress bar shows Current Line/file length. This should not be used to estimate total time left because command processing speeds vary). Once finished the user should get a message that loading has completed and the user may continue using the UI as desired.

The .pre input file is a batch of commands, so that all of the commands will be loaded and executed on the current model. Every .pre file has the ability to modify the current model. Any command that is valid in the UI can be used in a .pre file with no exception. Any .pre file written by the WRITE command is compatible with the previous ACS SASSI User Interface modules. However, there are new commands in the new UI that are not supported by the previous PREP and the SUBMODELER modules. Therefore any user written .pre file may contain commands GUI modules which are not recognized by older PREP and ACS SASSI User Interface module programs. If this happens both programs will issue a statement of "<X> Command not found " where <X> is the invalid command name. The .pre input file will continue to be processed if any error occurs and subsequent command will only be affected, if they required the command that was not recognized.

### **6.1.5 Converter for SASSI Model Fixed Format Input**

The ACS SASSI model converter for SASSI fixed format will attempt to take the input file for the HOUSE module (.hou file) convert that file into an usable .pre model input file. The user must fill the "Input File Name" box with the \*.hou filename to be converted. If the user fills in the .pre filename box the \*.pre file will be written, but this is optional. If the user puts a non-negative integer

number in the “Save Data to Model Number” box the converted model data will be stored to that model number otherwise the data will be stored to the current active model number.



Optionally the SITE (\*.sit) and POINT(\*.poi) module inputs are in the same folder as the HOUSE (.hou) module input, the UI converter will read those files and add the additional information to the SSI model. There should be no limitation to this converter.

For the older-style SASSI2000 formatted input files, before using the converter, the user to change the HOUSE input file name to a filename that includes extension .hou, and then, in the HOUSE input file need to add few zeros on the 3<sup>rd</sup> general information option line and delete any comment \$ sign. The converter can also include the information from the SITE and POINT modules if their name have that same name we the HOUSE input (.hou file) and with extensions .sit and .poi. Also, in the SITE formatted input file a new 1st input line has to be inserted with a zero number in column 5.

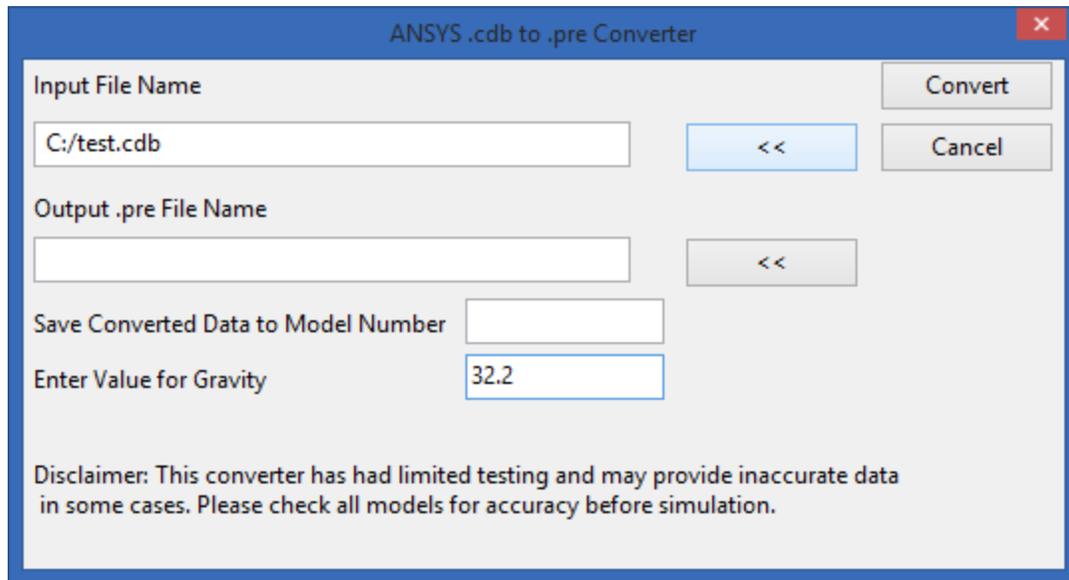
### 6.1.6 Converter from ANSYS .cdb File

The ANSYS file converter will take a Model in the .cdb file format and converts the data to a model that is usable by the UI.

**WARNING:** *The ACS SASSI converter only converts a subset of the ANSYS elements and the way that these elements are converted does not cover the full spectrum of how these elements may be defined by ANSYS for all keyopt options.*

When selected from the menu or using the CONVERT command with no arguments the user will see a popup that will ask them for an input model file and output file name along with model number to store and the value of gravity. The user is only required to give the input model name

and the value of gravity for the conversion to work properly. Once, the user click OK the model will be converted and stored into UI memory.



If the user does not provide an output file name to the UI converter, the model will only be stored in UI memory and not be written to an output file. If the user wants an output file after the file has been converted the WRITE command will allow the user to write the .pre file. If no model number is provided to the UI converter popup be for conversion the data will be stored to the current active model.

### ***ANSYS.cdb Model Conversion Restrictions***

The ACS SASSI UI ANSYS file converter will take a ANSYS Model in the .cdb file format and converts the data into a model that is usable by the ACS SASSI UI module. The UI converter only converts a subset of ANSYS elements and the way that these elements are converted does not cover the full spectrum of how these elements may be defined by ANSYS. The limitations below are for models that can be converted to run under ACS SASSI.

***WARNING:*** *If a particular option is not discussed, then it is not supported in the UI converter, the model will likely convert the model but the converted model will be incorrect for the ACS SASSI input. Any variance from these instructions will either cause an error in conversion or yield an incorrect model when conversion is complete.*

### ***Limitations for the ANSYS Model Conversion to the ACS SASSI Model***

Material data must be defined using the MPDATA command, unless otherwise specified. Any other way of defining or redefining this data is not recognized by the UI converter.

**BEAM4/BEAM44**

Reference to the beam cross-section commands in the limitations for the elements implies some combination of SECTYPE, SECBLOCK and SECDATA. Any other use of the section commands in the ANSYS model will not be transferred in the converted ACS SASSI model.

The BEAM4/BEAM44 elements are convertible with the following limitations: These beams must be defined with the K node definition. These beams must have properties defined by the RBLOCK and the RBLOCK entry for the property must have either 6, 8, 10, 12 or 19-24 fields defined in the RBLOCK for this element to be converted properly. The end releases are defined by KEYOPT 7 and 8. At this time, BEAM4/BEAM44 are legacy elements in ANSYS V15 or beyond.

**COMBIN14**

The COMBIN14 spring elements are convertible with the following limitations: KEYOPT 2 or KEYOPT 3 must be defined for the group. KEYOPT 2 will take precedence if both KEYOPT 2 and KEYOPT 3 are defined. The spring constant must be defined by the by RBLOCK entry. KEYOPT 2 options 1-6 are supported and options 1 and 2 for KEYOPT 3.

**MASS21**

The MASS21 elements are convertible with the following limitations: The mass data must be defined in the RBLOCK entry. KEYOPT (1) and KEYOPT (2) must be set to 0 or not defined in the .cdb for the mass group to be converted properly. Please note that in ACS SASSI the nodal mass units are controlled by the MUNITS command.

**SOLID45**

SOLID45 is fully convertible to ACS SASSI SOLID element with the following limitations: Some highly distorted pyramids might not be convertible. At this time, SOLID45 is a legacy element in ANSYS V15 or beyond.

**SHELL63**

The SHELL63 element is fully convertible to the ACS SASSI shell/plate element that has the same FE formulation with the following limitations: Shell thickness must be defined by RBLOCK entry. At this time, SHELL63 is a legacy element in ANSYS V15 or beyond.

**SHELL181**

The SHELL181 elements are convertible to the ACS SASSI shell/plate elements that is similar to the SHELL63 formulation, with the following limitations: The shell thickness must be defined using the section commands.

**SOLID185**

The SOLID185 elements are convertible to the ACS SASSI solid elements that is similar with SOLID45, with the following limitations: KEYOPT4 which sets the option for nonuniform materials is not defined or set to 0.

**BEAM188 & PIPE288**

BEAM188 & PIPE288 are convertible to the ACS SASSI beam elements with the following limitations: Pipes are converted to the equivalent straight beams. These elements must be defined with a K node definition. The end releases for these elements cannot be defined for these beams because ANSYS ENDRELEASE command creates new nodes and couples these node to simulate an end release. This cannot be properly represented in ACS SASSI. The beam properties must be defined by using the section commands. Beams must be defined with the ASEC or RECT type option to be convertible. The section offset must not be specified for pipes.

**Limitations for the ANSYS Model Conversion for Option AA**

In Option AA, the UI converter translates the ANSYS model only for the topology information, since the HOUSE module will use the .hou input file only for the node coordinates and element connections, and not for building the structure matrices for SSI calculations. The converted model will be flagged since potentially includes incompatible features with FE modeling in ACS SASSI. The UI AFWRITE command generates a .hou input file that will not be runnable directly by the HOUSE(FS) module.

If the HOUSE(FS) model isn't runnable, the user will receive a warning when the file is converted. The list of elements types below are the additional element types that can be converted for a display model. There are no limitations on these elements because only the element node connections need to be translated for topology information and model display.

Other ANSYS elements will be ignored by the UI converter and not be included in the model. The list below specifies the element type from ANSYS and the element type used to represent it in the ASC SASSI model.

**List of the ANSYS Elements Type Conversions in Option AA**

The following ANSYS to ACS SASSI element conversions are only for display purposes:

TRUSS180 will be displayed as a Beam  
MPC184 will be displayed as a Spring  
PIPE16 will be displayed as a Beam  
PIPE18 will be displayed as a Beam  
FLUID80 will be displayed as a Solid

### 6.1.7 GT-STRUDL Database Converter (Not Included)

This GT-STRUDL database converter will take a model in a specific GT-STRUDL database format and convert the model into an ACS SASSI model. GT-STRUDL data formats are highly customizable and any format change will cause the converter not to convert the model correctly.

### 6.1.8 Output

This Menu Output option provides the user the same functionality as the WRITE command does. When the menu option is used the user will be asked for the name and path of where the file is to be written in a popup box. When the Save button is clicked, the UI program will write the current model to a .pre file.

### 6.1.9 Export to ANSYS

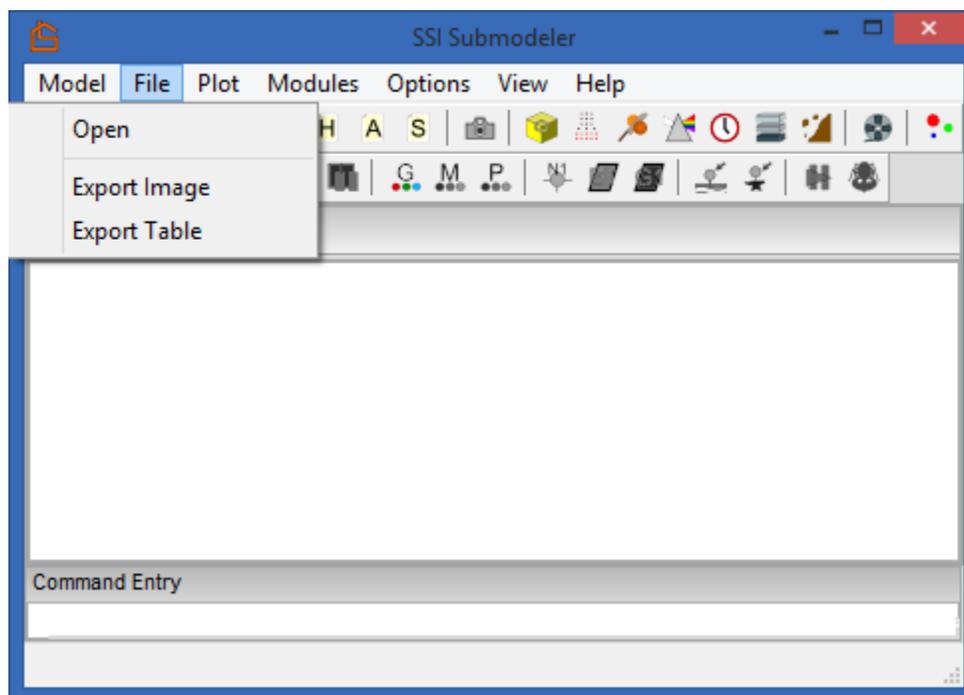
This Export to ANSYS option allows the user to output a model to an ANSYS ADPL formatted file for use with ANSYS. When the user selects this menu option the UI program will write an ADPL file with the name <modelname>.inp to the model directory. If the user has not specified the model name and model directory these file will be written however the file will have blank name (.inp file) and the file will reside in a working directory which is determined by a platform dependent process. The user can also create these files using the ANSYS command. Also, due to the differences in the group and element data structure of ACS SASSI and ANSYS, there have been some conversion problems with some models.

**WARNING:** *It is suggested that the user use the ANSYSREFORMAT command before converting a model which eliminates many of the issues. This functionality uses elements that are compatible with ANSYS V11-15. These elements have been depreciated as legacy elements in the newer versions of ANSYS. Therefore, the ANSYS models written by this functionality may not work latest ANSYS versions.*

### 6.1.10 Exit

The Exit option will close the UI and save the environment variables setting to SASSlini.xml. The Exit option does not save any models or plots stored in memory. The user should save the model changes in an appropriate way before closing the UI module.

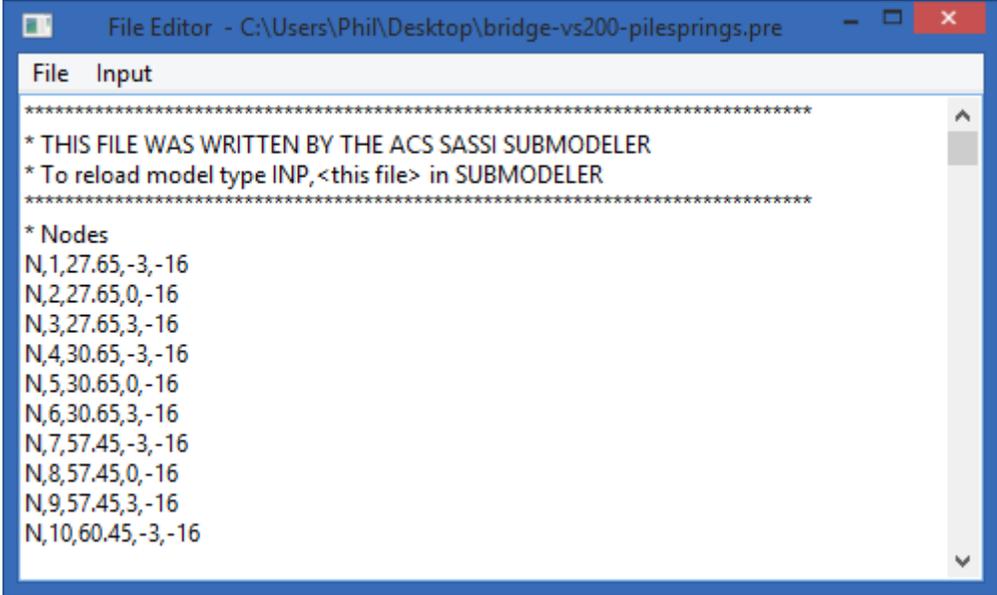
## 6.2 File Submenu



### 6.2.1 Open A File

The File>Open menu option allows the user to open edit and save an ASCII text format file from inside of the UI. When this menu option is selected a file selection popup opens and asks the user to select a file or the user can write a new file name into the box which will create a new file. Then, a simple text editor window will open and display the text inside the selected file.

The user can connect this window to the command output by selecting the Input > Connect to Command Entry. When the user selects this option any command entered into the command entry box that does not result in an error will be echoed into the text editor window while it is open. The user can save any changes to the text file by using the File> Save menu option.



```
File Editor - C:\Users\Phil\Desktop\bridge-vs200-pilesprings.pre
File Input
*****
* THIS FILE WAS WRITTEN BY THE ACS SASSI SUBMODELER
* To reload model type INP, <this file> in SUBMODELER
*****
* Nodes
N,1,27.65,-3,-16
N,2,27.65,0,-16
N,3,27.65,3,-16
N,4,30.65,-3,-16
N,5,30.65,0,-16
N,6,30.65,3,-16
N,7,57.45,-3,-16
N,8,57.45,0,-16
N,9,57.45,3,-16
N,10,60.45,-3,-16
```

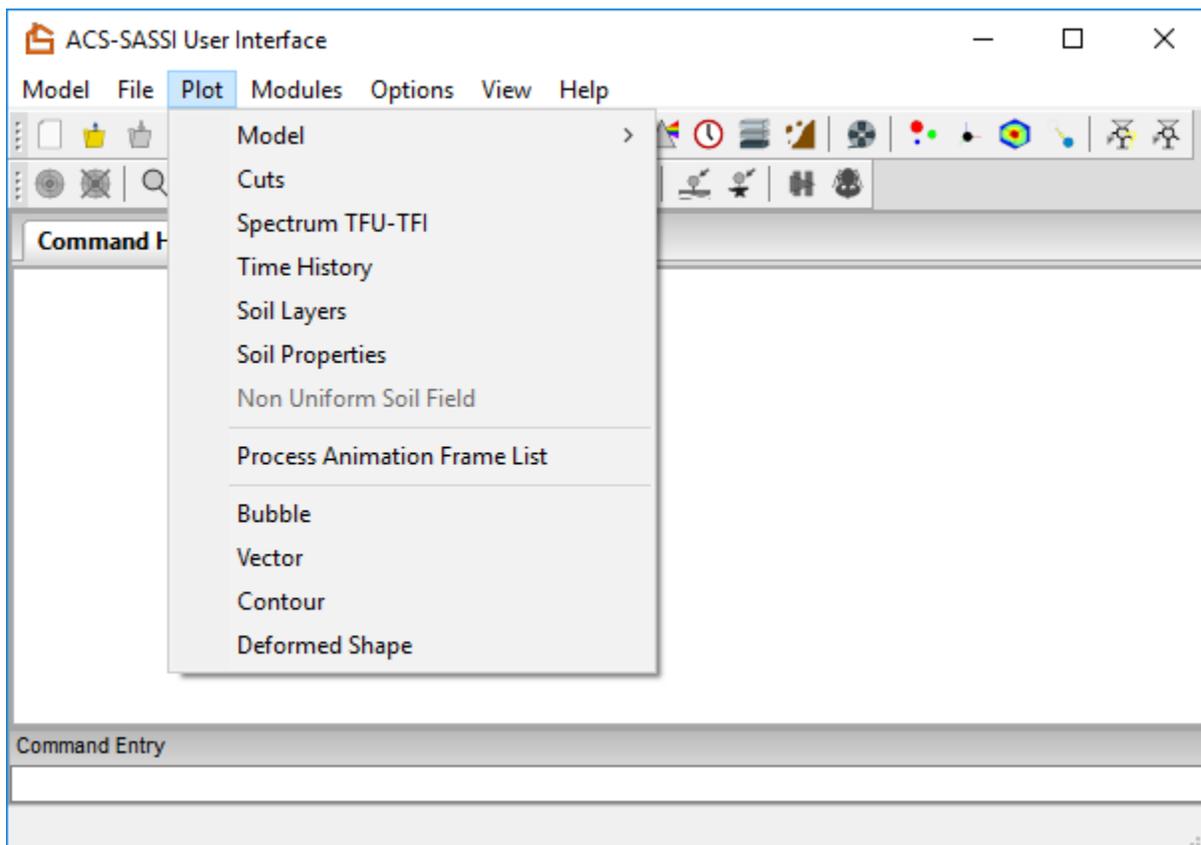
### 6.2.2 Export Image from a Plot

This functionality allows the user to capture an image of a plot in the UI. Once the menu option is selected the user will be asked for a file name for the captured image. After the user selects OK the image will be taken and stored in the user specified file in a .bmp image format. The menu option functions in a way similar to the CAPTUREPLOT command.

### 6.2.3 Export Table

This menu option allows the user to export data from certain 2D plots to a comma separated value table for use with different programs such as spreadsheets.

### 6.3 Plot Submenu



#### 6.3.1 Element Plot

See Section 7.1.2

#### 6.3.2 Node Plot

See Section 7.1.3

#### 6.3.3 Cut Plot

See Section 7.1.4

#### 6.3.4 Spectrum TFU-TFI

See Section 7.2.3

#### 6.3.5 Time History Plot

See Section 7.2.4

#### 6.3.6 Soil Layer Plot

See Section 7.2.5

### 6.3.7 Soil Properties Plot

See Section 7.2.6

### 6.3.8 Processing Animation for the ACS SASSI User Interface

See Section 7.3.1

### 6.3.9 Bubble Plot

See Section 7.3.4

### 6.3.10 Vector Plot

See Section 7.3.5

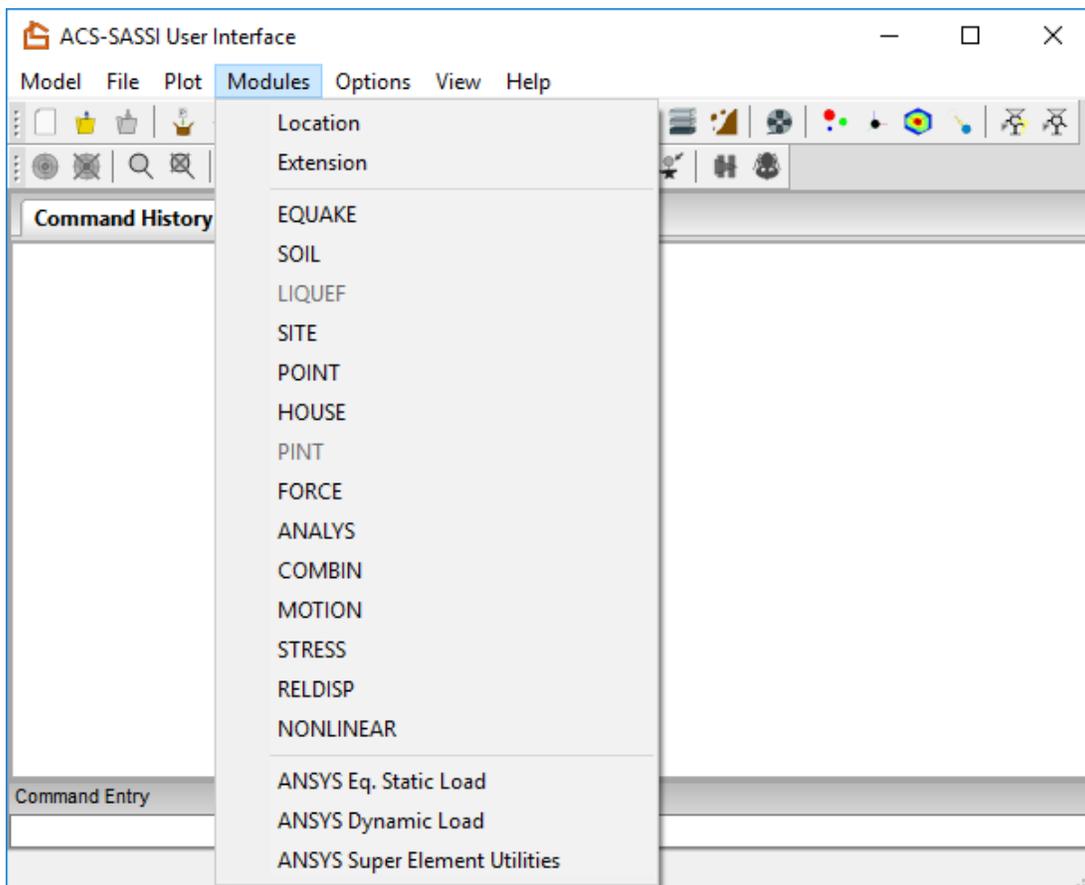
### 6.3.11 Contour Plot

See Section 7.3.6

### 6.3.12 Deformed Shape Plot

See Section 7.3.7

## 6.4 Modules Submenu



### 6.4.1 Location Window

This Option allow the user to choose the location of the modules that will be run when any of the module options are selected.

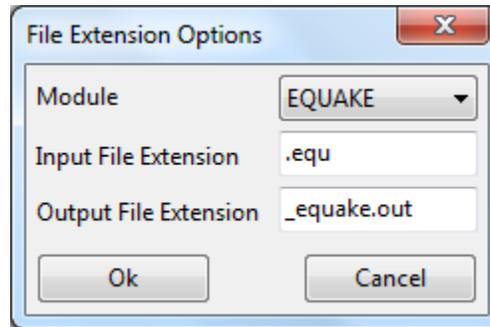
Module	Path	Navigation
EQUAKE Module	ler V3\FastSolver\EXEB\Equakeb.exe	<< Ok
SOIL Module	C:\SSI Files\SSI Installer V3\FastSolve	<< Cancel
LIQUEF Module		<<
SITE Module	C:\SSI Files\SSI Installer V3\FastSolve	<<
POINT Module	C:\SSI Files\SSI Installer V3\FastSolve	<<
HOUSE Module	C:\SSI Files\SSI Installer V3\FastSolve	<<
PINT Module		<<
FORCE Module	C:\SSI Files\SSI Installer V3\FastSolve	<<
ANALYS Module	C:\SSI Files\SSI Installer V3\FastSolve	<<
COMBIN Module	C:\SSI Files\SSI Installer V3\FastSolve	<<
MOTION Module	C:\SSI Files\SSI Installer V3\FastSolve	<<
STRESS Module	C:\SSI Files\SSI Installer V3\FastSolve	<<
RELDISP Module	C:\SSI Files\SSI Installer V3\FastSolve	<<
LOADGEN Module		<<
NONLINEAR Module		<<
SASSIANSYS Module		<<

When selected a window will appear and the user can either browse for modules or manually type in the full path location. After data entry is complete the user can then select ok and the module locations will be updated. This data is stored in a file named SASSIini.xml which is located in a default location on the user hard-drive. (The location of the SASSIini.xml varies depending on the computer platform). The initialization file will store the data, so that the user does not have to reenter the data in future uses of the UI. Currently, there are no modules for use with the UI.

### 6.4.2 Extension Window

The previous ACS SASSI MAIN module allowed the user to change the file extensions for the input and output files to the modules. The new UI module continues to allow the user modify the

extensions as desired. The extensions are stored in the SASSlini.xml file any modification will be reloaded when the UI is opened.



The user can select a module name from the drop down menu which will bring up input and output file extension. The user can then modify either extension in the text boxes then select another module from the drop down menu. All modifications to the extensions will be saved when the OK button is pressed. If the user presses the cancel button all of the file extension modifications will be canceled and the previously saved extensions will be used.

### 6.4.3 EQUAKE Module

Use this Menu Option to run the ACS SASSI EQUAKE module for the active model. The postfixes and extensions for the input and output files are set using the Modules->**Location** menu (section 6.4.1) and Modules->**Extension** (section 6.4.2). All files generated by ACS SASSI EQUAKE will be placed in the directory of the active model.

When EQUAKE starts the UI will open a new tab that shows the out of the module. This Option performs the same functionality as the RUNEQUAKE command.

The EQUAKE module generates earthquake accelerograms compatible with given ground spectra. A time-varying correlation can be specified between the horizontal components. The generated accelerograms can be input for the site response analysis and SSI analysis through SOIL, MOTION, RELDISP and STRESS modules.

### 6.4.4 SOIL Module

Use this Menu Option to run the ACS SASSI SOIL module for the active model. The postfixes and extensions for the input and output files are set using the Modules->**Location** menu (section 6.4.1) and Modules->**Extension** (section 6.4.2). All files generated by SOIL will be placed in the directory of the active model.

When SOIL starts the UI will open a new tab that shows the out of the module. This Option performs the same functionality as the RUNSOIL command.

The SOIL module performs a nonlinear site response analysis using an equivalent linear model for soil hysteretic nonlinear behavior. The iterated equivalent-linear or effective soil properties can be sequentially used in the SSI analysis. The units for the SOIL module are either in British system (feet for layer thickness, ksf for shear modulus, kcf for unit weight and ft/sec for wave velocity) or International system (m for layer thickness, kN/m<sup>2</sup> for shear modulus, kN/m<sup>3</sup> for unit weight and m/sec for wave velocity).

#### 6.4.5 SITE Module

Use this Menu Option to run the ACS SASSI SITE module for the active model. The postfixes and extensions for the input and output files are set using the Modules->**Location** menu (section 6.4.1) and Modules->**Extension** (section 6.4.2). All files generated by SITE will be placed in the directory of the active model.

When SITE starts the UI will open a new tab that shows the out of the module. This Option performs the same functionality as the RUNSITE command

The SITE module is a part of the site response problem. The control point and wave composition of the control motion has to be defined. The information needed to compute the free field displacement vector used is computed and saved on disk in FILE1.

The program also stores information required for the transmitting boundary calculations in FILE2. The actual time history of the control motion is not required in this program module, but later in the MOTION module. The soil motion incoherency is introduced elsewhere, in the HOUSE module.

#### 6.4.6 POINT Module

Use this Menu Option to run the ACS SASSI POINT module for the active model. The postfixes and extensions for the input and output files are set using the Modules->**Location** menu (section 6.4.1) and Modules->**Extension** (section 6.4.2). All files generated by POINT will be placed in the directory of the active model.

When POINT starts the UI will open a new tab that shows the out of the module. This Option performs the same functionality as the RUNPOINT command.

The POINT module consists of two subprograms, namely POINT2 and POINT3 for two- and three-dimensional problems, respectively. The POINT module computes information required to form the frequency dependent flexibility matrix. The results are saved on FILE3. FILE2 created by program module SITE is required as input. Thus, the SITE module must be run before the POINT2 or POINT3 module

### 6.4.7 HOUSE Module

Use this Menu Option to run the ACS SASSI HOUSE module for the active model. The postfixes and extensions for the input and output files are set using the Modules->**Location** menu (section 6.4.1) and Modules->**Extension** (section 6.4.2). All files generated by HOUSE will be placed in the directory of the active model.

When HOUSE starts the UI will open a new tab that shows the out of the module. This Option performs the same functionality as the RUNHOUSE command

The HOUSE module forms the mass and stiffness matrices of all the elements used in discretized model are determined and stored in FILE4 (named modelname.N4). The discretized model may include only the structure or also the irregular soil zone. The HOUSE module can be executed independent of SITE and POINT modules for surface models. The random field decomposition for incoherent motions is performed in this module. For incoherent analysis HOUSE also produces FILE77.

### 6.4.8 FORCE Module

Use this Menu Option to run the ACS SASSI FORCE module for the active model. The postfixes and extensions for the input and output files are set using the Modules->**Location** menu (section 6.4.1) and Modules->**Extension** (section 6.4.2). All files generated by FORCE will be placed in the directory of the active model.

When FORCE starts the UI will open a new tab that shows the out of the module. This option performs the same functionality as the RUNFORCE command

The FORCE module forms the load force vector for external load cases. This module is of no use for seismic soil-structure interaction problems. The loads may correspond to impact forces, rotating machinery, or simple unit forces to be used to determine the impedance of a flexible foundation. The results are stored in File 9. When foundation flexibility is not a significant concern the use of the direct impedance evaluation option included in the ANALYS module active for seismic option is clearly more efficient.

### 6.4.9 ANALYS Module

Use this Menu Option to run the ACS SASSI ANALYS module for the active model. The postfixes and extensions for the input and output files are set using the Modules->**Location** menu (section 6.4.1) and Modules->**Extension** (section 6.4.2). All files generated by ANALYS will be placed in the directory of the active model.

When ANALYS starts the UI will open a new tab that shows the out of the module. This Option performs the same functionality as the RUN ANALYS command

---

The ANALYS module computes the problem solution for the required frequency steps. FILE1, FILE3 and FILE4 (modelname.N4) are always required as input for seismic analysis option. For external load case option, FILE9, and for incoherence analysis FILE77 are also required as inputs.

ANALYS performs the following computational steps:

- Forms the soil flexibility matrix for interaction nodes
- Computes the soil impedance matrix for interaction nodes
- Determines the external load or seismic load vectors, including seismic motion incoherency effects
- Solves the SSI motion equation system for each frequency step, using LU decomposition and then back-substitution algorithm, to obtain the transfer functions for each degree of freedom; ATF for seismic input and DTF for external forces

The ANALYS module output contains the complex transfer functions which depending on the option required are from the control motion to the final motions or from external loads to total displacements. In either case, the results are stored in the FILE8 binary data file.

#### 6.4.10 COMBIN Module

Use this Menu Option to run the ACS SASSI COMBIN module for the active model. The postfixes and extensions for the input and output files are set using the Modules->**Location** menu (section 6.4.1) and Modules->**Extension** (section 6.4.2). When COMBIN starts the UI will open a new tab that shows the out of the module. This Option performs the same functionality as the RUNCOMBIN command

The COMBIN module combines results computed for different frequencies from two FILE8 data files. This module is useful when after the solution was obtained it is found that some additional frequencies are needed to be included. The COMBIN module requires as input two solution files, of FILE8 type, renamed FILE81 and FILE82. The output file of this module is a new FILE8 obtained by combining the two old solution files.

#### 6.4.11 MOTION Module

Use this Menu Option to run the ACS SASSI MOTION module for the active model. The postfixes and extensions for the input and output files are set using the Modules->**Location** menu (section 6.4.1) and Modules->**Extension** (section 6.4.2). All files generated by ACS SASSI MOTION will be placed in the directory of the active model.

When MOTION starts the UI will open a new tab that shows the out of the module. This Option performs the same functionality as the RUNMOTION command

The MOTION module reads the transfer functions from FILE8, and performs an efficient frequency domain interpolation using a two SDOF transfer function model with five parameters to compute the final response at a set of specified nodes selected by the user. Acceleration, velocity, or displacement response spectra may be requested in different location points and degrees of freedom. The MOTION module requires only FILE8 as input. Optional, baseline corrected nodal point motions are saved on FILE13 which is a formatted file.

#### **6.4.12 RELDISP Module**

Use this Menu Option to run the ACS SASSI RELDISP module for the active model. The postfixes and extensions for the input and output files are set using the Modules->**Location** menu (section 6.4.1) and Modules->**Extension** (section 6.4.2). All files generated by RELDISP will be placed in the directory of the active model.

When RELDISP starts the UI will open a new tab that shows the output of the module. This option performs the same functionality as the RUNRELDISP command.

This module calculates the relative displacement of the model and outputs the displacements of all points in the module in a frame format for animation.

#### **6.4.13 STRESS Module**

Use this Menu Option to run the ACS SASSI STRESS module for the active model. The postfixes and extensions for the input and output files are set using the Modules->**Location** menu (section 6.4.1) and Modules->**Extension** (section 6.4.2). All files generated by STRESS will be placed in the directory of the active model.

When STRESS starts the UI will open a new tab that shows the output of the module. This option performs the same functionality as the RUNSTRESS command.

The STRESS module computes requested stress, strain, and force time histories and peak values in the structural elements. The module STRESS requires FILE4 (modelname.n4) and FILE8 as inputs. Stress time histories are saved on FILE15. For beam elements, the transfer functions of sectional forces and moments are saved on FILE14. FILE15 and FILE14 are formatted files.

#### **6.4.14 NONLINEAR Module (Option NON)**

As shown in the ACS SASSI modular configuration chart in Figure 1.1, the NONLINEAR module is a part of the Option NON capability. This module is intended to be used in conjunction with other modules in an iterative procedure to produce a nonlinear analysis. The NONLINEAR module should run in batch mode, since the equivalent-linearization procedure involves an iterative scheme of running HOUSE-ANALYS(Restart)-MOTION-RELDISP-COMB\_XYZ\_THD-

---

NONLINEAR modules repeatedly until the convergence is reached. The NONLINBAT command creates a batch file that will set up a generic iterative batch loop for different nonlinear analyses.

Use this Menu Option to run the ACS SASSI NONLINEAR module for the active model. The postfixes and extensions for the input and output files are set using the Modules->**Location** menu (section 6.4.1) and Modules->**Extension** (section 6.4.2). All files generated by NONLINEAR will be placed in the directory of the active model.

The Demos 9 and 10 show how to run nonlinear structure SSI analysis using Option NON for a concrete structure and a base-isolated structure.

#### 6.4.15 LOADGEN Module (Option A)

**WARNING:** As shown in the ACS SASSI modular configuration chart in Figure 1.1, the LOADGEN module is a key module for the Option A capability. The LOADGEN module should be used only in the interactive mode. In this section only a brief description of the LOADGEN module functionality is included. The full details on the LOADGEN module functionalities and application are provided in “ACS SASSI-ANSYS Integration Capability” for Options A and AA, User Manual, Revision 4.

This UI interactive module is used in both the “ANSYS Eq. Static Analysis” and the “ANSYS Dynamic Analysis” menu options. The postfixes and extensions for the input and output files are set using the Modules->**Location** menu (section 6.4.1) and Modules->**Extension** (section 6.4.2). All files generated by LOADGEN module will be placed in the directory of the active model. LOADGEN does not have currently a command to run directly from the UI command line do to the complexity of the user options that affect the input data. The user needs to use the Modules menu selection for “ANSYS Eq. Static Load” or “ANSYS Dynamic Load”. A new window dialog will open. The detailed input description on how to use LOADGEN in Option A is provided in the “ACS SASSI-ANSYS Integration Capability” user manual.

#### **ANSYS Eq. Static Load (2<sup>nd</sup> Step is Equivalent Static Stress Analysis)**

When the user select this menu option the popup window show below will appear and request input from the user. The data required for an equivalent static load analysis will come from the UI AFWRITE command, running an analysis of the model and building a model in ANSYS.

Once the user has entered the data into the window and presses ok the UI will write an input file for the LOADGEN module and start the LOADGEN. Below is the LOADGEN input window.

Mass Data to add from the ACS SASSI to the ANALYS module – User selection of what data will be added to the LOADGEN module. This selection box will modify a flag in the LOADGEN input

to tell the module what type of analysis to run and disable data entry boxes in the “SASSI Model and Results Input” section of this window for data that will not be used.

**Use Multiple File List Inputs** – User option to generate multiple seismic load files for different selected critical time steps in a single LOADGEN run.

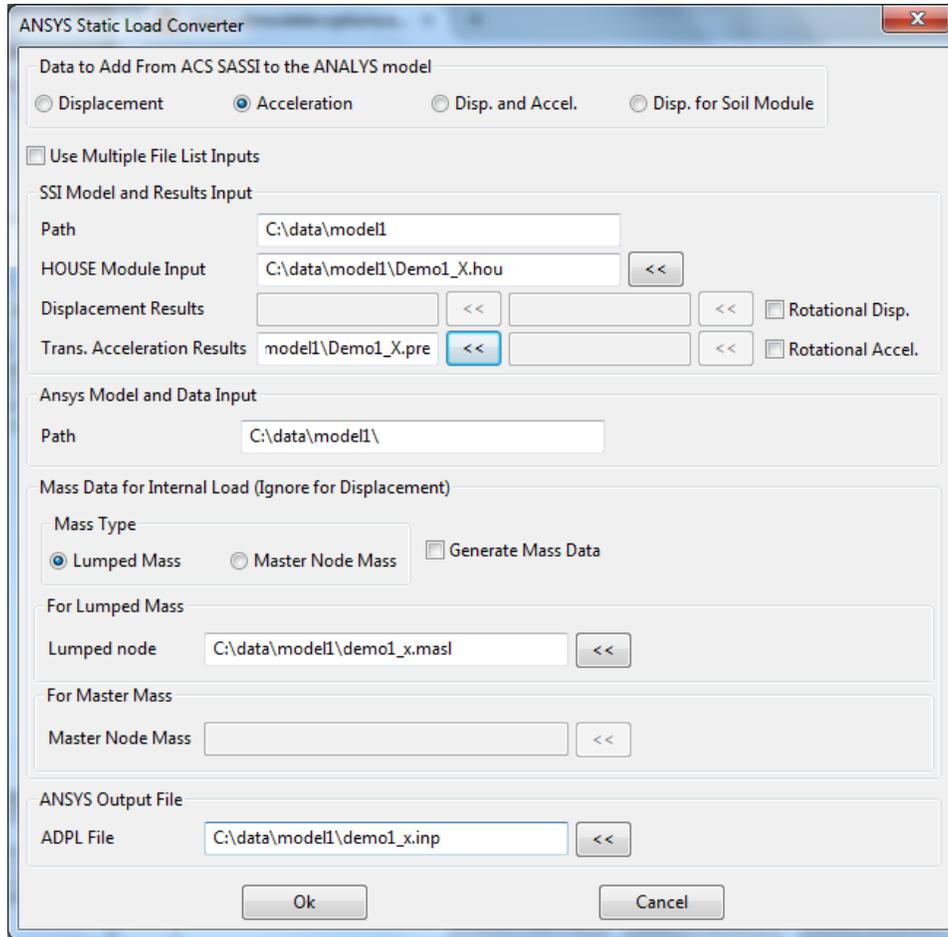
### **SASSI Model and Results Input**

**Path** - The path where the model data resides. The other SASSI Model inputs will use this directory if the other inputs in “SASSI Model and Results Input” do not specify the full path.

**HOUSE module Input** - Input file to the HOUSE module. This file is generated by the afwrite command when the user selects to generate an input file for the house module

**Displacement Results** – Add displacement frame data file if the box is enabled. The first edit box is to be used for translation displacement only. The second edit box is for rotational displacement data. If the user want to add rotational displacement the “Rotational Disp.” checkbox should be checked which will enable the second edit box for file name entry.

**Acceleration Results** - Add acceleration frame data file if the box is enabled. The first edit box is to be used for translation acceleration only. The second edit box is for rotational acceleration data. If the user want to add rotational acceleration the “Rotational Accel.” checkbox should be checked which will enable the second edit box for file name entry.



### Ansys Model and Data input

**Path** - The path for the ANSYS® input data.

### Mass Data for Internal Load

**Mass Type** – Flag to use Lumped or master mass type

**Generate Mass** – Flag to generate a mass file. If this is checked the Lumped mass or Master mass file boxes become the name of the mass file output and that file will be overwritten

**Lumped Mass** – Name of the lumped mass file being used or created

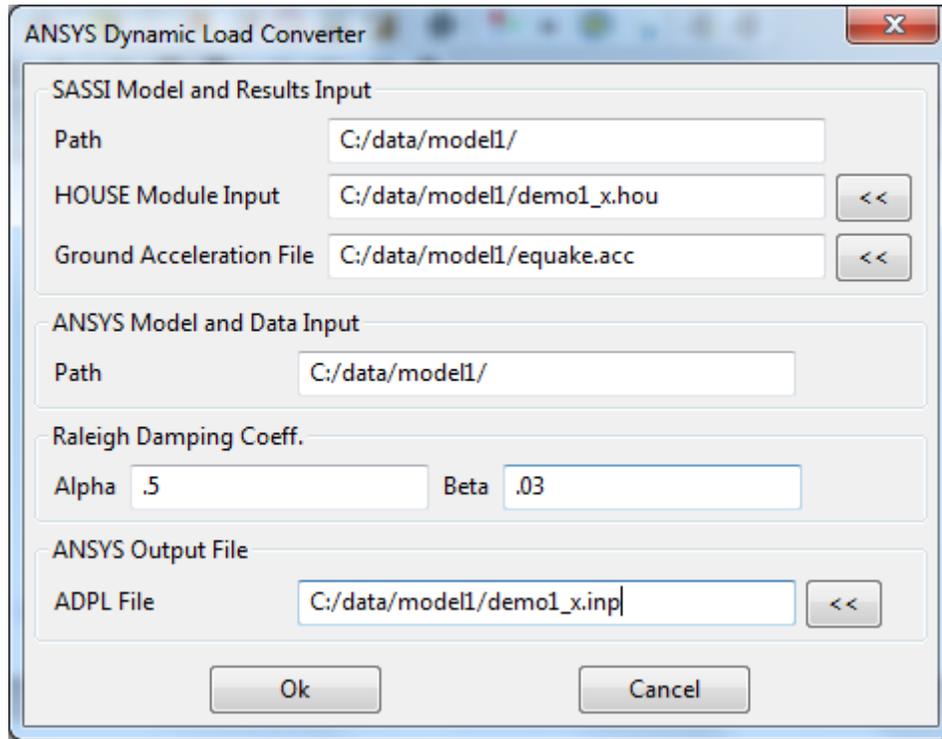
**Master Mass** – Name of the master mass file being used or created

### ANSYS Output File

**APDL File** – Full path name of the output file of the LOADGEN module. This file will be in an ANSYS APDL format.

### **ANSYS Dynamic Load (2<sup>nd</sup> Step is Dynamic Stress Analysis)**

When the user select this menu option the popup window show below will appear and request input from the user. Once the user has entered the data into the window and presses ok the UI will write an input file for the LOADGEN module and start the LOADGEN.



**SASSI Model and Results** - Full path of the folder that contains the displacement frames

**HOUSE Module Input** - File name of the .hou file, or browse to it by clicking on the arrow button next to the box

**Ground Acceleration File** – Ground (surface foundation) or kinematic SSI acceleration (embedded foundation) file name. This file will be prepared by the user for the ANSYS dynamic analysis.

**ANSYS Model and Data Input** - Path of the ANSYS file folder

**Rayleigh Damping Coeff** - Raleigh damping coefficients, alpha, beta, for the ANSYS dynamic analysis.

**ANSYS Output File** - name of the file that will contain the ANSYS APDL input commands for the dynamic analysis using the time-domain direct integration method. This is the file will be loaded in ANSYS.

During the generation of the dynamic step load files, the ANSYS dynamic load generator uses all the relative displacement data files with respect to the free-field motion from the SSI analysis results. The user should make sure these files have been copied to the ANSYS folder defined in the “Path” input box of “SASSI Model and Results Input” section.

**WARNING:** *The use of Rayleigh damping is viewed a significant limitation of the ANSYS dynamic analysis capability for performing the stress analysis in the 2<sup>nd</sup> step.*

Demo 5 and 6 show how to apply Option A for the superstructure stress analysis and the soil pressure computation including foundation-soil separation.

#### **6.4.16 SSI2ANSYS Module (Handling ANSYS Super-Elements via Option AA)**

As shown in the ACS SASSI modular configuration chart in Figure 1.1, the SSI2ANSYS module is a part of the Option AA capability. The SSI2ANSYS module should be used only in the interactive mode. In this section only a brief description of the SSI2ANSYS module functionality is included. Details on the SSI2ANSYS module functionalities and application are provided in “ACS SASSI-ANSYS Integration Capability” for Options A and AA, User Manual, Revision 4.

ANSYS MATRIX50 super-element is used in the ANSYS substructuring analysis, which condenses a set of finite elements into a single super-element (SE) represented by its dynamic matrices, K, M and C. ANSYS MATRIX50 element type is often used in the ANSYS modeling for various reasons.

The MATRIX50 SE capability is handled in ACS SASSI by the SSI2ANSYS module as described in Figure 1.1. This module provides new capabilities that let the user convert his ANSYS model having MATRIX50 into a complete ANSYS model or a ACS SASSI runnable model. The SSI2ANSYS Module allows the user two ways of converting the MATRIX50 elements, as follows:

Option 1: The SE-to-GE approach is to convert the ANSYS MATRIX50 elements into a set of ACS SASSI GM elements. Basically, the MATRIX50 dynamic matrices are ported to the ACS SASSI model via the conversion on the matrix components to general matrix stiffness/mass elements.

Option 2: The Add-SE approach is to add the MATRIX50 element K, M and C matrices (extracted using GENERAL PASS option in ANSYS to create the SE .sub files) to the main ANSYS model K, M and C matrices. This matrix additions are done outside of ANSYS.

**WARNING:** *The use of the MATRIX50 SE assembly directly in ANSYS does not provide accurate results for all problems.*

When the user selects the “ANSYS Super-Element Utilities” menu option, the popup window pictured below will appear. The user will select which type of conversion is to be performed from the radio box. Once the user has completed the data entry and clicks the OK button, The UI module will write to the file name given “SE Utility File Name” and then, launch the SSI2ANSYS module. The resultant files depend on the analysis type. For Option 1, SE-to-GE approach, the result is listed in the “General Element Output File” text file with extension .pre). For Option 2, Add-SE approach, the result are the modified COOSKI\_r, COOSK\_r, COOSMI\_r, COOSM\_r, COOCI\_r, COOSC\_r, and Node2Equ\_Stru.map ready for use in the HOUSE(FSA) and ANALYS(FSA) Option AA analysis.

Super Element Utility

ANSYS MATRIX50 Super Element Operation

Convert ANSYS SE Matrices to SASSI General Elements

Assemble SE Matrices into ANSYS Main Structure Matrices (Option AA)

SE Matrix Folder

Main Structure Matrix Folder

Number of Super Elements

General Matrix ID Start

Element Group ID Start

Input SE Files Names (.sub) One by One:

Add

Remove

General Element Output Folder

General Element Output File (.pre)

Ok Cancel

**ANSYS MATRIX50 Super-Element Operation** – Radio button option where the user can choose between the SE-to-GE and Add-SE approaches. Each option requires a different set of data. When the user selects a SE-to-GE conversion process any data entry that is not required by that conversion will be disabled.

**SE Matrix Folder** – Directory where the MATRIX50 elements reside. This directory is also where the SE utility input file will be written.

**Main Structure Matrix Folder** – Directory where the main structure data is residing (Add-SE approach only)

**Number of Super Elements** – The number of the .sub files to be processed

**General Matrix ID Start** – The matrix id number which starts the output file (SE-to-GE only)

**Element Group ID Start** – The group number which starts the output file (SE-to-GE only)

**Input SE File Names One-by-One (.sub)** – The names of the MATRIX50 .sub files to be converted to ACS SASSI GE elements, or considered further with main ANSYS module using Option AA

Add – Adds the name in the edit box next to the Add button to the list box below

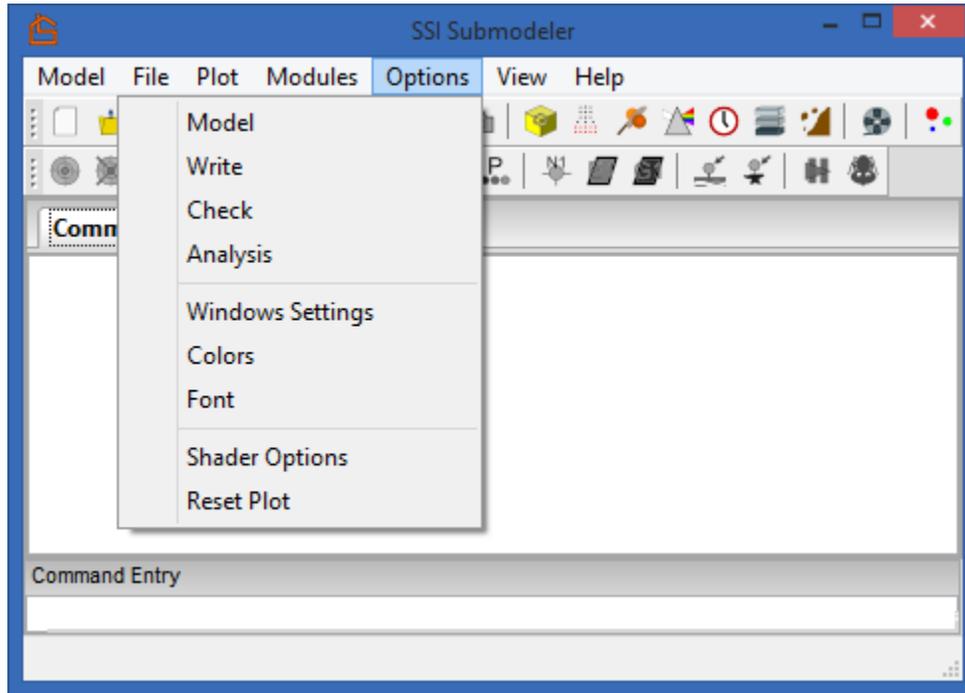
Remove – Removes the highlighted file name from the list box

**General Element Output Folder** – The directory where the results are stored (SE-to-GE only)

**General Element Output File (.pre)** – The name of the .pre file where the converted SE matrix data is stored. This should only be the filename without the .pre extension (SE-to-GE only)

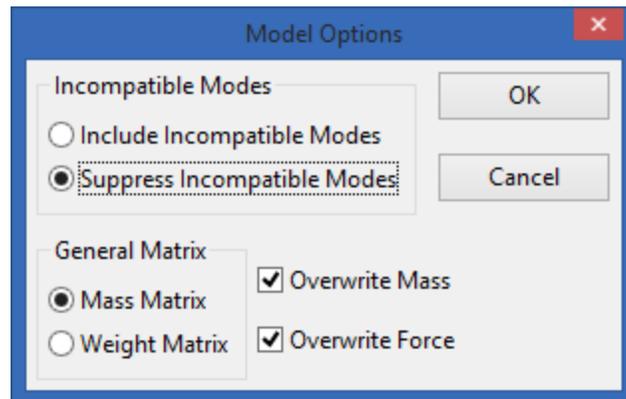
Demo 11 shows an example of how to use the ANSYS MATRIX50 super-elements for SSI analysis.

## 6.5 Options Submenu



### 6.5.1 Model

When the user selects this option a window will appear which allows the user to change the model options for the active model.



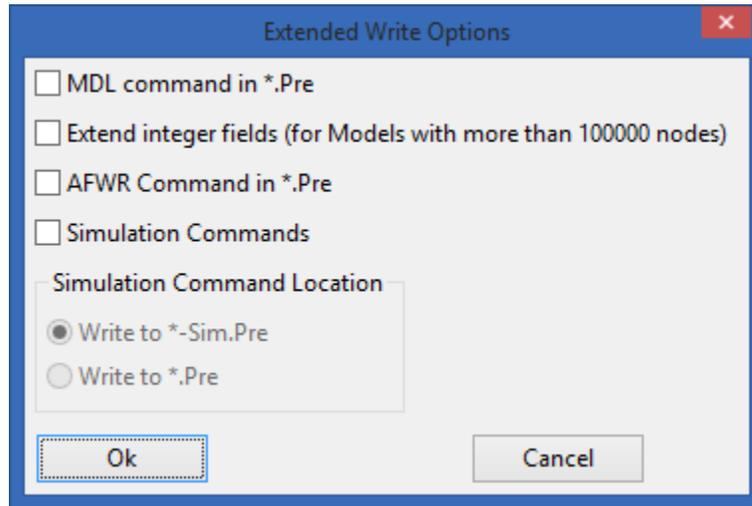
### 6.5.2 Write

The Write option window allow the user to select new capabilities for the WRITE and AFWRITE commands. Initially the UI module was designed to be a replacement of older PREP with no changes to the WRITE and AFWRITE commands.

For larger SSI model sizes with more than 99,999 nodes, it is necessary to expand the fixed integer field size for a number of SSI module inputs generated by the WRITE command. *This*

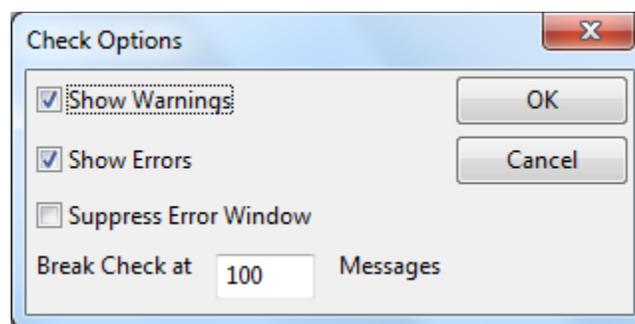
*integer filed extension is applicable only to the enhanced solver version that can run up 300,000 nodes SSI models (IKTR8\_300K).*

Options were also added to the WRITE command to help users with model analysis in batch. If no options from this window are selected the AFWRITE and WRITE command will write the same files as the commands in the previous PREP .pre files.



### 6.5.3 Check

This option will set the check option for the UI module. Changing these options will change the output to the .err file for a model. The user has the ability to hide or display errors and warnings. The user can also select how many of each type of message to print to the .err file per module. The Check functionality will still check and report the total number of error and warning for all of the modules selected in the command window, but the user will only see the messages until the user selected break number. Any messages after the break will not be reported. These options are global any change in these options will affect the next check output of all models. Currently these options are not saved and are reset to default every time the UI is opened.



### 6.5.4 Analysis

Select Analysis from the Options submenu. This menu item is used to set the analysis options for the active module.

The Analysis selection will bring up a window that allows the user to set options for the SSI models that will be used. The opened Analysis window is broken up into tabs by SSI modules. Some of the windows share variables and if changed in one window the variable will change in all windows

### ***EQUAKE Module Options***

Set the Analysis Options for the ACS SASSI EQUAKE Module:

The screenshot shows the 'Analysis Options' dialog box for the EQUAKE module. The dialog has a title bar with a close button and a tabbed interface. The 'EQUAKE' tab is selected. The 'Spectrum Files' section includes a 'Spectrum Number' dropdown set to 1, and three file input fields for 'Spectrum Input File', 'Spectrum Output File', and 'Acceleration Output File', all pointing to 'C:\Users\Owner\Desktop\Demo1s\'. The 'Optional Spectrum Files' section has checkboxes for 'Accel. Record' and 'External Accel', and an 'Acceleration Input File' field. The 'Target PSD' section has a checked 'Use Target PSD' checkbox and a 'PSD File' field. A 'Correlation' table is present with columns 'Time' and 'Corr.', and rows 1 through 7. The 'Number of Frequencies' is set to 24, 'Initial Random SEED' is 11975, 'Damping Value' is 0.05, 'Time Step' is 0.005, 'Total Duration' is 15, and 'Number of SEEDs' is 0. There is also a 'Spectra Title' field with the text 'Title for spectra'. 'Ok' and 'Cancel' buttons are at the bottom right.

	Time	Corr.
1	0	0
2	1	0
3	3	0
4	5	0
5	6	0
6	8	0
7	9	0

The following options allow you to specify the analysis options for the EQUAKE module:

**Spectrum Number** - Select the spectrum number (max. three spectra for the three translation in space can be defined).

**Spectrum Input File** - Type or select the given spectrum input file for the selected spectrum.

**Edit** - Click this button to edit the selected spectrum input file.

**Spectrum Output File** - Type or select the spectrum output file for the selected spectrum.

**Acceleration Output File** - Type or select the acceleration output file for the selected spectrum.

#### **Optional Acceleration Files**

**Accel. Record** - Select this option if you want to use “seed record” acceleration input files

**External Accel.** – Select this option if you want to compute acceleration response spectra for an external acceleration time-history. This acceleration should have same time step and number of data points.

**Acceleration Input File** - Type or select the acceleration input filename for the above selected option.

**Number of Frequencies** - Type the number of frequency steps for defining the given design spectrum.

**Initial Random Number** - Type the initial SEED random number with 5 digits.

**Damping Value** - Type the damping ratio value for the given input design spectrum.

**Time Step** - Type the time step (sec). The value should be the same as set in the Analysis Options for other SSI modules in the dialog box.

**Total Duration** - Type total motion duration (sec).

**Correlated** -Select this option for correlated X and Y motion components. If this option is selected, type the time-correlation coefficients pairs in the grid.

**Spectra Title** - Type the title.

**Number of Random SEEDs** - Number of random SEED number trials. It should a non-zero integer value. EQUAKE will retain the best response spectrum fitted acceleration history from

the set of simulated compatible accelerations. The PSD criterion is not checked for the simulated acceleration histories. Only the PSD .psd file is generated.

**Target PSD** - If check box is checked, then the user should provide the filename for the target PSD text file. The target PSD file format consists of two columns; the first column is frequency, and the second column is target PSD amplitude data. The PSD amplitude data should be either in the International units as  $\text{cm/sec}^3$  or British units as  $\text{inch}^2/\text{sec}^3$  depending on the unit system selection for Gravity Acceleration input.

**WARNING:** *The EQUAKE module uses the gravity acceleration units to determine velocity and displacement units and PSD units. The gravity acceleration value should be defined in any of the SOIL, SITE or HOUSE input windows. The gravity acceleration units should be either in the International System (IS)  $\text{m/sec}^2$  or the British System (BS)  $\text{ft/sec}^2$ , respectively. It should be noted that these are not the same units that are used for the simulated acceleration, velocity and displacement histories (acceleration is in g's), or the acceleration PSD (amplitude in the IS  $\text{cm/sec}^2$  or in the BS  $\text{ft/sec}^2$ ). Other response quantities that use the peak acceleration (PGA), velocity (PGV) and displacement (PGD) values are provided in the EQUAKE output in the same units as printed in the NUREG/CR 6728 Tables 3-5 and 3-6.*

**SOIL Module Analysis Options**
**Input Motion:**

**Nr. of Fourier Components** - Type the number of values to be used in the Fourier transform. The value is the same as set in the Analysis Options - SITE dialog box.

**Time Step of Control Motion** - Type the time step of control motion (sec). The value is the same as set in the Analysis Options - SITE dialog box.

---

**Number of Values** - Type the time number of acceleration values to be read from the time history file.

**Multiplication Factor** - Type the multiplication factor for scaling the time history. Use only if Max. Value for Time History is blank.

**Max. Value for Time History** (in g's) - Type the maximum value of time history to be used. The values of the time history will be scaled to this value. Use only if Multiplication Factor is blank.

**Acceleration of Gravity** - This acceleration value is used for free-field or site response analysis using SOIL module. The acceleration of gravity value should be in ft/sec<sup>2</sup> or in m/sec<sup>2</sup>, depending if the British System (BS) or International System (IS) is used. If BS is used, then the soil weight density should be input in lb/ft<sup>3</sup>, while if IS is used, then the soil weight density should be input in kN/m<sup>3</sup>. A change of this value does not affect the value used for the SSI analysis in SITE options.

**WARNING:** *The use of tones with meters is not permitted for the IS option.*

**Number of Header Lines** - Type the number of header lines at the beginning of the acceleration time history file.

**Input Direction** – This flag is used to specify if the motion direction is in the horizontal or the vertical direction. For horizontal selection, type 0, and for vertical, type 1. If vertical direction is selected, no iteration should be run. The shear wave velocity is  $V_s$  for horizontal selection and  $V_p$  for vertical selection. Please check SOIL output file with input echo to make sure that you selected the right option.

**Control Point Layer** - Type the layer number of the control point. The value is the same as set in the Analysis Options - SITE dialog box

**File** - Type the full path and name of the acceleration time history input file. To view the time history file, use the Plot/Time History command.

**Assign as Outcrop Motion** - Select this option if you want to assign the input motion as outcrop motion. Otherwise, the input motion is assumed to be an in-column motion.

**Equivalent-Linear Soil Behavior:**

**Save Strain-Compatible Soil Properties** – Save iterated soil properties in FILE88 for further used for the SSI analysis.

**Number of Iterations** – Input number of iterations (recommended is 8).

---

**Equiv. Uniform/Max. Strain** – Input ratio between the effective, equivalent-linear strain and maximum strain, assumed the same for all soil layers (recommended 0.60 to 0.70).

**Soil Profile:**

**Layer Number** - Select the active layer number. All following options apply to this layer.

**Property Number** - Type the soil layer property number for the active layer

**Dynamic Soil Property** - Select the dynamic soil property, i.e. normalized shear modulus and damping as functions of the effective soil shear strain for the active layer.

**Accelerations:** - Select the acceleration output options for the active layer.

**Response Spectrum** - Select response spectrum output options for the active layer.

**Stresses - Strains** - Select stress and / or strain output options for the active layer.

**Spectral Amplification Factor** - Select the spectral amplification factor (SAF) output options for the active layer.

**Fourier Spectrum** – Do not use this option. Please compute Fourier Spectrum using the EQUAKE module

**Nonlinear Soil Behavior:**

**Nonlinear Time Domain** – Select this option to use the hyperbolic model in time-domain.

**Subincrements Per Timestep** – Number of subincrements between time step.

**Displacement Convergence Error** - The acceptable error norm value for the displacements to determine if the solution has converged.

**Force Convergence Error** - The acceptable error norm value for the forces to determine if the solution has converged.

**Equilibrium Iterations** - Maximum number of equilibrium iterations performed in the Newton-Raphson integration scheme before the program terminates.

**Bedrock interface** - Layer interface/connection type between the last layer and the half space layer. Only Rigid option included in this version.

0 - Rigid bedrock (e.g. Bottom Layer is rigidly connected to the half space thus the bottom layer and the half space have the same acceleration as the input acceleration)

1 – Visco-elastic bedrock (e.g. Bottom Layer is connected to the half space through a visco elastic damper based on the half-space material properties. (Not applicable to this version)

**Damping Type** - Option to choose the small strain damping type for the model.

1 – Frequency independent damping

2 - Visco-elastic damping (requires viscosity input)

3 - Rayleigh damping (requires Rayleigh coefficients for Mass and Stiffness – see below)

**Mass Matrix Mult.** - Rayleigh Damping Mass Matrix Multiplier

**Stiff Matrix Mult.** - Rayleigh Damping Stiffness Matrix Multiplier

**Curve Fit Hyperbolic Parameters** - Option to determine if the Hyperbolic Parameters are directly input (if known) or found from a curve fit of the dynamic property stiffness curves. When using curve fit Beta, S exponent, Reference Strain and Viscosity are ignored.

0 – Parameters input by user

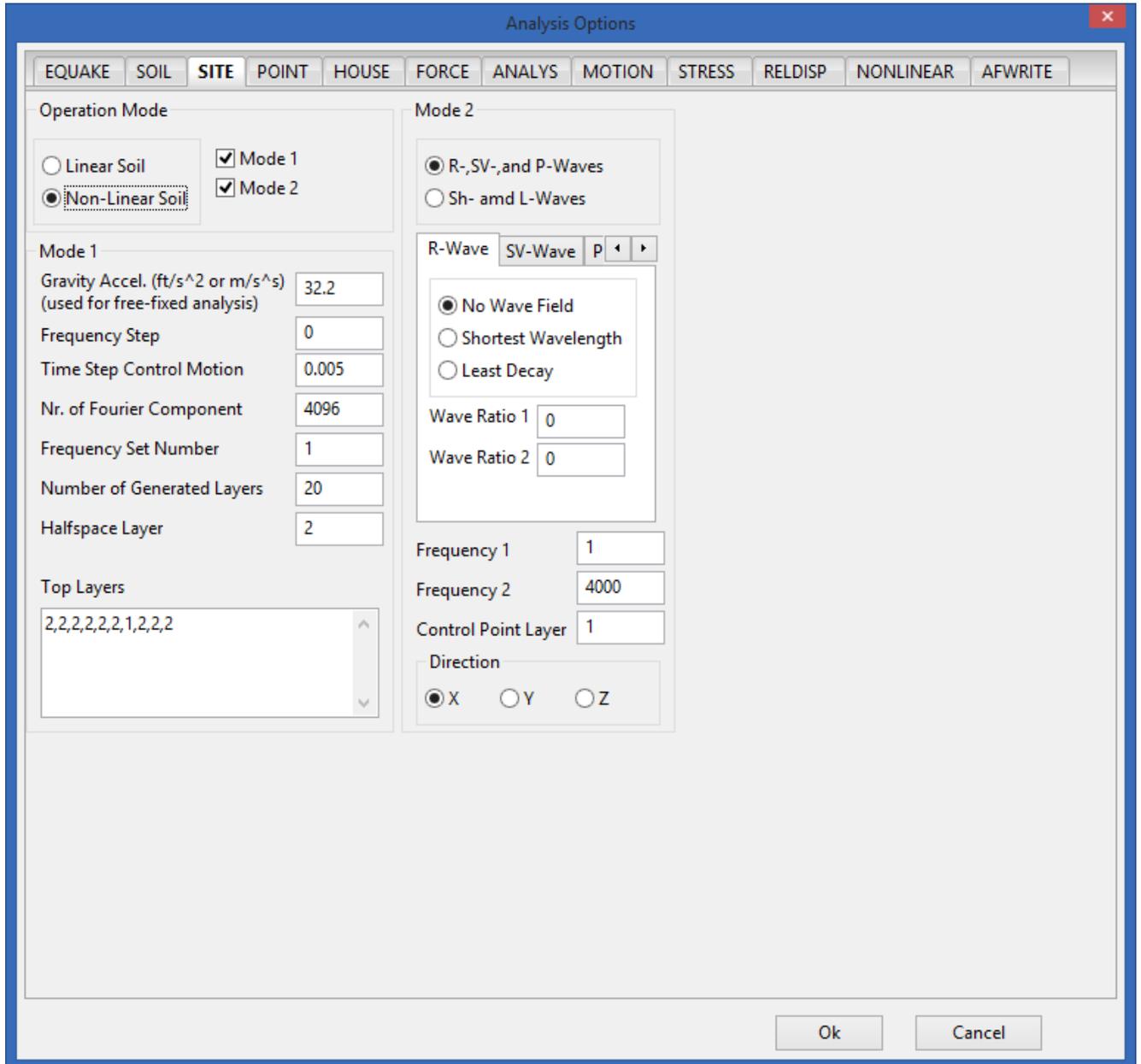
1 - Parameters computed from curve fit using a error minimization algorithm

**Beta** - Beta Parameter for soil layer

**S exponent** - Exponent Parameter for soil layer

**Reference Strain** - Yield shear strain for soil layer

**Viscosity** - Soil viscosity for soil layer

**SITE Module Options**

The SITE module has two basic operation modes:

Mode 1: Form and solve the transmitting boundary eigenvalue problem for POINT

Mode 2: Solve the linearized site response problem

**Mode 1:** The SITE module reads the soil layer properties and for each specified frequency forms the transmitting boundary submatrices for Rayleigh and Love wave cases. Then it solves the two eigenvalue problems, from which the eigenvalues and eigenvectors of the soil layering are

obtained. The results are then written to FILE2. The halfspace condition is also simulated at this stage. SITE automatically generates a specified set of sublayers whose thickness vary with frequency attached to viscous dashpots at the base. The generated sublayers and dashpots are then added to the fixed top layers.

FILE2 provides the information needed to run SITE in Mode 2 as well as to compute the transmitting boundary by POINT. Since the eigenvalue problems to be solved for an arbitrary three-dimensional horizontally layered site are the same as those to be solved for a plain strain model, the information from FILE2 can be used for both two- and three-dimensional cases as well.

**Mode 2:** The SITE module recovers the soil layer properties and the eigen solutions for the Rayleigh and Love surface wave modes from FILE2. Then, according to the existence of each wave type, SITE computes the mode shapes and wave numbers for each wave type in the defined coordinate system. Then, once the composition of the wave types causing the seismic environment and the nature of the control motion is known, SITE will scale and superimpose the results of all the wave types. These results are then stored in FILE1, which is used later for seismic analysis. Thus, this file will not be generated for foundation vibration analysis. If the seismic environment is the same for a two- and three-dimensional case, the information from FILE1 can be used for both problems.

The following options allow you to specify the analysis options for SITE module:

**Operation Mode** - Select the operation mode from Linear Soil and Nonlinear Soil. This selection will generate a linear or a nonlinear SSI analysis (primary soil nonlinearity is included based on SOIL equivalent-linear site response analysis). If the Nonlinear Soil option is selected then the dimension for the soil input data for SITE and SOIL should be consistent either in BS or IS of unities (kip, ft, ksf, kcf or kN, m, kN/m<sup>2</sup>, kN/m<sup>3</sup>). The check boxes Mode 1 and Mode 2 enable/disable the two basic operation modes of module SITE.

**Acceleration of Gravity** - Type the acceleration of gravity in ft/sec<sup>2</sup> for BS, or m/sec<sup>2</sup> for IS. This acceleration of gravity value is used for SSI analysis. A change of this value does not affect the value used for free-field analysis in SOIL options.

**Frequency Step** - Type the frequency step (Hz).

**Time Step of Control Motion** - Type the time step of control motion (sec).

**Nr. of Fourier Components** - Type the number of values to be used in the Fourier transform. The value should be a power of 2, otherwise the UI uses the closest power of 2 and generates a warning message.

---

For a time history analysis, the time step and number of Fourier components for the selected time history of the control motion must both be defined, and the frequency step may be left blank. SITE computes the corresponding frequency step =  $1/(\text{time step} * \text{number of Fourier components})$ . This frequency step may then be used to set up the frequency numbers in the specified frequency set.

For a single harmonic analysis the frequency step must be given and may be directly used to set up frequency numbers in the specified frequency set. In this case, the time step of control motion and the number of Fourier components are not used and therefore may be left blank.

**Frequency Set Number** - Type the number of the frequency set created with the FREQ instruction. The frequency numbers from the specified set must be positive integer numbers. SITE will automatically reorder the frequency numbers in ascending order and will stop if two or more equal frequency numbers are detected. The frequencies are obtained by multiplying the frequency number by the frequency step.

**Number of Generated Layers** (used for half-space simulation)- Type the number generated layers to simulate a half-space (between 10 and 20). Leave this edit box blank (or type 0) if you wish to suppress the half-space simulation. If the half-space simulation is suppressed, the soil profile will be assumed on rigid base; otherwise a specified number of sublayers whose thickness varies with frequency are generated to simulate the half-space. Also, SITE will add viscous boundary to account for the body wave radiation damping in the halfspace through the lower boundary. This number can't be larger than 20.

**Half-Space Layer** - Type the number of the soil layer property associated with the half-space or baserock properties (see L command)

**Top Layers** – This input define the free-field soil layering above half-space/baserock) - Type a list containing the numbers of the soil layers (created with the L command) defining the soil structure. The allowed separators are blank, tab, ',', ';', and enter. The maximum number of soil layer numbers is 200. The first number corresponds to the top most layer. To view the soil layers, select the Plot/Layers command. If two or more top layers have the same properties, simply use the same soil layer number instead of defining soil layers with the same properties. Soil properties should be consistent in SOIL and SITE modules if non-linear SSI analysis is performed.

**WARNING:** *For embedded models, the L layer numbers shall be equal to the embedment soil layer numbers that are the same with the far-field layer numbers. Otherwise the UI might generate incorrect L material for excavated soil layers included in HOUSE input (.hou file).*

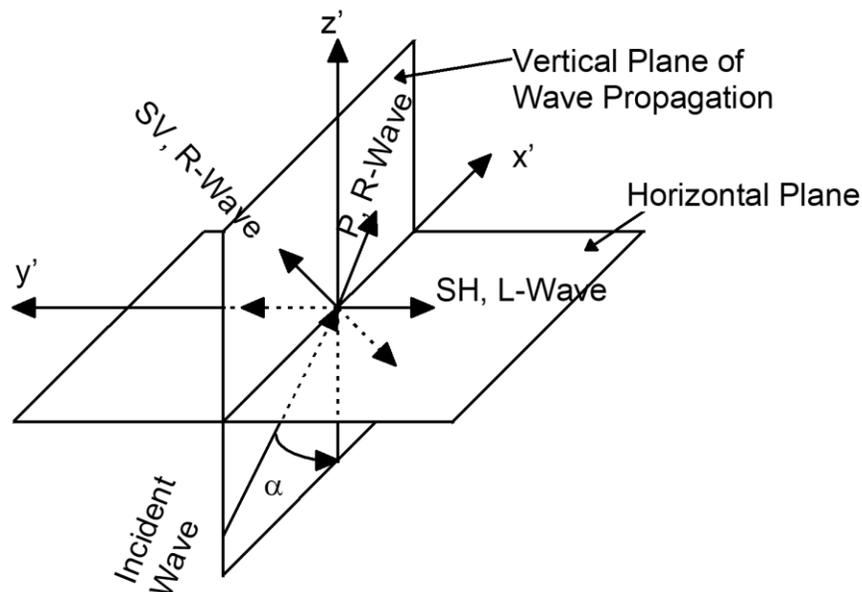
**R-, SV-, and P-Waves** - Select this option if your model uses a combination of P-, SV-, and R-waves (2 dofs per node).

**SH- and L-Waves** - Select this option if your model uses a combination of SH-, and L-waves (1 dof per node)

The seismic environment may be assumed to consist of one single wave type or several wave types. The basic wave types are P-waves and S-waves, which are also called body waves. When these waves impinge on the ground surface or layer interfaces, surface waves which include R-waves and L-waves may be generated.

P-waves involve motions in the direction of wave propagation. S-waves involve motions perpendicular to the direction of wave propagation. S-wave motions in the vertical plane are called SV-waves. Horizontal S-waves are called SH-waves. R-waves involve horizontally propagating elliptical motions in the vertical plane and L-waves consist of horizontal motions perpendicular to the horizontal direction of wave propagation.

With the above definitions, SITE sets up a coordinate system so that P-waves, SV-waves, and R-waves involve particle displacements in the  $x'z'$  plane while SH-waves and L-waves involve particle displacements along the  $y'$  axis. Therefore, the  $z'$  axis is always vertical up,  $x'$  is in the vertical plane of wave propagation, and  $y'$  is perpendicular to  $x'$  and  $z'$  following the right-hand rule.



**R-Wave** - Select this page to set the R-wave field options - choose between No R-Wave Field, Shortest Wavelength, and Least Decay.

**SV-Wave** - Select this page to set the SV-wave field options - choose between No SV-Wave Field, SV-Wave Field.

**P-Wave** - Select this page to set the P-wave field options - choose between No P-Wave Field, P-Wave Field.

**SH-Wave** - Select this page to set the SH-wave field options - choose between No SH-Wave Field, SH-Wave Field.

**L-Wave** - Select this page to set the L-wave field options - choose between No L-Wave Field, L-Wave Field.

**Wave Ratio 1** - Type the wave ratio for the selected wave type at the first frequency.

**Wave Ratio 2** - Type the wave ratio for the selected wave type at the second frequency.

In the case of seismic environment composed of two or more wave types, the ratio of participation of each wave type must be given. This ratio in general may be frequency-dependent and is defined at two discrete frequencies for each wave type. These frequencies must cover the frequency range of analysis. The ratio values for intermediate frequencies will be obtained by simple interpolation and therefore need not be given at exact frequencies for which complete solution is required.

In the case of seismic environments consisting of one simple wave type, the two frequencies (one in the beginning and the other at the end of the frequency range of analysis) with assigned ratio values of 1 are enough to define the ratio curve.

All ratio values are positive decimal numbers less than or equal to 1, and the addition of the ratio values of all the participating wave types at each frequency must be 1.

**Incident Angle** - Type the incident angle of the selected wave type. The incident angle is defined as the angle between the direction of propagation and the  $z'$  axis. For vertically propagating waves, this angle is zero.

**Frequency 1** - Type the first frequency to define ratio curve.

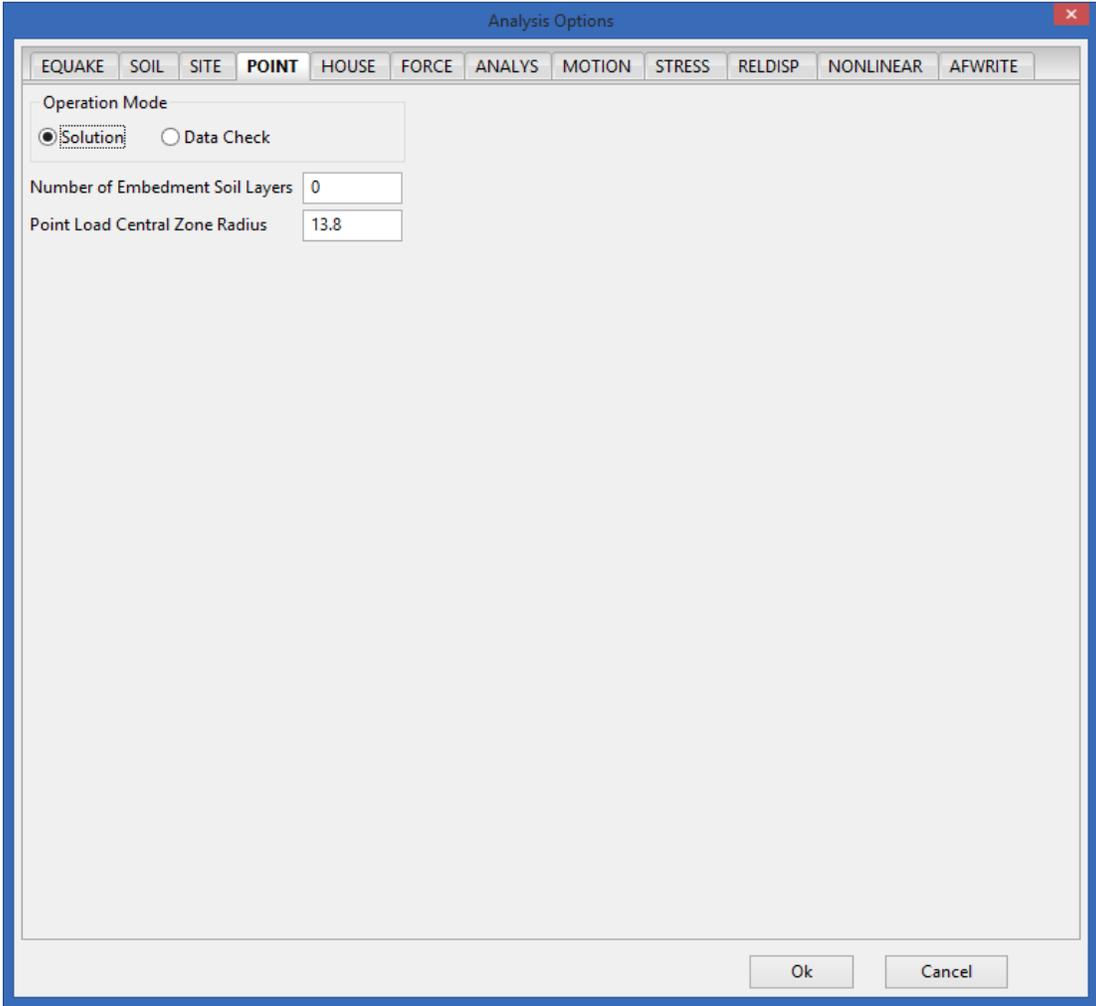
**Frequency 2** - Type the second frequency to define ratio curve.

**Control Point Layer** - Type the layer number of the control point. The control point is defined as the point where the control motion is specified. It will be located at the top of the specified layer number, e.g. 1 for the control point at the surface.

**Direction** - Select the direction of the control motion in the  $x'y'z'$  coordinate system. The transformation of the coordinates from the  $x'y'z'$  system to the final  $xyz$  coordinate system of soil structure will be done by the ANALYS module.

**WARNING:** It should be noted that if the “Simultaneous Cases” is selected, then, the SITE module should be run three times before ANALYS for all X, Y and Z direction inputs, and generate the FILE1X, FILE1Y and FILE1Z files that are copies of FILE1 after each of the three runs. The user should select the SV waves for the X-direction by selecting x' direction and 0 angle in the .sit SITE input file, the SH waves for the Y-direction by selecting y' direction and 0 degree angle and the P waves in the Z-direction by selecting z direction and 0 angle. The coordinate transformation angle in the .anl ANALYS input file should be 0.

### POINT Module Options



The screenshot shows the 'Analysis Options' dialog box with the 'POINT' tab selected. The 'Operation Mode' section has two radio buttons: 'Solution' (selected) and 'Data Check'. Below this, there are two input fields: 'Number of Embedment Soil Layers' with the value '0' and 'Point Load Central Zone Radius' with the value '13.8'. At the bottom right, there are 'Ok' and 'Cancel' buttons.

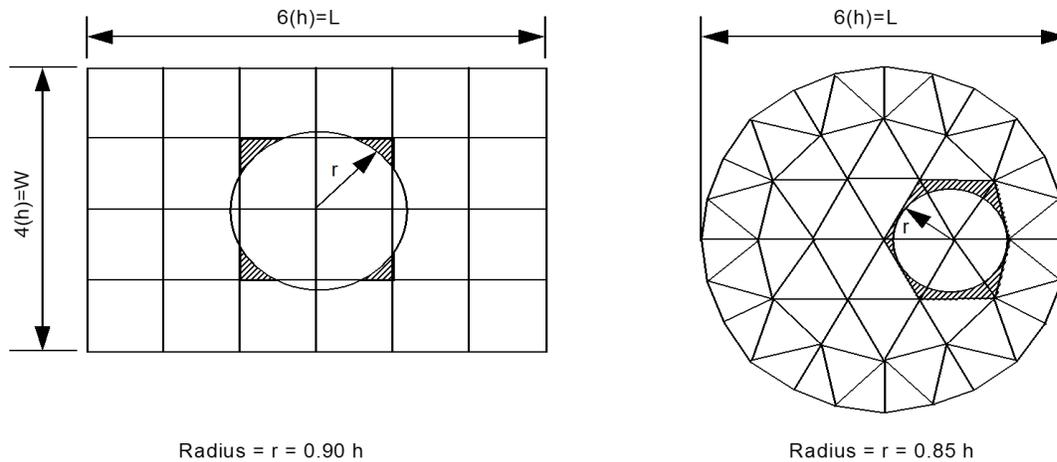
**Operation Mode** - Select the operation mode from Solution and Data Check.

**Last Layer Number in Near Field Zone** - Type the last layer number. This parameter is the maximum number of layers in the ground that the structure (including the irregular soil zone) is embedded into. The smaller this number, the less information is saved in FILE3, therefore less computation will be needed to form the flexibility matrix in the ANALYS module. However, this number must be large enough to ensure that the excavated soil region will not extend deeper than

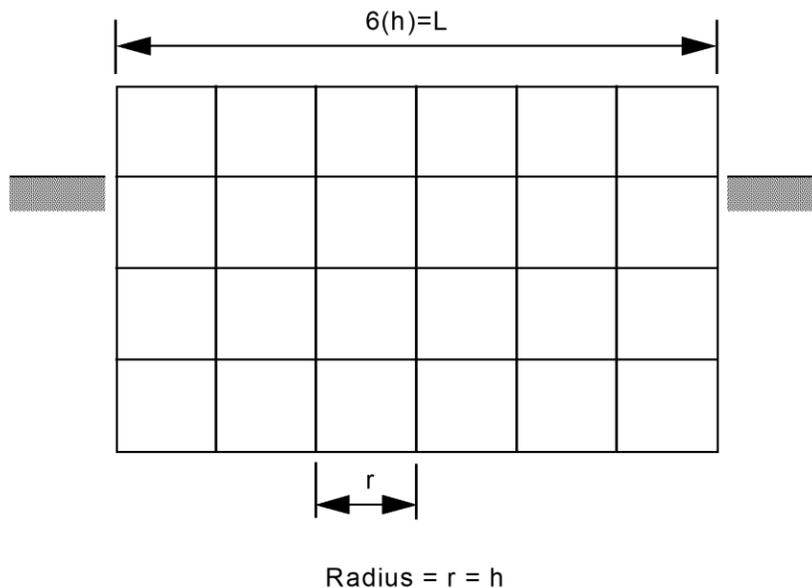
the specified layer number. For surface structures without assumed irregular soil zone, set this value to 0 or leave the edit box blank.

**Radius of Central Zone** - Type the radius of the central zone in the point load 2D or axisymmetric solution. The value must be positive and non-zero. The value of this parameter depends on the geometry of the foundation discretization by finite element method.

For two relatively uniform meshes this value is given in the following figure:



For 2D cases, the radius of the central zone shall be selected as shown above.



For nonuniform excavation meshes, an average value can be used. Uniform meshes are recommended as much as possible. For nonuniform excavation meshes, sensitivity studies are

always recommended. Transition meshes between structure and excavated soil are recommended to maintain a regular mesh for the excavation volume (Nie et., 2013)

**WARNING:** The SSI results are expected not to be relatively sensitive to changes in the assumed radius for the central zone for low and mid frequencies, as long as this radius is of the same order as the dimension of the finite elements in the interaction volume. However, for higher frequencies the radius approximation might produce larger deviations from exact solution. For nonuniform meshes, sensitivity studies are recommended for different radius values estimated based on the excavated soil mesh sizes as minimum radius, average radius and maximum radius values.

### HOUSE Module Options

The screenshot shows the 'Analysis Options' dialog box with the 'HOUSE' tab selected. The dialog is organized into several sections:

- Operation Mode:**  Solution,  Data Check
- Dimension of Analysis:**  1D,  2D,  3D
- Flexible Volume Method:**  Flexible Volume(FV),  Fast Flexible Volume(FFV),  Flexible Interface(FI)
- Acceleration of Gravity:** 32.2
- Ground Elevation:** -10
- Non-Linear SSI:** Input Data
- Optimize Model
- Wave Passage:**  Use Wave Passage, Angle Line D with X Axis: 0, Apparent Velocity for Line D: 1e+008, Unlagged Coherency Model: 3
- Motion Incoherency Simulation:**  Deterministic (Median) Incoherency Input,  Stochastically Simulated Incoherency Inupee
- Superposition Mode:**  Linear,  Quadratic
- Ansys Model Input, Ansys Model Type:  Embedded,  Surface
- Soil Motion:**  Coherent,  Incoherent
  - Coherence Parameter X Dir: 0.1
  - Coherence Parameter Y Dir: 0.1
  - Coherence Parameter Z Dir: 0.2
  - Alpha Directionality Factor: 1000
  - Number of Embedded Layers: 0
  - Time Step of Sismic Motion: 0.005
  - Nr. of Fourier Components: 4096
  - Frequency Set Number: 1
  - Number of Incoh. Modes: 0
  - Print Coherency Matrix
- Multiple Excitation:**  Use Multiple Excitation
  - Input Motion Number: 1
  - First Foundation Node: 1
  - Last Foundation Node: 69
  - X Coord. of Control Point: 0
  - Y Coord. of Control Point: 0
  - Z Coord. of Control Point: 0
- Spectral Amplification:** 1,1,1,1,1,1,1,1,1,1,1,1,1,1,1
  - Use Complex Spectral Amp.
  - Non-Uniform Motion
  - Non-Uniform Soil

Buttons: Ok, Cancel

The HOUSE module computes for the structure and excavation volume FE models the static stiffness and mass matrices. The matrix information is saved in the COOSK and COOSM binary files, while the topology information on node coordinates and element connections are saved in FILE4 (modelname.N4). For incoherent analysis, HOUSE also builds the coherency matrix and computes its eigen-solution that is further used to compute the free-field incoherent motion variations for each frequency which are save on FILE77.

Two separate FE models are constructed and included in the SSI model, one for the structure and the other for the excavated volume of soil. The interaction nodes belong to the excavated soil volume. For the FV method all excavated soil volume all nodes are interaction nodes. For the other FI and FFV methods only a part of the excavation nodes are interaction nodes.

The FE library includes the following basic elements (see GROUP):

<b>Type</b>	<b>Description</b>
SOLID	3D solid element (8 nodes, with/without incompatible modes)
BEAMS	3D beam element (3 nodes)
SHELL	3D thin plate element, Kirckhoff theory (4 nodes) *
TSHELL	3D thick plate element, Mindlin-Reissner theory (4 nodes) **
PLANE	2D plane strain solid element (4 nodes)
SPRING	3D spring element (2 nodes)
GENERAL	3D general stiffness/mass matrix element (2 or 3 nodes)

HOUSE reads the nodal point input data, nodal types, soil layer properties, and element data for the structural and excavated soil elements, then forms the element mass and stiffness matrices for these elements which are later assembled into corresponding the mass and stiffness matrices. These matrices are stored in compact format in preparation for the SSI solution.

**REMARKS:**

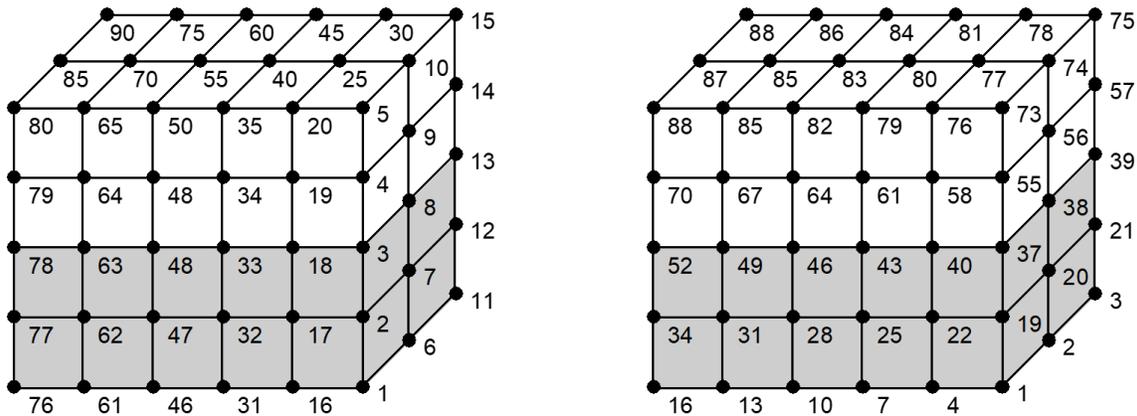
1) For the thin plate SHELL elements, it is always recommended to use FIXROT or FIXSHLROT commands for fixing the singularity for the drilling rotation dof by introducing a small rotational stiffness around the normal to the plate. This is not necessary for the newer thick TSHELL element, since its done automatically by the HOUSE module program.

2) For the SOLID elements incompatible modes could be included using UI menu selection. This is not applicable to excavated soil SOLID elements. For thick TSHELL elements, the user can select using either Reduced or Selective Gauss integration at element level using the EINT command.

3) The excavated soil volume is modeled using the 3D SOLID or 2D PLANE element types. The FE models of the structure and the excavated soil must be selected in such a way that every interaction node below the ground should lie on a soil layer interface.

4) The suggested node numbering should start with the nodes at the foundation level and then continue layer by layer up to the surface level. The figure below shows examples of node numbering. Thus, recommend the bottom-up node numbering as a standard convention.

**WARNING:** It should be noted that interaction nodes should be defined always in ascending order from deeper excavation level to ground surface level.



4) For element numbering, there is no special restriction in the element numbering for structures with no embedment. The element numbering has to be in continuous sequence and optimal in sense of stiffness matrix bandwidth. However, for structures with embedment, the excavation volume element numbering has to be based on the top-bottom element numbering scheme, from the ground surface to baserock, so that the far-field soil layers associated to the excavation elements are assigned in an increasing order.

**WARNING:** We recommend that excavation volume is modeled by a set of groups of elements that each contain a horizontal layer of elements that corresponds to a single far-field soil layer. Thus, the number of excavation volume groups should be equal to the number of embedment layers. The excavation groups should be numbered from top (surface) to the bottom of excavation volume (foundation level). Each excavation group soil layer will be defined using the L command with a layer number identical with the far-field soil layer associated to the excavation element group. This recommendation becomes a strict rule if the stress contour plots are required.

**WARNING:** We always recommend to the user to check his HOUSE FE model input by reviewing carefully the HOUSE output file.

The Demo 5 problem includes an illustrative SSI analysis input example for modeling for an embedded structure (please see carefully see the Demo5.pre file).

---

**WARNING:** *When building an embedded SSI model, the user should always check that the SOLID elements in the excavation volume have the correct soil layer number assignments in the HOUSE output file before the SSI analysis is run.*

It should be noted that the HOUSE modules, HOUSEFS and HOUSEFSA, include a node numbering optimization capability that improves the computational speed of the embedded structure SSI analysis and also reduces significantly the required RAM. This node numbering optimization is optional for HOUSEFS and can be handled through the UI Options/Analysis/HOUSE window (or directly by typing a 1 in the 1<sup>st</sup> line-1<sup>st</sup> column the HOUSE .hou input text file generated by the AFWRITE command). The Problem 31 in the Verification Manual describes and validates the use of the HOUSEFS optimizer. The HOUSE node numbering optimizer is automatically used in HOUSEFSA for the Option AA.

**WARNING:** *The node numbering optimizer is highly effective for large-size SSI models with significant embedment. We strongly recommend the use of the node numbering optimizer for any larger-size SSI model with embedment.*

**WARNING:** *If the automatic node numbering renumbering is used, then, the correct node numbering is provided in the renumbered model input file modelname.hownew. The mapping between old node numbers and new node numbers are provided in the modelname.map text file. The user shall use the new node numbering from .hownew file to select the post-processing node output requests for the MOTION and RELDISP modules. The STRESS output element results are also reported based on the new node numbering.*

The following options allow you to specify the analysis options for HOUSE module:

**Operation Mode** - Select the operation mode from Solution and Data Check.

**Dimension of Analysis** - Select the dimension of analysis: 2D or 3D (1D not available). If 2D is selected instead of 3D, the POINT2 module should be used instead of POINT3.

**Flexible Volume Methods** - Select either the Flexible Volume (FV) method, the Flexible Interface (FI) method (as FI-EVBN and FI-FSIN) or the Fast-Flexible (FFV) method. See Section 1.5.1 and 3.1.2 for additional SSI modeling details using the FV, FI and FFV methods. For surface foundation these methods are identical since there is no excavation volume defined and interaction nodes are placed only at the ground surface level. The interaction nodes can be automatically selected for each of these methods using the INTGEN command (the previous INT command still working was much simpler).

**WARNING:** *If other methods than FV are used, such as the FI and FFV methods, then, sensitivity studies for comparison against the FV method are required. These sensitivity studies are required per the new ASCE 04-2016 standard and the most recent USNRC SRP 3.7.2 requirements.*

---

**WARNING:** *It should be noted that the interaction nodes are related only to the nodes that belong to the excavated soil system. Structural nodes are not interaction nodes, except when the structure basement and the excavated soil models share common nodes on the excavation volume lateral and bottom surfaces.*

**Acceleration of Gravity** - Type the acceleration of gravity. The value is the same as set in the Analysis Options - SITE dialog box.

**Ground Elevation** - Type the ground surface elevation. This value is also used to determine whether an SOLID/PLANE element belongs to the structure or to the excavated soil (see *ETYPE* instruction 9.4.11). For SOLID/PLANE elements with *ETYPE*, 0 (default value) situated below ground surface, the UI module assumes that they are excavation volume elements. For SOLID/PLANE elements with *ETYPE*, 2, the UI assumes that they are also excavation volume elements and interprets their material assignments by the MSET command as the free-field soil layer assignments. The *ETYPE*,1 command is only for the element groups that belong to the structure FE model, that can also include a near-field backfill soil part.

**Input Data** - Click this button to enter data for the pile interface (PINT module is not available).

**Soil Motion** - Select the soil motion type that can be either Coherent or Incoherent.

The seismic motion incoherence option applies only to three-dimensional SSI models with no axis of symmetry, i.e. only for full models (not applicable for 2D models or half-models). It is required that the interaction node numbering to start from the bottom layer at baserock up to the ground surface.

The ACS SASSI code includes six incoherent SSI approaches, namely, five deterministic approaches and a stochastic approach. The deterministic approaches called AS/Linear and SRSS/Quadratic approaches were benchmarked in the 2007 EPRI studies in conjunction with the SRSS zero-phasing or AS phase adjustment assumptions (zeroing the phasing for the superposition of the incoherent modes in AS, or the ATF modal complex responses in SRSS). The stochastic simulation approach was used in the 2007 EPRI studies, as the reference approach called Simulation Mean approach (please also see 1.5 sections and 2 for details).

The incoherent SSI analysis includes different coherency model options:

- 1) Luco-Wong model (Luco and Wong, 1986), that is a physics-based, anisotropic parametric model with different coherence parameters for X, Y and Z motion components (not validated in practice by real records - see input below), and
- 2) Several Abrahamson models, as 2005 model for all sites and surface foundations, and 2006 model for all sites and embedded foundations (Abrahamson, 2005, 2006), and

---

2007 model for rock-sites for surface and embedded foundations, and 2007 model for soil-sites and surface foundations (Abrahamson, 2007)

- 3) User-defined plane-wave incoherency models defined for three-orthogonal X, Y and Z directions

The Abrahamson plane-wave incoherency models that are empirical-based models developed based on many earthquake records on different soil conditions, with no input parameters. Luco-Wong model can be applied with or without wave passage option checked.

**WARNING:** *The Abrahamson and User-defined coherency models are applied only when the wave passage selection is checked.*

The user defined model requires that the user defines the coherent functions for the X, Y and Z directions in separate text files called COHXUSER, COHYUSER and COHZUSER with no extensions. The user needs also to define the SSI frequency file, FREQCOH, and the relative distance file, DISTCOH.

**WARNING:** *It should be noted that the matrix size of 100 x 100 is the default dimension of the user defined coherent function matrices. However, the size of the coherent function matrix sizes could be smaller, if the sizes are defined by their integer values in the .hou file, as illustrated in the V&V Problem 39 of the NQA version where the sizes of the coherent function matrices were 61 frequencies x 81 distances. However, at this time the UI module does not permit changes in the sizes of the user defined coherent function matrices.*

**Coherence Parameter X Direction** (Horizontal Component, X) - Type the coherence parameter value for the Luco-Wong model. This parameter value lies usually between 0.10 and 0.30. Higher values can be used to determine upper-bounds of the incoherence motion effects on SSI response. The incoherent motion field can be isotropic or anisotropic for horizontal components.

**Coherence Parameter Y Direction** (Horizontal Component, Y) - Type the coherence parameter for the other horizontal motion component.

**Coherence Parameter Z Direction** (Vertical Component, Z) - Type the coherence parameter for vertical motion component.

**Alpha Directionality Factor** – This factor is 0.50 for isotropic or radial coherency models (with circular shape in horizontal plane), and different than 0.50 for anisotropic or directional coherency models (with ellipse shape in horizontal plane). For example, 0.1 indicates that the coherency ellipse has a three times shorter radius or decays three time faster in the X direction than in the Y direction. In contrast, a value of 0.90 indicates that the coherency ellipse has a three times shorter radius or decays three time faster in the Y direction than in the X direction.

The alpha directionality factor,  $\alpha$ , has values between 0 and 1. The alpha parameter is used to compute the relative distances D between interaction nodes based on the homotopic relationship  $D = (2(\alpha DX^2 + (1 - \alpha)DY^2))^{1/2}$  where DX and DY are the directional distances. The value of alpha = 0.5 that is in the center of the [0. 1.] interval should be used for radial, non-directional, or isotropic coherency models. The lower and upper values of alpha such as 0.1 or 0.9 correspond to directional, or anisotropic coherency models in which the distances in the Y-direction are weighted 3 times more than the distances in the X-direction, or vice versa, respectively.

**WARNING:** *It should be noted that in the 2007 EPRI validation studies (EPRI TR 1015111) used the 2005 Abrahamson isotropic or radial plane-wave coherency model. The 2007 Abrahamson coherency models for rock and soil sites were not available at the time of the EPRI studies (EPRI TR 1015110).*

**WARNING:** *For Luco-Wong model, the Alpha Directionality Factor has a different meaning than for the other models. It is the mean soil shear wave velocity parameter that is required for the Luco-Wong model.*

It should be noted that the principal axes of the coherency ellipse in the horizontal plane could be rotated, to become parallel to an arbitrary, user-selected direction that could be also selected as the wave passage direction – see below “Angle of Line D with X-axis”.

**Number of Embedment Layers** - Type the number of embedment layers defined by the set of interaction nodes. If the flexible interface method FI-EVBN or FI-FSIN is used then the internal nodes should be not defined as interaction nodes. If flexible volume is used then the interaction nodes are all the nodes that define the excavation volume.

**Time Step of Seismic Motion** - Type the time step of the control motion. The value is the same as set in the Analysis Options - SITE dialog box.

**Nr. of Fourier Components** - Type the number of values to be used in the Fourier transform. The value is the same as set in the Analysis Options - SITE dialog box.

**Frequency Set Number** - Type the number of the frequency set. The value is the same as set in the Analysis Options - SITE dialog box.

**Number of Incoherency Modes** - Type the number of incoherent spatial modes to be used in the spectral factorization of the coherence matrix. If a zero value is input, the code assumes that all incoherency modes are used; if a positive number n is input, the code assumes that only n modes are used; and if a negative number -n is used, then, the code assumes that only the mode n is used. The negative input is used in conjunction with the Quadratic (SRSS) deterministic approach that requires a separate SSI analysis for each incoherency modes. For stochastic simulation and the linear (AS) deterministic approach, we recommend to use all the incoherency

modes that is the default option. Including all modes offers the best accuracy without having any non-negligible impact on the incoherent SSI runtime. If Quadratic (SRSS TF) approach is used, additional information is required by the MOTION module (that is incorporated in the SRSSTF.TXT file).

**Incoherent Mode Contributions** – If this is checked, then, all the incoherent mode percent contributions to the reconstruction of the free-field coherency matrix will be printed in the HOUSE module output. This option is provided for checking the numerical accuracy of the random field decomposition and evaluate the incoherent spatial mode contributions.

To check the eigen-expansion convergence criterion, in ACS SASSI we used as a norm the trace of the transformed matrix  $\Lambda$  that is equal to the trace of the original  $\Sigma$  and equal to  $N$  (the number of all interaction nodes). This happens since the diagonal elements of the two matrices have all a value equal to 1 (for the same location that means for a zero distance between two locations, the coherent function has value 1 for any node location). Thus, for a given frequency if all incoherent eigenvectors (modes) are considered, then, the following equality shall hold

$$\sum_{j=1}^N \lambda_j = N \quad (6.1)$$

Please note that the eigenvalue  $\lambda_j$  correspond to the variance associated to the mode shape  $j$ , while  $N$  corresponds to the total variance of the incoherent amplitude variation field at given frequency. If a reduced number of modes are used, say  $m < N$ , then, the total variance of the incoherent amplitude variation field will be not recovered since

$$\sum_{j=1}^m \lambda_j < N \quad (6.2)$$

For each frequency, we define the percent contribution of incoherent mode  $j$  to the total incoherent motion field variance by

$$v_j = \frac{\lambda_j}{N} 100 \quad (6.3)$$

For  $m$  incoherent modes that are ordered with increasing contributions, the cumulative percent contribution to the total variance of the incoherent field variance is calculated in HOUSE by

$$\sum_{j=1}^m v_j = \sum_{j=1}^m \frac{\lambda_j}{N} 100 \quad (6.4)$$

If the user uses Stochastic Simulation (SS) approach, then all incoherent modes (equal to the number interaction nodes for each direction) are considered. Basically, SS is a Monte Carlo simulation of the incoherent wave field in horizontal plane. For each wave field simulation a SSI analysis is performed for modified free-field seismic load vectors. Recently, we included in the most recent ACS SASSI implementations after IKTR4 the capability of running tens of incoherent samples in all three X, Y and Z directions in a single SSI analysis run with no restart needed. This reduces drastically the runtime for performing the incoherent SSI analysis, for example, for 20 simulations to a fraction of the initiation SSI analysis runtime (about 15-20 times faster than the previous 2015 ACS SASSI IKTR4).

For the SRSS TF the above equation 5.4 is similar to the cumulative modal mass contribution used in structural dynamics. For structural dynamics problems, a 90% cumulative mass contribution criterion (based on the sum of squared modal participation factors) is acceptable. We consider that the same 90% cumulative modal contribution criterion (based on the sum of modal mean-square amplitudes or variances) could be used in practice as a convergence criterion for how many incoherent modes should be considered at each frequency. Such criterion is independent of the building foundation flexibility, and therefore, applicable to all types of structures and foundations. For extremely rigid foundations, in the horizontal plane, this free-field coherency matrix convergence criterion might be conservative.

**WARNING:** *Using the incoherent mode contribution useful accuracy checking of the of the coherence matrix eigen-decomposition implemented in ACS SASSI is to verify that the equation 5.1 holds for each frequency. If not, then, the coherency matrix is numerically ill-conditioned, most likely due to the space distribution of the interaction nodes.*

**WARNING:** *For flexible foundations, the number of required incoherent spatial modes need much larger than for rigid foundations, in order of tens or even hundreds on modes. This makes the SRSS approach impractical for flexible foundation problems. The SRSS TF and FRS approaches were implemented in ACS SASSI only for benchmarking purposes for stick models with rigid basemats, rather than for practicality for realistic problems. The SRSS approaches may not include all the “artifacts” inherent to these approaches as implemented in the SASSI2010 and EPRI INCOH codes, respectively.*

**Wave Passage** - Select this button for activating the wave passage option. This selection is required for Abrahamson and user-defined coherency models and nonuniform/multiple excitation option.

**Apparent Velocity for Line D** - Type the apparent velocity for line D (if wave passage option is enabled). Use a large value for apparent speed for traveling waves, say 1.0e9, to make negligible the wave passage effects, if this is desired (as in the 2007 EPRI studies for rock sites).

**Angle of Line D with X Axis** - Type the angle of horizontal line D with the x axis (if wave passage option is enabled). This Line D will be also used for directional Abrahamson or user-defined plane-wave coherency models.

**Unlagged Incoherency Model** - Seven plane-wave coherency models can be selected:

Type 1 - 1986 Luco-Wong model

Type 2 - 1993 Abrahamson model for all soil types (similar with 2005)

Type 3 - 2005 Abrahamson model for all sites and surface foundations

Type 4 - 2006 Abrahamson model for all sites and embedded foundations

Type 5 - 2007 Abrahamson model for hard-rock sites and surface/embedded foundations

Type 6 - 2007 Abrahamson model for soil sites and surface foundations.

Type 7 – User defined coherent functions for X, Y and Z directions

It should be noted that Models # 2-7 can be rotated using the wave passage direction to define the principal axes of motion through Line D. For the case of directional coherency models, the alpha parameter can be also different than 0.50, say 0.1 or 0.9. For the radial coherency models, alpha is 0.5.

If the option for the user defined coherent functions is used, then, the user is required to define the values of the frequency point and distance point vectors in FREQCOH and DISTCOH text files and the coherent functions for the X, Y and Z directions in the COHXUSER, COHYUSER and COHZUSER text files. The user defined coherent function matrices shall have a size of 100 x 100. This implies that user shall define the coherent function variations for 100 frequencies and 100 relative distances in FREQCOH and DISTCOH files. The selected frequencies and relative distances should cover the frequency range of interest and the foundation size of the SSI model to be used for the incoherent analysis.

**WARNING:** *It should be noted that the matrix size of 100 x 100 is the default dimension of the user defined coherent function matrices. However, the size of the coherent function matrix sizes could be smaller, if the sizes are defined by their integer values in the .hou file, as illustrated in the V&V Problem 39 of the NQA version where the sizes of the coherent function matrices were 61 frequencies x 81 distances. However, at this time the UI does not permit changes in the sizes of the user defined coherent function matrices that are assumed to be 100 x 100.*

**WARNING:** *The 2007 Abrahamson model for soil sites (Model 6) is applicable to the surface foundations or shallowly embedded. For the deeply embedded foundations in soils, as SMRs, the coherency model input at the foundation level should be different (with a decay), than that defined at the surface. Therefore, for deeply embedded structures in soil deposits, a multilevel incoherent SSI analysis is required. This can be done if the HOUSE eigen-solution for coherency matrix is run separately per level of interaction node layers, in which case only the interaction nodes at the selected node layer level are defined. After getting all the FILE77s for all embedment levels from*

---

*separate HOUSE incoherent runs, a final HOUSE coherent run with all interaction nodes is required to construct correctly the FILE4 file to be used by ANALYS, Then, the BuildFILE77.exe DOS auxiliary program should be used to combine the FILE77s of all embedment node layers in a single FILE77 that includes the incoherent eigen-solution information for all interaction nodes. However, BuildFILE77 will provide accurate results ONLY if the incoherent modes at different levels maintain the same node numbering pattern, so that the incoherent eigen-modes are not flipping their signs randomly at different levels. This may imply a significant checking effort for each mode at each level. Additional tech support is highly recommended for such special, complex situations for which additional checking efforts are required.*

**Motion Incoherency Simulation** - This option is used for simulating the seismic motion incoherent field at the interaction nodes. The user has the options to use either **Deterministic (Mean) Input** assuming zero phase angles between different motion spatial wavelength components, or **Stochastically Simulated Input** assuming random phase angles in the interval [-180, 180] degrees. If the user selects a pair of a random, arbitrary SEED numbers for the Horizontal and the Vertical components, and a 180 degree angle for the Random Phase angle for different wavelength components, the Number of Simulations, then, a single stochastic motion incoherency field or many fields are simulated. If the user selects zero values for the two SEED numbers and the Phase Angle, then, a deterministic (median) motion incoherency field is generated and used in SSI analysis. The number of simultaneous simulations should be no larger than 50. These simulations are executed in a single HOUSE run.

**Stochastic approach** is based simulating random incoherent motion realizations (Simulation Mean in EPRI studies). Using stochastic simulation algorithms, a set of random incoherent motion samples is generated at each foundation SSI interaction nodes. For each incoherent motion random sample an incoherent SSI analysis is performed. The final mean incoherent SSI response is obtained by statistical averaging of SSI response random samples.

**Deterministic approach** approximates the mean incoherent SSI response using simple superposition rules of the incoherent modal effects, such as the algebraic sum (AS) (AS in EPRI studies) and the square-root of the sum of square (SRSS) (SRSS in EPRI studies). In HOUSE, four deterministic approaches could considered: i) and ii) Linear superposition, or Algebraic Sum of the scaled incoherent spatial modes (AS in EPRI studies) with or without ATF phase adjustment (only the approach with phase adjustment used in EPRI studies), iii) and iv) Quadratic superposition of the incoherent modal SSI complex response ATF amplitudes (acceleration transfer function amplitudes) assuming a zero-phase for the incoherent SSI complex response phase (used in EPRI studies and called herein SRSS TF) and quadratic superposition of the incoherent modal SSI complex response ATF amplitudes assuming a non-zero phase for the incoherent SSI complex response that is equal to coherent SSI complex response phase (not used in EPRI studies). The SRSS phase option for the SRSS TF approach is selected inside the

---

STRSSTF.txt input file for the MOTION module by a flag variable that can be 0 (zero-phase) or 1 (non-zero phase).

The fifth deterministic approach that is implemented in ACS SASSI is an alternate version of the SRSS approach that does not neglect the complex response phase. This deterministic approach is called herein the SRSS FRS and is applied to the SSI end response quantity that could be either ATF, ISRS or ZPA. To apply the SRSS FRS approach the user has to select deterministic “Linear” option (instead of Quadratic option) in the HOUSE input window and, then, run MOTION repeatedly for each selected incoherent mode. The user will need to SRSS the SSI response end results of interest provided MOTION (or STRESS).

**WARNING:** *The SRSS FRS approach was not validated in the 2007 EPRI studies (Short et al., 2007) and also not endorsed by USNRC in the ISG-01 position document. It should be noted that the way the incoherent mode contributions are computed depart considerably from the other 2007 EPRI validated approaches. The SRSS FRS approach was originally implemented in the 1997 EPRI INCOH code. It should be noted that the SRSS FRS implementation in ACS SASSI follows the same way of computing mode contributions as for the other approaches in the 2007 EPRI studies. Thus, the SRSS FRS approach in ACS SASSI has a different implementation than in original 1997 EPRI INCOH. The SRSS FRS approach is not recommended for any real nuclear project, but only for benchmark comparisons and/or research projects.*

For *rigid foundations* the incoherency-induced stochasticity of the basemat motion is driven by the global or rigid body spatial variations (integral variations) of free-field motion and, therefore, is less complex and random than free-field motion. The rigid foundation motion has a linearized spatial variation pattern. Thus, the differential free-field motions are highly constrained by the rigid basemat, and because of this, the rigid foundation motion complexity is highly reduced in comparison with the complexity of an elastic foundations that is soil motion spatial wavy variations. The rigid foundation assumption creates an unrealistically large kinematic SSI interaction effects that overestimates the effects of incoherency.

For *flexible foundations*, the incoherency-induced stochasticity of the basemat motion is driven by the local spatial variations of free-field motion. The flexible foundation motion has a less smoothed spatial variation pattern since kinematic SSI is reduced. Thus, the differential free-field motions are less constrained by the basemat, and because of this, the (flexible) foundation motion complexity is similar to the complexity of the local motion spatial variations.

Based on a number of investigations for incoherent SSI analyses for different nuclear island (NI) designs and sizes, we suggest that the deterministic SSI approaches should be limited to rigid foundation applications, as demonstrated in the 2007 EPRI studies (Short et al., 2007, Ghiocel, 2013b). For flexible foundations, we recommend the stochastic simulation approach since it accurately captures the statistical nature of the free-field motion spatial variations and include correctly all incoherent mode contributions and the mode random combination pattern.

---

**WARNING:** *The 2007 EPRI validated approaches based on zeroing the incoherent mode phasings are not capable of capturing in all detail the differential phasing of the free-field motions at the interaction nodes. Based on a number of case study investigations (Ghiocel 2013a, 2014b, 2015b), it appears that the 2007 EPRI validated approaches when applied to typical nuclear island elastic foundation models produce overly conservative incoherent ISRS results that are closer to coherent ISRS in the mid frequency range, and potentially unconservative incoherent ISRS results in the high frequency range if only a limited number of incoherent modes is included. For the vertical direction, even a number of 40-50 incoherent modes for the SRSS approaches might not be sufficient for an accurate result convergence, depending on the foundation stiffness*

For the elastic foundation SSI models, the accuracy of the SRSS approaches depends significantly on the number of incoherent modes that is used (Ghiocel, 2013b, 2014a, 2015b). It should be noted that the SRSS approaches requires a separate SSI restart analysis for each incoherent mode that makes the SRSS approaches highly impractical for application to SSI models with elastic basemats under high-frequency incoherent inputs.

Consideration of all incoherent spatial modes as in the stochastic simulation approach, improves the incoherent SSI accuracy and produces an “exact” recovery of the free-field coherency matrix at the interaction nodes.

**WARNING:** *Before using the SRSS approach, a preliminary investigation to establish the required number of incoherent modes is recommended. The HOUSE output shows the cumulative modal mass/variance contributions at all SSI frequencies – search for the string “I N C O”. in the HOUSE output. For a limited number of incoherent modes, the cumulative mode contributions could be much less than 100%. We suggest to consider a number of incoherent modes that satisfies the 90% mode contribution criterion that is similar to the cumulative modal mass criterion used in structural dynamics.*

**Use Multiple Excitation** - Select this button for activating the multiple excitation or the nonuniform input option. To use this option the wave passage option should be also selected. This option is applicable to a single continuous foundation or multiple foundations. The nonuniform amplitude seismic input is introduced by a variable motion Fourier amplitude in the free-field at different locations under foundation. The multiple excitation or nonuniform input motion option could be applied in conjunction with motion incoherency and wave passage for creating a more realistic, randomly spatially varying seismic ground motion environment.

**WARNING:** *For the nonuniform input case with a single foundation, the foundation should be partitioned in up to 5000 zones. Please note that the node numbering for each foundation should be in a continuous sequence. For the moment this can be a highly inconvenient restriction for practice.*

---

**Input Motion Number** - Select the number of the active input motion. All following data refers to this motion.

**First Foundation Node** - Type the number of the first interaction node for the selected isolated foundation.

**Last Foundation Node** - Type the number of the last foundation node for the selected foundation. These foundation nodes have to be defined in a sequential node order with unit increment. Thus, it is required that the interaction nodes to be defined in a sequential order for each foundation, so that ranges of node numbers can be defined for each isolated foundation, i.e. do not skip node numbers from one foundation to another foundation.

**WARNING:** *The foundation node numbering has to be defined in a continuous sequence for each foundation to apply the multiple excitation input analysis option which can be inconvenient in practice. The requirement with the bottom-up numbering for interaction nodes still remain in effect.*

**X Coord. of Control Point** - The X-coordinate is used to define the input motion location application (not used).

**Y Coord. of Control Point** - The Y-coordinate is used to define the input motion location application (not used)

**Z Coord. of Control Point** - The Z-coordinate is used to define the input motion location application (not used)

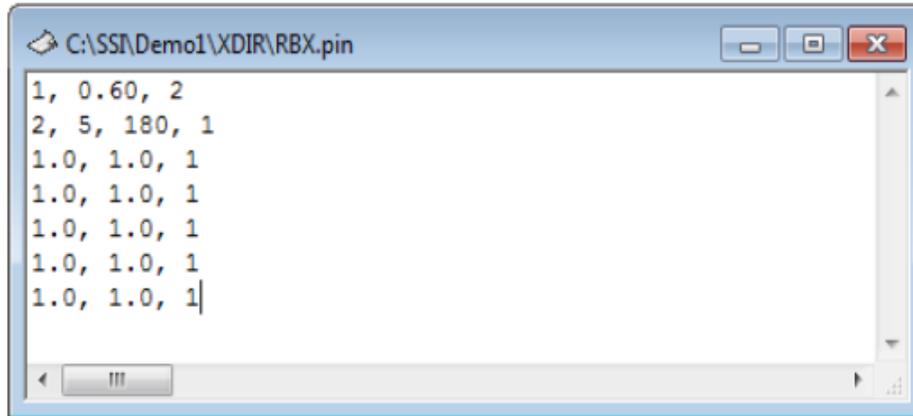
**Spectral Amplification Ratios (SAR)** - Type the spectral amplification ratios applicable to complex transfer functions of the foundation motion at different support points. These SAR have real values complex values.

The SAR define the ratios (at each frequency solution points) between the complex ATF of the “local” motion for each isolated foundation or zone with respect to the “reference” motion computed by SSI analysis for the single input control motion. The number of ratios must be equal to the number of SSI frequencies. Use blank or coma as separators. The phasing effects due to the differential motions can be included by using complex SAR.

An example of the use of the multiple seismic input excitation option is shown in the V&V Problem 35 or the Verification Manual of the NQA software version.

**WARNING:** *Usually the complex amplitude amplification SAF could be different for X, Y and Z inputs depending on the soil layering configuration below foundation. Thus, in these cases the HOUSE input files (.hou files) for the X, Y and Z inputs should be different since they include different SAF values. The SSI analysis can be run separately for each direction using HOUSE and ANALYS (restart) runs for each direction.*

**Nonlinear SSI Analysis Input** - If the nonlinear SSI analysis is used, then the user needs to click on the Nonlinear SSI Input Data to define the input for the initial soil properties for the near-field soil element groups. By clicking the Nonlinear SSI Input Data a new input file is opened for editing. This file has extension .pin. The user needs to input in a free-format the following information (see example shown below in the figure):



```

C:\SSA\Demo1\XDIR\RBX.pin
1, 0.60, 2
2, 5, 180, 1
1.0, 1.0, 1
1.0, 1.0, 1
1.0, 1.0, 1
1.0, 1.0, 1
1.0, 1.0, 1
1.0, 1.0, 1

```

1<sup>st</sup> line, 3 items: Number of nonlinear soil element groups, effective strain factor, number of soil material curves defined in SOIL (soil constitutive model);

2<sup>nd</sup> line, 4 items: Number of the nonlinear soil element group, number of materials (could be equal with the number of layers or not) in the group, number of solid elements in the group and the flag for effective shear calculation. This effective shear strain calculation flag input is:

0 – for maximum component shear strain value between X, Y and Z

1 – for octahedral shear strain for 3D SSI models or maximum shear strain in 2D SSI models

**WARNING:** An auxiliary DOS batch program called *COMBIN\_XYZ\_STRAIN* is available to combine the effective shear strain computed for X, Y and Z inputs for each SSI iteration, so that that nonlinear soil behavior is computed based on the three-directional seismic input, not separate for each unidirectional input. The aux program uses the SRSS rule to determine the overall effective shear strain based on the computed shear strains for the X, Y and Z inputs. The SRSS calculations are performed based on the directional FILE74 text files computed for each input direction, X, Y and Z.

3<sup>rd</sup> line, 3 items, within a loop over the number of soil materials, with each line including: Initial shear modulus reduction factor (1.00 indicates same shear modulus as in free-field), initial damping ratio factor (1.00 indicates the same damping as in free-field), soil material curve order number (defined in pairs for shear modulus and damping from the FILE73 file). The block of lines after 1st line, needs to be input for all nonlinear soil element groups. The figure below is shown

---

an example with a single group of nonlinear soil elements, an effective strain factor of 0.60 and 2 soil material curves. The order number of the nonlinear soil group is 2, the number of soil materials in the group is 5, and total number of elements in the group is 180. The last number that is equal to 1 is the shear strain calculation option that is based on i) octahedral shear-strain for 3D SSI models, or ii) maximum shear strain for 2D SSI models.

**WARNING:** *This nonlinear soil capability could be extended to any nonlinear material modeled by solid elements (massive concrete blocks, rubber material, etc.) not for soil material only. In this case the user has to add a new material pair of constitutive curves, G-gamma and D-gamma, in the text file FILE73 produced by the SOIL module, or define a new material in the SOIL module. The STRESS module will use these new curves for performing the directional nonlinear analysis using the iterative equivalent linear procedure. The COMBIN\_XYZ\_STRAIN.exe aux program should be run in batch mode for each SSI iteration after each the directional STRESS runs for X, Y and Z inputs are completed.*

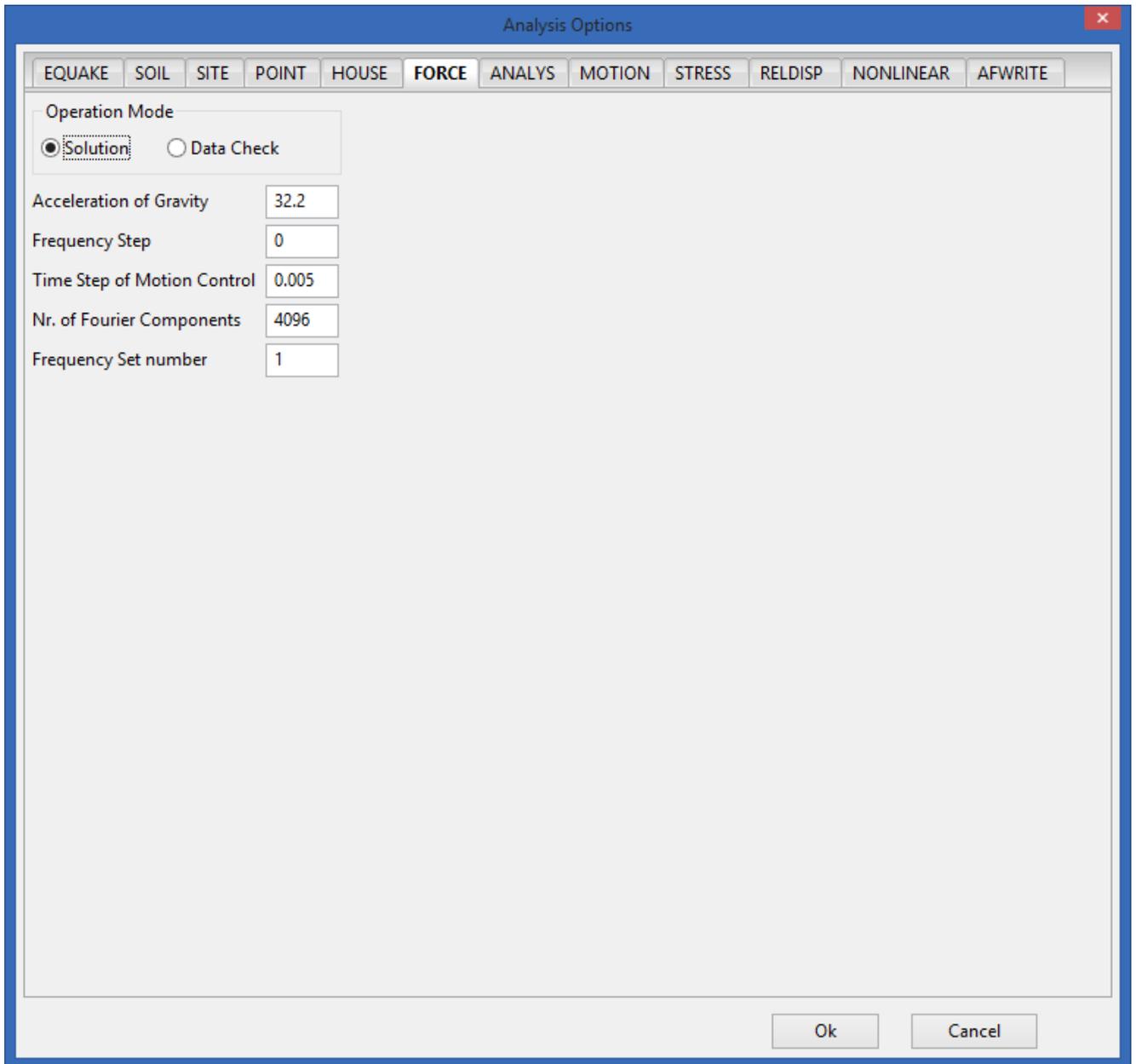
**Optimize Model** - Check box to enable the node numbering optimization.

**WARNING:** *If the automatic node numbering renumbering is used, then, the correct node numbering is provided in the renumbered model input file modelname.hownew. The mapping between old node numbers and new node numbers are provided in the modelname.map text file. The user shall use the new node numbering from .hownew file to select the post-processing node output requests for the MOTION and RELDISP modules. The STRESS output element results are also reported based on the new node numbering.*

**ANSYS Model Input** - Check box if the ANSYS structural model is directly used for the SSI analysis as described in Option AA (Advanced ANSYS). Option AA is available only for the fast-solver version. The user needs to further select if the **ANSYS Model Type** that will be used for SSI analysis in ACS SASSI is a surface or an embedded model. The user can also use the ANSYSMODELTYPE command to set the input for Option AA. We recommend use the UI selection to avoid input mistakes.

**WARNING:** *If the Option AA is used, the user is required to copy all the ANSYS model related files in the SSI working directory as described in detail in the “ACS SASSI-ANSYS Integration Capability” User Manual.*

**WARNING:** *If the Option AA is used, the .hou file obtained using the AFWRITE command is fake and do not include the real materials and constants. The .hou file is used only to topological information and element connection informations. IT IS NOT A CORRECT RUNNABLE FILE AS A REGULAR .hou INPUT FILE USING HOUSE WITHOUT OPTION AA!*

**FORCE Module Options**

For each specified frequency, the FORCE module forms the elements of the load vector which correspond to external forces such as impact and rotating machinery acting directly on the structure. The results are stored in FILE9.

The following options allow you to specify the analysis options for FORCE module:

**Operation Mode** - Select the operation mode from Solution and Data Check.

**Acceleration of Gravity** - Type the acceleration of gravity. The value is the same as set in the Analysis Options - SITE dialog box.

**Frequency Step** - Type the frequency step (Hz). The value is the same as set in the Analysis Options - SITE dialog box.

**Time Step of Control Motion** - Type the time step of control motion (sec). The value is the same as set in the Analysis Options - SITE dialog box.

**Nr. of Fourier Components** - Type the number of values to be used in the Fourier transform. The value is the same as set in the Analysis Options - SITE dialog box.

**Frequency Set Number** - Type the number of the frequency set. The value is the same as set in the Analysis Options - SITE dialog box.

All forces and moments defined within the UI modules (see F, MM, FREAD, MREAD section 9.5) will be written to the input file for the FORCE module.

Dynamic loads with similar time history but different maximum amplitude and arrival time (time lag that corresponds to complex force phasing) may be applied at nodal points. Reference time history is defined as having maximum reference amplitude and zero arrival time ( $t_0$ ), which means the load starts acting on the nodal point at time zero. All other time histories must be given relative to reference time history by defining load factor and arrival time. The load factor is defined as the ratio between the maximum amplitude of the applied load and the maximum reference amplitude.

**ANALYS Module Options**

Analysis Options

EQUAKE SOIL SITE POINT HOUSE FORCE **ANALYS** MOTION STRESS RELDISP NONLINEAR AFWRITE

Operation Mode  
 Solution  Data Check

Type of Analysis  
 Seismic  
 Foundation Vibration

Mode Of Analysis  
 Initiation  
 New Structure  
 New Seismic Environment  
 New Dynamic Loading

Frequency Numbers  
 Take Frequency Numbers from File1 / File9  
 Frequency Set Number

Control Motion Foundation Reference Point  
 X-Coordinate of Control Point   
 Y-Coordinate of Control Point   
 Z-Coordinate of Control Point   
 Coordinate Transformation Angle

Coherent  Incoherent

Wave Passage Effects Included

Free-Field Load  Free-Field Motion

Multiple Excitation  
 Use Multiple Excitation  
 Input Motion Number   
 First Foundation Node   
 Last Foundation Node   
 X Coord. of Control Point   
 Y Coord. of Control Point   
 Z Coord. of Control Point

Global Impedance Calculations  
 No Impedance Calculations  
 Only Decoupled (Diagonal) Impedances  
 Full Rigid Body Impedance Matrix 6X6

Simultaneous Cases   
 Save Restart Files  Delete Restart Files  
 Print Amplitude Only

Ok Cancel

The ANALYS module is the heart of the ACS SASSI code. This module computes soil layering flexibility and impedance matrices, assembles all SSI system matrices and then, solves the SSI problem to get the complex amplitude responses.

The ANALYS module also controls the SSI analysis restart modes. ANALYS has four different running modes - the first mode is the initiation mode and the other three are the restart modes.

The following options allow you to specify the analysis options for ANALYS module:

**Operation Mode** - Select the operation mode from Solution and Data Check.

---

**Type of Analysis** - Select the type of analysis from Seismic and Foundation Vibration.

**Mode of Analysis** - Select the mode of analysis from Initiation (Mode 1), New Structure (Mode2), New Seismic Environment (Mode3), and New Dynamic Loading (Mode 3).

**Initiation** (Mode 1): This is the first mode to be executed for a new problem. In this mode, ACS SASSI ANALYS basically reads the three input files - FILE1 (or FILE9), FILE3, and FILE4 or modelname.N4 and COOSK and COOSM FE model matrix files. - and generates the result files – COOXxxx and COOTKxxx (xxx is the frequency order number) and the FILE8. The FILE8 is the SSI solution database file that include complex transfer functions at all frequencies for all SSI system dofs.

**New Structure** (Mode 2, Restart): If the physical properties of the structure(s) are changed or the geometry of the structure is altered, then, as long as other interaction nodes and soil layer data remain the same, then, ANALYS may be restarted using Mode 2.

In order to use this mode, a new FILE4 has to be generated by re-executing module HOUSE. Since the geometry and numbering of the nodal points below the ground have not changed, the impedance matrices can be recovered. The information on the two input files is then used to compute the new reduced modified stiffness of the structure.

**New Seismic Environment** (Mode 3, Restart): If the seismic environment is changed, the information from FILE1 changes, but the other information written in the restart files COOXxxx and COOTKxxx files is the same, ANALYS can be run in Mode 3 (xxx is the frequency order number).

**New Dynamic Loading** (Mode 3, Restart): If the problem is to be analyzed for a new set of external forces, then the program module ANALYS can be restarted in Mode 3. Only restart files and a new FILE9 are needed for this restart analysis. The “New Dynamic Load” is in Mode 3, being identical with the “New Seismic Environment Restart”

**New Load Vector** (Mode 6, Restart): This restart is rarely used, and is only in the batch mode. It is not included in the GUI input. It is special restart analysis related to changes in the seismic load vector. It could be useful for some special application, as per level incoherent SSI analysis for embedded models. To use this restart additional information needs to be provided by tech support.

**Simultaneous Cases** – Number of dynamic load cases to be run in a single SSI analysis

To save significant SSI runtime, ANALYS can solve simultaneously all three X, Y and Z input directions for seismic analysis (seismic option), or to up to 500 load cases of external forcing function (vibration option). For incoherent SSI analysis up to 50 stochastic simulations can be included in a single ANALYS run as in the HOUSE module (that produces files FILE77xxx, where

is number of simulations). However, the number of generated FILE8s will be 3 times larger, up to 150 since there are three direction FILE8s for each simulation. The FILE8s will be saved as FILE8xxx, where xxx could go up to 150 for 50 simulations including all X, Y and Z inputs.

**WARNING:** *The number of load cases or incoherent simulations depends on the SSI model size and available RAM of the workstation. If the resources are insufficient then an “access violation” message will be shown on the screen.*

For *incoherent seismic analysis* the number of simulations should be the same as considered for the HOUSE input. ANALYS can run to get all three X, Y and Z direction results for up to 50 simulations in a single run.

For *coherent seismic analysis* the number of simultaneous cases should be 1. The ANALYS run results will be saved in three FILE8 files computed for the X,Y and Z inputs named FILE8X, FILE8Y and FILE8Z. The ANALYS output will include ATF computed for all three directions.

**WARNING:** *To use the “Simultaneous Cases” option for seismic analysis, the SITE module should be run before ANALYS for X, Y and Z direction inputs to generate a FILE1 for each run that has to be copied as FILE1X, FILE1Y and FILE1Z. The user should select the SV waves for the X-direction by selecting x' direction and 0 angle in the .sit SITE input file, the SH waves for the Y-direction by selecting y' direction and 0 degree angle and the P waves in the Z-direction by selecting z direction and 0 angle. The coordinate transformation angle in the .anl ANALYS input file should be 0.*

For *external force or vibration analysis* to use multiple load cases, up to 500 load cases in a single ANALYS run, the FORCE module should be run before ANALYS for all considered load cases and generate all needed the FILE9xxx files, where xxx can be up to 500. For the vibration analysis, ANALYS produces a FILE8 for each load case, named FILE8xxx, where xxx can be up to 500.

**Save Restart Files** - Selecting the save restart files enables the saving of COOXxxx and COOTKxxx files for the “New Seismic Environment” or “New Dynamic Loading” restart options, or only COOXxxx files for “New Structure” option. Additional required restart files for ANALYS are DOFSMAP, FILE90 and FILE91 produced by HOUSE. If the flag is not checked, then, COOXxxx and COOTKxxx files are not saved.

**Print Amplitude Only** - If this option is set, ANALYS will print out the ATF response amplitude for all nodal points and all frequencies for which the solution has been obtained. If the option is not checked, then, ATF real and imaginary part will be printed separately for all 6 dofs per node. It should be noted that complex ATF amplitude and phase values can be output directly using MOTION.

## Frequency Numbers

**Take Frequency Numbers from File1/File9** - Check box if you wish to load the frequencies from FILE1 or FILE9. Otherwise, type the frequency set number in the Frequency Set Number edit box. The value is the same as set in Analysis Options - SITE dialog box. Frequencies for which a complete solution is desired must be specified at this stage. The program automatically surveys these frequencies to make sure that they reside in the input files. If one or more frequencies are not found on the input files, ANALYS stops. It is also possible to break the complete frequency set into smaller subsets and then, run each set separately on the same computer or different computers. The results of these separate runs can be combined into the complete solution. This has the advantage that the runs are smaller and the created files occupy smaller memory space during the execution.

You may specify a new frequency set number to solve the problem for new frequencies and combine the results with those of old frequencies if the analysis demands it at a later time.

Since more than 90% of the execution time of all ACS SASSI modules is spent by the ANALYS module, assessment of the final time of the run job can be made in advance by estimating the time of solving one single frequency and multiplying it by the number of frequencies for which a solution is desired.

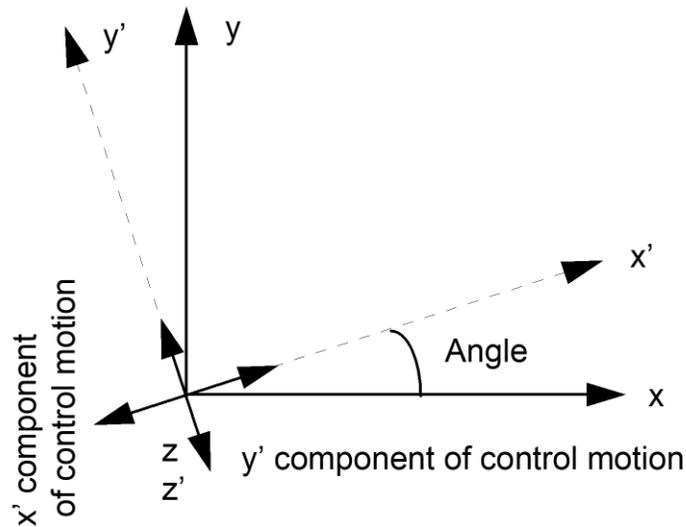
### Control Motion Foundation Reference Point:

**X-Coordinate of Control Point** - Type the X-coordinate of oblique input motion control point, and/or the foundation reference point for global, rigid-body foundation impedance calculations.

**Y-Coordinate of Control Point** - Type the Y-coordinate of oblique input the control point and/or foundation reference point for global, rigid-body foundation impedance calculations.

**Z-Coordinate of Control Point** - Type the Z-coordinate of foundation reference point for global, rigid-body foundation impedance calculations.

**Coordinate Transformation Angle** - Type the coordinate transformation angle (degrees) - the angle between the x' axis of the local coordinate system, as defined in the SITE module (see Analysis Options - SITE dialog box - see section 6.5.4) and the x axis of the global coordinate system.



<u>Coordinate System</u>	<u>Program</u>
x y z	HOUSE
x' y' z'	SITE

**Wave Passage** - Select this button to enable the wave passage option. This option is the same as set in the Analysis Options - HOUSE dialog box.

**Free-Field Load (FFL)** – This FFL user selection option is for applying the motion incoherency randomization to the seismic load vector defined at the interaction nodes. This option provides slightly conservative results for the elastic foundation than the “Free-Field Motion” (FFM) since is reducing the coupling between the forces applied different location points in the horizontal plane that provide the appearance that makes the incoherent effects slightly less incoherent. This option was included in the earlier versions of ACS SASSI and was validated during the 2007 EPRI studies (Short et al.,2007). The slight conservatism of this implementation also covers some uncertainties related to the use of the standard Abrahamson isotropic coherency model. i. e. coherence function is the same in all directions.

**Free-Field Motion (FFM)** - This user selection is for applying the motion incoherency randomization to the soil motion defined at interaction nodes. This option computes the seismic load vector by multiplying the free-field soil impedance matrix with the incoherent motion amplitude vector defined at the interaction nodes. The results obtained using this option are identical with the “Free-Field Load” (FFL) option for surface SSI models with rigid foundations founded on rock sites, and only slightly reduced for SSI models with elastic foundations.

The FFM is a recent added option to ANALYS based on some users' request. The FFM is based on the "free-field acceleration randomization" at interaction nodes. Apparently, the FFM is an alternative to the FFL, that is theoretically even more attractive. However, the fact that the free-

field motion amplitude is randomized to become *incoherent*, and then is multiplied by the *coherent* soil impedance matrix to compute the seismic force vector might create a problem for deeply embedded structures. Mixing the incoherent motion with the coherent soil impedance trends to create a much larger wave scattering within the excavation volume than should be.

**WARNING:** *The FFM randomization approach is not recommended for embedded structures to avoid producing artificial wave scattering effects due the embedment cavity. The newer FFM approach is highly accurate only for surface structures.*

We recommend using the FFL approach as the official incoherent randomization approach (accepted EPRI and USNRC) for all types of incoherent SSI applications from the surface to deeply embedded structures. The FFM is useful for surface structure only. Sometimes, in the low frequency range, the FFL might provide a small bias in results, so that some incoherent ATF have slightly larger amplitudes than the coherent ATF amplitudes – usually this bias is zero or quite insignificant. In such cases, for the surface structures, the FFM approach provides a slightly more accurate ATF in low frequency, not larger than coherent ATF. These very low frequency differences are expected to be quite negligible for ISRS obtained for the high-frequency seismic inputs on rock sites.

**WARNING:** *The complex response phase adjustment is an intrusion in the computed physics-based SSI response that in most cases provides conservative incoherent ISRS results. It should be noted the 2007 EPRI validation studies (Short et al., 2007) included the stochastic simulation approach with the phase adjustment option that provided incoherent ISRS results matching closer with the deterministic SRSS approach ISRS results based on zeroing the complex response phasing, as implemented in the other SASSI-based and CLASSI-based industry codes, and by this offered a “consensus” of the EPRI team of investigators, including Bechtel Co. and ARES Co. teams. Thus, the phase adjustment option provides conservative incoherent ISRS for SSI models with rigid basemats as shown in the 2007 EPRI studies (Ghiocel, 2007a, 2013b). However, the use of the phase adjustment or the zeroing of the complex response phasing in the SRSS approach, does not guarantee obtaining conservative incoherent ISRS at all locations for SSI models with elastic foundations (Ghiocel, 2016a).*

**Global Impedance Calculations** – The options are between No Calculations, Calculate Only Diagonal Impedances and Calculate the Full-Rigid Body 6x6 Impedance Matrix for soil layering. The second choice produces four text files FOUNSTIF, FOUNDASH, FOUNDAMP and FOUNIMPD that includes the computed dynamic stiffnesses, viscous damping coefficients, effective damping ratios and absolute value of impedance functions, respectively. Calculations of the global soil impedances are based on the integration of node point impedances assuming rigid-body linear displacement shapes for the foundation motion in 3D space. This option is active only for 3D SSI models with or without embedment.

---

**WARNING:** For embedded SSI models the global soil impedance calculations works only for the FI-FSIN method. **It should be noted that the “unconstrained” soil global impedances are different than the rigid foundation global impedances.** Only for surface models the “unconstrained” global impedances are the same with the rigid foundation based global impedances. The “unconstrained” global impedances include only the contribution of the distributed soil stiffnesses; basically it is like assuming that the foundation is infinitely flexible. If the two step SSI analysis is desired, then, we strongly recommend the use of the Option A capability that is a highly accurate theoretical approach, as described in the “ACS SASSI-ANSYS Integration Capability” User Manual document.

**Use Multiple Excitation** -Select this button for activating the multiple excitation option. This option is the same as set in the Analysis Options - HOUSE dialog box.

**Input Motion Number** - Select the number of the active input motion. All following data refers to this motion. The motion data is the same as set in the Analysis Options - HOUSE dialog box.

**First Foundation Node** - Type the number of the first foundation node for the active motion.

**Last Foundation Node** - Type the number of the last foundation node for the active motion. (see comments for the HOUSE module)

**X Coord. of Control Point** - The X-coordinate is used to define the input motion location application.

**Y Coord. of Control Point** - The Y-coordinate is used to define the input motion location application.

**Z Coord. of Control Point** - The Z-coordinate is used to define the input motion location application.

The ANALYS results are the computed complex transfer functions in degrees of freedom of the SSI model that are saved on FILE8. This FILE8 is used by MOTION and STRESS modules to compute the SSI responses.

**MOTION Module Options**

The main function of the MOTION module is to compute SSI acceleration motion time histories, transfer functions and in-structure response spectra (ISRS). For seismic problems, the ACS SASSI MOTION module reads the acceleration time history of the control motion from the input file and transforms it to the frequency domain using the Fast Fourier Transform technique. It then reads the computed transfer functions from FILE8 for the selected output nodal points, performs the interpolation and, then, convolve the transfer function with the control motion Fourier transform, and, finally, returns to the time domain using the inverse Fast Fourier Transform algorithm. The resulting time histories of acceleration may be output directly or converted to output response spectra.

---

The following options allow you to specify the analysis options for MOTION module:

**Operation Mode** - Select the operation mode from Solution and Data Check.

**Type of Analysis** - Select the type of analysis from Seismic and Foundation Vibration. This option is the same as set in the Analysis Options - ANALYS dialog box.

**Baseline Correction** - Select if the motion module will generate a baseline correction. The baseline correction is based on the classical Hudson-Housner time-domain procedure. It computes only approximate absolute displacements. The user has to check if the ground displacement reasonable by checking if it is zero at the end of earthquake motion. The relative displacements between different locations can be computed more accurately by the difference absolute displacements.

**WARNING:** *For computing relative displacements, we recommend the use of the RELDISP module instead of the baseline correction option. RELDISP uses a refined, analytical approach to compute the relative displacement histories based on the absolute acceleration transfer functions (ATF) obtained by MOTION in complex frequency.*

**Output Only Transfer Functions** - If this option is set, only transfer function are extracted at the selected nodal points.

**Save Complex Transfer Function** - If this option is set then amplitude and phase of the computed transfer functions are saved in TFU and TFI files. This option is not recommended for typical SSI analysis since the information on the Fourier phase is often difficult to interpret from engineering point of view.

**Save FILE13 or FILE12** - Type 1, if you would like to save the baseline corrected absolute acceleration, velocity and displacement time histories in FILE13. If no baseline correction is selected no FILE13 will be generated. For full post-processing of large-size models the FILE13 size can become extremely large. Therefore, for typical SSI analyses, we recommend not to save FILE13. Type 2, if you would like to save acceleration histories and response spectra in the FILE12 file.

**Total Duration** for Extracted Time Histories - Input the total duration of input acceleration time history to be extracted/plotted. The duration will be 20 percent larger than the selected duration to include also a part of the free vibration of the structure.

**Smoothing Parameter** - The smoothing parameter is used to filter the potential spurious spectral sharp peaks and valleys that could be artificially introduced by the original SASSI methodology interpolation schemes for computing complex ATF responses. This is applicable to the first six interpolation options from 0 to 5. Using the UI "Spectrum TFU-TFI" plot capability, the analyst could investigate the effects of smoothing on the SSI results by comparing for selected nodes the

computed and the interpolated amplitude transfer functions that are saved on hard-drive in text files with extension TFU and TFI. This smoothing parameter has no effect for the bicubic complex spline interpolation scheme that is the interpolation option equal to 6.

**Incoherent SRSS** - Allows the user to create a text file “SRSSTF.txt” that will be read by the MOTION module to apply the SRSS TF approach. This SRSSTF.txt file indicates the number of incoherency modes and the name of each FILE8 obtained from the incoherent SSI analysis for each considered mode by a separate SSI analysis (or restart SSI analysis using “New Seismic Environment” option after HOUSE is run repeatedly for each incoherent mode).

The format of the SRSSTF.txt file is as follows:

```
[# of modes] [phase option]
[FILE8 for 1st mode or FILE8 for Coherent if phase option = 1]
[FILE8 for 2nd mode]
...
[FILE8 for nth mode]
```

The SRSS phase option inputs are:

- 1) Type 0 for SRSS TF with zero-phase approach (validated by 2007 EPRI studies) and
- 2) Type 1 for SRSS TF with the coherent phase (not validated by 2007 EPRI studies)

The user is required to rename the FILE8 after each ANALYS run. See example below; FILE8\_coh is for the coherent input, and FILE\_0J is for each J-th mode of the total of 10 incoherency modes considered. Example of the SRSSTF.txt for the SRSS TF approach with the coherent phase option for a set of 10 selected incoherency modes:

```
10 1
FILE8_coh
FILE8_01
FILE8_02
FILE8_03
FILE8_04
FILE8_05
FILE8_06
FILE8_07
FILE8_08
FILE8_09
FILE8_10
```

---

This input uses the coherent phase for the overall SRSS ATF response, so that the first FILE8\_coh in the above list is from the coherent SSI analysis. For zero-phase option, the second line with FILE8\_coh should be skipped.

**WARNING:** *If the analyst desires to use the SRSS approach, he should be aware that ACS SASSI implementation respects the basic SRSS theoretical aspects, but may not include all the “artifacts” implemented in SASSI2010 and EPRI INCOH codes. Therefore the SRSS approaches implemented in ACS SASSI may not produce necessarily identical results with the SASSI2010 and EPRI INCOH codes.*

**Interpolation Option** - To get accurate acceleration transfer function (ATF) computation, seven complex frequency interpolation schemes can be selected:

- 0 - SASSI2000, dense overlapping windows (weighted averaging)
- 1 - Original SASSI 1982, non-overlapping windows (no averaging)
- 2 - Dense overlapping windows (averaging).
- 3 - Only three overlapping windows (averaging)
- 4 - Non-overlapping windows with one position shift (no averaging)
- 5 - Non-overlapping windows with two position shift (no averaging)
- 6 - Complex bicubic spline interpolation (no windowing, need a denser frequency point grid)

**WARNING:** *The spline interpolation option is recommended for performing incoherent SSI analysis since avoids any overshooting for the ATF interpolation. Before using the spline option for incoherent SSI analysis, the user has to check that the number of selected frequencies is sufficiently large, so that using the spline interpolation will not smooth or clip any significant incoherent ATF spectral peak at any node locations. Typically, for a number of frequencies larger than 200 as recommended for incoherent analysis, the complex bicubic spline interpolation should be accurate.*

**Phase Adjustment** - This option could be used for incoherent SSI analysis with the stochastic approach (Simulation Mean in 2007 EPRI studies) or the deterministic linear approach (AS in 2007 EPRI studies) is selected. The phase adjustment option provides an approximate upper-bound SSI solution by zeroing the ATF phasing for incoherent SSI responses.

The following phase adjustment input options could be selected:

0 - *No phase adjustment.* It maintains untouched the complex ATF response phasing coming from the SSI physics. Not used in the 2007 EPRI studies since it provided lower ISRS results than other industry SRSS approaches implemented by ARES Co. and Bechtel Engineering.

1 - *Phase adjustment* option that reduces largely to almost zero the differential phase differences between Fourier components, depending on the smoothing parameter selected, to avoid

---

counteracting effects for neighboring frequencies (it creates a critical, “minimum delay” motion). It is an approximate approach that is expected to be generally conservative, as indicated by the 2007 EPRI validation studies (Short et al., 2007). It produces results that are very close to the results obtained using the SRSS TF approach, as shown in the 2007 EPRI studies for stick models with rigid basemats. *The phase adjustment option received a “consensus” industry agreement among the technical investigators of the 2007 EPRI studies since it provided the best matching between the different US industry approaches tested during the 2007 EPRI research project. The application of the phase adjustment in stochastic simulation approach or the zeroing of the complex response phasing in the SRSS approach provided also conservative results for the EPRI investigated stick model with rigid base (Ghiocel, 2007a). However, the phase adjustment or zeroing introduces a departure from the physics-based SSI modeling since it artificially affects the complex ATF response phasing. A “theoretical exact” approach will be based on the stochastic simulation approach without phase adjustment. It should be understood that the 2007 EPRI validated approaches are conservative approaches for computing ISRS for rigid base SSI models.*

**WARNING:** *The use of the phase adjustment, similar to the use of the SRSS approaches, affects the cross-correlation between the SSI response motions at different locations. Thus, if phase adjustment option or the SRSS methods are used, then, the response motions computed at different locations should be not used for “Multiple Time History Analysis” of secondary systems.*

**Smoothing Parameter** -The smoothing parameter is the potential spurious spectral sharp peaks and valleys that could be artificially introduced by the classical SASSI interpolation scheme for complex transfer functions (interpolation options 0 to 5). Using the US “Spectrum TFU-TFI” plotting capability, the analyst can investigate the effects of interpolation error smoothing on the SSI results by comparing for selected nodes the computed and the interpolated amplitude transfer functions saved in the text files with extensions TFU and TFI. The smoothing parameter values can vary typically between 10 and 1000.

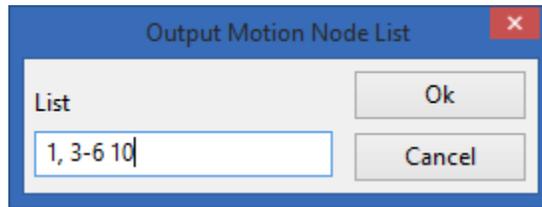
**WARNING:** *Smoothing parameter should be zero for coherent analysis, but non-zero for incoherent analysis, except for the interpolation option = 6. Sensitivity studies are recommended for incoherent analysis before final smoothing parameter is decided for SSI production runs.*

The 2007 EPRI studies (Short et al., 2007) showed that for a smoothing parameter with values from 10 to 500, the ISRS results were practically the same. The 2007 EPRI incoherent SSI validation report (Short et al., 2007) includes additional details on how to use interpolation ATF error smoothing.

**WARNING:** *The smoothing option should not be applied if the spline interpolation was selected. The smoothing parameter should be zero, if the spline interpolation is used.*

**Nodal Output Data** - The Node List contains lists of nodes having the same output request. Output requests in constrained directions are ignored. Press the Add button to add a new node

list, the Edit button to edit the nodes of the selected list, or the Delete button to delete the selected list.



You may use blank, tab, ',' , or ';' as separators for the node numbers. You may use '-' to add a set of nodes to the node list.

For each node list and for each direction (buttons X, Y, Z, XX, YY, ZZ), select the requested output from the check boxes placed on the right side of the Node List:

Printed Plot of Transfer Functions \*

Save Time History of Requested Response \*\*

Plot Time History of Requested Response \*\*

Plot Acceleration and Velocity R. S. \*\*\* (this does not work in newer versions; it will be taken out)

Save Acceleration and Velocity R. S. \*\*\*

Print Maximum Requested Response \*\*

- 
- \* The transfer function for seismic problems is defined for total acceleration response while for foundation vibration problems, it is defined for total displacement response.
  - \*\* The requested response for seismic problems is acceleration; for foundation vibration problems, it is determined by the selected option from the Output Control group box.
  - \*\*\* The response spectra are computed independent of the option from the Output Control group box. Therefore, displacement response spectra cannot be requested.

**WARNING:** The definition of an output node twice times will produce incorrect results. To prevent this, the UI deletes duplicate nodes when the AFWRITE command is executed for MOTION.

**Response Spectrum Data** - The user can compute the response spectra for selected nodal locations using the setting described next.

**First Frequency** - Type the first frequency used in response spectrum analysis (Hz).

**Last Frequency** - Type the last frequency used in response spectrum analysis (Hz).

**Total Number of Freq. Steps** - Type the total number of frequency steps used in response spectrum analysis. Per US NRC SRP 3.7.1 requirements, we recommend for nuclear safety-related projects to use a frequency range from 0.1 Hz to 100 Hz with at least 301 frequency steps.

---

**WARNING:** *A reduced number of frequency steps could affect the accuracy of the UI spectrum broadening algorithm. Use 301 frequencies as recommended.*

**Damping Ratios** - Type the damping ratios used in response spectra. You may use blank, tab, ‘,’ or ‘;’ as separators between damping ratios.

**Nr. of Fourier Components** - Type the number of values to be used in the Fourier transform. The value is the same as set in the Analysis Options - SITE dialog box.

**Time Step of Control Motion** - Type the time step of control motion (sec). The value is the same as set in the Analysis Options - SITE dialog box.

**Multiplication Factor** - Type the multiplication factor for scaling the time history. Use only if Max. Value for Time History is blank.

**Max. Value for Time History** - Type the maximum value of time history to be used. The values of the time history will be scaled to this value. Use only if Multiplication Factor is blank.

**First Record** - Type the first record from the time history file to be used (default: first).

**Last Record** - Type the last record from the time history file to be used (default: last).

**Title** - Type the title of the acceleration time history.

**Acceleration History File** - Type the full path and name of the input time history file. To view the time history file, use the Plot / Time History command.

**File Contains Pairs Time Step-Acceleration** - Select this option if your time history file contains pairs of time step and acceleration values on each line. Otherwise, the file is expected to contain the time step on the first line, and on the following lines one acceleration value.

### **Convert Time History to Response Spectrum**

This option is only for the user convenience and has nothing to do with the SSI analysis. Using it the user can compute response spectra for an external time history file (not for a selected nodal output). This option is convenient to be used after we compute new acceleration time histories based on algebraic summation outside the code. The resultant time histories can be input to the code to compute their response spectra. The setting for response spectrum calculations are assumed to the same as for the nodal outputs (same calculation frequencies, damping and number of time steps).

To use this option the user needs to check the option. Then, the user needs to browse for the acceleration time histories he wants to select by using for computing their response spectra. All

---

the external file names are written in the CONTTRS.txt file (can be edited by the user). The external time history files are required to have the same one column format, same time step and duration (not smaller or longer than the computed accelerations). The external files should have name format and the extension .ACC that is similar to the acceleration histories calculated by the MOTION module.

**Post Processing Options** - The three options in this box are to allow the user to generate data for Bubble, Vector TF and Displacement plot in all nodes. These options can be used in tandem with the options in the Node Output data fields.

**Save TF in All Points** - Save computed and interpolated acceleration TF in all translational DOFs (.TFU files)

**Save ACC in All Points** - Save computed acceleration histories in all translational DOFs (.ACC files)

**Save RS in All Points** - Save computed acceleration RS for selected dampings in all translational DOF (.RS files)

**Save Rotations for ANSYS** - Save computed acceleration histories in all rotational DOFs (.ACC files)

**Restart for TF** - Compute and save frames for complex TF vector plots (animated) (\TFU subdirectory)

**Restart for ACC** - Compute and save frames for acceleration time history deformed shape (animated) or bubble plots (static) (\ACC and \ACCR subdirectories)

**Restart for RS** - Compute and save frames for RS deformed shape (animated) or bubble plots (static) (\RS subdirectory)

The three options in this box are to allow the user to generate data for Bubble, Vector TF and Deformed Shape plots in all nodes. The “Save” options are to save SSI responses in all DOFs. The “Restart” options generate frame files for graphical animations. These frame files are saved in subdirectories \TFU, \RS and \ACC (and \ACCR for rotational accelerations). The “Save” options can be used in tandem with the options in the Node Output data fields.

**Save Binary Database** – This option saves acceleration time histories in a single binary database for fast post-processing. See also the BINOUT command.

To combine acceleration history binary databases with default name Modelname\_ACC.bin for the X, Y and Z input directions, the COMBACCDB command should be used.

### STRESS Module Options

The main function of the STRESS module is to compute and output maximum stresses, forces, or moments in the elements. The user may also request time histories of those components to be printed and saved. The program reads the acceleration (displacement) transfer functions from FILE8 and information about elements from FILE4. Then, for each requested elements, the STRESS module computes the stress, force, or moment components at each frequency, performs interpolation and convolution with the control motion, and finds the corresponding time histories by returning in the time domain using the inverse Fast Fourier Transform algorithm.

The screenshot shows the 'Analysis Options' dialog box with the 'STRESS' tab selected. The 'Element Output Data' section contains a table with the following data:

Group	Element List	Output Code
10	1-28	000000000000

The 'Components' section has the following options:

- Force NXX
- Force NYY
- Force NXY
- Force QXZ
- Force QYZ
- Moment MXX
- Moment MYY
- Moment MXY

The 'Component Request' section has the following options:

- No Request
- Print Only Maximum
- Print Maximum and Save Time History Response

The 'Post Processing Options' section has the following options:

- Save Max Value
- Save Time History
- Restart for Nodal Stress Contours
- Restart for Soil Pressure Contours

The 'Section Cut Options' section has the following options:

- Save Time History

The 'Acceleration Time History Data' section has the following values:

- Nr. of Fourier Components: 8192
- Time Step of Control Motion: 0.005
- Multiplication Factor: 1
- Max Value for Time History: 0
- First Record: 1
- Last Record: 5000

The 'Title' field contains 'acc\_X\_8192' and the 'File' field contains 'C:/test/tshell/acc\_X\_8192.acc'. The 'File Contains Pairs Time Step - Accel.' checkbox is unchecked. The 'Binary Processing Option' section has the following options:

- Save Binary Database

The 'Operation Mod' section has the following options:

- Solution
- Data Check

The 'Type of Analysis' section has the following options:

- Seismic
- Foundation Vibration

The 'Output Control' section has the following options:

- Auto Computation of Strains in Soil El.
- Save Stress Time Histories
- Output Transfer Function

The 'Phase Adjustment' field contains '0', the 'Interpolation Option' field contains '1', and the 'Smoothing Option' field contains '0'. The 'Frame Selection' button is visible in the 'Post Processing Options' section.

The captured screen above shown is an example for selecting element forces and moments for the new thick plate TSHELL element.

---

For nonlinear soil SSI analysis, the time history of the shear strain is calculated in time-domain for each time interval defined by the time step. From this, the effective shear strain that is 0.6-0.7 pf the maximum shear strain for the soil elements can be estimated and used to define new strain-compatible soil properties.

**WARNING:** *The STRESS module is only for directional inputs. Auxiliary DOS batch program called COMB\_XYZ\_STRAIN is available to combine the co-directional effects after each SSI iteration, so that that nonlinear soil behavior is computed based on the three-directional seismic input, not separate for each unidirectional input. Please see “Nonlinear SSI” input for HOUSE to select which soil shear-strain you selected for considering the nonlinear soil behavior (either components, maximum shear strain for 2D SSI models and octahedral shear strain for 3D SSI models).*

The following options allow you to specify the analysis options for the STRESS module:

**Operation Mode** - Select the operation mode from Solution and Data Check.

**Type of Analysis** - Select the type of analysis from Seismic and Foundation Vibration. This option is the same as set in the Analysis Options - ANALYS dialog box.

**Auto Computation of Strains in Soil Elements** - Set this option if you want STRESS to automatically compute the strains in all soil elements (if secondary non-linear effects are considered). This option should be used together with the save option of transfer functions of forces and moments in beam elements (see next two save options).

**Save Stress Time Histories** - Set this option if you want STRESS to save the stress time histories in .ths files. This save option should be used in conjunction with component output request per element.

**Save Transfer Functions** - Set this option if you want STRESS to output transfer functions for beam element nodal forces and moments. The .TFU and .TFI files will be saved.

**Skip Time History Steps** -Type the number of steps to be skipped for the output time history.

**Phase Adjustment** -This parameter is defined in the same way as for the MOTION input – see previous section. There is no literature publication in the support of using or not the phase adjustment. If no phase adjustment is selected for the stochastic simulation approach, then, this approach corresponds exactly to the Monte Carlo simulation approach applied to the incoherent SSI analysis to get the statistical mean of the seismic in-structure responses of interest.

**Interpolation Option** - To improve accuracy of the stress transfer function (STF) computation, several complex frequency interpolation schemes can be used (same as in MOTION):

- 
- 0 - SASSI2000, dense overlapping windows (weighted averaging)
  - 1 - Original SASSI 1982, non-overlapping windows (no averaging)
  - 2 - Dense overlapping windows (averaging).
  - 3 - Only three overlapping windows (averaging)
  - 4 - Non-overlapping windows with reduced shift (no averaging)
  - 5 - Non-overlapping windows with large shift (no averaging)
  - 6 - Complex bicubic spline interpolation (no windowing, need a denser frequency point grid)

**WARNING:** *The spline interpolation option is recommended for performing incoherent SSI analysis since avoids any overshooting during the STF interpolation process. However, before using the spline option for incoherent SSI analysis, the user has to check that for coherent SSI analysis the number of selected frequencies is sufficiently large, so that using the spline interpolation will not smooth or clip any significant STF spectral peaks.*

**Smoothing Parameter** - The smoothing parameter is used optionally to filter the potential spurious spectral sharp peaks and valleys that could be artificially introduced by the original SASSI interpolation scheme for transfer functions. Using the ACS SASSI spectrum plot capability, the analyst could investigate the effects of smoothing parameter values on the SSI results by comparing for selected nodes the computed and the interpolated amplitude transfer functions that are saved on hard-drive in text files with the extensions TFU and TFI. Smoothing parameter values vary typically between 10 and 1000.

**WARNING:** *Smoothing parameter should be zero for coherent analysis, and non-zero for incoherent analysis, except Option = 6. Sensitivity studies are recommended for incoherent analysis before the final smoothing parameter value is decided for the SSI production runs.*

**Acceleration History Data** – This information is the same as for SITE or MOTION. Please see the MOTION input.

**Nr. of Fourier Components** - Type the number of values to be used in the Fourier transform. The value is the same as set in the Analysis Options - SITE dialog box.

**Time Step of Control Motion** - Type the time step of control motion (sec). The value is the same as set in the Analysis Options - SITE dialog box.

**Frequency Set Number** - Type the number of the frequency set. The value is the same as set in the Analysis Options - SITE dialog box.

**Multiplication Factor** - Type the multiplication factor for scaling the time history. The value is the same as set in the Analysis Options - MOTION dialog box.

**Max. Value for Time History** - Type the maximum value of time history to be used. The value is the same as set in the Analysis Options - MOTION dialog box.

**First Record** - Type the first record from the time history file to be used. The value is the same as set in the Analysis Options - MOTION dialog box.

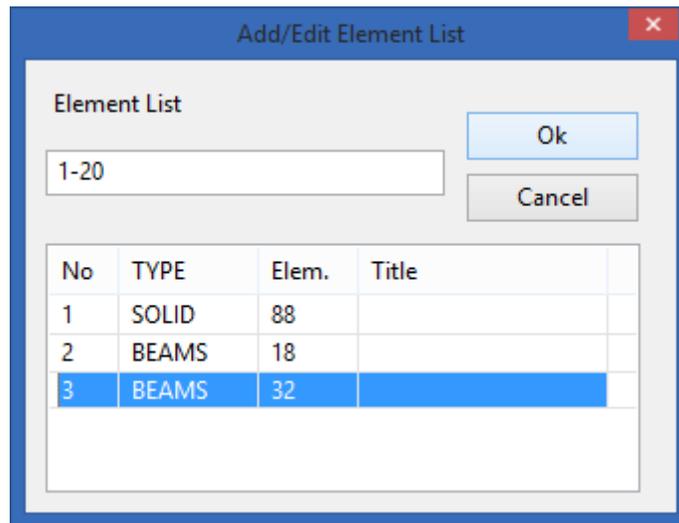
**Last Record** - Type the last record from the time history file to be used. The value is the same as set in the Analysis Options - MOTION dialog box.

**Title** - Type the title of the acceleration time history. The title is the same as set in the Analysis Options - MOTION dialog box.

**Acceleration History File** - Type the full path and name of the time history file. The file name is the same as set in the Analysis Options - MOTION dialog box.

**File Contains Pairs Time Step-Acceleration** - Select this option if your time history file contains pairs of time step and acceleration values on each line. The option is the same as set in the Analysis Options - MOTION dialog box.

**Element Output Data** - The list box contains groups, lists of elements having the same output request and the output request for the group. Press the Add button to add an element list, the Edit button to edit the elements of the selected list, or the Delete button to delete the selected list.



You may use blank, tab, ',' , or ';' as separators for the element numbers. You may use '-' to add a set of elements to the element list. Select the group from the Groups list box.

For each element list and for each component (buttons from the Components group box), select the requested output from the buttons placed in the Component Request group box.

The group element numbers should be defined in the ascending order.

**WARNING:** For mixed models that include BEAM elements but also SOLIDS and/or SHELL, if the mixed STRESS output post-processing is used, then, it should be checked against separate post-processing STRESS output for the BEAM elements only. There is an unconfirmed reporting that indicates that could be sometime differences in the printed maximum axial forces for the first elements of the BEAM groups if the output requests include mixed element groups vs. only BEAM element groups. However, this reporting was not confirmed.

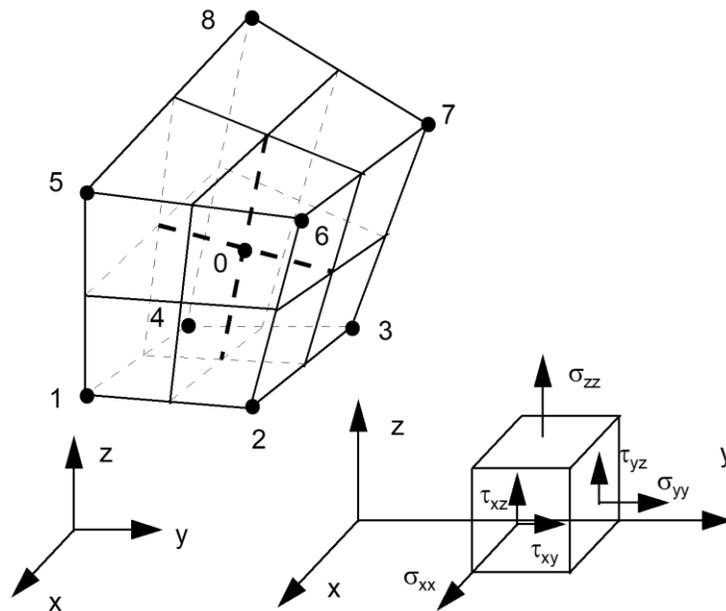
If the nonlinear soil SSI analysis option is used, then the nonlinear soil element group (including all solid elements for 3D models or plane elements for 2D models) has to be selected by the user. The nonlinear SSI results are saved at each iteration in FILE74.

### SOLID elements

The available components are:

- Stress / Strain XX Direction
- Stress / Strain YY Direction
- Stress / Strain ZZ Direction
- Stress / Strain XY Direction
- Stress / Strain XZ Direction
- Stress / Strain YZ Direction
- Octahedral Shear Stress

The stresses in the 3D solid elements are computed at the centroid of the element and are referred to in global axes. The stresses are shown in the following figure:



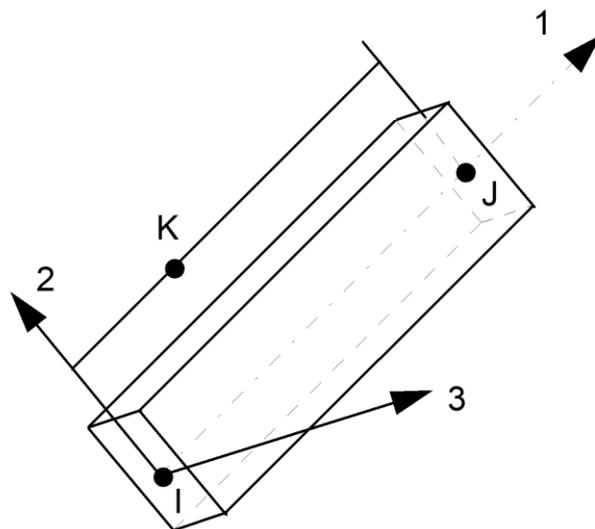
In addition to the user-requested response, MOTION may also output other components of the response if calculation of these components is necessary in order to determine the requested response. For example, in order to output maximum octahedral shear stress for 3D solid elements, all six components of stress must be computed. Therefore, MOTION also outputs the maximum response of these components. However, the corresponding time histories are not to be saved on FILE12, unless they are specifically requested by the user.

### BEAMS elements

The available 12 components are:

- Force 1-Direction - Node I
- Force 2-Direction - Node I
- Force 3-Direction - Node I
- Moment 1-Direction - Node I
- Moment 2-Direction - Node I
- Moment 3-Direction - Node I
- Force 1-Direction - Node J
- Force 2-Direction - Node J
- Force 3-Direction - Node J
- Moment 1-Direction - Node J
- Moment 2-Direction - Node J
- Moment 3-Direction - Node J

The forces and moments in beam elements are computed at the end and are reference in local beam axes:



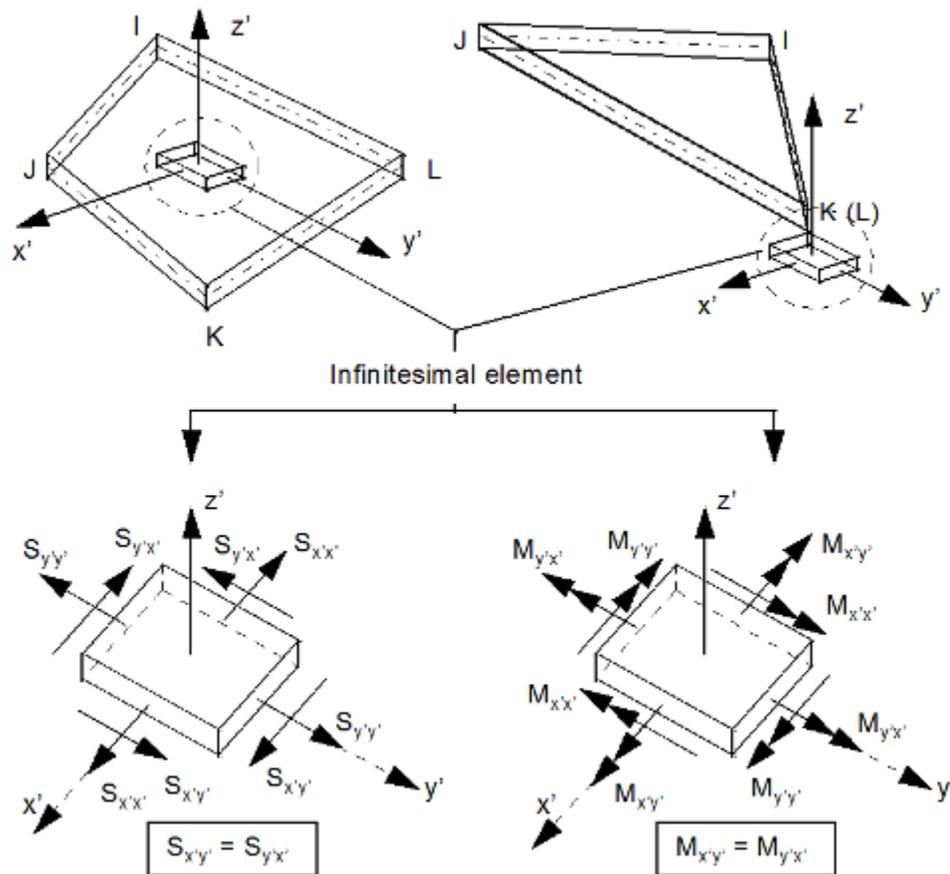
## SHELL elements

The available 6 components are:

- Force XX-Direction ( $S_{x'x'}$ )
- Force YY-Direction ( $S_{y'y'}$ )
- Force XY-Direction ( $S_{x'y'}$ )
- Moment XX-Direction ( $M_{x'x'}$ )
- Moment YY-Direction ( $M_{y'y'}$ )
- Moment ZZ-Direction ( $M_{x'y'}$ )

The membrane stresses and plate bending moments are computed in respect to the local element coordinate system. The forces are in units of force/length/length (F/L/L) or force over area, and the bending moments are in units of moment/length or force-length/length (FL/L).

The location of the infinitesimal elements where the forces and moments are computed are shown in the following figure:

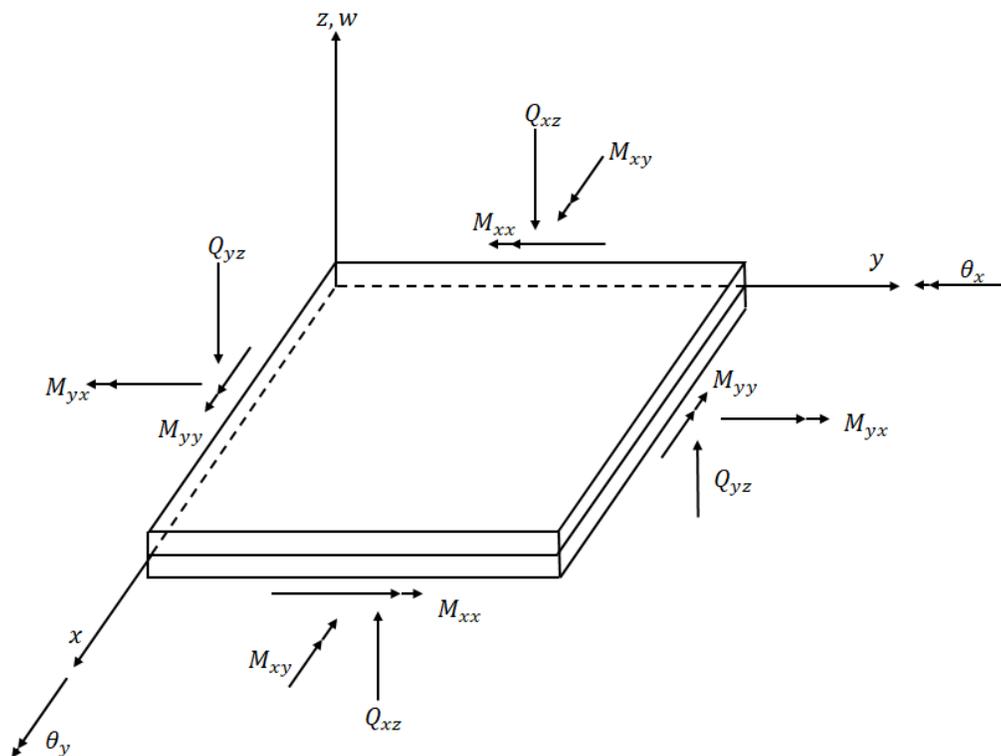


## TSHELL elements

The available 8 components are:

- Force XX-Direction ( $N_{xx}$ )
- Force YY-Direction ( $N_{yy}$ )
- Force XY-Direction ( $N_{xy}$ )
- Force XZ-Direction ( $Q_{xz}$ )
- Force YZ-Direction ( $Q_{yz}$ )
- Moment XX-Direction ( $M_{xx}$ )
- Moment YY-Direction ( $M_{yy}$ )
- Moment ZZ-Direction ( $M_{xy}$ )

The membrane forces and bending moments are computed in respect to the local element coordinate system. The forces are in units of force/length (F/L) or force over area, and the bending moments are in units of moment/length or force-length/length (FL/L). The location of the infinitesimal elements where the forces and moments are computed are shown in following figure:



To get more element output including the top and bottom face maximum stresses, the THSHLSTR command should be used. The maximum in-plane component stresses are obtained based on the sign permutations for the maximum in-plane force and maximum out-of-plane moment contributions for these stress components, as ++, --, +- and -+ for the signs of  $N_{xx}$  and  $M_{xx}$ , and

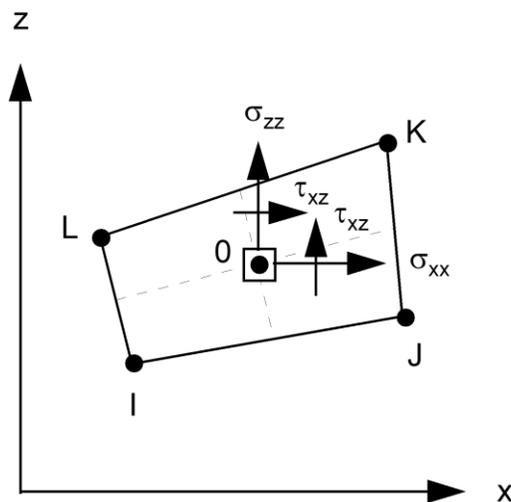
$N_{yy}$  and  $M_{yy}$ , assuming that the in-plane shear stress is the largest, corresponding to signs ++ for  $N_{xy}$  and  $M_{xy}$ . The principal stresses and strains on the faces are computed based on the maximum component stresses.

### PLANE elements

The available 3 components are:

Sigma XX  
Sigma ZZ  
Tau XZ

The stresses in 2D PLANE elements are computed at the center of the element and are referred to in global axes. These stresses are shown in the following figure:



### SPRING elements

The available 6 components are:

Force X-Direction  
Force Y-Direction  
Force Z-Direction  
Moment XX-Direction  
Moment YY-Direction  
Moment ZZ-Direction

The available requests for each component are:

---

No Request  
Print Only Maximum Response  
Print Maximum and Save Time History

**WARNING:** *The signs of the computed spring forces should be defined based on the relative displacements between the spring end nodes that can be computed using the RELDISP module. The way how the STRESS spring forces are computed could be confusing to the users, since the spring force sign does not reflect that the spring is in tension or compression.*

### Post Processing Options

The Options in this box to allow the user to save the stress data in all of the node so it can be plotted in the contour plot. These Options also allow the user to specify a restart of motion to calculate soil pressure after the motion has initially been run. The Frame Selection button opens a file "Frames.txt" which is used by the stress module.

**Save Max Values** - Save only maximum values in all elements (.sig, .tau, .bdsig, .bdtau files)

**Save Time Histories in All Elements** - Save only time histories in all elements (.sig, .tau, .bdsig, .bdtau files)

**Restart for Nodal Stress Contours** - Compute and save frames for contour plots (static or animated) (\NSTRESS subdirectory)

**Restart for Soil Pressure Contours** - Compute and save nodal pressure frames for contour plots (static or animated) (\SOILPRES subdirectory).

The options in this box to allow the user to save stress data in all of the nodes, so that they can be plotted as stress contours. The "Save" options are to save SSI responses in all DOFs. The "Restart" options generate the frame files for graphical animations. These frame files are saved in subdirectories \NSTRESS and \SOILPRES. The "Save" option can be used in tandem with options for Element Output data fields.

These post-processing "Restart" option for computing soil pressures on foundation walls and mat (adjacent SOLID groups need to be defined for near-field soil). The Restart for soil pressures could be run first time to get only the seismic pressures and the second time to get the total soil pressures including both the static and seismic pressures (before this the user should combine the seismic soil pressure frames in time or SRSS to include the three-directional input effects).

Please note that to run the Restart options a "Frame.txt" needs to be input:

**Frame Selection** - The Frame Selection button opens a file "Frames.txt" which is used by the STRESS module to generate stress and soil pressure frames.

The Frames.txt file has the following format:

[# of Frames]

List of Frame Numbers

[# of Soil Pressure Groups]

List of Soil Pressure Groups (Enter all on the same line)

Frames.txt Example

```
10
1
2
3
4
5
6
7
8
9
10
3
8 9 10
```

This input specifies to save frames 1-10 for stresses and calculate and save frames for soil pressures for 3 near-field soil SOLID groups, numbered group # 8, 9 and 10.

### Section Cut Options

**Save Time History** - If the box is checked, then, the STRESS module will save the frames all the element center stresses at all time steps. The element stress frame files named `estress_framenumber.ess` will be saved in the `.\NSTRESS\` subfolder. Using the UI module, the user is able to compute the section-cut forces and moment time histories for all time steps.

Please see for details the Demo 8 and the Problem 47 in the Verification Manual.

**Save Binary Database** – This option saves element stress/force component time histories in a binary database with a default name called `Modelname_STRESS.bin` for fast post-processing. See also the BINOUT command.

To combine STRESS element binary databases for the X, Y and Z input directions, the COMBTHSDB command should be used.

### RELDISP Options

Use this Dialog box to change the options for the RELDISP Module. These Options will be written to a .rdi file in the model directory. To run RELDISP, first the complex TFI files at the selected output nodes need to be computed before by using MOTION.

**Reference Location And Direction** - Use the reference location and direction (for the reference node/DOF). Input the reference node transfer function file (TFI file for reference node/DOF) for the calculation of the relative displacements in other nodes for the same DOF. This implies that RELDISP needs to be run six times to get all the six DOF relative displacement components. For all six DOFs, the reference TFI extension file names should include in i all six DOFs, namely, TR\_X, TR\_Y, TR\_Z, R\_XX, R\_YY, R\_ZZ.

Analysis Options ×

EQUAKE SOIL SITE POINT HOUSE FORCE ANALYS MOTION STRESS **RELDISP** NONLINEAR AFWRITE

Reference Location and Direction

Complex TF File Name

Output Control

Save Rel Disp Complex TF

Acceleration Time History Data

Nr. of Fourier Components

Time Step of Control Motion

Multiplication Factor

Max Value for Time History

First Record

Last Record

Title

File

File Contains Pairs Time Step - Accel.

Nodal Output Data

Node Num...	X	Y	Z	XX	YY	ZZ
1	✓	✓	✓			

Post Processing Options

Save Relative Displacement in All Nodes  Restart For Frame Generation

Save Rotations for ANSYS V11.0

Binary Disp. Option

No Binary

TFD Binary

THD Binary

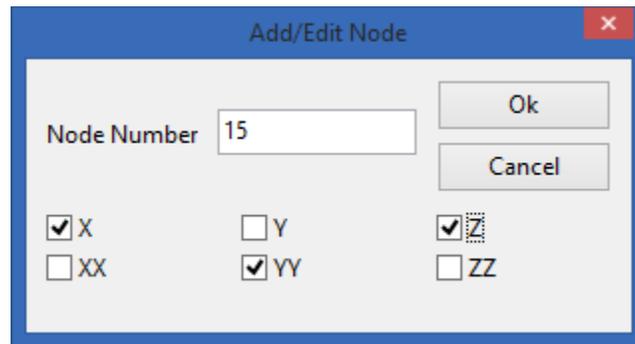
To compute relative displacement with respect to free-field input motion, the *reference* TFI files should be developed by the user. These TFI free-field reference files will have a TFI with amplitude 1.0 in the input direction amplitude, and 0 in the other directions for all frequencies. For coherent inputs, the phases will be zero for all directions and all frequencies. For incoherent stochastic simulations the zero-frequency phase should be considered in the *reference* TFI files.

**Output Control** - Allows the user to choose to write the complex or amplitude relative displacement transfer functions and allows the user to select whether .TFI or .TFU files will be used when generating the output data.

**Acceleration Time History Data** - The Acceleration Time History Data options are the same options as for MOTION or STRESS. Please see MOTION input.

**Node Output Data** - The User can select a list of nodes and degrees for freedom for the Relative Displacement Module to Analyze.

When entering a node or editing an entry in the node list the window above appears which allow the user to select the node and the degrees of freedom that the RELDISP module will output when run.



### Post Processing Options:

Quick option that allows the user to save the Relative Displacement Data in all node and set a flag generate Frames for the Contour/Deformed Shape plots, restart the RELDISP module for frame generation and save the rotational displacements for ANSYS.

**Save Relative Displacements in All Nodes** - Calculate and save relative displacements with respect to reference node/DOF in all nodes in THD files

**Save Rotations for ANSYS** - Calculate and save relative rotational displacements with respect to reference node/DOF in all nodes in THD files

---

**Restart for Frame Generation** - Compute and save frames for relative displacements with respect to the reference node/DOF (in \THD subdirectory)

**Binary Disp. Option** – THD Binary should be selected to create a relative displacement history binary database for a given input reference node and response direction. The default names for the binary databases for the three response directions X, Y and Z is Modelname\_TR\_X\_THD.bin, Modelname\_TR\_Y\_THD.bin, and Modelname\_TR\_Z\_THD.bin.

The COMBDISPDIR command should be used to combine the principal and coupling response binary databases is a single database for each input directions X, Y and Z. Then, the COMDISPDB command should be applied to combine the X-input, Y-input and Z-input response binary databases in a single binary database.

### ***NONLINEAR Module Options***

For the nonlinear structure SSI analysis, the Option NON (NONLINEAR and COMB\_XYZ\_THD modules) is required. The Option NON can address the reinforced concrete cracking and post-cracking behavior for low-rise shearwalls, nonlinear rubber base-isolators behavior, nonlinear pile-soil interface behavior, and even limited foundation sliding effects. The Option NON includes the two modeling choices: i) nonlinear reinforced concrete (RC) wall/panel elements (macroshell elements) and ii) nonlinear spring elements with translational dofs.

To understand in sufficient detail how to perform the nonlinear structure SSI analysis, please see the Demo 9 and 10 problems that show the use of the NONLINEAR and COMB\_XYZ\_THD modules for nonlinear structure SSI analysis. Demo 9 shows the nonlinear structure SSI analysis for a reinforced concrete low-rise shearwall building with walls modeled with nonlinear wall panels, subjected to a beyond design-level seismic input of 0.60g. Demo 10 shows the nonlinear SSI analysis for a seismically base-isolated building that uses lead-rubber bearing (LRB) isolators that are modeled with nonlinear shear spring elements.

***WARNING:*** To help the user to run nonlinear structure SSI analysis using NONLINEAR module, the UI module can provide a batch file for performing the entire analysis, optionally including the combination of X, Y and Z input direction results using the COMB\_XYZ\_THD.exe aux program. To generate the nonlinear SSI analysis batch file input, the NONLINBAT command should be used.

For the reinforced concrete shear walls, the BBC should be given by the user. For the shear deformation BBC the in-plane shear force-shear strain relationship is required, while for the bending deformation BBC the in-plane bending moment-section strain rotation or curvature relationship is required. Then, the user has to select the hysteretic model: i) Cheng-Mertz Shear (CMS) hysteretic model, Cheng-Mertz bending (CMB) hysteretic model (not included) or Takeda (TAK) hysteretic model to compute nonlinear time domain in-plane shear forces or bending

moments and strains, and based on these results to compute the equivalent-linear stiffness and damping properties for each wall.

**WARNING:** To create the correct input data files for MOTION and RELDISP that contains the output requests for all the panel corner nodes, before using the AFWRITE command, the user should use the NONLINMODISP command. The NONLINMOTDISP command ensures that the corner nodes are appropriately selected. If warped or non 4-noded panels are used then panel the NONLINMOTDISP command or the NONLINEAR module run might fail.

The screenshot shows the 'Analysis Options' dialog box with the 'NONLINEAR' tab selected. The 'Global Modeling Options' section includes input fields for 'Disp. Factor' (0.8), 'Damping Cutoff %' (0), 'Damping Scale Factor' (0), and 'Material Parameter'. Checkboxes for 'Use Non-linear Panels', 'Use Non-linear Springs' (checked), and 'Use Non-linear Beams' are present, along with 'Include Elastic Damping'. The 'Backbone Curve Data' section features a table with columns for 'X' and 'Y' values, and a 'Yield Num.' column. The 'Panel Data', 'Spring Data', and 'Beam Data' sections each have several input fields for numerical values.

	X	Y
1	0.01	100
2	0.0223	220
3	0.0232	226
4	0.0244	232
5	0.0265	238
6	0.0302	244
7	0.0374	251

The nonlinear springs under either the shear deformation or the axial deformation use the General Massing Rule (GMR) hysteretic model.

The BBC input by the user is generally in terms of a force-displacement, shear force-shear strain or moment-strain rotation/curvature relationship. Thus when the secant shear stiffness of the curve is found, this value of stiffness is normalized with the elastic stiffness. This normalized curve is then scaled with the elastic modulus value and this allows the stiffness-damping curves to be in terms of elastic modulus and damping. After each iteration the elastic modulus and damping are modified in the HOUSE input (.hou file).

The NONLINEAR module requires the input files: *modelname.sit*, *modelname.hou*, *modelname.EQL*, and the corresponding nodal displacement files (*xxxxxTR\_[X,Y,Z].THD*, *xxxxxR\_[XX,YY,ZZ].THD*) for the nodes in the shear walls and springs. The *modelname* is the name of the active model. The *.eql* file is written from the UI. The NONLINEAR module reads the corresponding *.hou*, *.sit*, *.eql*, and corresponding *.THD* files to generate several output files and a new updated *.hou* file that can be re-run for the next iteration.

The NONLINEAR module generates a *Panelxxxx.crv*, *Panelxxxx.thd*, *Panelxxxx\_AXIAL.thd*, *Panelxxxx.ths*, *Panel\_EQL\_Matl\_Prop.txt*, and *Panel.fmu* files as outputs for the reinforced concrete shearwall panels, and *SPRINGxxxx.crv*, *SPRINGxxxx.thd*, *SPRINGxxxx.ths*, *SPRING\_EQL\_Matl\_Prop.txt*, and *Spring.fmu* files as outputs for the nonlinear spring elements. As well as a general output text file which is named by the user and contains nearly all the input and output data from the program.

The NONLINEAR module provides a number of useful text files that are saved after each SSI iteration. The *Panelxxxx.crv* file contains the equivalent-linear elastic modulus-damping curves for each nonlinear wall panel, as function of the panel strain amplitude after the rigid body motions are extracted from the panel node displacement results. The *Panelxxxx.crv* are computed during the initial run.

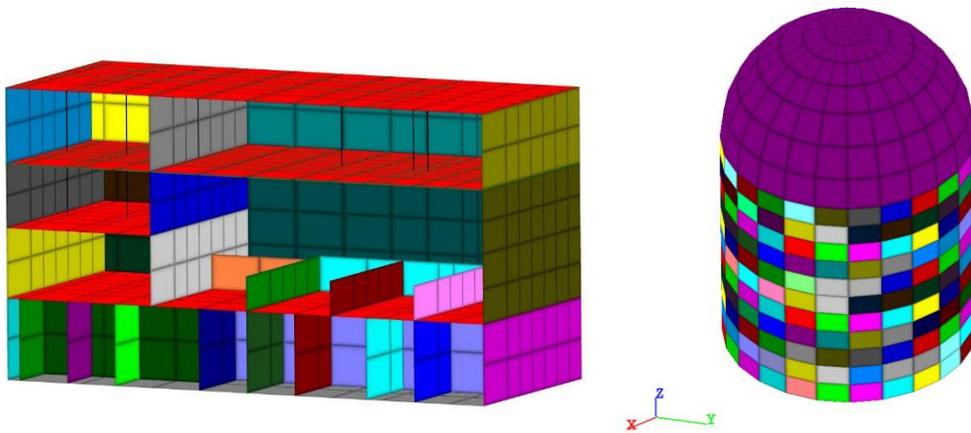
The *Panelxxxx.thd* and *Panelxxxx.ths* files contain the panel equivalent-linear strain and the nonlinear force time histories, respectively. These two files include the needed information to plot the panel hysteretic loops.

The *Panel\_EQL\_Matl\_Prop.txt* file contains the new material properties of the shear walls. The *Panel.fmu* file contains the ductility ( $\mu$ ) and force reduction factors ( $F\mu$ ) of the shearwalls. The ductility value of the shearwalls is found by dividing the maximum value of nonlinear displacement by the cracking displacement (not the yielding displacement that is not obvious to define for many realistic BBC shapes), or the displacement point on the BBC where the elastic region ends. The force reduction factor is found by dividing the initial linear elastic force by the final nonlinear force.

The NONLINEAR module provides similar files for the SPRING output files that use instead of “panel” the “spring” name.

**WARNING:** It should be noted that that for the initial NONLINEAR run for elastic model, the text file PANEL.NON or SPRING.NON should not exist in the working directory. If the the .NON extension file exists and it contains the number 1, then, NONLINEAR will use the previous iteration SSI results and run a nonlinear analysis for a new SSI iteration. If .NON is not existent, then the NONLINEAR module will run a linear elastic analysis and will create the .NON file with number 1 included.

**WARNING:** The PANEL\_EQL\_MATL\_PROP.txt file or SPRING\_EQL\_MATL\_PROP.txt file exist in the working directory, then, the NONLINEAR module will run the next SSI iteration using the equivalent-linear stiffness and damping values included in this file. If the user desires to run the initial elastic analysis then both the .NON file and the PANEL\_EQL\_MATL\_PROP.txt or the SPRING\_EQL\_MATL\_PROP.txt should be deleted.



Nonlinear Shearwall Building Model (left) and Reinforced Concrete Containment Model (right)

Before the nonlinear SSI analysis, an important preliminary step is to divide the structure in nonlinear wall panels as shown in the above figure. To help the users for this step, and make it almost automatic, the UI has a number commands such as MERGEPANEL, WALLFLR, PANELIZE, MERGEGROUP, and SPLITGROUP, EDGEMODEL, EDGE, UNIPNL. An explanation of all these commands can be found in the command reference section of this manual. An example for using these commands is provided in Demo 9.

Two application examples of nonlinear structure models are shown above for a low-rise shearwall building (left) including 40 nonlinear wall panels (front wall is not shown) and a reinforced concrete containment (only cylinder part is nonlinear). Both structures are considered low-rise with governing lateral deformation governed by the shear deformation.

It should be noted that for concrete curved walls as the containment perimeter wall, the nonlinear wall panel mesh should be sufficiently refined mesh, so that the angle between to wall panels is less than 15 degree angle. In the case shown below for the containment, each nonlinear wall

---

panel is a single shell element. To create the nonlinear containment model from a one-material elastic model, the UNIPNL command can be used. This command splits the model in panels with one shell element for each panel. It should be noted that the experimental test information is very minimal. Therefore, the elementary wall panel shear capacities are more difficult to define. However, if pushover test data or analysis data exist that describes the nonlinear containment system static behavior under increasing lateral load, then, this information can be used to adjust the elementary wall panel BBCs to reflect the overall nonlinear behavior of the containment structure.

Demo 12 shows an example of a pushover analysis for typical RC containment structure based on the test and analysis results obtained at the DOE Sandia National Labs (SNL) as described in NUREG/CR-6783. The demo describes how to use ACS SASSI with Option NON to perform the pushover analysis of the containment structure, and then, based on the obtained analysis results, how to simply adjust the elementary wall panel BBCs to better match the SNL test results of the overall nonlinear containment structure behavior.

Given the strain/relative displacement time-history of a nonlinear element, wall panel or spring, the equivalent-linear strain/relative displacement is found by multiplying the maximum strain/relative displacement by a reduction factor. For typical random seismic input histories, for the best fitting of the true nonlinear hysteretic responses, this reduction ratio is around 0.80, or somewhere in the 0.7-0.9 range. In general, the use of a value of 1.0 produces too soft elements, and a value of 0.6 too stiff elements. Then, the equivalent-linear strain/displacement values can be used to find the linear-equivalent elastic modulus and damping that will be used in next SSI iteration. The new material properties are written in the new HOUSE input file.

### **Global Modeling Options:**

This section includes general input data information using the UI window for performing the nonlinear structure SSI analysis using the NONLINEAR module.

It should be noted that for practical problems that typically involve a larger number of wall panels or nonlinear springs it is more convenient to use different UI commands rather than the UI interactive input for preparing the NONLINEAR module input file (.eql file). Useful UI commands include the EQL, P, S, BBCX and BBCY commands. The user is also suggested to understand and use the newer PANELGEN, EDGEMODEL, SHEAR, and BBCGEN commands that are much more efficient when dealing with complex nuclear island models having thousands of concrete wall panels with nonlinear behavior.

**Disp. Factor** – Equivalent-Linear Displacement Factor (EDF). It has typical values between 0.7-0.9. For most of applications, the best-fitting value appears to be close 0.80. This factor determines the equivalent-linear strain/displacement that is used to determine the new iterated material properties

---

**Damping Cutoff** – User specified percentage value of the selected damping cut-off. The cut-off damping value can be used to be in accordance with the new ASCE 4 standard and USNRC damping limitation requirements based on the seismic response levels obtained by linearized SSI analysis: for *Level 1* use uncracked concrete stiffness and 4% damping; for *Level 2*, use 0.50 of the uncracked shear/bending stiffness and 7% damping; for *Level 3* use 0.50 of the uncracked shear/bending stiffness and 7% damping, Please see new ASCE 4-17 Section C.3.3.2 for more details.

**Damping Scale Factor** – User specified Scale factor for damping. This scale factor can be to adjust/calibrate the damping value based on specific test data.

**Material Parameter** – (disabled in this version).

**Include Elastic Damping** – This selection implies that the total damping is obtained by adding the calculated hysteretic damping at each SSI iteration with the initial elastic viscous damping. If not selected only hysteretic damping is considered. If only the hysteretic damping is considered, this should be not smaller than the initial viscous-elastic damping.

**Use Nonlinear (Element) Options** – Select nonlinear element types to be used, wall panels,. This is controlled by the check boxes in panel.

#### **Backbone Curve Data:**

This section includes inputs for defining the backbone curves (BBC). Please see also the SHEAR and BBCGEN commands to efficiently generate the BBCs for many wall panels.

**Backbone Curve** - BBC number being displayed

**Type** – The BBC type for hysteretic model (used only for hysteretic model titles in the printout):

- 1 - Cheng-Mertz Shear (CMS) hysteretic model for shear deformation
- 2 - Cheng-Mertz Bending (CMB) hysteretic model for bending deformation
- 3 - Takeda (TAK) hysteretic model for both bending and shear deformation

**Yield Number** – Order number of the curve point that is the yield point withing the BBC data points. .

**X and Y** – This input is to define the BBC data points. From this panel the user can view and edit backbone curves. This window is not capable of adding new or removing data points from the curve. Curve size and curve points must be initially defined using the BBC commands, however the user can modify a point on the BBC curve from this box.

---

**Panel Data:** - This section is only for wall panel data. Please also see the P and PANELGEN commands that helps the user to deal with complex FE models with many wall panels.

**Panel** – Wall panel number in the nonlinear FE model

**Group Num.** - Corresponding group number for the wall panel

**BBC Num.** - Define which BBC is associated with the wall panel

**Disp. Opt.** - Displacement calculation option is:

1 – Shear-strain in wall panel (based on the average of the relative story drift for the two lateral edges after removing rigid body motion). See Figure 1.3.

2 – Bending section rotation of wall panel (based on the relative cross-sectional strain rotation of the top and bottom edges). See Figure 1.3.

**Force Opt.** - This force calculation option is to define which of the hysteretic model is used to calculate nonlinear force for the active panel. It is the same as the BBC type.

- 1 - Cheng-Mertz Shear (CMS) hysteretic model for shear deformation
- 2 - Cheng-Mertz Bending (CMB) hysteretic model for bending deformation
- 3 – Takeda (TAK) hysteretic model for both bending and shear deformation

**WARNING:** *In this version, the Takeda model can be only used with an equivalent damping equal to the constant elastic damping.*

**WARNING:** *The first point that is in the list is assumed to be the cracking displacement and force. Thus the zero point on the curve is omitted in the list. Also the yield point needs to be defined. The yield point force/moment is used in the Takeda model for the wall panel peak capacity. Thus, a lower yield point will produce a lower panel capacity.*

**WARNING:** *The user must ensure proper units are used for the backbone curves and panel options. For example, if the user chooses displacement option 1 (i.e. shear strain calculation) the shear BBC used must have units of the shear strain - force (decimal-force). For the displacement option 2 for the bending BBC, the panel edge relative strain rotation or curvature in the vertical plane is needed, that have units of the section strain rotation/curvature - moment (radian/length-moment). Essentially ACS SASSI does not change any units of the BBC or displacements based on the user inputs and assumes the user has input the appropriate properties, options, and units. The BBC units should be consistent with elastic stiffness. For shear BBCs, the elastic stiffness units is based on the GAShear product units, while for bending BBCs, the elastic stiffness is based on the EI product units.*

**Spring Data** - This section is only for wall spring data

**Spring** – Spring Number

**Group Num.** - Corresponding group number for the nonlinear spring

**Elem. Num.** - Corresponding element number for the nonlinear spring

**BBC Num.** - Option to define which BBC is used for active spring

**Disp. Type** – Option to define the active spring dofs

- 1 – Translation in X direction
- 2 – Translation in Y direction
- 3 – Translation in Z direction

**Force Opt.** - This force calculation option is to define which of the hysteretic model is used to calculate nonlinear force for the active panel. This is the same as the BBC type.

- 1 – Not applicable for Cheng-Mertz Shear (CMS) hysteretic model
- 2 – Not applicable for Cheng-Mertz Bending (CMB) hysteretic model
- 3 – Not applicable for Takeda (TAK) hysteretic model
- 4 - General Massing Rule (GMR) hysteretic model

#### **Beam Data - NOT AVAILABLE IN THIS VERSION**

**Beam** – Beam Number

**Group Num.** - Group of the nonlinear Beam

**Spring Gr.** - Spring group for the beam ends

**BBC Num.** - Option to define which BBC is associated with the active beam

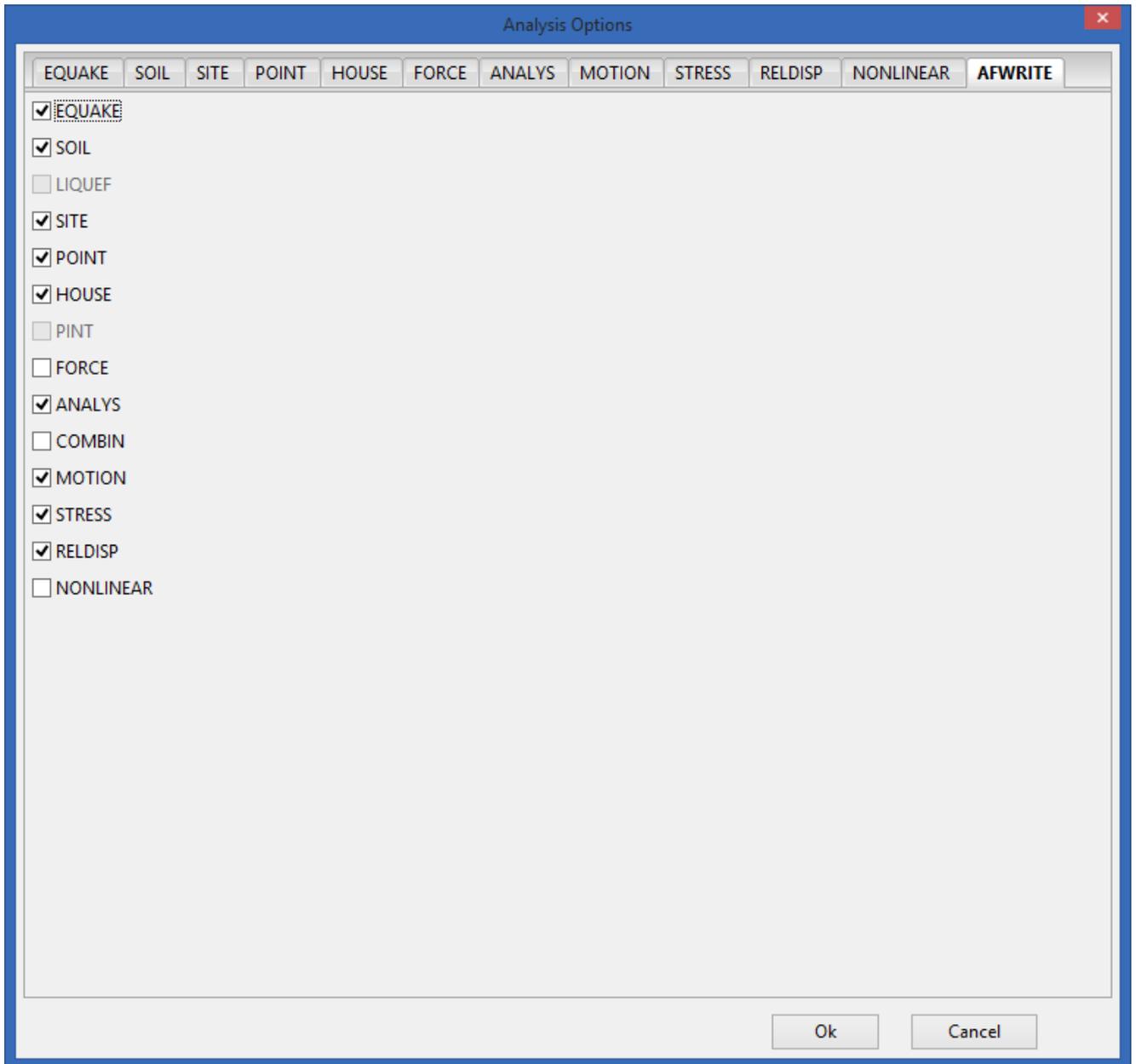
**Force Opt.** - This force calculation option is to define which of the hysteretic model is used to calculate nonlinear force for the active panel. Typical is the same as the BBC type.

- 1 - Cheng-Mertz Shear (CMS) hysteretic model for low-rise shearwalls (not applicable)
- 2 - Cheng-Mertz Bending (CMB) hysteretic model for low-rise shearwalls (not applicable)
- 3 – Takeda (TAK) hysteretic model for bending and shear deformation (not applicable)
- 4 - General Massing Rule Hysteresis Model

**Beam End 1** – Element that defines the node at the end of the beam

**Beam End 2** – Element that defines the node at the other end of the beam

### ***AFWRITE and CHECK Options***



Use this dialog box to enable / disable each of the ACS SASSI modules for the AFWRITE and CHECK instructions. The generated input files should be placed in the active model's directory

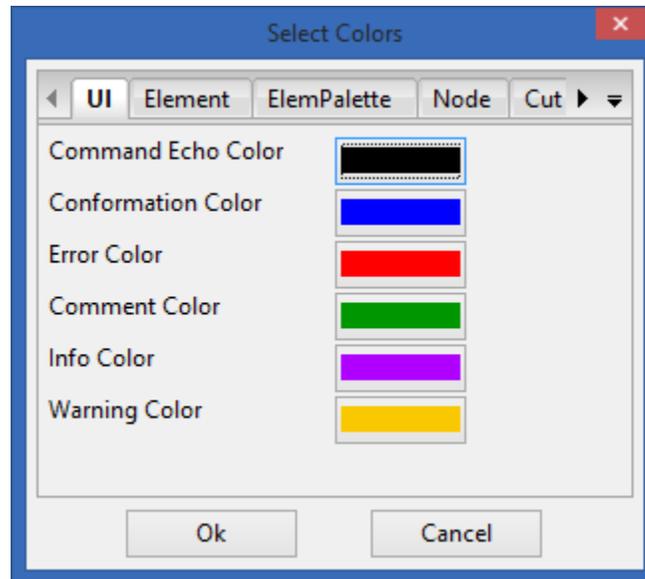
and their names will begin with the model's name followed by the corresponding postfix and extension as set from the UI module.

### 6.5.5 Windows Settings

This menu item will bring up the window setting for the active plot. These windows are shown and described in the plotting reference.

### 6.5.6 Colors

This window brings up a color pallet that will allow the user to see and modify the colors of many of the graphs and text throughout the UI module. This window is broken into tab table based on graph or window. These options were in UI however there were spread across multiple windows and the user could only modify the color options for the active window.



Once the window is open the user can select the Tab to change a color, the user can click on the color button next to the color name. When the button is clicked the user will see a standard color selection screen which is then used to choose the new color. Once the user selects the new color the selection window will close and the color button in the pallet will change.

**UI** – Change the colors of the Command history text. Each type of text in the command history has a code that can be represented by a different color

**Element** – Change the colors of the element plot except for the element colors. The element colors are controlled by the element palette.

**ElemPalette** – Change the colors of the elements in the element plot. This palette will be populated as the user plots elements with different groups, materials or properties. The element Palette can have up to 128 colors. This single set of colors is used to show the groups, materials

or beam/spring properties. If a model has more than 128 groups, materials or properties the colors will be assigned by the equation ( $Elemnum \equiv Colnum(mod128)$ ).

**Node** – Change the colors on the node plot

**Cut** – Change the colors of the cut plot

**Bubble** – Change the colors of the bubble plot.

**Vector** – Change the colors of the vector plot.

**Contour** – Change the colors of the contour plot.

**Deformed** – Change the colors of the deformed shape plot.

**Spec** – Change the colors of the spectrum plot axis. Spectrum axis line colors are controlled by the SpecLines color palette.

**SpecLines** – Change the color of the spectrum lines. This palette will be populated as lines with different line numbers are plotted.

**TimeHist** – Change the colors of the time history plot axis. Time history line colors are controlled by the THLines palette.

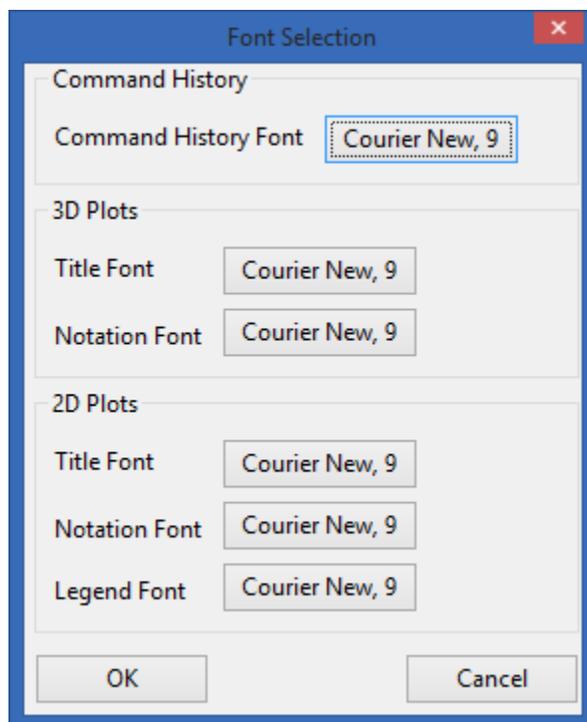
**THLines** – Change the color of the time history lines. This palette will be populated as lines with different line numbers are plotted.

**SoilLayer** – Change the colors of the Soil Layers plot.

**SoilProp** – Change the colors of the Soil Properties plot.

### 6.5.7 Font

This Menu Option allows the user to change the fonts on the plots, command history and the command entry window. When the user selects this option a window will appear like the one shown below. The user can click on any of the font description buttons to bring up a platform dependent Font Change Dialog window to select a new font. Once the user has selected the desired font the user can then click OK button to apply the fonts to the UI.



The 2D and 3D fonts are applied to all 2D and 3D plots in the UI including the plots that the user currently has open.

### 6.5.8 Shader Options

This item will open the Shader Options window to set the following options:

- max point size in node/bubble plots
- adjust outline width of soli
- adjust amount of shrink in element plot
- adjust the scale factor for animations

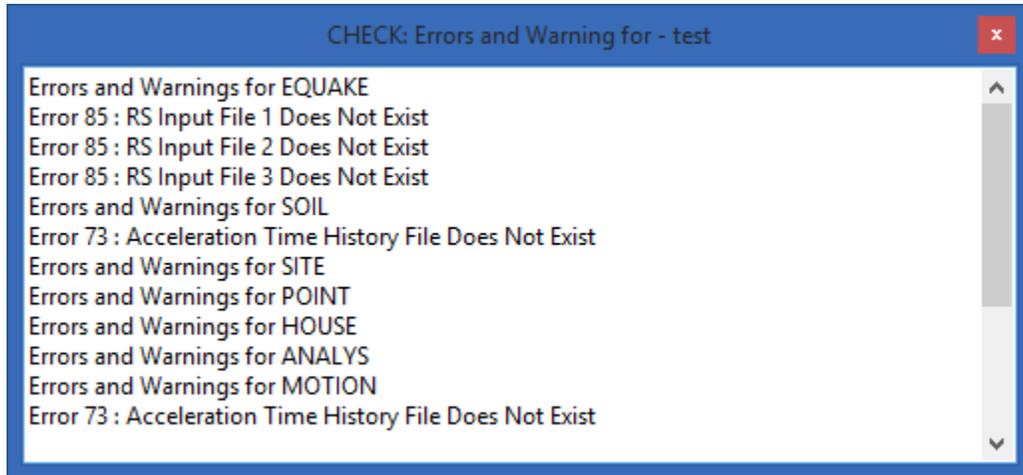
### 6.5.9 Reset Plot

This item will reset the 3d plot view to its default view and zoom.

## 6.6 View Submenu

### 6.6.1 Check Errors

This Menu option will open the error listing window for the active model. This window reads the .err file produced by CHECK and displays the contents of the file in the window. If the active model does not have a name and path or has not been checked the program will not find the proper .err file and the Check window will either be blank or a file not found error message box will be shown.

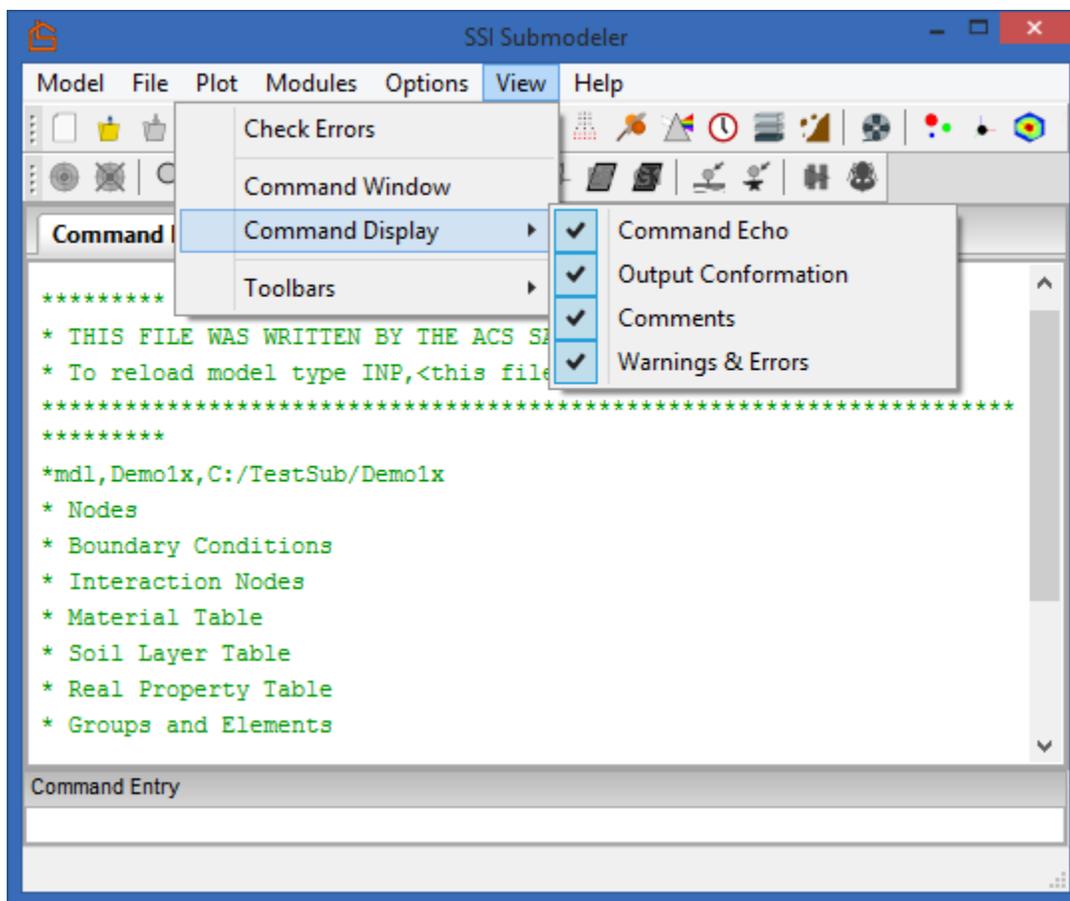


### 6.6.2 Command Window

This menu option will open the command window or make the command window tab active to show the command window. When the command history is reopened the command history output will be cleared. The clearing of the command history window does not affect any models in memory.

### 6.6.3 Command Display

The updates to the UI makes it easy for a user to copy and paste data out of a command history window into a file for a custom .pre. A problem with copying and pasting is that a user would have to copy and paste around the error and command acceptance messages. The command display options allow the user to turn off the echoes to the screen of 5 out of 6 types of messages that show up in the command history (The information type [Default Color: Purple] cannot be turned off)

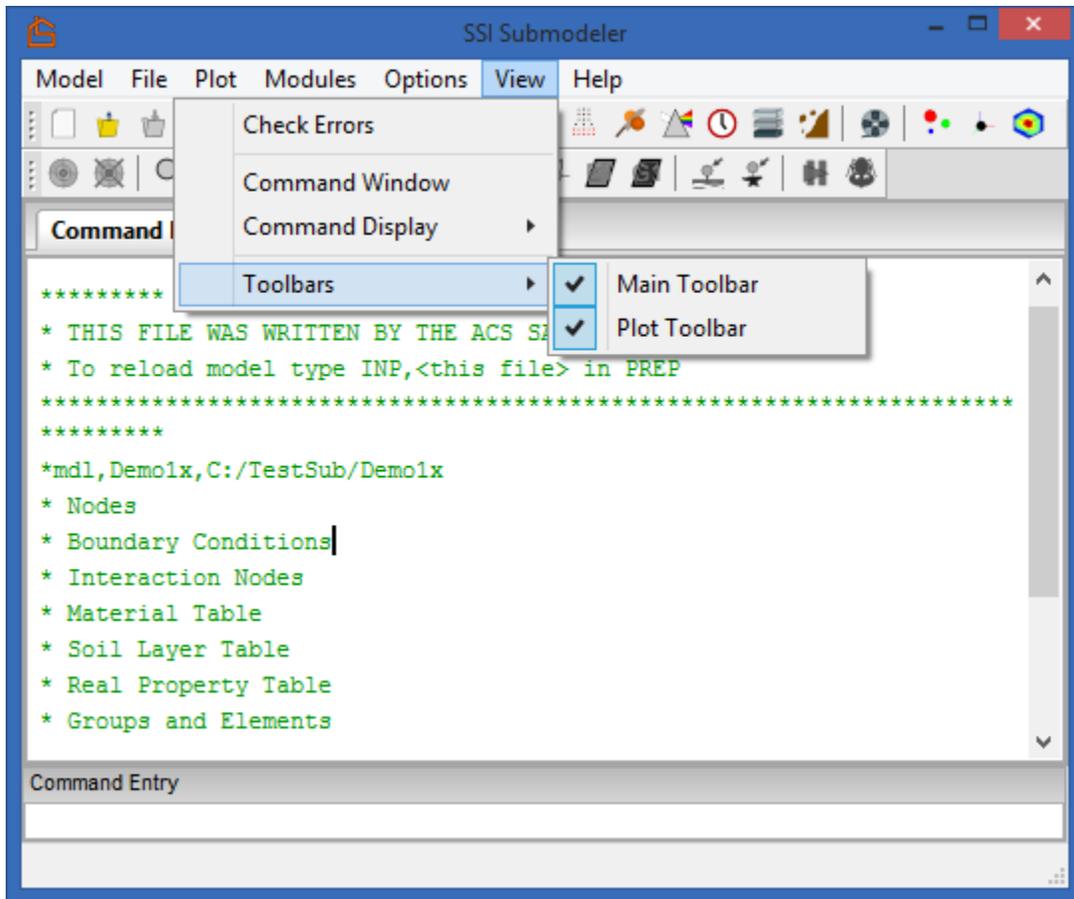


When the user first opens the UI on a user account all of the command display options in the menu will have a check mark next all of the message types. If the user clicks on a menu display option that will disable/enable that type of command message. The user selections for Command Display options are stored in SASSlini.xml

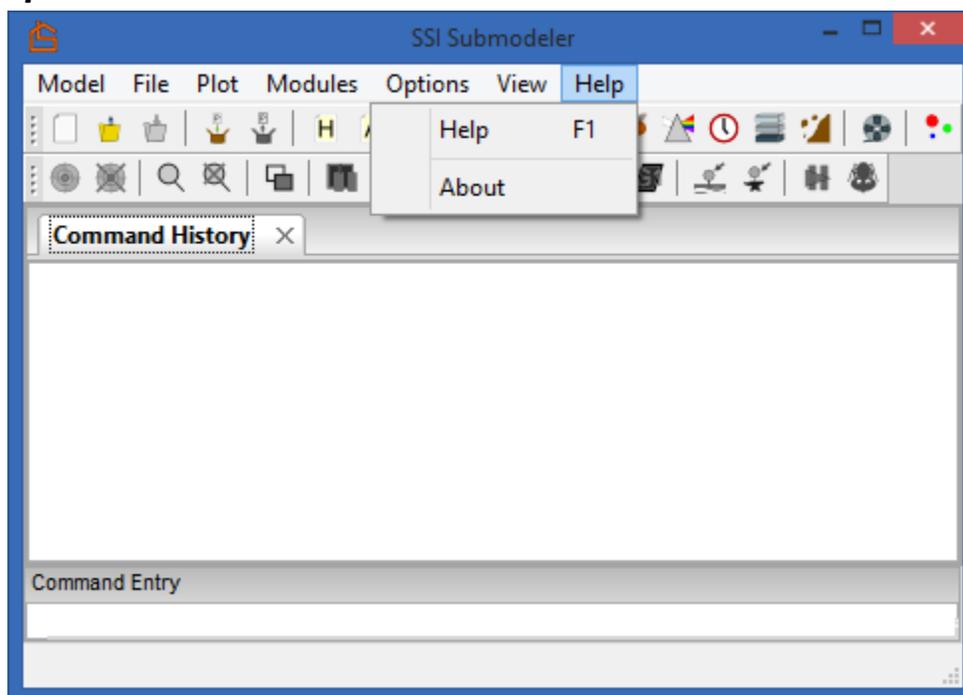
When inputting a .pre file the command echo and command acceptance are suppressed. This cannot be changed using the command display options. The error and comment output can be modified during .pre file input using these options. Forcing suppression of command echo and command acceptance during .pre input sped up the parsing of the .pre file dramatically while the volume of information made the output from the .pre file useless.

#### 6.6.4 Toolbars

This Submenu option allows the user to hide either of the toolbars. If the toolbar is checked the toolbar is visible otherwise it will be hidden. The default is for these toolbars to be shown. These selections currently are not save in the SASSlini.xml so the toolbar setting will return to the default when UI is initially opened.



## 6.7 Help Submenu



### 6.7.1 Help

The SASSI Help menu option will direct the online help Overview page. Help was switched to the wiki format due to the history of proprietary help file formats not being supported in future operating systems that ACS SASSI may use.

### 6.7.2 About Box

The about box will show the current version number of the UI along with the release date and some other information.

---

# 7 Plotting Operations and Examples

## 7.1 *Three-dimensional Plots*

### 7.1.1 3D Plot Controls

All of the 3D plots feature a set of mouse controls mouse and keyboard controls that allow the user to manipulate the model based on dragging and clicking the mouse in certain instances. These controls will work for all of the model plots and animations. There are also Plotting Commands and plot toolbar which are discussed in Section 7 and Section 8.2.

#### ***Mouse Controls***

Right Click and Drag – Rotate Model around the center

Left Click and Drag (Vertically) – Zoom Model in and out

Middle Click and Drag – Move model across the plot area

Shift+Middle Click & Drag – Create a Zoom Box area. Zoom to area on middle mouse button release

Left Double Click – Highlight node in plot and Identifies node number in the Command history

Right Double Click – Highlight element in plot and Identifies element number in the Command history

#### ***Keyboard Commands***

Insert/Delete – Rotate model around X axis

Home/End – Rotate model around Y axis

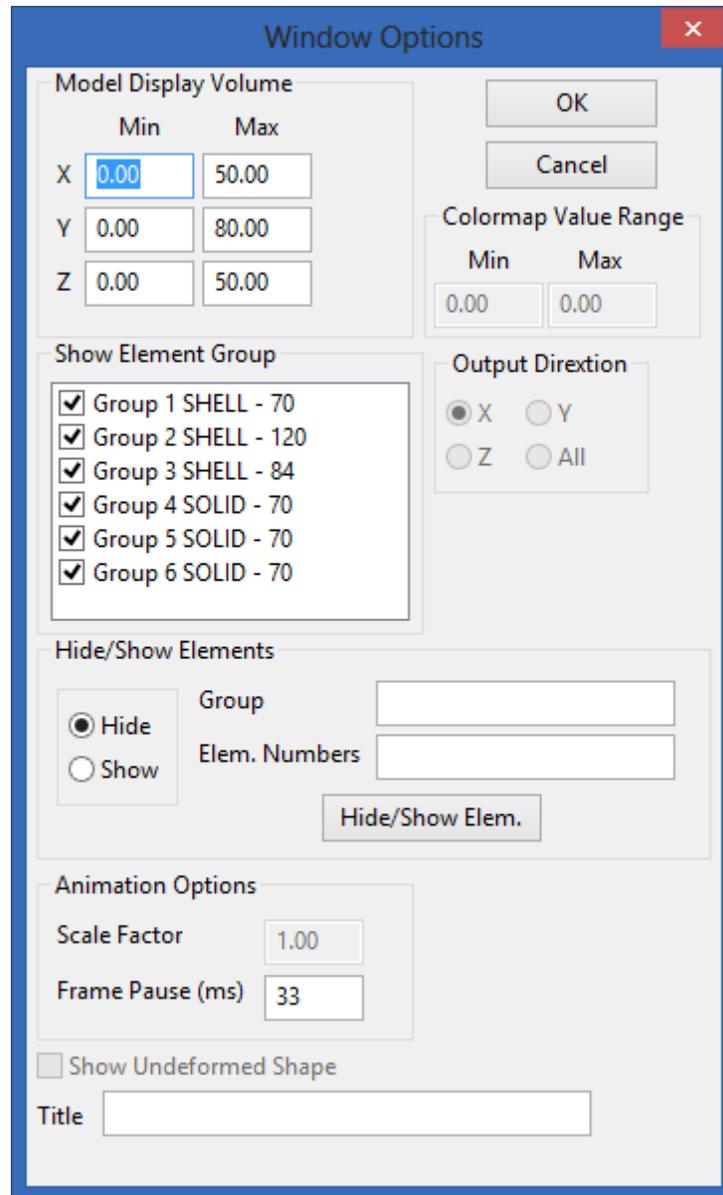
PageUp/PageDown – Rotate model around the Z Axis

Pause - Start Stop Animation

+/- - Increment/Decrement Frames (Note: Should only be used while animation is paused)

#### ***Windows Settings***

This is the UI “Windows Option” window will be shown when any 3D plot is the current active plot. Unlike the previous version PREP module user request to hide groups/elements/nodes are stored in the model data so if the user will not have to copy the hide request for each plot



**Model Display Volume** – The user can view part of the model by defining the minimum and maximum of the view volume. When redefined the program will hide any geometry outside of the view volume. By default the volume will always be set to a box that surrounds the model

**Show Element Group** – Allow the user to hide or show groups from a list of groups in the model. The user can check or uncheck groups to show or hide. If the user wants to select/deselect multiple groups the user can highlight the groups to be changed and then press the spacebar.

**Colormap Value Range** (Bubble and Contour only) – Change the range of values that the colormap represents by modifying the values in the edit boxes. If a color falls outside of the user defined range the color shown on the plot for that value will be the min color (Dark blue) for smaller

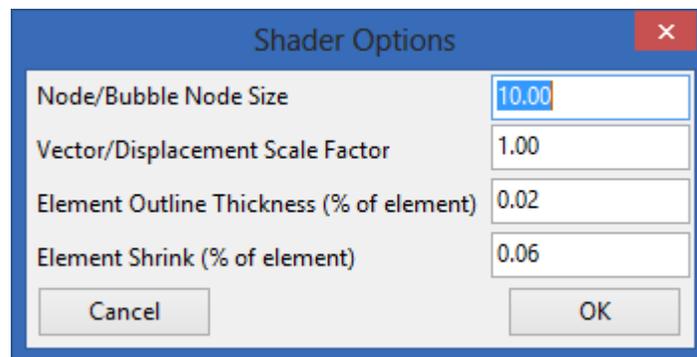
values or the max color (Dark Red) for larger values. Output Direction (Vector only) – Show vectors in the selected direction or all directions at once.

**Hide/Show Elements/Nodes** - The user can hide or show elements or node (depending on the plot) by clicking the Hide/Show button after the modifying the edit boxes and radio buttons in this section. The user can enter in a list of disjointed node or element numbers with a space divided list. Also if the user has a continuously numbered range of nodes/elements using a dash “-” between the first number and the last number in the block will hide all nodes/elements in the range. The user can Hide/Show as many ranges or list as desired by using pushing the Hide/Show Button multiple times but the data in the Element/Node Number box can only contain a list or a single range when the button is pressed. When hiding elements only one group can be entered in the group edit box at a time. Only Once the Hide/Show button is pressed those elements/nodes will be hidden/shown even if the user does not click OK on the popup window.

**Animation Options** – Allows the user to change the scale factor for the vector and deformed shape plots and the refresh rate for all graphs by changing frame pause for all graphs(the non animated graphs as well)

**Show Undeformed Shape** (Deformed Shape Plot Only)– Show the wire frame of the original Unreformed model with Deformed model when checked. Title - Add a title to the animation plot.

### **Shader Options**



The Shader Options screen provides additional plot option for 3D plots. These options are used to change variables in the OpenGL shaders that control the plot display on the computer. These options were placed in this window instead of the Windows Settings to keep the plot options window more consistent with the previous version PREP module of the window. When OK is pressed the current active plot will be updated with the values in the Shader Options Window.

The Node Bubble size controls the maximum node or bubble size of the plot for node bubble and vector plots.

The vector/displacement scale factor control the scale factor for those two plot types

Element outline thinness is the percentage of the element edge that is the outline color instead of the element color for Element, Cut, Contour, and Deformed Shape plots.

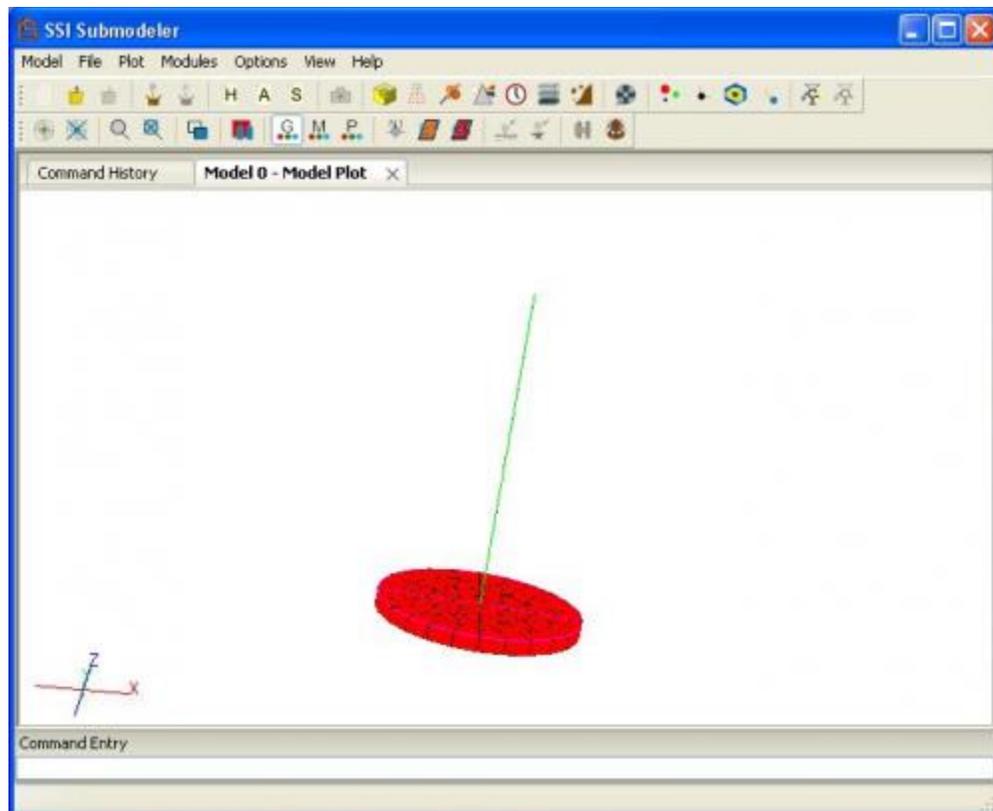
The element shrink controls the amount of face shrink for the Element plot

### ***Plot Toolbar***

See the toolbar reference in section 8.2. Different plotting commands can be executed from the toolbar, or from the menu selections, or from the command line.

### **7.1.2 Element Plot**

When this option is selected the user will see a plot of the active model with all of the element shaded in different colors based on either group, material or property. This plot can be modified by mouse click commands, command entry and, the plot toolbar. This Plot is intended to allow the user to view the model geometry in a way to identify where the elements are in relation to one another check connections between elements, and correct problems with the model in a visual way.

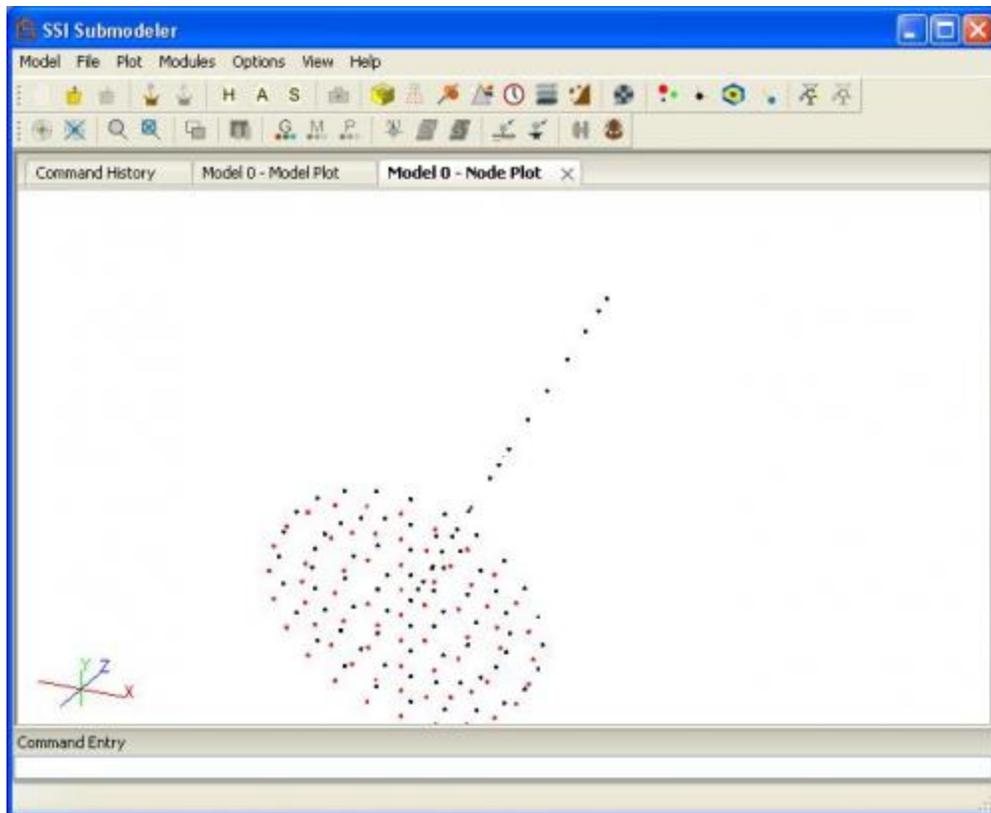


### **Command**

Selecting this menu option is the same as the MODEL PLOT command. Many of the 3D Plotting commands will be able to modify the plot from the command line or allow the user to do batch image manipulation using a .pre File

#### **7.1.3 Node Plot**

When this option is selected the user will see a plot of the active model similar to that of the Model element plot however the user will only see nodes that are connected to elements in the model. The plot shows all of the non-interaction nodes in black while the interaction nodes are in red. When the user select show fixed degrees of freedom a green border will surround each node that has the fixed degree of freedom that the user requested. When the user selects the Show mass option the node will be surrounded by a purple border. If the user selects a node by using the mouse selection or the NODESEL command the node the user selected will be surrounded by a blue square border. All of the colors described in this section are the default colors and can be changed by using the Colors menu option or the COLOR command

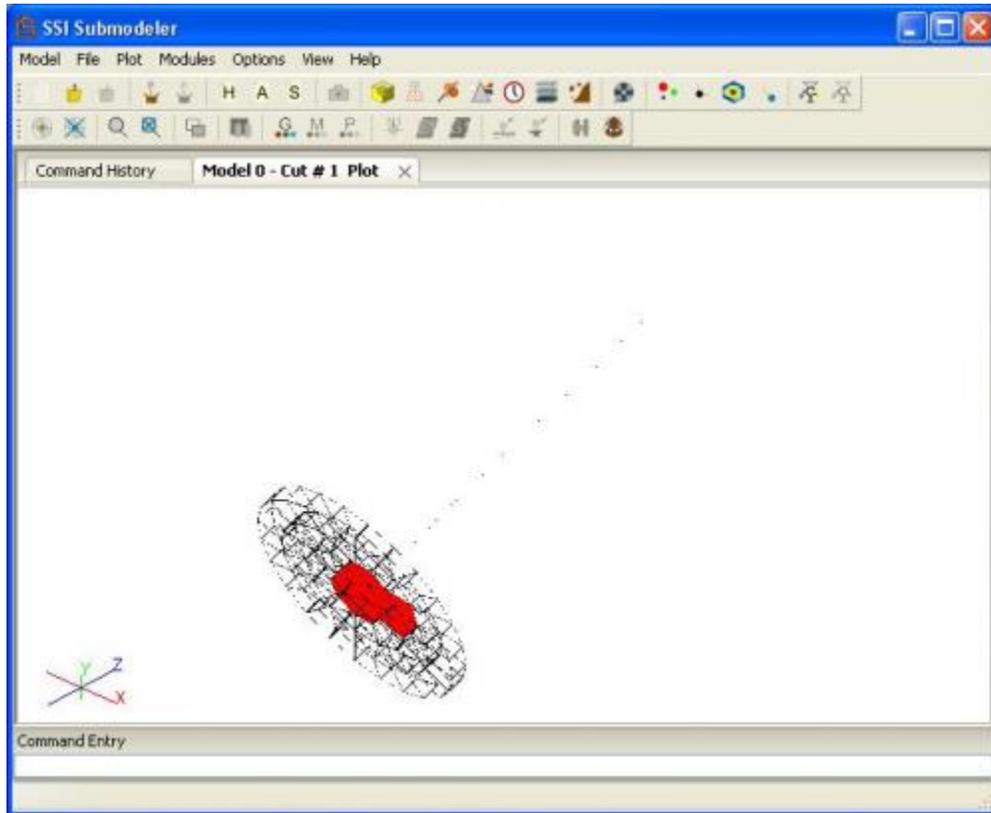


### **Command**

Selecting this menu option is the same as the NODEPLOT command. Many of the 3D Plotting commands will be able to modify the plot from the command line or allow the user to do batch image manipulation using a .pre file.

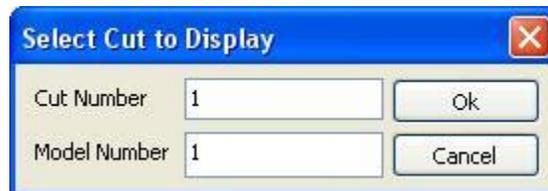
### 7.1.4 Cut Plot

This Plot will show the user a wireframe of the entire model with the elements of the cut filled in. This Plot is intended to allow the user to view the cut for completeness and modify what is in the cut before the user creates a Submodel or a cross sectional model from the cut.



#### Opening

When opening the cut plot the user must provide the cut number and the model number of model to be plotted. (**Note:** the cut plot is the only plot type that can plot a model that is not the current active model) When using the menu option or the CUTPLOT command with no arguments a popup window will appear asking the user to provide a cut and model number. Once OK is pressed the user will see the requested cut plot.



**Command**

Selecting this menu option is the same as the CUTPLOT command. Many of the 3D Plotting Commands will be able to modify the plot from the command line or allow the user to do batch image manipulation using a .pre file.

## 7.2 Two-dimensional Plots

### 7.2.1 Opening Spectrum & Time History Plot

The user can then load lines into UI memory from files or select lines currently in memory and plot those lines. To load lines into memory the user will need input a file name into the Input Line File section of the window by typing or browsing the file system. Once the user has a file selected the user must then enter the number of lines to be loaded from the file and the starting line reference number.

The starting line number is used to keep track of lines in memory much like the model reference number is used to keep track of models. (**Note:** If the user wants to load multiple lines from a single file there is no way for the user to choose which columns to load. The user must load all of the line between the first line in the file and the last required line for the plot. All extra lines added will have a reference number incremented from the starting line number). (**Note:** Loading a line into a reference number that already has a line in it overwrites the line currently in memory) Once this information is entered the user can press the add lines button which will tell the UI to read the file and store the lines in memory. At completion the user should see line information in the selection box at the top of the window.

The screenshot shows a dialog box titled "Line Selection". At the top, there is a list box containing one entry: "1: Test2.rs". To the right of this list box are two buttons: "Ok" and "Cancel". Below the list box, there are three sections for configuring the plot axes:

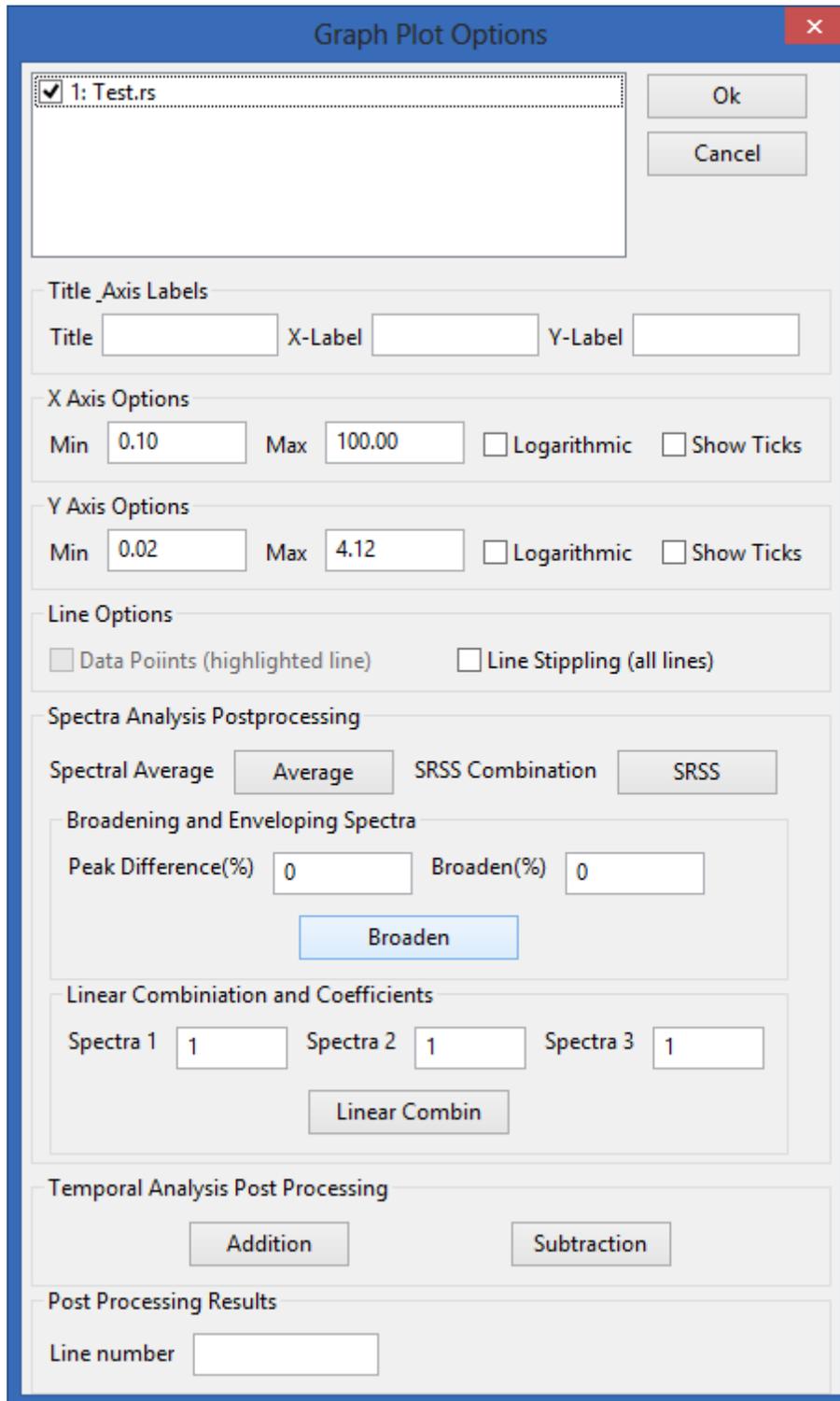
- Title Axis Labels:** Three text input fields labeled "Title", "X-Label", and "Y-Label".
- X Axis Options:** Two text input fields for "Min" and "Max", and two checkboxes labeled "Logarithmic" and "Show Ticks".
- Y Axis Options:** Two text input fields for "Min" and "Max", and two checkboxes labeled "Logarithmic" and "Show Ticks".

At the bottom of the dialog is the "Input Line File" section, which includes:

- A "File Name" text input field followed by a "<<" button.
- Two text input fields for "Starting Number" and "Lines in file", followed by an "Add Line(s)" button.

Once the user has all of the lines for plotting the user can then select the lines by the box at the top of the screen and checking the line needed for plot. At this time the user can also modify the graph properties using the data entry below the line selection box. The user can modify these properties after the graph has been plotted by using the Plotting commands, or by using the Windows Settings menu option. The user has the ability to set the plot titles, change the graphs minimum and maximum extent in the X and Y axis, and set the axis to logarithmic scale or show minor ticks.

## 7.2.2 Spectrum & Time History Options



The image shows a software dialog box titled "Graph Plot Options" with a close button (X) in the top right corner. The dialog is organized into several sections:

- Plot Selection:** A list box contains "1: Test.rs" with a checked checkbox. To the right are "Ok" and "Cancel" buttons.
- Title\_Axis Labels:** Three text input fields for "Title", "X-Label", and "Y-Label".
- X Axis Options:** "Min" (0.10) and "Max" (100.00) input fields, with checkboxes for "Logarithmic" and "Show Ticks".
- Y Axis Options:** "Min" (0.02) and "Max" (4.12) input fields, with checkboxes for "Logarithmic" and "Show Ticks".
- Line Options:** Checkboxes for "Data Points (highlighted line)" and "Line Stippling (all lines)".
- Spectra Analysis Postprocessing:** "Spectral Average" (Average) and "SRSS Combination" (SRSS) buttons.
- Broadening and Enveloping Spectra:** "Peak Difference(%)" (0) and "Broaden(%)" (0) input fields, with a "Broaden" button.
- Linear Combination and Coefficients:** "Spectra 1", "Spectra 2", and "Spectra 3" (all 1) input fields, with a "Linear Combin" button.
- Temporal Analysis Post Processing:** "Addition" and "Subtraction" buttons.
- Post Processing Results:** "Line number" input field.

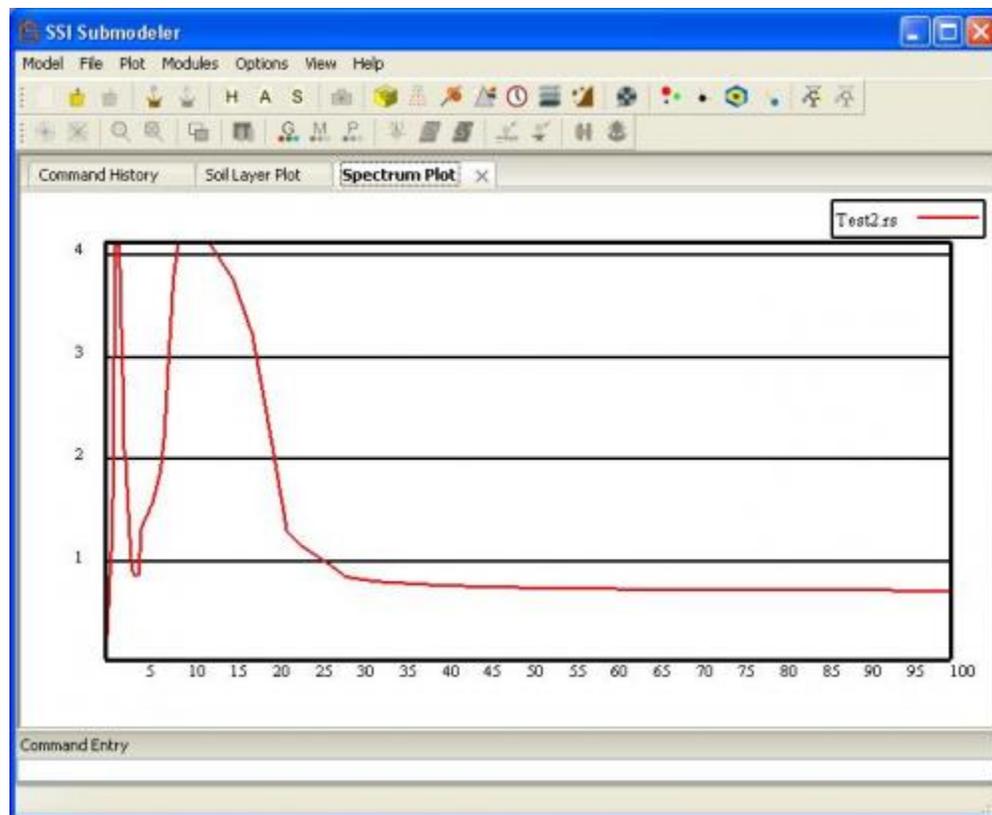
The Line Settings Window is used for both the Transfer function and Time history plots. This window allows the user to select lines that are in memory to display on the current active plot as well as change the scale and extent of active plot.

The user can also create new lines using the line calculation buttons in the setting window. These buttons use the same functionality as the buttons in UI window settings and the commands ADDITION, AVERAGE, BROADEN, LINECOMBIN, SRSS and SUBTRACTION. To create the new line the user must select lines in the line selection check box list to be processed. Then for Broaden or Linear Combination the Text boxes in the section with the button need to be modified to the appropriate value before the button is pressed (**Note:** the Linear Combination will only work for up to 3 lines using the window while the command can combine more that 3 lines). When the button is pressed, a new line will appear at the end of the selection list. The user can select this new line to be show on the current graph. The buttons do not write the line to a data file like the calculation buttons did in PREP. The user must use either the WRITESPEC, or WRITETH command to write the resultant of the calculation.

### 7.2.3 Spectrum Plot

This Plot allows the user to show plots that use the Spectrum file format.

In UI the graphs for Spectrum, Impedance and TFU-TFI Curves all used the same backing axis, but required modifications to the file reading code and line display that made it impossible to combine the options without rewriting the way that the spectrum plot worked. Now to show these types of graphs in UI the user can use the Plotting Commands to modify the look of the graph to obtain each of these types of graphs.

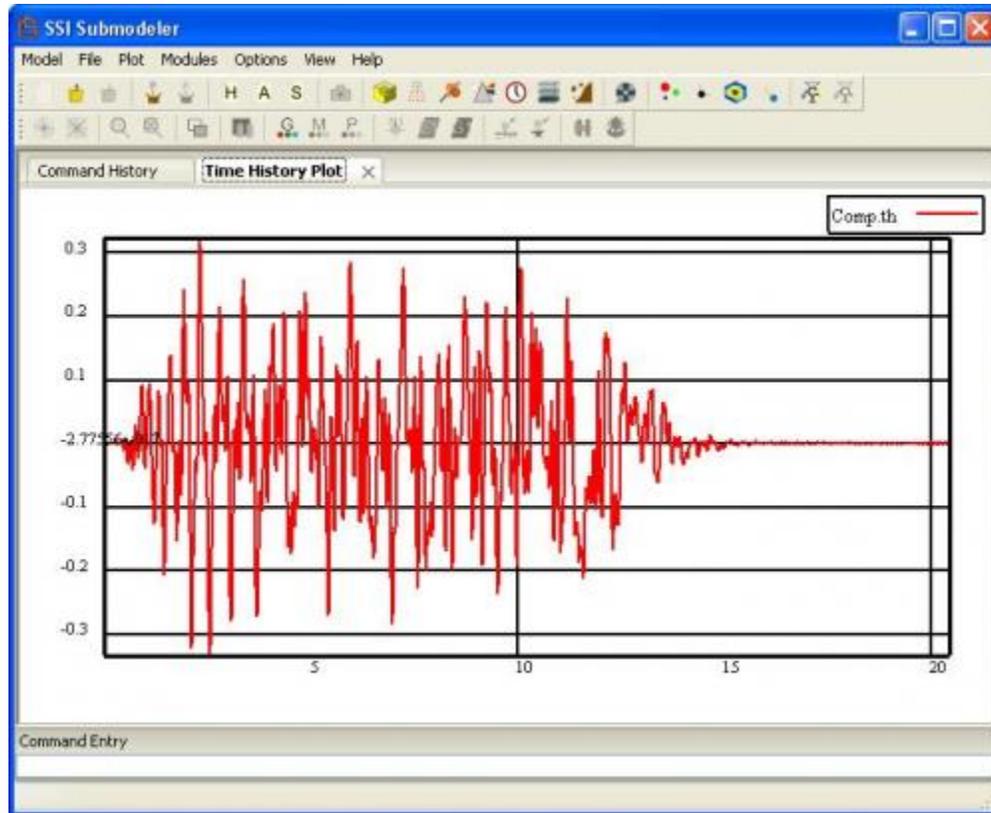


### **Command**

Selecting this menu option is the same as the SPEC PLOT command. Many of the 2D Plotting Commands will be able to modify the plot from the command line or allow the user to do batch image manipulation using a .pre file.

### **7.2.4 Time History Plot**

The Time History plot option is intended to allow the user to load and show lines in the Time History file format. The time history plot looks different from the previous version PREP time history plot because backing Axis for the Time History plot is now the same axis as the Spectrum plot.

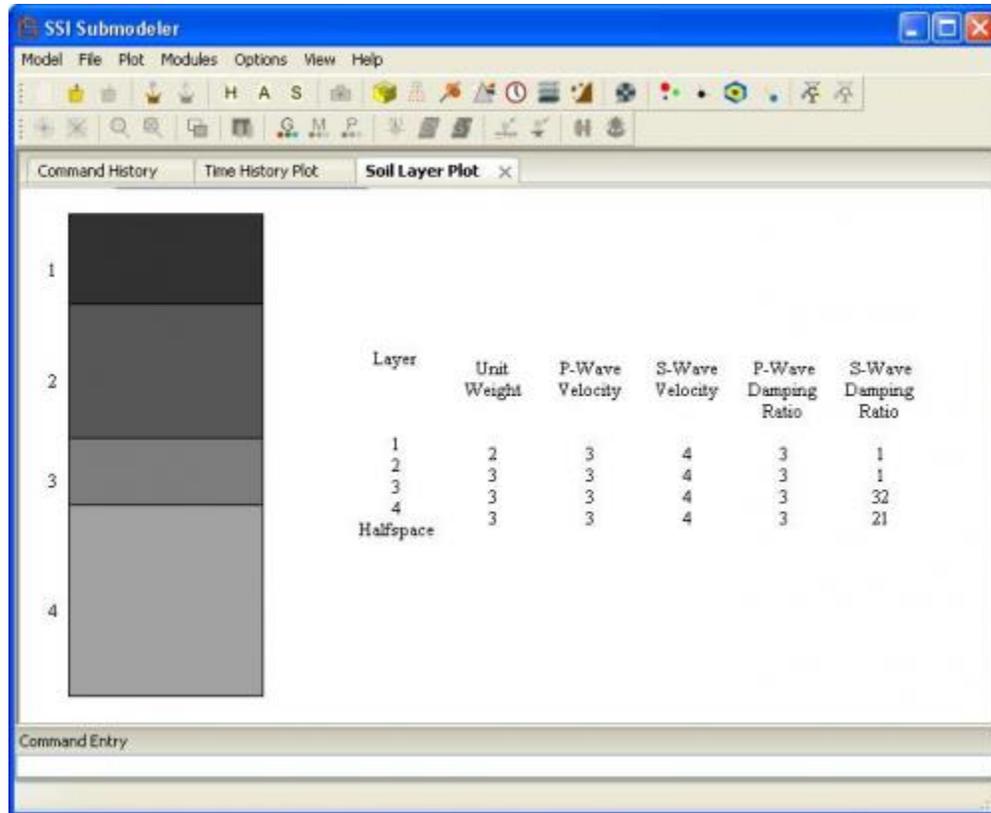


### **Command**

Selecting this menu option is the same as the THPLOT command. Many of the 2D Plotting Commands will be able to modify the plot from the command line or allow the user to do batch image manipulation using a .pre file.

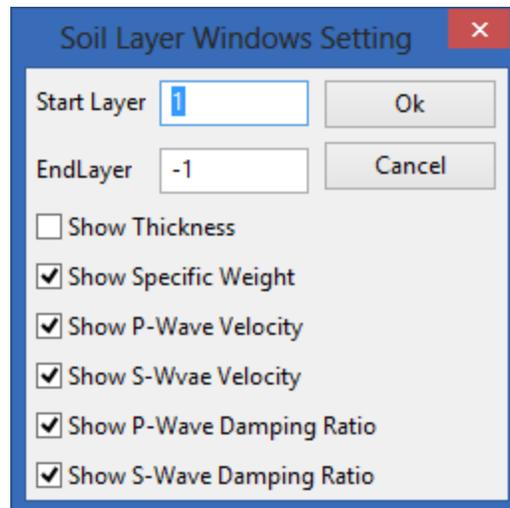
### **7.2.5 Soil Layer Plot**

This plot will show a graph that depicts the models layer information. The graph depicts a cross-section of the layers with the relative thickness as well as information about each layer depicted. This information is set by the L command and the TOPL command. The user can change the color of these layers by using the Options>Colors Menu and changing the Soil Layers Palette.



### Controls

The window options for the Soil Properties allow the user to select logarithmic axes for either X or Y and Show minor ticks along the graph. This window also allow the hide or show the 2 line that make up the soil properties graph.

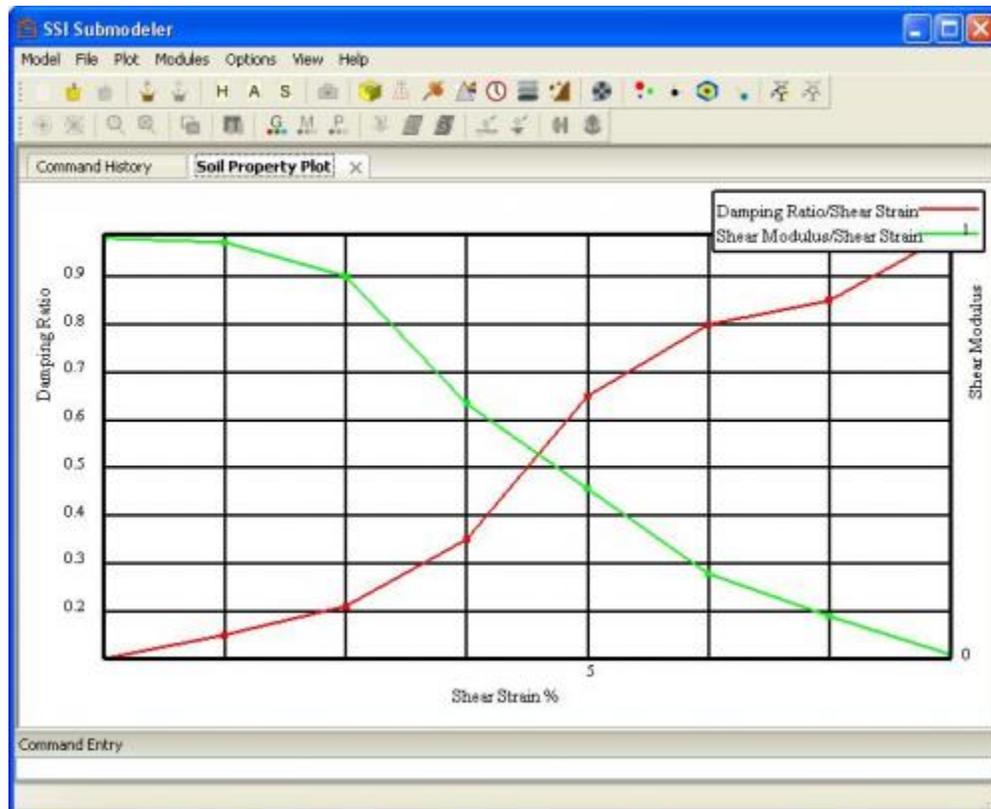


### Command

Selecting this menu option is the same as the LAYERPLOT command. Many of the 2D Plotting Commands will be able to modify the plot from the command line or allow the user to do batch image manipulation using a .pre file.

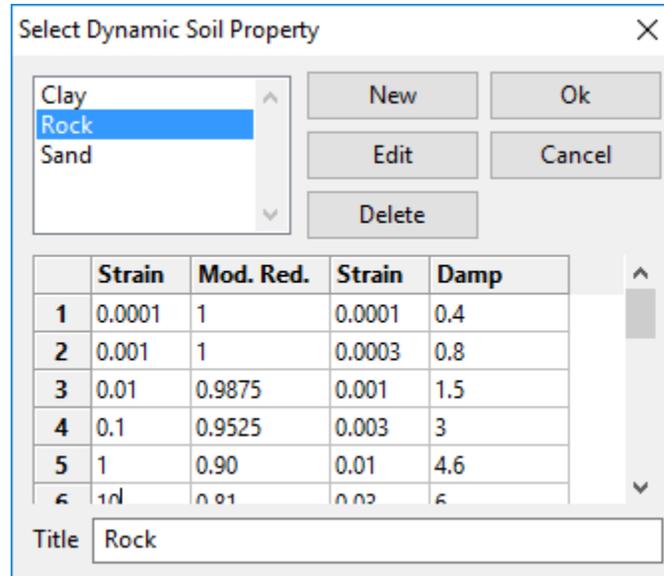
### 7.2.6 Soil Properties Plot

The Soil Layers Property plot show the Damping ratio and the Shear modulus vs. Shear strain for a selected soil property.



### Opening the Plot

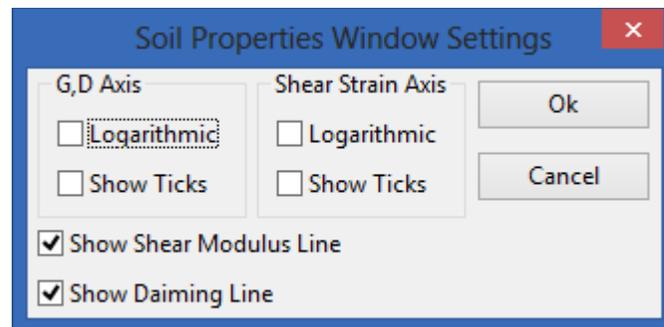
When the user first clicks on the soil properties Plot menu a pop up window appears so the user can select the soil property to be plotted. This window allow the user to choose and Soil property that has been defined in the model as well as creating a new soil property or modifying an existing property before plotting.



The new button will ask the user to specify a new soil property will pop up a window that will ask the user to specify a new property name. Once the user clicks OK on the popup the name will be added to the property selection list in the top left corner of the window. If the user selects edit He can modify the name of the currently selected property while delete will remove the currently selected property from the list. Once a property is selected the user can modify the property data by changing the number in the table window at the bottom of the popup window.

### Controls

The soil layer settings window will appear when the user has selected a soil layer plot as the current plot. This window allows the user to select which layers will be plotted and which data from the layers will show up in the table. The defaults for the layers numbers are show in the example image of this window. 1 is the minimum default which represents Surface Level. The -1 max tells the program to find the deepest layer in the model data and use that number as the maximum layer number.



### Command

Selecting this menu option is the same as the SOILPROPLOT command. Many of the 2D Plotting Commands will be able to modify the plot from the command line or allow the user to do batch image manipulation using a .pre file.

## 7.3 Animated Plots

### 7.3.1 Processing Animation for the ACS SASSI User Interface

Later versions of PREP allowed the user to view data from the spectrum and time history graphs on the model surface in a time stepped animation and static 3D models. These animations gave users an understanding of what is going on that the line graphs simply can't capture. These animations have been ported to the new UI. The new UI uses a newer version of OpenGL than was used in previous PREP to take advantage of the hardware advancements made in the current video cards. The data format that the video card requires for animation cannot be written directly by the SSI modules. Therefore, the frame output of the modules was slightly modified for use by the UI and this extra processing step has been added.

Parse Frame Data

List File Name  << Ok

Frame Storage Dir  << Cancel

Data Description

Plot Type

- Bubble
- Vector
- Contour
- Time History
- Stress DB (Binary)
- ACC DB (Binary)
- RelDisp DB (Binary)

The user must provide an animation list file and a frame storage directory by either browsing or manually typing the location of the File and the desired directory. The animation list file format is the same format frame listing file format from previous PREP (all \*.dispani, \*.tfiani, and \*.impani). The file will be read in a similar fashion in the new UI however the first line header information will be ignored because the processing step can take a long time based on the size of the model and the number of frames to be processed, so the user should only do this once.. The user can then modify the frames to be used in the animation on the animation loading screen. The data

description and the plot type box are for informational purposes. However, the user should fill these in will help in using the animation in future runs.

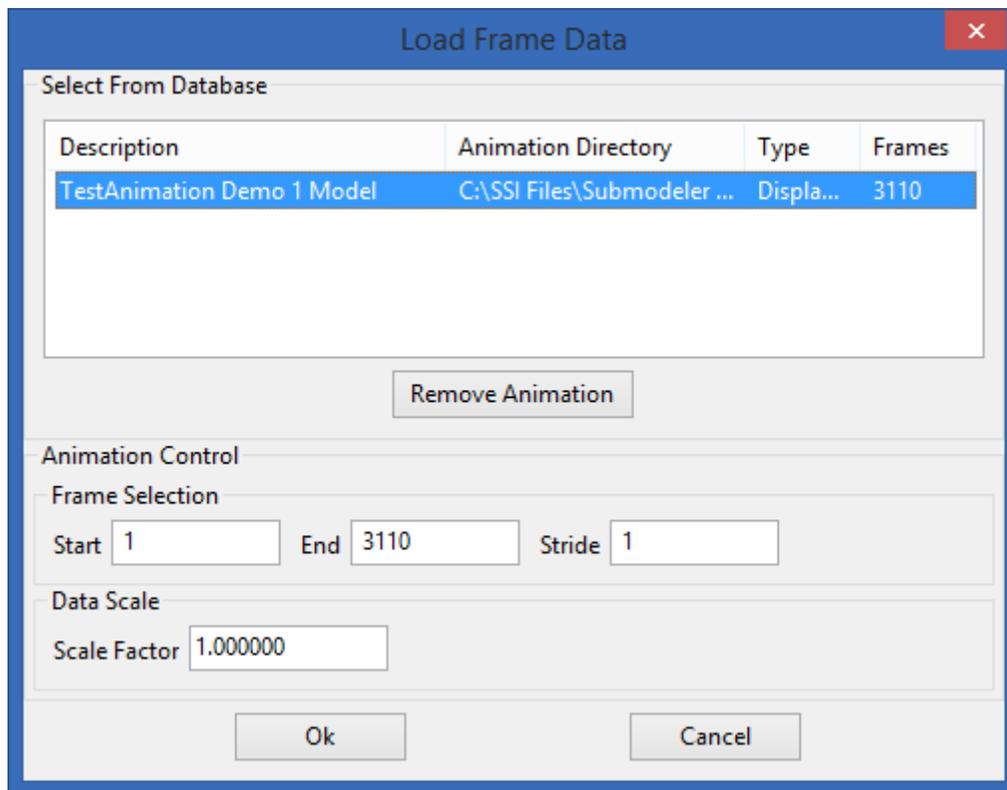
When the user selects the Process Animations Frame List from the menu a window will pop-up and as the user to input data. Once the data has been input the user will press the OK button and the processing will begin. A Progress bar will appear in the lower right hand side of the screen and tell the user what percentage for frame from the animation have been converted at this point.

After the frames have been converted an entry will be made into SASSlani.xml which keeps track of the location of the animations on the system along with some information for the animation. The frame storage directory will be filled with frame files that are now in the proper binary format for animation. The user should not attempt modify these binary files other than to delete them.

The last three items that contains the word binary in parenthesis are for animations obtained based on the SSI response databases.

### 7.3.2 Opening Animations

When the user first selects any of animation plots the user will see a pop-up window. This window will ask the user to select an animation from a list of animations that the user has previously processed. The user must select a plot out of the list by clicking on a line in the list box.



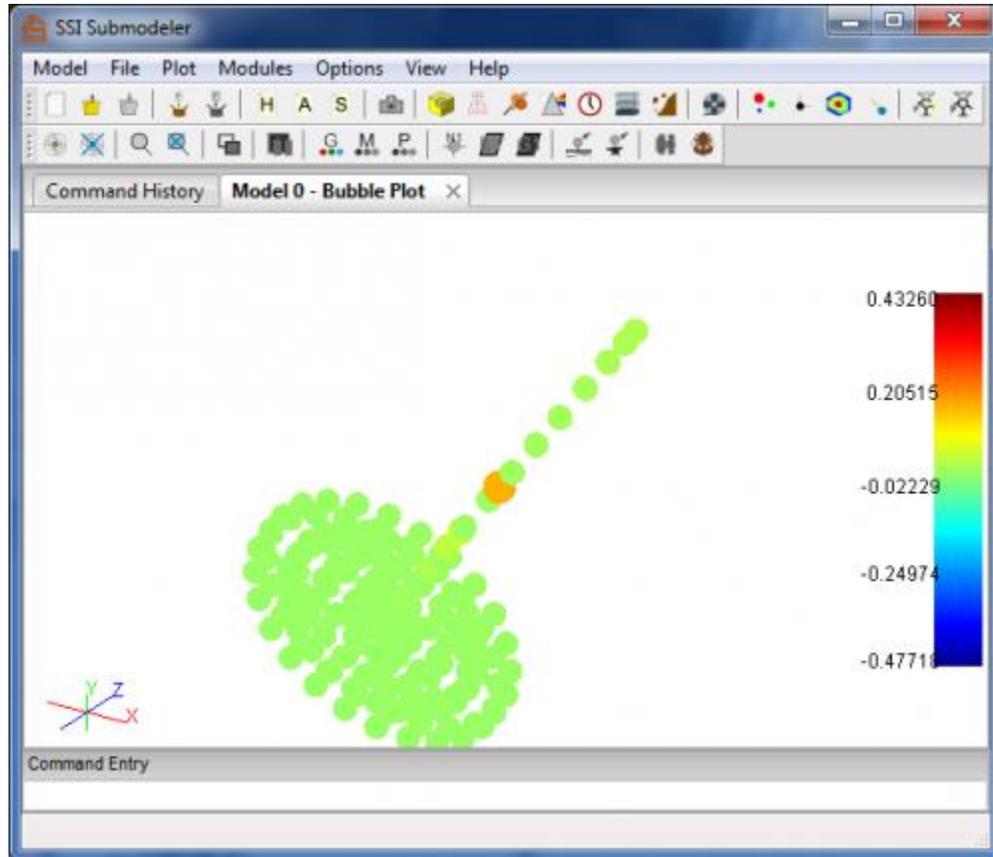
The list-box is populated by processing animations as discussed in the process frame animation section. Information about the animations is stored in a SASSlani.xml file that is used by the new UI to find and load animations. SASSlani.xml is stored in a system dependent default location so the user will never have to search for this file. Once the user selects a file from the list the animation control input boxes should populate with default data stored in the SASSlani.xml for the animation. The user to adjust which frame in the animation are to be displayed by adjusting the start end and stride inputs. The user will also have control over the data scale factor or the colormap range limits depending on the plot. The remove animation button will delete the animation from the file list and remove the data files that were produced by the process animation functionality.

### 7.3.3 Animation Controls

See 3D Plot Controls 7.1.1

### 7.3.4 Bubble Plot

The bubble plot will show the user an animation of all the nodes used in the model. (**Note:** Unused nodes can fall into the following categories, nodes that aren't connected to elements, orientation nodes like K-nodes, or nodes on element that have been simplified after being converted from an outside program). The size and the color of the bubbles are associated with the data for the node in the model and the color bar shown on the right of the graph.

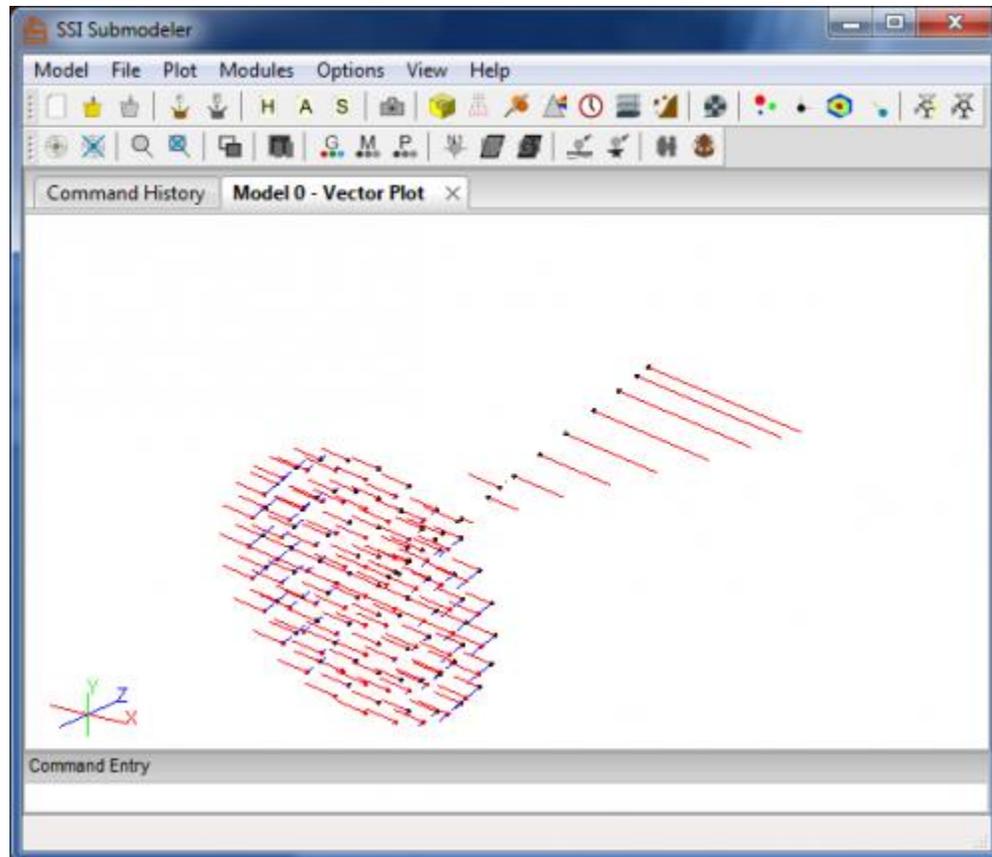


### **Command**

Selecting this menu option is the same as the BUBBLEPLOT command. Many of the 3D Plotting Commands will be able to modify the plot from the command line or allow the user to do batch image manipulation using a .pre file.

### **7.3.5 Vector Plot**

The vector plot animation show the user the direction of a force on each of the used nodes in the model. Each node will show 3 vectors on the plot. (**Note:** Unused nodes can fall into the following categories, nodes that aren't connected to elements, orientation nodes like K-nodes, or nodes on element that have been simplified after being converted from an outside program). The red vector is the data based on the x direction, green y direction and blue the z direction. If there is no imaginary component to the vector data each vectors direction will be in plane with its directional axis. If there is imaginary component with a vectors data the vector out of plane in an equivalent amount in the other 2 axis (if the x vector is  $5+.3i$   $x=5$ ,  $y = z = .3$ )

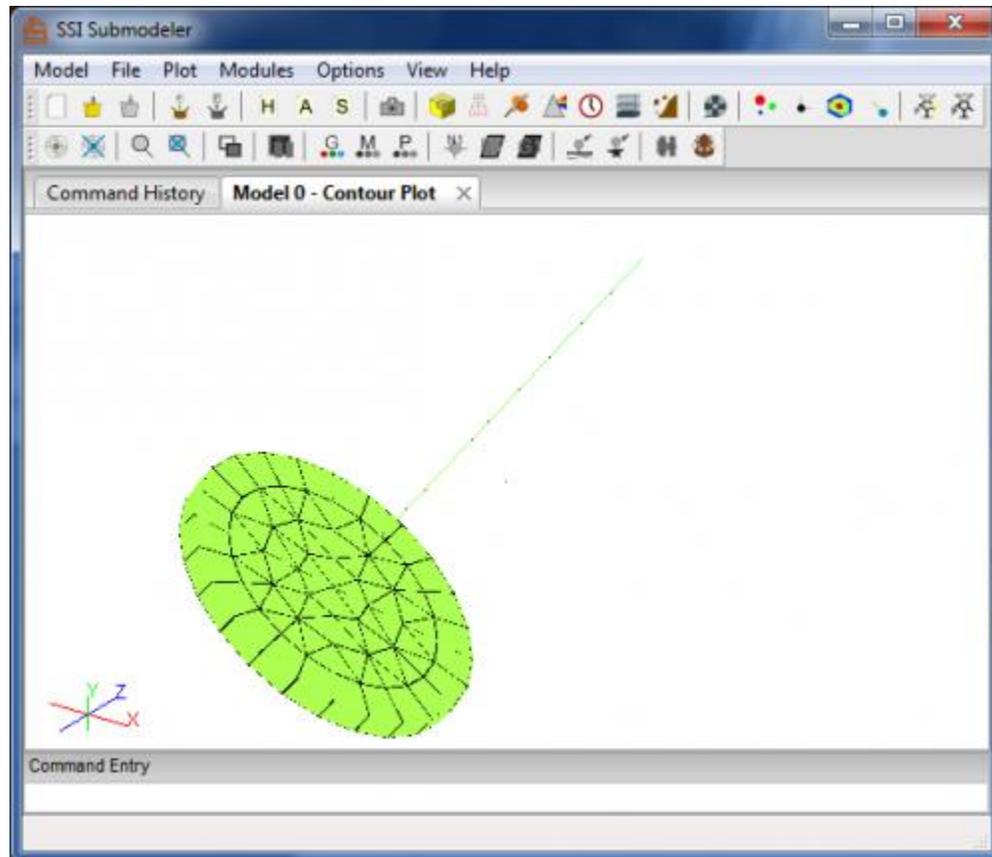


### **Command**

Selecting this menu option is the same as the VECTORPLOT command. Many of the 3D Plotting Commands will be able to modify the plot from the command line or allow the user to do batch image manipulation using a .pre file.

### **7.3.6 Contour Plot**

This Graph allows the user to show contour data on the actual model elements. The data for the contours is base on the nodes in the elements and the colors are linearly interpolated across the faces of the elements.

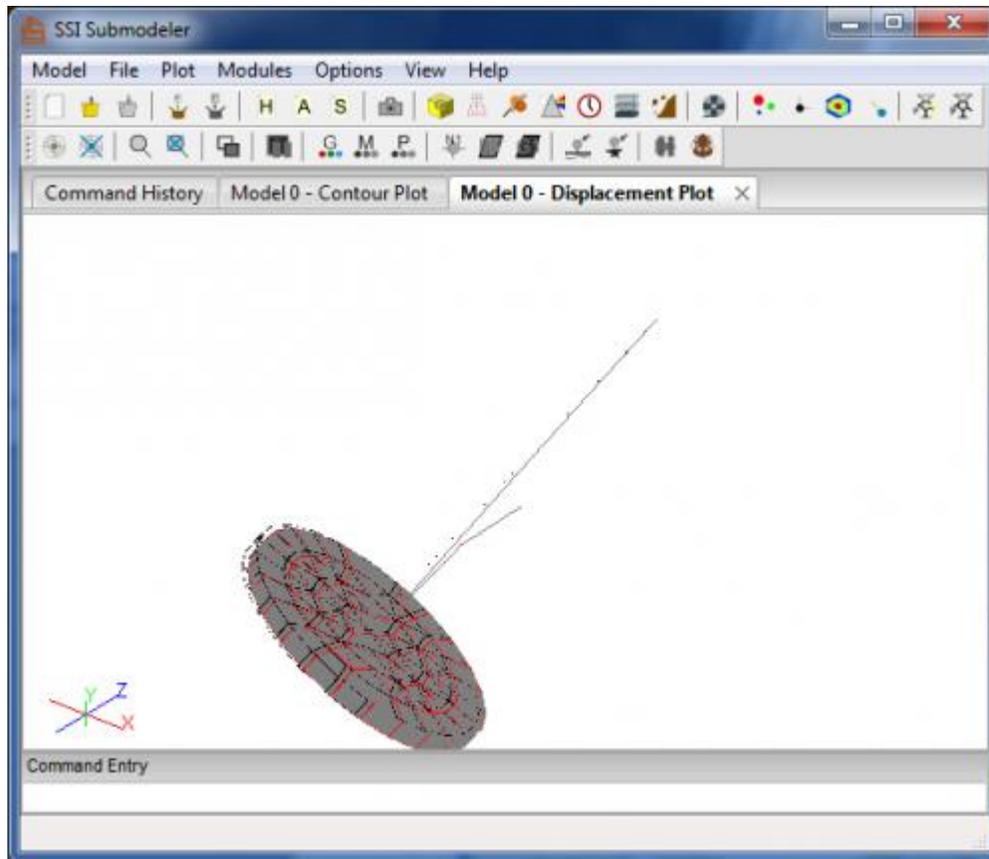


### **Command**

Selecting this menu option is the same as the CONTOURPLOT command. Many of the 3D Plotting Commands will be able to modify the plot from the command line or allow the user to do batch image manipulation using a .pre file.

### **7.3.7 Deformed Shape Plot**

The deformed shape plot is an animation of the model based on displacement data for every node in the model at a given time. The user can choose to show an unreformed wireframe of the model as a comparison of the deformed model.

**Command**

Selecting this menu option is the same as the DEFORMPLOT command. Many of the 3D Plotting Commands will be able to modify the plot from the command line or allow the user to do batch image manipulation using a .pre file.

## 8 Toolbar Reference

This section contains the reference tables for the two UI toolbars. The tables will show the toolbar icons with a short description of what each tool does. Also there will be links to any menu option or command that is closely associated with the tool which will provide a more in depth description of the actions of the tool.

### 8.1 Main Toolbar

Icon	Description	Menu Option	Command
	Create a New Model	Section 6.1.1	
	Open the Model Database	Section 6.1.2	
	Save a Model	Section 6.1.3	Section 9.2.29
	Output model to .pre	Section 6.1.8	Section 9.2.47
	output model to ANSYS	Section 6.1.9	Section 9.11.1
	use .hou converter	Section 6.1.5	Section 9.11.4
	use ANSYS® converter	Section 6.1.6	Section 9.11.4
	use GT STRUDL converter	Section 6.1.7	Section 9.11.4
	capture a plot image	Section 6.2.2	Section 9.14.6
	Element Plot	Section 7.1.2	Section 9.14.23
	Node plot	Section 7.1.3	Section 9.14.25
	Cut Plot	Section 7.1.4	Section 9.14.12
	Spectrum Plot	Section 7.2.3	Section 9.14.40
	Time History Plot	Section 7.2.4	Section 9.14.44
	Soil Layer Plot	Section 7.2.5	Section 9.14.19
	Soil Properties Plot	Section 7.2.6	Section 9.14.39
	Process Animation Frames	Section 7.3.1	Section 9.14.30
	Bubble Plot	Section 7.3.4	Section 9.14.5
	Vector Plot	Section 7.3.5	Section 9.14.45
	Contour Plot	Section 7.3.6	Section 9.14.11
	Deformed Shape Plot	Section 7.3.7	Section 9.14.14
	Change Analysis Options	Section 6.5.5	
	Afwrite current model		Section 9.2.3

## 8.2 3D Plot Toolbar

Icon	Description	Command
	Change View of Current Plot	Section 9.14.9
	Reset current plot view to default	Section 9.14.34
	Change center of current plot	Section 9.14.8
	Reset Center of current plot to default	Section 9.14.33
	Wireframe view	Section 9.14.47
	Shrink faces of elements	Section 9.14.38
	Colors of elements determined by group	Section 9.14.15
	Colors of elements determined by material	Section 9.14.15
	Colors of elements determined by property	Section 9.14.15
	Show node labels	Section 9.14.24
	Show element labels	Section 9.14.16
	Show group labels	Section 9.14.17
	show fixed degrees of freedom	Section 9.14.36
	show lumped masses	Section 9.14.37
	Pause Start Animation	Section 9.14.27
	Show Debug info	Section 9.14.13

## 9 Command Reference

The commands are presented in this chapter using the Backus-Naur form. <value> must be replaced with the actual value (number or string), all other characters must be typed as they are presented. Items enclosed by straight brackets are optional.

Many of the command in the new UI are from previous version PREP module. PREP only considered up to 4 letters in an instruction name while new UI uses the entire instruction name. The instructions that are valid in the UI will have an underlined section of the command name. If a user entered the whole command name or just underlined section the instruction will be interpreted properly. Any other variation will not be interpreted as a valid command. Command names typed at the instruction line are not case sensitive.

Due to differences in the between previous PREP and new UI a very few commands are not supported by new UI because they are either superfluous or cannot be supported under the new program architecture.

Refer to the online help documentation for additional explanation of specific commands, as well as examples. The online help is the most current reference documentation, as it is updated frequently. To access the online help, select Help from the menu bar in the ACS SASSI UI and select "Help" in that menu. This will open the default web browser, and load the online help site.

### 9.1 *Previous PREP Commands Included in New User Interface*

The next four tables list all of the commands that were active in both the previous PREP and are active also now in the new UI. Commands that are longer than four characters have underlined sections in the command description. A user can either enter the entire command name or just the underlined section of these commands.

### 9.2 *General Commands Compatible with Previous PREP Versions*

Command	Action	Description
ACCIN	Sets the acceleration time history input file for ACS SASSI EARTHQUAKE module.	Section 9.2.1
ACCOUT	Sets the acceleration time history output file for ACS SASSI EARTHQUAKE module.	Section 9.2.2
AFWRITE	Writes the analysis files.	Section 9.2.3
AMP	Defines spectral amplification ratios for ACS SASSI HOUSE module.	Section 9.2.4
ANALYS	Defines analysis options for ACS SASSI ANALYS module.	Section 9.2.5
AOPT	Sets options for AFWRITE and CHECK.	Section 9.2.6

<b>Command</b>	<b>Action</b>	<b>Description</b>
CHECK	Checks data.	Section 9.2.7
CORR	Sets spectra correlation values for ACS SASSI EARTH module.	Section 9.2.8
DAMP	Adds / resets damping ratios for RS analysis.	Section 9.2.9
DYNP	Defines data for dynamic soil properties.	Section 9.2.10
EOUT	Adds element output request for ACS SASSI STRESS module.	Section 9.2.11
EQTIT	Defines spectra title for ACS SASSI EARTH module.	Section 9.2.12
EARTH	Defines analysis options for ACS SASSI EARTH module.	Section 9.2.13
FORCE	Defines analysis options for ACS SASSI FORCE module.	Section 9.2.14
FREQ	Adds / deletes frequency numbers to / from frequency set.	Section 9.2.15
HOUSE	Defines analysis options for ACS SASSI HOUSE module.	Section 9.2.16
INCOH	Defines incoherence analysis options for ACS SASSI HOUSE module.	Section 9.2.17
INP	Switches input to file.	Section 9.2.18
LFREQ	Lists frequency sets.	Section 9.2.19
ME	Defines input motion data for ACS SASSI HOUSE module.	Section 9.2.20
MOPT	Changes the model options.	Section 9.2.21
MOTION	Defines analysis options for ACS SASSI MOTION module.	Section 9.2.22
NOUT	Adds nodal output request for ACS SASSI MOTION module.	Section 9.2.23
POINT	Defines analysis options for ACS SASSI POINT module.	Section 9.2.24
RESUME	Re-loads the active model.	Section 9.2.25
RSIN	Sets the response spectrum input file for ACS SASSI EARTH module.	Section 9.2.26
RSOUT	Sets the response spectrum output file for ACS SASSI EARTH module.	Section 9.2.27
SACC	Sets the acceleration output options for ACS SASSI SOIL module.	Section 9.2.28
SAVE	Saves active model.	Section 9.2.29
SFOU	Sets the Fourier spectrum output options for ACS SASSI SOIL module.	Section 9.2.30
SITE	Defines analysis options for ACS SASSI SITE module.	Section 9.2.31
SOIL	Defines analysis options for ACS SASSI SOIL module.	Section 9.2.32
SPRO	Defines soil profile data for ACS SASSI SOIL module.	Section 9.2.33
SRS	Sets the response spectrum output options for ACS SASSI SOIL module.	Section 9.2.34
SSAF	Sets the spectral amplification factor output options for ACS SASSI SOIL module.	Section 9.2.35

Command	Action	Description
SSTR	Sets the stresses and strains output options for ACS SASSI SOIL module.	Section 9.2.36
STATUS	Lists general information.	Section 9.2.37
STRESS	Defines analysis options for ACS SASSI STRESS module.	Section 9.2.38
SYMM	Sets information for symmetry / anti-symmetry plane / line	Section 9.2.39
THFILE	Sets acceleration time history file.	Section 9.2.40
THTIT	Sets title for acceleration time history.	Section 9.2.41
TPSD	Use the Target PSD functionality of ACS SASSI EARTHQUAKE	Section 9.2.42
TIT	Sets the model title.	Section 9.2.43
TOPL	Adds / deletes top layers for ACS SASSI SITE module.	Section 9.2.44
WAVE	Defines wave information for ACS SASSI SITE module.	Section 9.2.45
WPASS	Defines wave passage data for ACS SASSI HOUSE module.	Section 9.2.46
WRITE	Writes model data to an input file.	Section 9.2.47
RELD	Writes Relative Displacement Options	Section 9.2.48
RELFILE	Writes the Relative Displacement Reference File.	Section 9.2.49
RDND	Write a node to the Relative Displacement Output Node List	Section 9.2.50

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

### 9.2.1 ACCIN Command

ACCIN,<no>,<file>

sets the acceleration time history input file

<no> – file number

<file> – full path file name

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI EARTHQUAKE module from the ACS SASSI PREP desktop or from the ACS SASSI User Interface, use the Options / Analysis

### 9.2.2 ACCOUT Command

ACCOUT,<no>,<file>

sets the acceleration time history output file number <no> for ACS SASSI EQUAKE module to <file>.

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI EQUAKE module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis.

### 9.2.3 AFWRITE Command

#### AFWRITE

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

**Note:** Before writing the analysis file, the program first runs a CHECK instruction. If there were any errors found (excluding warnings), the affected analysis files will not be written.

### 9.2.4 AMP Command

AMP,<no>,<a1>,<a2>,<a3>,<a4>,<a5>,<a6>,<a7>,<a8>,<a9>,<a10>...<a100>

provides the spectral amplification ratio list to be applied to reference input motion <no> for the multiple excitation option - ACS SASSI HOUSE module or deletes spectral amplification ratios if <a1> is zero.

**Note:** This instruction is provided for input files. To change spectral amplification ratios from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis command.

### 9.2.5 ANALYS Command

ANALYS,<opmode>,<type>,<mode>,<save>,<prnt>,<fopt>,<ang>,<xc>,<yc>,<zc>,<impe>,[simul]

defines the following analysis options for ACS SASSI ANALYS module:

<opmode> - operation mode

0 - complete solution

1 - data check only

<type> - analysis type

0 - seismic

1 - foundation vibration

<mode> - analysis mode

0 - initiation

1 - new structure  
 2 - new seismic environment  
 3 - new dynamic loading  
 <save> - file save option  
     0 - do not save analysis restart files  
     1 - save analysis restart files  
 <prnt> - print option for transfer functions  
     0 - print complex transfer functions  
     1 - print transfer function amplitude only  
 <fopt> - frequency option  
     0 - take frequencies from frequency set  
     1 - take frequencies from File1 (or File9)  
 <ang> - coordinate transformation angle  
 <xc> - x-coordinate of foundation reference/control point  
 <yc> - y-coordinate of foundation reference/control point  
 <zc> - z-coordinate of foundation reference/control point  
 <impe> - calculation mode for the global impedances of the foundation  
     0 - no calculations  
     1 - calculate the global soil impedances for the six diagonal terms  
     2 - calculate the global impedances as rigid body matrix with the size 6x6  
 <simul> - the number of simultaneous cases to be simulated (default = 0)

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI ANALYS module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis command.

### 9.2.6 AOPT Command

AOPT,<EQUAKE>,<SOIL>,<DEP>,<SITE>,<POINT>,<HOUSE>,<DEP>,<FORCE>,<ANALYS>,<COMBIN>,<MOTION>,<STRESS>,<RELDISP>,<PANEL>

sets the AFWRITE and CHECK options for modules. If <opt> = 0, the corresponding module will not be taken into account by the AFWRITE and CHECK instructions.

<EQUAKE> – Flag for EQUAKE module  
 <SOIL> – Flag for SOIL module  
 <DEP> – Flag for module not included in the current version of ACS-SASSI should always be 0  
 <SITE> – Flag for SITE module  
 <POINT> – Flag for POINT module  
 <HOUSE> – Flag for HOUSE module  
 <FORCE> – Flag for FORCE module  
 <ANALYS> – Flag for ANALYS module

<COMBIN> – Flag for COMBIN module  
 <MOTION> – Flag for MOTION module  
 <STRESS> – Flag for STRESS module  
 <RELDISP> – Flag for RELDISP module  
 <PANEL> - Flag for the RELDISP module

**Note:** This instruction is provided for input files. To set the analysis options for the AFWRITE and CHECK instructions from the ACS SASSI PREP and ACS SASSI User Interface use the Options / Analysis command.

### 9.2.7 Check Command

#### CHECK

Checks all data for errors and warnings that can affect the analysis. Check is automatically run before writing the analysis file AFWRITE, but CHECK can be run explicitly for finding any errors, even with a partial model. The analysis parameters must be set before running check, because this instruction simulates the writing of the analysis files. After the check procedure is complete, and if there are any errors or warnings, they will be listed in the Check Errors window.

### 9.2.8 CORR Command

CORR,<no>,<time>,<val>

sets the <no>-th pair of spectra correlation values to <time> and <val> for the ACS SASSI EARTHQUAKE module.

**Note:** This instruction is provided for input files. To change correlation values from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis.

### 9.2.9 DAMP Command

DAMP,<d1>,<d2>,<d3>,<d4>,<d5>,<d6>,<d7>,<d8>,<d9>,<d10>

adds non-zero parameters to damping ratio list or deletes damping ratios if <d1> is zero.

**Note:** This instruction is provided for input files. To change damping ratios from the ACS SASSI PREP and ACS SASSI User Interface desktop, use the Options / Analysis.

### 9.2.10 DYNP Command

DYNP,<no>,<sg>,<g>,<sd>,<d>,<label>

sets the values for the <no>-th pair of the shear strain-shear modulus curve to <sg> and <g> and of the shear strain-damping curve to <sd> and <d> for the dynamic soil property <label>.

**Note:** This instruction is provided for input files. To change dynamic soil property data from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis.

### 9.2.11 EOUT Command

EOUT,<code1>,...<code12>,<group>,<element list>

adds the element output request for ACS SASSI STRESS module for the elements from group <group> and belonging to <element list>. The parameters <code1> to <code12> are the output options.

**Note:** This instruction is provided for input files. To change element output requests from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.12 EQTIT Command

EQTIT,<title>

sets the spectra title for ACS SASSI EQUAKE module to <title>.

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI EQUAKE module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.13 EQUAKE Command

EQUAKE, <accopt>,<nrfreq>,<rand>,<damp>,<dur>,<corr>,<seeds>,[tpsdl]

defines the following analysis options for the ACS SASSI EQUAKE module:

<accopt>- option for acceleration input files

0 - disabled

1 - enabled

<nrfreq>- number of frequencies in spectra files

<rand>- initial random number

<damp>- damping value

<dur>- total duration

<corr>- correlated spectra option

0 - disabled

1 - enabled

<seeds> - number of random seeds per Acceleration file

<tpsdl> - Option to use target PSD files (default = 0)

0 - disabled

1 - enabled

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI EARTHQUAKE module from the User Interface, use the Options / Analysis menu option.

### 9.2.14 FORCE Command

FORCE,<opmode>

defines the following analysis options for ACS SASSI FORCE module:

<opmode>- operation mode  
 0 - complete solution  
 1 - data check only

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI FORCE module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.15 FREQ Command

FREQ,<ndx>,<f1>,<f2>,<f3>,<f4>,<f5>,<f6>,<f7>,<f8>,<f9>,<f10>

adds non-zero parameters to frequency list of frequency set <ndx> or deletes the frequency set <ndx> if <f1> is zero. The list includes maximum ten data for each line. If more data need to be input the FREQ commands is repeated, for example FREQ,<ndx>,<f11>,<f12>,<f13>,<f14>,<f15>,<f16>,<f17>,<f18>,<f19>,<f20> is repeated with new frequencies,

### 9.2.16 HOUSE Command

HOUSE,<gravity>,<gelev>,<opmode>,<dim>,<imp>,<coh>,<wpass>, <me>,<cmplxspec>

defines the following analysis options for ACS SASSI HOUSE module:

<gravity> - acceleration of gravity  
 <gelev> - ground elevation  
 <opmode>- operation mode  
 0 - complete solution  
 1 - data check only  
 <dim>- analysis dimension  
 0 - 1D  
 1 - 2D  
 2 - 3D  
 <imp>- method of computing impedance matrix  
 0 - Flexible Volume method  
 1 - Fast Flexible Volume method

2 – Flexible Interface method

<coh>- soil motion  
 0 - coherent  
 1 - incoherent

<wpass> - wave passage  
 0 - disable  
 1 - enable

<me> - multiple excitation  
 0 - disable  
 1 - enable

<cmplxspec> - complex spectral amplification ratio  
 0 - disable  
 1 - enable

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI HOUSE module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.17 INCOH Command

INCOH,<gammax>,<gammay>,<gammaz>,<alpha>,<ngp>,<ipr>,<nmodes>,<met>,<HSeed>,<VSeed>,<RandPhz>

defines the following incoherence analysis options for ACS SASSI HOUSE module:

<gammax>- coherence parameter for x direction  
 <gammay>- coherence parameter for y direction  
 <gammaz>- coherence parameter for z direction  
 <alpha> - directionality parameter for the Abrahamson or user-defined models; or mean shear wave velocity for the Luco-Wong model  
 <ngp>- number of mesh points per each embedment level  
 <ipr>- flag for printing incoherent mode contributions for each frequency  
 0 - no print  
 1 - print  
 <nmodes>- number of incoherent modes  
 0 - for all modes, N  
 -k – for only mode k  
 k – for all modes up to k only, where k < N  
 <met>- Flag for use of metric Units  
 0 – British units (ft)  
 1 – International units (m)  
 <Hseed>- Horizontal Seed

<Vseed> - Vertical Seed

<RandPhz>- Random Phase Angle in degrees

**Note:** This instruction is provided for input files. To set the incoherence analysis options for the ACS SASSI HOUSE module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.18 INP Command

INP,<filename>

switches input to file <filename>. When the input file reaches EOF, input will be switched automatically to keyboard. The default path for <filename> is the path of the active model. The Input file format is a \*.pre file that has either been created by the user or has been written by the Write Command

### 9.2.19 LFREQ Command

LFREQ,[start],[end],[step]

lists the frequency sets from <start> (default: 1) to <end> (default: last) with step <step> (default: 1).

### 9.2.20 ME Command

ME, <no>,<nfirst>,<nlast>, <xc>,<yc>,<zc>

defines the following multiple excitation options for ACS SASSI HOUSE module:

<no> - number of input motion (between 1 and 10)

<nfirst> - first foundation node for input motion <no>

<nlast> - last foundation node for input motion <no>

<xc> - x-coordinate of control point for input motion <no> - not user in this version

<yc> - y-coordinate of control point for input motion <no>- not user in this version

<zc> - z-coordinate of control point for input motion <no>- not user in this version

**Note:** This instruction is provided for input files. To set the multiple excitation analysis options for the ACS SASSI HOUSE module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.21 MOPT Command

MOPT, <incomp>,<matrix>,<mass>, <force>

defines the following model options:

<icomp> - incompatible mode for solid elements

- 0 - include
- 1 - suppress

<matrix> - matrix option for general elements

- 0 - mass units
- 1 - weight units

<mass> - overwrite masses option

- 0 - add
- 1 - set

<force> - overwrite forces option

- 0 - add
- 1 - set

**Note:** This instruction is provided for input files. To set the model options from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Model menu option.

### 9.2.22 MOTION Command

MOTION,<opmode>,<out>,<step>,<dur>,<res>,<freq1>,<freq2>,<fstep>,<mult>,<max>,<rec1>,<rec2>,<fopt>,<bl>,<smo>,<cplx>,<cnvrt>,<pzadj>,<interp>

defines the following analysis options for ACS SASSI MOTION module:

<opmode> - operation mode

- 0 - complete solution
- 1 - data check only

<out> - output option

- 0 - full output
- 1 - only transfer functions to be output

<step> - output time history step

- 0 - print only table
- >1 - print every <step>-th point

<dur> - total duration of time histories to be plotted

<res> - not used for Data Check in this version

<freq1> - first frequency used in response spectrum analysis

<freq2> - last frequency used in response spectrum analysis

<fstep> - total number of frequency steps for response spectra

<mult> - multiplication factor for scaling time history

<max> - maximum value of time history to be used. The values of time history will be scaled to the given maximum value.

<rec1> - first record of time history file to be written to analysis file

<rec2> - last record of time history file to be written to analysis file (default: last record from file)

<fopt> - time history file option

- 0 - file contains time step on the first line, one acceleration on each other line
  - 1 - file contains pairs of time step and acceleration on each line
- <bl> - baseline correction
- 0 - time domain
  - 1 - frequency domain
- <smo> - Smoothing parameter; values can be between 1 and 1000 – see Section 6
- <cplx> - Determine if the saved TFU and TFI are complex quantities (amplitude and phase)
- 0 – Only amplitude
  - 1 – Complex; amplitude and phase (radians)
- <cnvrt> - Compute RS for external acceleration histories files – see Section 6
- 0 – No RS – see Section 6
  - 1 – Compute RS – see Section 6
- <pzadj> - Phase Adjustment parameter – see Section 6
- <interp> - Motion Interpolation Option– see Section 6

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI MOTION module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.23 NOUT Command

NOUT,<dir>,<code1>,...<code6>,<node list>

adds the nodal output request for ACS SASSI MOTION module for nodes belonging to <node list>. The parameter <dir> specifies the direction (1-x, 2-y, 3-z, 4-xx, 5-yy, 6-zz), and <code1> to <code6> are the output options.

**Note:** This instruction is provided for input files. To change nodal output requests from the ACS SASSI PREP or ACS SASSI User Interface use the Options / Analysis menu option.

### 9.2.24 POINT Command

POINT,<opmode>,<layer>,<rad>

defines the following analysis options for ACS SASSI POINT module:

<opmode> - operation mode

- 0 - complete solution
- 1 - data check only

<layer> - last layer number in near field zone, equal to the number of the embedment layers

<rad> - radius of the central zone

---

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI POINT module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.25 RESUME Command

#### RESUME

re-loads the model data. You may use this instruction to cancel the instructions which were run after the last save action. This command reloads the binary data created by the SAVE command. This command requires the model information to be set before use.

When an existing model is activated, ACS-SASSI PREP loads the model data automatically. The ACS SASSI User Interface will load this data automatically when a model is opened using the model menu.

### 9.2.26 RSIN Command

RSIN,<no>,<file>

sets the response spectrum input file number <no> for ACS SASSI EQUAKE module to <file>.

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI EQUAKE module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu Option.

### 9.2.27 RSOUT Command

RSOUT, <no>,<file>

sets the response spectrum output file number <no> for ACS SASSI EQUAKE module to <file>.

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI EQUAKE module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.28 SACC Command

SACC,<layer>,<opt>,<outcrop>

defines the acceleration output options for ACS SASSI SOIL module:

<layer> - layer number – acceleration defined at the top of the layer

<opt> - output option

0 - no computation

1 - compute maximum

2 - compute maximum and save time history

<outcrop> - outcrop option

0 - disabled

1 - enabled

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI SOIL module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.29 SAVE Command

SAVE

saves the active model's data needed for later resuming or loading in a later working session. This command will save the model data in a binary format that can be loaded with the RESUME command. This command requires the model information to be specified before use.

**Note:** The format of the binary data in PREP not compatible with the ACS SASSI User Interface. Transferring models between the two interface requires a Pre file which can be obtained using the WRITE command.

### 9.2.30 SFOU Command (Not usable in this version)

SFOU,<layer>,<out>,<save>,<outcrop>,<smooth>,<nrvl>

defines the Fourier spectrum output options for ACS SASSI SOIL module:

<layer> - sublayer number

<out> - output option

0 - no computation

1 - computation

<save> - save option

0 - disabled

1 - enabled

<outcrop> - outcrop option

0 - disabled

1 - enabled

<smooth> - number of times to be smoothed

<nrvl> - number of values to be saved

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI SOIL module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.31 SITE Command

SITE,<opmode>,<mode1>,<fstep>,<nl>,<hs>,<mode2>,<wopt>,<freq1>,<freq2>,<cl>,<cm>,<delt>,<nft>,<freq>

defines the analysis options for ACS SASSI SITE module:

<opmode> - operation mode

0 - complete solution

1 - data check only

<mode1> - mode 1 switch

0 - skip

1 - write

<fstep> - frequency step

<nl> - number of layers to simulate halfspace

<hs> - halfspace layer number

<mode2> - mode 2 switch

0 - skip

1 - write

<wopt> - wave combination option

0 - R-, SV-, and P-waves

1 - SH- and L-waves

<freq1> - frequency 1 that is the lowest for defining wave contribution (= 1)

<freq2> - frequency 2 that is the highest for defining wave contribution (= > NFFT/2)

<cl> - layer number of control point

<cm> - control motion direction

0 - X

1 - Y

2 - Z

<delt> - time step of seismic motion

<nft> - number of Fourier components (power of 2)

<freq> - frequency set number

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI SITE module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.32 SOIL Command

SOIL,<nrvl>,<grav>,<header>,<outcrop>,<save>,<iter>,<ratio>,<gravmult>,<cof>

Defines the analysis options for ACS SASSI SOIL module. This command arguments have changed since the original versions of PREP. Users should check and modify older models that

---

use the soil module for the old version of this command. The old command has a format flag as the last argument. The format flag identifiable because it is surrounded by parenthesis.

<nrvall> - number of acceleration values to be read from the time history file

<grav> - value of gravity for the model

<header> - number of header lines at the beginning of the acceleration time history file

<outcrop> - outcrop motion option

0 - disabled

1 - enabled

<save> - save strain-compatible soil properties option

0 - skip

1 - save

<iter> - number of iterations

<ratio> - ratio between the equivalent uniform and maximum strain

<gravmult> - multiplier for the acceleration of gravity

<cof> - cut-off frequency(disabled); always is assigned to Nyquist frequency for more accuracy

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI SOIL module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.33 SPRO Command

SPRO,<layer>,<prop>,<dynprop>

defines the soil profile for ACS SASSI SOIL module:

<layer> - sublayer number

<prop> - number of soil layer property

<dynprop>- label of dynamic soil property

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI SOIL module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.34 SRS Command

SRS,<layer>,<save>,<outcrop>

defines the response spectrum output options for ACS SASSI SOIL module:

<layer> - layer number; ARS defined at the top of the layer

<save> - save option

0 - disabled

1 - enabled  
 <outcrop> - outcrop option  
 0 - disabled  
 1 - enabled

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI SOIL module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.35 SSAF Command

SSAF,<layer>,<save>,<outcrop1>,<outcrop2>,<layer2>,<freqstep>,<title>

defines the spectral amplification factor output options for ACS SASSI SOIL module:

<layer> - sublayer number  
 <save> - save option  
 0 - disabled  
 1 - enabled  
 <outcrop1> - outcrop option for first sublayer  
 0 - disabled  
 1 - enabled  
 <outcrop2> - outcrop option for second sublayer  
 0 - disabled  
 1 - enabled  
 <layer2> - number of second sublayer  
 <freqstep> - frequency step  
 <title> - title for spectral amplification factor output

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI SOIL module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.36 SSTR Command

SSTR,<layer>,<opt1>,<opt2>,<opt3>,<opt4>

defines the stresses and strains output options for ACS SASSI SOIL module:

<layer> - layer number; stresses/strains are computed in the center of the layer  
 <opt1> - stress computation option  
 0 - no computation  
 1 - computation  
 <opt2> - save stress time history option

- 0 - disabled
- 1 - enabled
- <opt3> - strain computation option
  - 0 - no computation
  - 1 - computation
- <opt4> - save strain time history option
  - 0 - disabled
  - 1 - enabled

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI SOIL module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.37 STATUS Command

#### STATUS

displays global variables and general information for the active model.

### 9.2.38 STRESS Command

*STRESS*,<opmode>,<iter>,<save>,<itran>,<interopt>

defines the following analysis options for ACS SASSI STRESS module:

- <opmode> - operation mode
  - 0 - complete solution
  - 1 - data check only
- <iter> - iteration control key
  - 1 - automatic computation of strains in all soil elements
  - 0 - otherwise
- <save> - save option
  - 1 - save stress time histories in .ths files
  - 0 - do not save stress time histories
- <itran> - output option for transfer functions
  - 1 - output transfer functions
  - 0 - no transfer function output
- <interopt> - Interpolation option for stress

**Note:** This instruction is provided for input files. To set the analysis options for the ACS SASSI STRESS module from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.39 SYMM Command

SYMM,<no>,<type>,<node1>,<node2>,<node3>]

---

sets the parameters for symmetry/anti-symmetry plane/line number <no> (maximum 2), where: <type> = 0 - symmetry, <type> = 1 - anti-symmetry, and <node1>, <node 2>, and <node3> define the line / plane. The three nodes defining a plane must not lie on a straight line. To reset the plane / line, set <node1> to 0. To list the active planes/lines, use the STATUS command.

Any combination of a maximum of two structural planes / lines of symmetry which are symmetric or anti-symmetric relative to the loading can be considered. In case of a 3D analysis, the planes of symmetry or anti-symmetry must be parallel to the xz or yz planes. In case of 2D analysis, the line of symmetry / anti-symmetry must be parallel to the z-axis. Also note that the name symmetry or anti-symmetry is used in relation to the loading.

#### 9.2.40 THFILE Command

THFILE,<file>

sets the acceleration time history file to <file>.

**Note:** This instruction is provided for input files. To set the time history file from the ACS SASSI PREP or ACS SASSI UI, use the Options / Analysis menu option.

#### 9.2.41 THTIT Command

THTIT,<title>

sets the title for the acceleration time history to <title>.

**Note:** This instruction is provided for input files. To set the time history title from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

#### 9.2.42 TPSD Command

TPSD,<num>,<file>

Assign target PSD to spectrum number in EQUAKE

<num> - input number for the target PSD file. (corresponds to spectrum number in the UI and is the same number used in other EQUAKE commands ACCIN,ACCOUT, RSIN, and RSOUT)

<file> - Full path file name of the target PSD file to be used.

#### 9.2.43 TIT Command

TIT,<title>

sets the active model's title to <title>.

### 9.2.44 TOPL Command

TOPL,<l1>,[l2],[l3],[l4],[l5],[l6],[l7],[l8],[l9],[l10].....[l200]

adds non-zero parameters to top layers list for ACS SASSI SITE module or deletes the list if <l1> is zero.

For embedded models, the layers defined using the L command should in the same order as is used in the TOPL command. Soil layers shall not be repeated in the TOPL command for the embedment layers.

**Note:** This instruction is provided for input files. To set the top layers from the ACS SASSI PREP or ACS SASSI UI use the Options / Analysis menu option..

### 9.2.45 WAVE Command

WAVE,<type>,<opt>,<ratio1>,<ratio2>,<angle>

defines the following wave data for ACS SASSI SITE module:

<type> - wave type

- 1 - R-wave
- 2 - SV-wave
- 3 - P-wave
- 4 - SH-wave
- 5 - L-wave

<opt> - wave field option

- 0 - no wave field
- 1 - wave field or shortest wave length method for R-waves
- 2 - least decay method for R-waves only

<ratio1> - wave type fraction contribution in wave composition at the lowest frequency 1

<ratio2> - wave type fraction contribution in wave composition at the highest frequency 2

<angle> - incident angle of wave (degrees)

**Note:** This instruction is provided for input files. To wave information from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.46 WPASS Command

WPASS,<appv>,<ang>,<cohf>

defines the following wave passage data for ACS SASSI HOUSE module:

<appv> - apparent velocity for line D

<ang> - angle of line D with x axis

<cohf> - directional coherence factor

**Note:** This instruction is provided for input files. To define wave passage data from the ACS SASSI PREP or ACS SASSI User Interface, use the Options / Analysis menu option.

### 9.2.47 WRITE Command

Write Model data to a PREP Input file.

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

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

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

### 9.2.48 RELD Command

RELD,<RelDisOutput>,<RelDispSAll>,<RelDispNumFiles>

Sets the options for the ACS SASSI RELDISP module

<RelDisOutput> - User requested output type

<RelDispSAll> - Flag that overrides the output node list and puts all node components into the output list when the .rdi is written also tells the RELDISP module to make frames for displacement.

<RelDispNumFiles> - the number of file names in the Output node list.

### 9.2.49 RELFILE Command

RELFILE,<FileName>

Sets the reference node .TFI file for the ACS SASSI RELDISP module

<FileName> - Reference node .TFI file name

### 9.2.50 RDND Command

RDND,<NodeNum>,<X>,<Y>,<Z>,<XX>,<YY>,<ZZ>

This command creates the user output request for a node in the RELDISP module input file.

<NodeNum> is the number of the node to be added to the Output List.

<X><Y><Z><XX><YY><ZZ> are the Degrees of Freedom to be Listed. If < 1 the DOF is to be ignored. Otherwise, the DOF will be considered by the RELDISP module..

### 9.3 Node Commands

Command	Action	Description
CSYS	Activates a coordinate system.	Section 9.3.1
D	Sets boundary conditions.	Section 9.3.2
FILL	Generates a node line.	Section 9.3.3
GLOBAL	Transforms nodes into the global coordinate system.	Section 9.3.4
INT	Sets interaction, interface, intermediate, or internal nodes.	Section 9.3.5
INTLIST	Lists interaction, interface, intermediate, or internal nodes.	Section 9.3.6
LMOVE	Generates a node list by translation.	Section 9.3.7
LOC	Defines a local coordinate system with Euler angles.	Section 9.3.8
LOCAL	Defines a local coordinate system with nodes.	Section 9.3.9
N	Defines a node.	Section 9.3.10
NDEL	Deletes nodes.	Section 9.3.11
NGEN	Generates nodes by copying a node pattern.	Section 9.3.12
NLIST	Lists nodes.	Section 9.3.13
NMED	Defines a node with average coordinates.	Section 9.3.14
NMOVE	Generates a node list by scaling.	Section 9.3.15
NSCALE	Scales nodal coordinates.	Section 9.3.16
SDEL	Deletes coordinate systems.	Section 9.3.17
SLIST	Lists coordinate systems.	Section 9.3.18

#### 9.3.1 CSYS Command

CSYS,<ns>

activates the local coordinate system number <ns>. To activate the global system, <ns> must be set to 0.

#### 9.3.2 D Command

D,<n1>,<n2>,[<inc>],[<val>],[<label1>],[<label2>],...<label6>]

where <label i> can be UX, UY, UZ, DISP, ROTX, ROTY, ROTZ, ROT, or ALL

Defines the boundary conditions for the nodes between <n1> and <n2> with step <inc>, <val>=0 for free DOF and <val>=1 for fixed DOF. This instruction is used to generate the kinematic boundary conditions after the nodes have been previously defined. This fact allows a greater flexibility for the node generation procedure. The default values are: 0 for <val> and 1 for <inc>.

**Note:** Concentrated masses (MT and MR Commands), forces (F Command), or moments forces (MM Command) described in Section 9.5 shall be applied at the free degrees of freedom. A fixed degree of freedom does not allow the node to translate or rotate in that direction. Any concentrated masses or forces assigned to this degree of freedom are ignored by ACS SASSI User Interface.

Removing unwanted degrees of freedom has the advantage of reducing the size of the set of equations that must be solved. The following table lists the degrees of freedom that are defined for each element type. (GROUP Command)

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

Note, for example, that for solid elements, only the translations are defined at the nodes. If a node is common to two or more element types, then the non-trivial degrees of freedom are found by combination. For example, all six components are possible at a node common to both beam and solid elements; i.e., beam governs. Symmetrical structures (with symmetrical loading only) may also be analyzed by modeling only one half or one quarter of the structure and constraining appropriate degrees of freedom on the planes of symmetry. (SYMM Command)

### 9.3.3 FILL Command

FILL,[<n1>],[<n2>],[<nr>]

generates a line of nodes between two existing nodes (by interpolation); <nr> = number of nodes to be filled-in between <n1> and <n2>; if <nr> is not specified, default value will be used: <nr> = <n2>-<n1>-1; if <n1> and <n2> are not specified, these will be set to the latest two consecutively defined nodes.

### 9.3.4 GLOBAL Command

GLOBAL,<n1>,<n2>,<inc>

transforms the node set between <n1> and <n2> with step <inc> into the global coordinate system. This operation can be made using the GLOBAL instruction, or is made automatically when writing the analysis file.

**Note:** The GLOBAL instruction does not deactivate the local systems.

### 9.3.5 INT Command

INT,<n1>,<n2>,<inc>,<set>,<code>

sets interaction, intermediate, interface, or internal nodes. The node set is defined by start node number <n1>, end node number <n2>, and step <inc>. Depending on the value of parameter <set>, this instruction sets (<set>=1) or resets (<set>=0) the nodes as interaction (<code>=0), intermediate (<code>=1), interface (<code>=2), or internal (<

### 9.3.6 INTLIST Command

INTLIST,<n1>,<n2>,<step>,<c1>,<c2>,<c3>,<c4>

lists the set of interaction (if <c1>=1), intermediate (if <c2>=1), interface (if <c3>=1), and internal (if <c4>=1) nodes defined by <n1>, <n2>, and <step>. The default values are: the first node for <n1>, the last node for <n2> and 1 for <step>. If all codes <c1> - <c4> are zero, all interaction, intermediate, interface, and internal nodes are listed.

### 9.3.7 LMOVE Command

LMOVE,<dx>,<dy>,<dz>,<nd>,<l1>,<l2>,<l3>,<l4>,...,<l15>

defines new nodes beginning with <nd>, <nd+1>, <nd+2>,... according to the nodes belonging to the list <l1>, <l2>, <l3>, <l4>,...,<l9> by translating them with <dx>, <dy>, <dz>. The default values for <dx>, <dy>, <dz> are 0. The node list must contain at least one node.

### 9.3.8 LOC Command

LOC,<ns>,<type>,<x0>,<y0>,<z0>,<txy>,<tyz>,<txz>

defines the local coordinate system number <ns> (using Euler angles) with the type <type>, having the origin coordinates <x0>, <y0>, <z0> regarding to the general system, and whose axes are rotated with the angles <txy>, <tyz>, <txz>. (the angles are in degrees)

<type>

0 - Cartesian

1 - cylindrical (z is rotation axis) (not included in this version)

See also: LOCAL command for defining coordinate systems using nodes.

### 9.3.9 LOCAL Command

LOCAL,<ns>,<type>,<n1>,<n2>,<n3>

defines the local coordinate system number <ns> (using nodes) with the type <type>, with the origin in <n1>, and the local axis orientation  $X=\langle n1 \rangle \langle n2 \rangle$  and <n3> in the positive x0y quadrant. The Z local axis orientation will result from the vectorial product  $\langle n1 \rangle \langle n2 \rangle \times \langle n1 \rangle \langle n3 \rangle$  and Y local from  $X \times Z$ .

<type>

0 - Cartesian

1 - cylindrical (z is rotation axis) (not included in this version)

See also: LOC Command for defining coordinate systems using Euler angles.

### 9.3.10 N Command

N,<nd>,<x>,<y>,<z>

defines node <nd> with coordinates <x>, <y>, <z> in the active coordinate system. (global or local) The default values for <x>, <y>, <z>, <pile>,<control node> are 0. See also CSYS, LOC, and LOCAL instructions for local coordinate systems. (Pile and Control Nodes are depreciated in this version of the ACS SASSI User Interface) Note: The node numbering is arbitrary. However, in order to minimize storage and block operations as well as to provide flexibility for restart analysis with a new superstructure and for the incoherence analysis option, it is required to number the nodes at or below ground surface first, layer by layer, starting from the bottom. The z-coordinate must always be chosen vertical upward and the right-hand-rule must be used to set X and Y coordinates. For 1D and 2D analysis, the program ignores the y-coordinate of the nodal points.

### 9.3.11 NDEL Command

NDEL,<n1>,<n2>,<inc>

deletes the node set defined by <n1>, <n2> with step <inc>. The default value for <n2> is <n1>, and the default value for <inc> is 1.

**9.3.12 NGEN Command**

NGEN,[itim],[step],[n1],[n2],[inc],[dx],[dy],[dz]

generates a set of node lists by copying a node pattern, where:

<itim> - total number of sets of nodes to be generated, not including the original pattern (default = 1)

<step> - increment to be applied to generated node numbers. ( default = <n2>-<n1>+1 )

<n1> - Start node of the pattern to be copied (default = second to last node to be defined )

<n2> - End node of the pattern to be copied (default = last node to be defined)

<inc>- Increment for the pattern to be copied (default = 1)

<dx> - increments to be applied to the X node coordinates (default = 0)

<dy> - increments to be applied to the Y node coordinates (default = 0)

<dz> - increments to be applied to the Z node coordinates (default = 0)

**9.3.13 NLIST Command**

NLIST,[<n1>],[<n2>],[<inc>]

lists the nodes between <n1> (default: first node) and <n2> (default: last node) with increment <inc> (default: 1).

**9.3.14 NMED Command**

NMED,<nd>,<n1>,[<n2>,<n3>,<n4>,...,<n8>]

defines node <nd> with the coordinates x,y,z as the average value of the coordinates of the nodes contained in the list <n1>,<n2>,<n3>,<n4>,...,<n8>. The node list must contain at least one node.

**9.3.15 NMOVE Command**

NMOVE,[<dx>],[<dy>],[<dz>],[<nd>,<l1>,[<l2>,<l3>,<l4>,...,<l15>]

defines new nodes beginning with <nd>, <nd+1>, <nd+2>,... according to the nodes belonging to the list <l1>, <l2>, <l3>, <l4>,...,<l9> by scaling them with <dx>, <dy>, <dz>. The default values for <dx>, <dy>, <dz> are 1. The node list must contain at least one node.

**9.3.16 NSCALE Command**

NSCALE,<n1>,<n2>,[<inc>],[<sfx>],[<sfy>],[<sfz>]

scales the coordinates of the nodes between <n1> and <n2> with step <inc> (default: 1) by the scaling factors <sfx>, <sfy>, <sfz> (if one of the scaling factors is 0.0, it will be set to 1.0).

**9.3.17 SDEL Command**

SDEL,<s1>,<s2>,<inc>

deletes the coordinate systems from <s1> to <s2> (default value: <s1>) with step <inc> (default: 1).

**9.3.18 SLIST Command**

SLIST,<s1>,<s2>,<inc>

lists the coordinate systems from <s1> (default: first system) to <s2> (default: last system) with step <inc> (default: 1).

**9.4 Element Commands**

Command	Action	Description
DELL	Deletes soil layers.	Section 9.4.1
DELM	Deletes materials.	Section 9.4.2
DELR	Deletes real properties.	Section 9.4.3
DELSC	Deletes spring properties.	Section 9.4.4
E	Defines an element.	Section 9.4.5
ECOMPR	Compresses elements.	Section 9.4.6
EDEL	Deletes elements.	Section 9.4.7
EGEN	Generates elements by translation	Section 9.4.8
EINT	Sets integration order for SOLID elements.	Section 9.4.9
ELIST	Lists elements.	Section 9.4.10
ETYPE	Sets type for SOLID, PLANE, SHELL/TSHELL elements.	Section 9.4.11
GDEL	Deletes groups.	Section 9.4.12
GLIST	Lists groups.	Section 9.4.13
GROUP	Creates or activates a group.	Section 9.4.14
GTIT	Sets group title	Section 9.4.15
KI	Defines end release code in node I of BEAMS elements.	Section 9.4.16
KJ	Defines end release code in node J of BEAMS elements.	Section 9.4.17
L	Defines a soil layer.	Section 9.4.18
LLIST	Lists soil layers.	Section 9.4.19
M	Defines a material.	Section 9.4.20
MACT	Sets active material / soil layer index.	Section 9.4.21
MLIST	Lists materials.	Section 9.4.22

Command	Action	Description
MSET	Sets element material / soil layer index.	Section 9.4.23
MTYPE	Modifies the group type.	Section 9.4.24
MXDEL	Delete matrix properties.	Section 9.4.25
MXI	Set terms for matrix property - imaginary part of stiffness matrix.	Section 9.4.26
MXLIST	List matrix property.	Section 9.4.27
MXM	Set terms for matrix property - mass / weight matrix.	Section 9.4.28
MXR	Set terms for matrix property - real part of stiffness matrix.	Section 9.4.29
R	Defines a real property.	Section 9.4.30
RACT	Sets active real / spring / matrix property index.	Section 9.4.31
RLIST	Lists real properties.	Section 9.4.32
RSET	Sets element real / spring / matrix property index	Section 9.4.33
SC	Defines a spring property.	Section 9.4.34
SCLIST	Lists spring properties.	Section 9.4.35
THICK	Sets thickness for SHELL elements.	Section 9.4.36

#### 9.4.1 DELL Command

DELL,<m1>,<m2>,<inc>

deletes the soil layers with index from <m1> to <m2> (default value: <m1>) with step <inc> (default: 1).

#### 9.4.2 DELM Command

DELM,<m1>,<m2>,<inc>

deletes the materials with index from <m1> to <m2> (default value: <m1>) with step <inc> (default: 1).

#### 9.4.3 DELR Command

DELR,<r1>,<r2>,<inc>

deletes the real properties with index from <r1> to <r2> (default value: <r1>) with step <inc> (default: 1).

#### 9.4.4 DELSC Command

DELSC,<r1>,<r2>,<inc>

deletes the spring properties with index from <r1> to <r2> (default value: <r1>) with step <inc> (default: 1).

### 9.4.5 E Command

E,<ne>,<n1>,<n2>,...,<n8>

defines the element number <ne> and element nodes <n1>, <n2> ... <n8> (the total node number changes from one group type to another). An element group must be active; element numbers start with 1 for each element group.

Prisms and pyramids may be obtained by repeating node numbers.

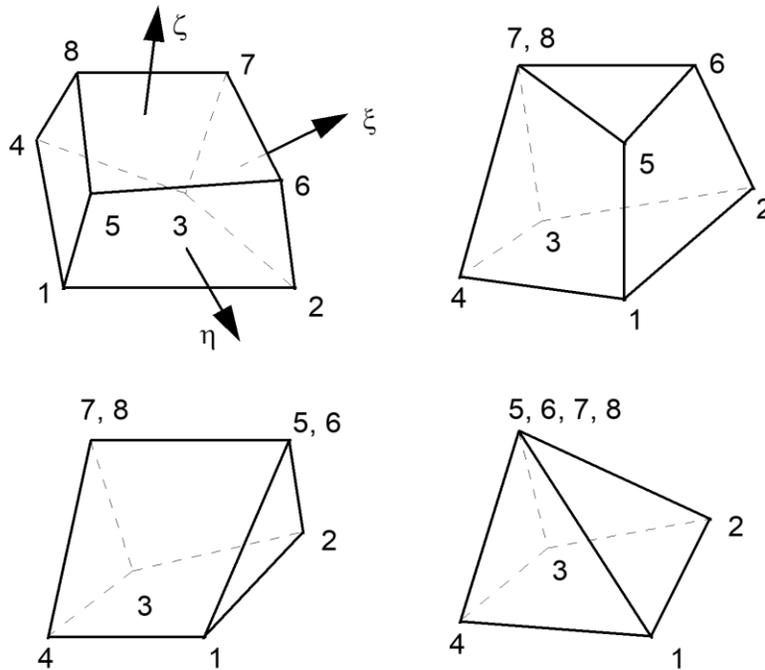


Figure 9.1: Node Numbering for SOLID elements

For SHELL/TSHELL, PLANE triangular elements do not define node L (see GROUP instruction)

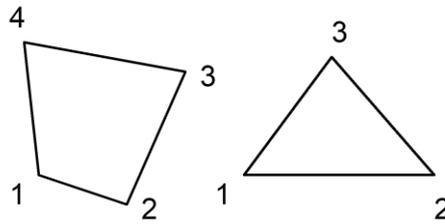


Figure 9.2: Node Numbering for 2D Elements

The nodes I, J, K, L must be in sequence in a counter-clockwise direction around the element. Local coordinates in a four-node SHELL, PLANE element are shown below:

$x'$  - specified by LI-JK, where LI and JK are midpoints of sides L-I and J-K.

$z'$  - normal to  $x'$  and to the line adjoining midpoints IJ and KL.

$y'$  - normal to  $x'$  and  $z'$  to complete the right-handed system used to compute the resultant forces.

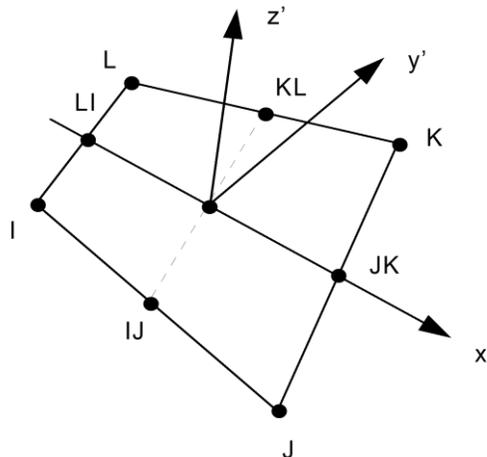


Figure 9.3: Node numbering and local coordinate system for SHELL and PLANE elements

Node K from the BEAMS elements is a geometry reference point which is used to define the local axes 1, 2, 3 of the beam element. Node K, which may be any other nodal point in the system, must not lie on the local axis 1:

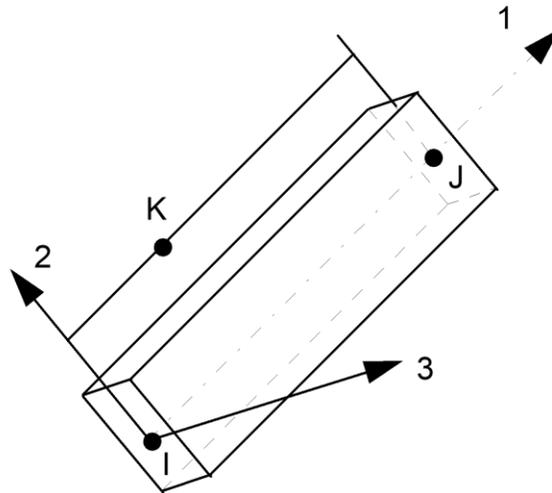


Figure 9.4: Local coordinate system for BEAMS elements

#### 9.4.6 ECOMPR Command

ECOMPR

compresses elements from the active group (removes the element numbering gaps).

#### 9.4.7 EDEL Command

EDEL,<e1>,<e2>,<inc>

deletes elements between <e1> and <e2> with step <inc>. The default values are: <e1> for <e2> and 1 for <inc>.

#### 9.4.8 EGEN Command

EGEN,<itim>,<ninc1>,<e1>,<e2>,<inc>,<ee>

generates elements by copying a pattern of elements, where:

<itim> total number of element sets to be generated not including original pattern (default: 1)

<ninc1> increment to be applied to node numbers for each element set

<e1> first pattern element

<e2> last pattern element (default: <e1>)

<inc> pattern step (default: 1)

<ee> first element number of the generated elements (default: Max element number +1)

**9.4.9 EINT Command**

EINT,<e1>,<e2>,<inc>,<order>

For SOLIDS sets the integration order for elements <e1> and <e2> with step increment <inc> (default: 1) to rectangular (<order> = 0), skewed (<order> = 1), or extremely distorted (<order> = 2). However, using distorted elements should be avoided as much as possible for stress analysis. Default integration is 2x2.

For TSHELL sets the integration type for elements <e1> and <e2> with step increment <inc> (default: 1) to (<order> = 0) for Reduced integration (1x1 for plate bending and transverse shear stiffness) and (<order> = 1) for Selective integration (2x2 for plate bending and 1x1 for transverse shear stiffness). Default integration is Reduced integration.

**9.4.10 ELIST Command**

ELIST,<e1>,<e2>,<inc>

lists the elements belonging to the active group starting with <e1> to <e2> with step <inc>. If the parameters are not specified, all elements from the active group will be listed.

**9.4.11 ETYPE Command**

ETYPE,<e1>,<e2>,<inc>,<type>

sets the element type to <type> for the elements between <e1> and <e2> with step <inc> (default: 1), where <type> is

- 0 – default; SOLIDs below ground surface are excavation volume, otherwise structural elements; SHELL are structural elements
- 1 – structural elements (SOLID or PLANE or SHELL/TSHELL)
- 2 – excavated soil element (SOLID or PLANE) or embedded shells (SHELL/TSHELL)

**9.4.12 GDEL Command**

GDEL,<g1>,<g2>,<inc>

deletes all groups between <g1> and <g2> with step <inc>. The default values are: <g1> for <g2> and 1 for <inc>.

**9.4.13 GLIST Command**

GLIST,<g1>,<g2>,<inc>

lists all groups between <g1> and <g2> with step <inc>. If no parameter is specified, all created groups will be listed.

### 9.4.14 Group Command

**GROUP**,<ng>,<type>

activates or creates the element group number <ng>. The parameter <type> can be typed as a number or a string.

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

The excavated soil zones may be modeled using either SOLID or PLANE element types.

The information (excepting nodes) which may be assigned to elements of each type is shown in the following table:

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

**WARNING:** All the elements in the system must be grouped separately according to their type. Gaps in element numbering are not allowed. Use the ECOMPR instruction to compress groups with element gaps. It is possible to use more than one group for an element type. For example, all structural brick elements may be considered as one group and all excavated soil elements as another group.

#### 9.4.15 GTIT Command

GTIT,[gr],<title>

sets the title of group <gr> ( Default = active group ) to <title>.

#### 9.4.16 KI Command

KI,<e1>,[<e2>],[<inc>],<k1>,<k2>,...,<k6>

defines the end release code for the BEAMS I node for the element set defined by <e1>, <e2> (default: <e1>), and <inc> (default: 1) from the active group. The active group must be of type BEAMS. The six values <k1> ... <k6> correspond to the force components P1, P2, P3, M1, M2, M3 at the I node and may take the values 1 and 0 only. If one of the element end forces is known to be zero (hinge or roller), the corresponding digit should be set to 1.

#### 9.4.17 KJ Command

KJ,<e1>,[<e2>],[<inc>],<k1>,<k2>,...,<k6>

defines the end release code for the BEAMS J node for the element set defined by <e1>, <e2> (default: <e1>), and <inc> (default: 1) from the active group. The active group must be of type BEAMS. The six values <k1> ... <k6> correspond to the force components P1, P2, P3, M1, M2, M3 at the J node and may take the values 1 and 0 only. If one of the element end forces is known to be zero (hinge or roller), the corresponding digit should be set to 1.

#### 9.4.18 L Command

L,<nm>,<thick>,<weight>,<pveloc>,<sveloc>,<pdamp>,<sdamp>

defines the soil layer number <nm> with thickness <thick>, specific weight <weight>, P-wave velocity <pveloc>, S-wave velocity <sveloc>, P-wave damping ratio <pdamp>, and S-wave damping ratio <sdamp>.

For embedded models, the layers defined using this command should in the same order as is used in the TOPL command. Layers should not be repeated in the TOPL command for embedded models.

**Note:** Soil layers may be assigned to excavated soil elements (see MSET) or may be used to define soil layering or halfspace properties for the ACS SASSI SITE module.

**9.4.19 LLIST Command**LLIST,<m1>,[<m2>],[<step>]

lists the table of soil layer properties between <m1> and <m2> (default: last defined) with step <step> (default :1).

**9.4.20 M Command**

M,&lt;nm&gt;,&lt;val1&gt;,&lt;val2&gt;,&lt;weight&gt;,&lt;pdamp&gt;,&lt;sdamp&gt;,&lt;type&gt;

defines the material number <nm>. If <type> = 1, <val1> is the elasticity modulus and <val2> is the Poisson coefficient; if <type> = 2, <val1> and <val2> are the constrained and shear moduli; and if <type> = 3, <val1> and <val2> are the P- and S-Wave velocities. For all cases, <weight> is the specific weight, <pdamp> is the P-wave damping ratio, and <sdamp> is the S-wave damping ratio.

Note: <pdamp> and <sdamp> must be equal for materials assigned to SHELL elements.

**9.4.21 MACT Command**

MACT,&lt;index&gt;

sets the active material / soil layer index to <index>. All elements defined after this command and before another MACT command will have the material / soil layer index automatically set to <index>

**9.4.22 MLIST Command**MLIST,<m1>,[<m2>],[<step>]

lists the table of material properties between <m1> and <m2> (default: last defined) with step <step> (default :1).

**9.4.23 MSET Command**

MSET,&lt;e1&gt;,[&lt;e2&gt;],[&lt;inc&gt;],&lt;index&gt;

sets the material / soil layer index to <index> for the elements in the active group from <e1> to <e2> (default: <e1>) with step <inc> (default: 1). If an element belongs to a SOLID or PLANE element group, and by default or by explicit setting is of type excavated soil, the index refers to the soil layer table, otherwise the index refers to the material table.

**9.4.24 MTYPE Command**MTYPE,[<gr>],<type>

---

modifies the type of group <gr>. If <gr> is not specified, it is set to the active group. The parameter <type> can be typed as a number or a string. See also GROUP.

#### 9.4.25 MXDEL Command

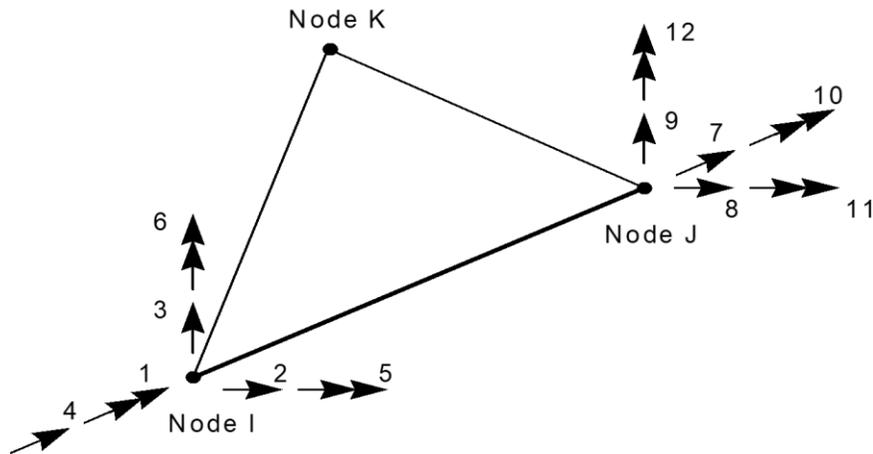
*MXDEL*, <p1>, [<p2>], [<step>]

deletes the matrix properties with index from <m1> to <m2> (default value: <m1>) with step <inc> (default: 1).

#### 9.4.26 MXI Command

*MXI*, <p>, <row>, <t1>, <t2>, ... <t12>

sets the terms belonging the imaginary part of the stiffness matrix of matrix property <p>, row <row> to <t1> - <t12>. If the matrix property <p> does not exist, it will be created by this command. Since the matrix is upper-diagonal, enter only the required number of terms from the diagonal to the right, i.e., for the first row use <t1> - <t12>, for the second row <t1> - <t11>, and for the last row ( <row> = 12), only <t1>.



	1	2	3	4	5	6	7	8	9	10	11	12
1												
2												
3												
4												
5												
6												
7												
8												
9												
10												
11												
12												

GENERAL Matrix Element and Corresponding Property Matrix

**WARNING:** If only the 2 nodes, I and J, are defined that then the MXI, MXR and MXM stiffness and mass inputs are provided in global coordinate systems. If 3 nodes, I, J and K, are defined then, the MXI, MXR, and MXM inputs are provided in local coordinate system as shown in the above figure.

**9.4.27 MXLIST Command**

MXLIST, <p>

lists the matrix property <p> - the real and imaginary parts of the stiffness matrix and the mass / weight matrix.

**9.4.28 MXM Command**

MXM,<p>,<row>,<t1>,<t2>,...<t12>

sets the terms belonging the mass / weight matrix of matrix property <p>, row <row> to <t1> - <t12>. If the matrix property <p> does not exist, it will be created by this command. Since the matrix is upper-diagonal, enter only the required number of terms from the diagonal to the right, i.e., for the first row use <t1> - <t12>, for the second row <t1> - <t11>, and for the last row (<row> = 12), only <t1>.

**Note:** Enter the terms in mass or weight units, as set from the Options / Model menu or the MOPT command.

**WARNING:** *If only the 2 nodes, I and J, are defined that then the MXI, MXR and MXM stiffness and mass inputs are provided in global coordinate systems. If 3 nodes, I, J and K, are defined then, the MXI, MXR, and MXM inputs are provided in local coordinate system.*

#### 9.4.29 MXR Command

MXR,<p>,<row>,<t1>,<t2>,...<t12>

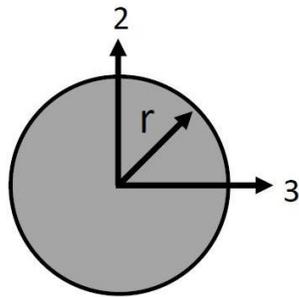
sets the terms belonging the real part of the stiffness matrix of matrix property <p>, row <row> to <t1> - <t12>. If the matrix property <p> does not exist, it will be created by this command. Since the matrix is upper-diagonal, enter only the required number of terms from the diagonal to the right, i.e., for the first row use <t1> - <t12>, for the second row <t1> - <t11>, and for the last row (<row> = 12), only <t1>.

**WARNING:** *If only the 2 nodes, I and J, are defined that then the MXI, MXR and MXM stiffness and mass inputs are provided in global coordinate systems. If 3 nodes, I, J and K, are defined then, the MXI, MXR, and MXM inputs are provided in local coordinate system as shown in the above figure.*

#### 9.4.30 R Command

R,<nm>,<axial>,<shear2>,<shear3>,<tors>,<flex2>,<flex3>

defines the real property set number <nm> with axial area <axial>, shear area for local axis 2 <shear2>, shear area for local axis 3 <shear3>, torsional inertia moment <tors>, flexural inertia moment for local axis 2 <flex2>, and flexural inertia moment for local axis 3 <flex3>.

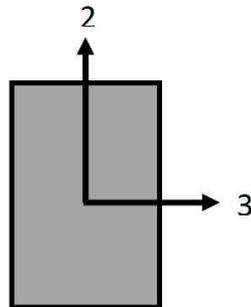


$$I_2 = I_3 = \pi r^4 / 4$$

$$f_2 = f_3 = 10/9$$

$$A = \pi r^2$$

$$J = \pi r^4 / 2$$



$$I_2 = hb^3 / 12$$

$$I_3 = bh^3 / 12$$

$$f_2 = f_3 = 6/5$$

$$A = bh$$

$$J = [1/3 - 0.21(bh)(1 - b^4/12h^4)]hb^3$$

Flexural area moment of inertia ( $I_2, 3$ ), cross sectional area ( $A$ ), torsional area moment of inertia ( $J$ ), and shear correction coefficient ( $f_{2,3}$ ) for solid rectangular and circular beam cross sections. The section properties of some mostly used cross-sections are given above. If shear deformations are not going to be included in the analysis, let  $\langle \text{shear}_2 \rangle$  and  $\langle \text{shear}_3 \rangle$  be zero.

#### 9.4.31 RACT Command

RACT,  $\langle \text{index} \rangle$

sets the active real / spring / matrix property index to  $\langle \text{index} \rangle$ . All elements defined after this command and before another RACT command will have the property index automatically set to  $\langle \text{index} \rangle$ .

#### 9.4.32 RLIST Command

RLIST,  $\langle r1 \rangle$ ,  $\langle r2 \rangle$ ,  $\langle \text{step} \rangle$

lists the table of real properties between  $\langle r1 \rangle$  and  $\langle r2 \rangle$  (default: last defined) with step  $\langle \text{step} \rangle$  (default :1).

#### 9.4.33 RSET Command

RSET,  $\langle e1 \rangle$ ,  $\langle e2 \rangle$ ,  $\langle \text{inc} \rangle$ ,  $\langle \text{index} \rangle$

sets the real / spring / matrix property index to  $\langle \text{index} \rangle$  for the elements in the active group from  $\langle e1 \rangle$  to  $\langle e2 \rangle$  (default:  $\langle e1 \rangle$ ) with step  $\langle \text{inc} \rangle$  (default: 1). If the active group is of type BEAMS, then the index refers to the real property table. If the active group is of type SPRING, then the

index refers to the spring property table. If the active group is of type GENERAL, then the index refers to the matrix property table.

#### 9.4.34 SC Command

`SC,<nm>,<scx>,<scy>,<scz>,<scxx>,<scyy>,<sczz>,<damp>`

defines the spring property set number <nm> with the translational spring constants <scx>, <scy>, and <scz> and the rotational spring constants <scxx>, <scyy>, and <sczz>. <damp> defines the damping ratio of the spring.

**Note:** Spring constants are directly added to the global stiffness matrix. Thus, these constants must be given in the global xyz directions. The spring constants in the six global directions are uncoupled.

#### 9.4.35 SCLIST Command

`SCLIST,<r1>,<r2>,<step>`

lists the table of spring properties between <r1> and <r2> (default: last defined) with step <step> (default :1).

#### 9.4.36 THICK Command

`THICK,<e1>,<e2>,<inc>,<thick>`

sets the element thickness to <thick> for the elements between <e1> and <e2> with step <inc> (default: 1) from the SHELL active group.

### 9.5 Load Commands

Command	Action	Description
F	Defines a force.	Section 9.5.1
FDEL	Deletes forces.	Section 9.5.2
FLIST	Lists forces.	Section 9.5.3
FSCALE	Scales forces.	Section 9.5.4
MM	Defines a moment.	Section 9.5.5
MMDEL	Deletes moments.	Section 9.5.6
MMLIST	Lists moments.	Section 9.5.7
MR	Defines a rotational mass.	Section 9.5.8
MRGEN	Generates rotational masses by translation.	Section 9.5.9
MRDEL	Deletes rotational masses.	Section 9.5.10
MRSCALE	Scales rotational masses.	Section 9.5.11

Command	Action	Description
MSCALE	Scales moments.	Section 9.5.12
MT	Defines a translational mass.	Section 9.5.13
MTDEL	Deletes translational masses.	Section 9.5.14
MTGEN	Generate translational masses by translation.	Section 9.5.15
MTLIST	Lists translational and rotational masses.	Section 9.5.16
MTSCALE	Scales translational masses.	Section 9.5.17
MUNITS	Sets units for translational and rotational masses.	Section 9.5.18

### 9.5.1 F Command

`F,<n>,<fx>,<fy>,<fz>,<tx>,<ty>,<tz>`

defines the force with force factors `<fx>`, `<fy>`, `<fz>` and force arrival times `<tx>`, `<ty>`, `<tz>` in node `<n>`.

### 9.5.2 FDEL Command

`FDEL,<n1>,<n2>,<inc>`

deletes the forces from the nodes between `<n1>` to `<n2>` (default: `<n1>`) with step `<inc>` (default: 1).

### 9.5.3 FLIST Command

`FLIST,<n1>,<n2>,<inc>`

lists the forces from the nodes between `<n1>` and `<n2>` with step `<inc>` (default: 1). If no parameters are specified, all forces will be listed.

### 9.5.4 FSCALE Command

`FSCALE,<n1>,<n2>,<inc>,<sx>,<sy>,<sz>`

scales the forces belonging to the node set defined by `<n1>` to `<n2>` with step `<inc>`, multiplying the force factors with the scaling factors `<sx>`, `<sy>`, `<sz>`. If one scaling factor has the value 0.0, it will be set to 1.0. If `<n1>` and `<n2>` are not specified, they will be set to the last 2 defined nodal forces. The default value for `<inc>` is 1. This instruction works similarly to the NSCALE instruction from the Node Command group, with the difference that forces are modified.

**9.5.5 MM Command**

MM,<n>,<fxx>,<fyy>,<fzz>,<txx>,<tyy>,<tzz>

defines the moment with moment factors <fxx>, <fyy>, <fzz> and moment arrival times <txx>, <tyy>, <tzz> in node <n>.

**9.5.6 MMDEL Command**

MMDEL,<n1>,<n2>,<inc>

deletes the moments from the nodes between <n1> to <n2> (default: <n1>) with step <inc> (default: 1).

**9.5.7 MMLIST Command**

MMLIST,<n1>,<n2>,<inc>

lists the moments from the nodes between <n1> and <n2> with step <inc> (default: 1). If no parameters are specified, all moments will be listed.

**9.5.8 MR Command**

MR,<n>,<mxx>,<myy>,<mzz>

defines the rotational masses <mxx>, <myy>, and <mzz> in node <n> in weight units.

**9.5.9 MRGEN Command**

MRGEN,<itim>,<ninc>,<n1>,<n2>,<inc>,<mxx>,<myy>,<mzz>

generates <itim> (default: 1) sets of rotational masses in the nodes specified by the numbers obtained by incrementing with <ninc> (default value: <n2>-<n1>+1) the node numbers belonging to the node pattern defined by <n1> to <n2> with step <inc> (default: 1); while at the same time incrementing the pattern masses with <mxx>, <myy>, and <mzz> (default: 0). This instruction works similarly to the NGEN Command, with the difference that rotational masses and not coordinates are generated.

**9.5.10 MRDEL Command**

MRDEL,<n1>,<n2>,<inc>

deletes the rotational masses from the nodes between <n1> to <n2> (default: <n1>) with step <inc> (default: 1).

**9.5.11 MRSCALE Command**

MRSCALE,<n1>,<n2>,<inc>,<sx>,<sy>,<sz>

scales the rotational masses belonging to the node set defined by <n1> to <n2> with step <inc>, with the scaling factors <sx>, <sy>, <sz>. If one scaling factor has the value 0.0, it will be set to 1.0. If <n1> and <n2> are not specified, they will be set to the last 2 defined rotational masses. The default value for <inc> is 1. This instruction works similarly to the NSCALE command, with the difference that rotational masses are modified.

### 9.5.12 MSCALE Command

MSCALE,<n1>,<n2>,<inc>,<sx>,<sy>,<sz>

scales the moments belonging to the node set defined by <n1> to <n2> with step <inc>, multiplying the moment factors with the scaling factors <sx>, <sy>, <sz>. If one scaling factor has the value 0.0, it will be set to 1.0. If <n1> and <n2> are not specified, they will be set to the last 2 defined nodal moments. The default value for <inc> is 1. This instruction works similarly to the NSCALE command, with the difference that moments are modified.

### 9.5.13 MT Command

MT,<n>,<mx>,<my>,<mz>

defines the translational masses <mx>, <my>, and <mz> in node <n>

### 9.5.14 MTDEL Command

MTDEL,<n1>,<n2>,<inc>

deletes the translational masses from the nodes between <n1> to <n2> (default: <n1>) with step <inc> (default: 1).

### 9.5.15 MTGEN Command

MTGEN,<itim>,<ninc>,<n1>,<n2>,<inc>,<mx>,<my>,<mz>

generates <itim> (default: 1) sets of translational masses in the nodes specified by the numbers obtained by incrementing with <ninc> (default value: <n2>-<n1>+1) the node numbers belonging to the node pattern defined by <n1> to <n2> with step <inc> (default: 1); while at the same time incrementing the pattern masses with <mx>, <my>, and <mz> (default: 0). This instruction works similarly to the NGEN command, with the difference that translational masses and not coordinates are generated.

### 9.5.16 MTLIST Command

MTLIST,<n1>,<n2>,<inc>

lists the translational and rotational masses of the nodes between <n1> and <n2> with step <inc> (default: 1). If no parameters are specified, all masses will be listed.

### 9.5.17 MTSCALE Command

MTSCALE,[<n1>],[<n2>],[<inc>],[<sx>],[<sy>],[<sz>]

scales the translational masses belonging to the node set defined by <n1> to <n2> with step <inc>, with the scaling factors <sx>, <sy>, <sz>. If one scaling factor has the value 0.0, it will be set to 1.0. If <n1> and <n2> are not specified, they will be set to the last 2 defined nodal translational masses. The default value for <inc> is 1. This instruction works similarly to the NSCALE command, with the difference that translational masses are modified.

### 9.5.18 MUNITS Command

MUNITS,<n1>,[<n2>],[<step>],<units>

sets the units for the translational and rotational masses belonging to the node set defined by start node <n1>, end node <n2> (default: <n1>), and <step> (default: 1) to mass units if <units> = 0 or weight units if <units>=1.

## 9.6 ACS SASSI User Interface Commands

The commands in the next sections are only valid in the ACS SASSI User Interface, and will not work in PREP in older version of ACS SASSI. The user must use the full command name in all of the commands in these tables.

### 9.7 ACS SASSI User Interface General Commands

Command	Action	Description
ACTM	Change the Active model	Section 9.7.1
AFWRBAT	Split SSI analysis set into multiple sets	Section 9.7.2
CPMODEL	copy the active model to another model	Section 9.7.3
CRITFREQ	frequencies where the interpolation and actual simulated results differ	Section 9.7.4
DMODEL	Delete a model from ACS SASSI User Interface memory	Section 9.7.5
ETYPEGEN	Modify the element types for the models	Section 9.7.6
FIXSLDROT	Fix the solid rotations in a model	Section 9.7.7
FIXSHLROT	Fix the shell rotations in a model	Section 9.7.8
FIXSPRROT	Fix the spring rotations in a model	Section 9.7.9
FIXROT	Fix the rotations in a model	Section 9.7.10
FRAMECOMBIN	Combine frame files	Section 9.7.11
FRAMESEL	Identify local max/min of time history	Section 9.7.12
GCOM	Compress group numbers	Section 9.7.11
GETENV	Display solver environment variables	Section 9.7.14
GLB2LOC	Transform a global node to a local coordinate system	Section 9.7.15
GRAVITY	Sets Gravity value for the model	Section 9.7.16
GROUNDELEV	Sets Ground Elevation for the model	Section 9.7.17
GROUPMAT	Sets a material for each group	Section 9.7.18
INTGEN	generate interaction node if embedment is explicitly	Section 9.7.19

Command	Action	Description
	defined	
MDL	Change the Model name and path for uses with save, resume and AFWRITE	Section 9.7.20
MDLNAME	Change the Model Name without changing the path or model title	Section 9.7.21
NCOM	Compress Node Numbers in a Model	Section 9.7.22
MODELLIST	Show the models that have been defined and their names	Section 9.7.23
MODFRAMES	Modify legacy frames files for the new frame format	Section 9.7.24
RADIUS	Write a radius for non uniform soil	Section 9.7.25
RMVUNUSED	Remove Unused nodes from the model	Section 9.7.26
SETENV	Set solver environmental variables	Section 9.7.27
TPSD	Assign target PSD to spectrum number in EQUAKE	Section 9.7.28

### 9.7.1 ACTM Command

ACTM,<Model>

Changes the active model in the ACS-SASSI UI. The active model is the model on which user wants to operate. All subsequent commands are executed on the selected model number defined in <model> argument.

If the model already exist in memory this command will not change any model data. If the model does not exist the ACTM command will create an empty model (No nodes, elements, ... and default simulation setting)

<Model> – The integer number assigned to the model

### 9.7.2 AFWRBAT Command

AFWRBAT,<plits>

The AFWRBAT command allows the user split a simulation across multiple systems by frequency set into multiple models each with a separate frequency subset in separate folders. The folders can then be transferred to different systems and batch files created by this command run the necessary modules, then the data from each model can be combined by another batch file created by this command. The command uses model data defined by MDL command to determine the name and location of the new folders that are created. This command also uses the Module locations defined in the User Interface location window. The location of the ACS SASSI installation must be the same on all systems used or the user must modify the batch file to reflect the ACS SASSI installation on each system.

<plits> - number sections the frequency set will be split into.

### 9.7.3 CPMODEL Command

CPMODEL,<Mdl>

This command copies the active model into the model number referenced by the user input

Mdl - Destination Model Number of the active model copy

### 9.7.4 CRITFREQ Command

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

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

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

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

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

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

### 9.7.5 DMODEL Command

DMODEL,<Mdl>

Deletes the model specified by reference number in the <Mdl> argument.

<Mdl> – Model Number of the model to be deleted, Removes the model from ACS SASSI User Interface memory only, and does not delete any saved files or databases

### 9.7.6 ETYPEGEN Command

ETYPEGEN,<type>

The SOLID in 3D SSI models and the PLANE elements in 2D SSI models are considered to be either a part of the Structure or a part of the Excavated soil based on their assigned type. The type is implicitly defined by the zero default value (ETYPE = 0). Based on the zero default value, ACS SASSI assumes that any SOLID or PLANE element below ground surface is an Excavation element. The element type can be also defined explicitly: EYPE = 1 for Structure an ETYPE = 2 for Excavation.

For SHELL elements, the ETYPE = 2 implies that these elements are buried or sitting on the soil, so that their nodes are interaction nodes that are not included in the Excavation volume.

The AFWRITE command determines if implicitly defined elements are structural or excavation \*.hou file based on the ground surface level. However, when using WRITE the implicit ETYPE definitions are maintained in the \*.pre file. The ACS SASSI User Interface has commands that require explicitly defined types.

Please note that all elements that are not defined to be structural will be ignored by the ANSYS® interface in Option A. This command will define explicitly the type elements.

<type> - The element type selector.

- 0 - element type is defined by the location with respect to ground surface level
- 1 - change all elements to Structure
- 2 - change all elements to Excavated Soil

### 9.7.7 FIXSLDROT Command

FIXSLDROT

This command fixes the rotational degrees of freedom of all the nodes that are only connected to Solid elements. This command in combination with FIXSHLROT, and FIXSPRROT is an alternate to FIXROT that more closely approximates stiffness matrix for a models that are generated by ANSYS®.

### 9.7.8 FIXSHLROT Command

FIXSHLROT,[stiff]

This command applies soft rotational springs in shell planes to all nodes that are connected to coplanar shells. The overall spring stiffness is determined by the stiffness argument of the command and applied along the normal shell's plane. This command in combination with FIXSLDROT, and FIXSPRROT is an alternate to FIXROT that more closely approximates stiffness matrix for a models that are generated by ANSYS®.

---

<stiff> – Stiffness of springs added to shells to remove the singularities. (default stiffness = 10)

### 9.7.9 FIXSPRROT Command

FIXSPRROT

This command fixes the rotational degrees of freedom of all the translational spring elements that are connected to only springs and springs/solid node connections. If the node is only connected to a spring the fixed degrees of freedom are determined by the spring stiffness. If the node is connected to springs and solids the rotation degrees of freedom are determined by the springs rotational stiffness. This command in combination with FIXSHLROT, and FIXSLDROT is an alternate to FIXROT that more closely approximates stiffness matrix for a models that are generated by ANSYS®.

### 9.7.10 FIXROT Command

FIXROT,[Stiff]

Command to fix rotations of solid elements connected only with solid elements and in-plane rotations for shell that are connected only with shells. This command will attempt to fix rotations by applying automatically the D command to fix rotations that shouldn't be free. However, shell faces that are not parallel to a global coordinate system plane cannot be fixed in this way. Therefore, this command will also add soft rotational springs at the shell nodes in the plane of these shells. The user can control the stiffness of this spring by the stiff parameter of the command.

<Stiff> – Stiffness of springs added to shells to remove the singularities. (default stiffness = 10)

### 9.7.11 FRAMECOMBIN Command

FRAMECOMBIN,<op>,<num>,<InFile1>, ...,<InFileX>,<Outfile>

Combine ASCII frame files generated by modules and combine them to make a new animations. This command requires the header of frame files to specify the number of rows columns in the frame file. This is written by default in the current version of ACS SASSI, however frames written in previous version will have the legacy header format, and will need to be converted. If the frames use the legacy header format, use the MODFRAMES command to change the frame headers of the files.

- op - operation code controls the way frames are to be combine.
  - 0 - SRSS
  - 1 - sum
  - 2 - average
- num - number of input frames to be combine.

- Infile - full path of the input frame files.
- Outfile - full path of output frame file.

### 9.7.12 FRAMESEL Command

FRAMESEL,<tol>,<Acc>,<Var>

This command allows the users to find local maximums/minimums in an acceleration time history and store these in a variable. Based on the list in the variable, the user can determine where in an animation the critical frames are found.

- tol - percentage of the global maximum below which local maximums will be ignored as critical frames
- Acc - Acceleration file to be processed
- Var - Variable Name to store the list of critical frames

### 9.7.13 GCOM Command

GCOM

The GCOM command will compress group number so there are no gaps in the group numbering. The new group numbering will start a 1. The relative order of the groups will not change and this command does nothing to compress the element numbers in each group (use ECOMPR to compress element numbers inside of groups).

### 9.7.14 GETENV Command

GETENV

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

### 9.7.15 GLB2LOC Command

GLB2LOC,<Start>,<End>,<Stride>,<Sysno>

Changes the coordinate system from global to a local coordinate system. Currently this command only allows for transformation into a local Cartesian system. The command will transform all requested nodes in local systems into the global system before applying the local system transformation.

- Start - the first node to transform
- End – The last node to transform
- Stride – the stride increment between nodes
- Sysno – The local system number to use in the transformation

**9.7.16 GRAVITY Command**

GRAVITY,&lt;grav&gt;

This command sets the gravity constant for the model. This constant is also set by the HOUSE command. This allows the user to modify gravity without overwriting all of the other options set by the HOUSE command.

<grav> - The gravity constant for the model.

**9.7.17 GROUNDELEV Command**

GROUNDELEV,&lt;elev&gt;

This command sets the ground elevation constant for the model. This constant is also set by the HOUSE command. This allows the user to modify ground elevation without overwriting all of the other options set by the HOUSE command.

<elev> - The ground elevation for the model.

**9.7.18 GROUPMAT Command**

GROUPMAT

This Command will create a new material for each group and assign it to all of the elements in the group. The default values for the new materials are based on the original material of the first element in each group. After execution the original material list will have been deleted and the new material list will consist of the group materials. This command is intended to be used during the Nonlinear Modeling process.

**9.7.19 INTGEN Command**

INTGEN,&lt;type&gt;,[level skip]

Automatically generates interaction nodes for different substructuring approaches FV, FI-FSIN (SM), FI-EVBN (MSM), Surface model and Fast FV. Options 1-5 do not remove any interaction nodes that have been previously defined. The excavation volume must be explicitly defined by ETYPE or ETYPEGEN command for options 1-3 and 5. If the ETYPE of the elements is left to default this command will not work. It should be noted that for buried shell elements defined with an ETYPE value of 2, the shell nodes will be considered interaction nodes. The ETYPE value of 2 has no consequence for beam elements.

---

<type> – Type of iteration node generation

- 0 = Set all nodes to non-interaction nodes
- 1 = Embedded Foundation - Flexible Volume (FV)
- 2 = Embedded Foundation - Flexible Interface with Excavation Volume Boundary Nodes, denoted FI-EVBN or Modified Subtraction Method (MSM)
- 3 = Embedded Foundation - Flexible Interface with Foundation-Soil Interface Nodes, denoted FI-FSIN or Subtraction Method (SM)
- 4 = Surface Foundation (interaction nodes are only at the ground surface level)
- 5 = Embedded Foundation - Fast FV including multiple layers of internal interaction nodes

[level skip] - For option 5 number of levels skipped between interaction node levels (default = 1).

### 9.7.20 MDL Command

MDL,<Model>,<Path>

Command to define the model name and path. This was done in the previous PREP by the database and can still be done by database in the ACS SASSI User Interface. However, the ACS SASSI User Interface does not require the user to load the model into a database before using a model. Some commands require the name and path to be defined before they work (ex. AFWRITE). Therefore, this command makes the use of the database optional yet the user still can use the full functionality of the ACS SASSI User Interface without the database.

<Model> – Model Name

<Path> – Model Directory Path

All files written using the WRITE and AFWRITE commands will have the name specified in the <Model> argument with the appropriate extensions in the <Path> folder.

### 9.7.21 MDLNAME Command

MDLNAME,<name>

This command will change the model name without changing the model path or model title assigned with the TIT command. All files written using the WRITE and AFWRITE commands will have the name specified in the <name> argument with the appropriate extensions..

<name> - the new model name. All file WRITE and AFWRITE will have this name with the appropriate extensions

### 9.7.22 NCOM Command

NCOM

---

This command compresses the node list so there are no gaps in the defined node numbering in the current model. The command also update the node element connections in the model to reflect the new node numbering in the model. The command will maintain relative order of the nodes in the model.

ACS-SASSI has a modeling limitation of 99,999 as the maximum node number. Users may need to compress the numbering in node list for models converted from ANSYS® or when working with combined models (MERGE, MERGESOIL or SOILMESH) in ACS-SASSI. It is suggested that when a model is completed the user first remove any unused nodes from the model using the RMVUNUSED command then using the NCOM command it compress the model

### 9.7.23 MODELLIST Command

MODELLIST

List the models in memory by reference number and model name (if defined).

### 9.7.24 MODFRAMES Command

MODFRAMES,<cols>,<framelist>

This command is useful when old PREP frames are available for animations. It modifies the header of the frame files in an animation frame list. The command reads each frame in the <framelist> file, then changes the second number in the header to the number of columns specified by the user in the <cols> argument. The rest of the data frame file and the framelist file will remained unchanged. The <framelist> file used in this command is the same file that is used to load animation frames for processing (zpani, thiani, contani, thani file extensions).

<cols> - Number of columns in the frame files.

<framelist> - full path of the animation framelist file

### 9.7.25 RADIUS Command

RADIUS,<Scale>,<FileName>

This command writes a file containing the element radii for elements in the excavation module, as well as the average radius, to the file specified by <FileName>. This can be used to determine an appropriate radius for the POINT module for models with irregular excavation volumes.

<Scale> – Scale factor for the radius file

<FileName> – The full path name for the radius file

### 9.7.26 RMVUNUSED Command

RMVUNUSED

The RMVUNUSED Command checks element nodes and interaction nodes in the current model to see which nodes are being used. All unused, non-interaction nodes will be removed from the model. This command does not compress node numbers or change element node connections. The command should be used in conjunction with NCOM to compress the node list of a complete model.

### 9.7.27 SETENV Command

SETENV,<mem>

This command shall be used after software installation when the UI is launched for the first time. Sets the environment variables for the fast solver modules. The command sets 3 environment variables in the users registry. The environment variables are local to each user account and are persistent once they have been set. This command should be run by each user account that will use the fast solver. This command should only need to be run once per user unless the systems physical memory (RAM) size has changed.

<mem> - The Memory size limit that the fast solver is allowed to use in megabytes. It is suggested that user set the size limit at 90 - 95% of physical memory on the system. This is to allow the computers Operating System to allocate memory to background task without interfering with the solver. If the user attempts to allocate more than 100% of physical memory the fast solver has shown to return incorrect results.

### 9.7.28 TPSD Command

TPSD,<num>,<file>

Assign target PSD to spectrum number in EQUAKE

<num> - input file number to the target PSD file. (corresponds to the number other EQUAKE commands ACCIN,ACCOUT, RSIN, and RSOUT)

<file> - Full path file name of the target PSD file to be used.

## 9.8 Model Checking Commands

Command	Action	Description
EXCSTRCHK	Checks excavation nodes for potential errors	Section 9.8.1
FIXEDINT	Find Fixed Interaction Nodes	Section 9.8.2
FREESPRING	Find Free Spring Nodes	Section 9.8.3
HINGED	Check model for possible hinged connections	Section 9.8.4
INTCOUNT	Display a count of interaction nodes	Section 9.8.4
KINT	Find K node that are interaction nodes	Section 9.8.5
USED	Check and fix Unused nodes	Section 9.8.7

### 9.8.1 EXCSTRCHK Command

EXCSTRCHK

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

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

### 9.8.2 FIXEDINT Command

FIXEDINT

This command will search the model and find all interaction nodes in the active model that have fixed translational degrees of freedom. The results of this command will either tell the user that there are no fixed interaction nodes or the command will provide a list of all the fixed interaction nodes in the model. If there are fixed interaction nodes it is up to the user to modify the model so these conditions don't exist.

### 9.8.3 FREESPRING Command

FREESPRING

This command will find all unconstrained nodes that are only connected to a spring and warn the user about this condition at the nodes where it occurs.

### 9.8.4 HINGED Command

HINGED

This command checks for possible hinges in a model and warns the user about the places where this condition may occur. It is up to the user to check these locations in the model and determine if any action should be taken to correct the model.

Hinged conditions exist when two element types are connected, and one of those element types has more active node degrees of freedom than the other. This is common in models that connect beams and shell elements (6 DOFs) to solid elements (3 DOFs). In these cases, the 6 DOF elements should penetrate the solids along an edge or side to properly resolve to rotational degrees of freedom.

### 9.8.5 INTCOUNT Command

INTCOUNT

This command prints the number of interaction nodes to the command history. Interaction nodes can be defined with or deleted with commands INT and INTGEN. The number of interaction nodes tends to heavily influence stimulation run time.

### 9.8.6 KINT Command

KINT

The command will check the K-nodes of all beams to see if they are also defined as interaction nodes. K node that are interaction nodes may cause incorrect simulation results in cases where they are only used as K-nodes for beams. This command will report the nodes that have this issue to the user. This command does not change the model the user must determine if, or how the model is to be changed.

### 9.8.7 USED Command

USED

This command checks to find unused nodes and fixes them in the same way the using D command with the ALL keyword would fix the nodes.

## 9.9 Model Generation & Combination Commands

Command	Action	Description
EXCAV	Create an Excavation volume for a model	Section 9.9.1
MERGE	Merge 2 models	Section 9.9.2
MERGEGROUP	Merge 2 groups together	Section 9.9.3
MERGE PANEL	Merge a Panel Model to the Solids and beams of the original model	Section 9.9.4
MERGE SOIL	Merge a Structural Model With a Matching Soil	Section 9.9.5

Command	Action	Description
ROTATE	Rotate the model around a point	Section 9.9.6
SOILMESH	Create a soil mesh for the active model	Section 9.9.7
TRANSLATE	Translate all nodes a specified distance	Section 9.9.8
WELD	Combine nodes that share the same location	Section 9.9.9

### 9.9.1 EXCAV Command

EXCAV,<model>,[delta]

This command will create an excavation volume model using the geometry of an existing model. The excavation model created from this command will be stored in the model number assigned in the <model> argument. This command is used in conjunction with the MERGESOIL command to join the excavation and structural models.

The command will use the lowest z-level grid as template a to create a homogenous mesh model up to ground surface. If there is outcropping beneath ground surface that does not extend to the bottom z-level, the code will not generate excavation volume for these areas. The ground surface level must be defined properly in the model used in the generation. The delta is a factor used to match slight variations in z-level used in some models for the same embedment level. Models that don't have uniform Z coordinates across the floor should use a delta > 0 so the command doesn't generate multiple levels for the small variations of Z.

<model> - Model number to store the excavation volume

<delta> - allowable distance of z-level variation on a single level (Default = 0). Delta should be entered as a positive floating point number or else the default will be used.

### 9.9.2 MERGE Command

MERGE,<Mdl1>,<Mdl2>,<X>,<Y>,<Z>

Merges two models in memory and stores the resulting model in the current active model. In the resulting merged model, the <Mdl1> model will have all the same nodes, elements, etc. , while the <Mdl2> data will be appended to <Mdl1> data with proper renumbering and offsets. The <X>, <Y>, and <Z> arguments allow the user to move the Mdl2 nodes with a constant offset so <mdl2> can be repositioned in relation to <mdl1>. If the models need to be rotated use the ROTATE command prior to merging the models. See the MERGE example in the online help for an application of this command.

<Mdl1> - Model Number 1 to be combined

<Mdl2> - Model Number 2 to be combined

<X> - X offset for the second models nodes

<Y> - Y offset for the second models nodes

---

<Z> - Z offset for the second models nodes

### 9.9.3 MERGEGROUP Command

MERGEGROUP,<dest>,[G1], ... ,[G10]

This command allows the user to merge up to 11 groups together into a single group. The destination group (first argument) original element numbers will be unchanged. Each subsequent groups' elements number will be offset by destination groups maximum element number when it is added to the destination. All of the merge groups (G1-G10) will be removed at the end of the merge leaving only the destination group in the model.

All of the groups to be merged must be of the same element type.. After the command execution, the models group numbering will have gaps due to the deleted groups. Users may want to compress group numbers using the GCOM command after merging groups.

<dest> - Group Number for the destination group. This group will contain all of the elements of the groups that are to be merged as a result of this command.

[G1] - [G10] - Groups that are to be added to dest group. these groups will be erased as a result of this command.

### 9.9.4 MERGEPANEL Command

MERGEPANEL,<Panel>

This command is a special merge command to combine panel model with the non shell elements of the original model. The user should set the active model to the original model using ACTM. The user should delete the shell group(s) from the original model because these shells will be replaced by the panel model. The user should then use the MERGEPANEL command with the model number for the panel model as the argument. The result will be the combination of the original model with the panel model groups and materials added to the end of the group and material list of the original model.

<Panel> - the model number of the panel model

### 9.9.5 MERGESOIL Command

MERGESOIL,<Struct>,<Soil>,[Mode],[Stiff],[Stiff2],[SepLevel],[Mapping]

Merge a Structural and excavation volume model together in an active model. All elements in the soil model will be converted into soil elements and the materials will be converted to soil layers. See the MERGESOIL Example in the online help for usage. During the node merging process, the higher number node will no longer be used, and can be removed with the RMVUNUSED command.

<Struct> – Model Number of the Structure

<Soil> – Model Number of the Excavation volume

<Mode> - merging the nodes on the FE model and the excavation volume interface

0 = Fully unbounded interface for all sides

1 = Fully bounded interface for all sides by merging nodes (default)

2 = Stiff spring interface connection for all sides

3 = Stiff spring interface connection below the <SepLevel> separation level and soft springs above this level

<Stiff> - Stiffness constant for stiff springs in Mode = 2 and =3. (Default =  $10^7$ )

<Stiff2> - Stiffness constant for soft springs above separation level (Default = 10)

<SepLevel> - Global Z coordinate for soil separation level

<Mapping> - Mapping file for nodes of the excavation volume model, old node and new node.

### 9.9.6 ROTATE Command

ROTATE,<x>,<y>,<z>,<rx>,<ry>,<rz>

This command will rotate a model's nodes based on a central point and a rotation around each axis in degrees. This command can be used with the MERGE command to orient models relative to each other for SSSI analysis.

<x> – center of the X rotation

<y> - center of the Y rotation

<z> - center of the Z rotation

<rx> – rotation around Z axis

<ry> – rotation around the X axis

<rz> – rotation around the Y axis

### 9.9.7 SOILMESH Command

SOILMESH,< dest >,< sX >,< sY >,< hori > ,<vert >,< xAdj>,<yAdj>,<Zdepth>,<contact>,<rNum>

This command allow the user to make a soil field mesh for a model. This mesh is used to analyze pressure on the excavation volume. The basement must be a continuous convex hull that has uniform external wall along all layers.

- dest – destination model number
- sX - floating point percentage of growth in the X direction of each level
- sY – floating point percentage of growth in the Y direction of each level
- hori – number of new horizontal layers
- vert – number of vertical vertical layers
- xAdj – floating point Centroid correction in the X direction

- yAdj – floating point Centroid correction in the Y direction
- Zdepth - floating point Thickness of each new level in the Z direction
- contact – if equal 0 do not use contact surfaces, else include contact surfaces in the model.
- rNum – the real property number for the contact surface

**Note:** This command created before MERGE and other model combination commands existed. It is recommended that the model created by this command should be saved using WRITE then loaded over the original model to create the soil pressure model. MERGE can be used to combine the two models, however both MERGE and SOILMESH both apply an offset to the group numbers, element number, node numbers, etc. Using MERGE and SOILMESH together will double the offset causing a gap in the models group numbers, element number, node numbers, etc.

### 9.9.8 TRANSLATE Command

TRANSLATE,<x>,<y>,<z>

This command move all of the nodes in the models node list the specified amount. This command does not change any model data other than the node locations.

<x> - amount of translation in the X direction (default = 0)

<y> - amount of translation in the Y direction (default = 0)

<z> - amount of translation in the Z direction (default = 0)

### 9.9.9 WELD Command

WELD

The WELD command will combine nodes that occupy the same location. The command does this by searching the node list for node that occupy the same location to create a substitution table. Then the SASSI User Interface will use the table to change the node connections of the elements. This command was designed to work with excavation only and structure only models before using MERGESOIL.

**Note:** The node that have been substituted will still be in the Node list but they will not be attached to any elements. The WELD command does not change the mass, boundary condition, or interaction status of any node. The user may need to readjust these parameters before simulation. This command will change the element connections on all coincident nodes even in intended situations such as models with structural and excavation elements, or models that have springs applied by the FIXROT command.

## 9.10 Cut & Submodeling Commands

Command	Action	Description
CSECT	create a cross sectional model	Section 9.10.1
CUT2SUB	Transfer Elements in a cut to another model	Section 9.10.2
CUTADD	Add elements from a group to a cut	Section 9.10.3
CUTCLR	Clear a cut in memory	Section 9.10.4
CUTRMV	Remove elements from a cut	Section 9.10.5
CUTVOL	Add elements in a volume to a cut	Section 9.10.6
EXTRACTEXCAV	Make a submodel of the excavation volume	Section 9.10.7
SLICE	Add elements to a cut that lie along a plane	Section 9.10.8
SPLITGROUP	Split a group into 2 different groups	Section 9.10.9
TRANELEM	Transfer a list of element in the current model to another model	Section 0
TRANVOL	Transfer all elements in a volume to another model	Section 9.10.11

### 9.10.1 CSECT Command

CSECT,<dest>, <cutnum>,<pointx>,<pointy>,<pointz>,<normalx>,<normaly>,<normalz>

Creates a cross-section model for design calculations. The user must define a cut to be used and the destination of the cross-section in the first two argument. The rest of the arguments define an infinite plane by a point on the plane and the normal to the plane. The ACS SASSI UI will then take a cross-section of all the elements in the cut at the infinite plane. The elements cross-section then will be expanded to a unit thickness so the user can plot the cross-sectional model. This model is used to perform design calculations on the cross section using the commands found in the Cut & Submodeling commands section of the command reference.

<dest> – The model number the new model is to be stored in

<cutnum> – number of the cut

<pointx> – x of point on the plane

<pointy> - y of point on the plane

<pointz> - z of point on the plane

<normalx> – x of the normal of the plane

<normaly> – y of the normal of the plane

<normalz> – z of the normal of the plane

### 9.10.2 CUT2SUB Command

Cut2Sub,<cutnum>,<dest>,[solid]

This command make a submodel out of a cut defined by the user. The active model and the user defined cut data are used to create the submodel. The user also has the option to create a thick

shell model using the solid option. The thick shells should be used for plotting only because the geometry generation algorithm has no way to test if the newly formed solids intersect each other.

<cutnum> – Number of the cut to be used

<dest> – number of the destination model

<solid> – Turn all shells in to solid elements if solid  $\geq 1$  (Default solid = -1)

### 9.10.3 CUTADD Command

CutAdd,<cutnum>,<group num>,<elem 1>, ... <elem N>

CutAdd,<cut num>,<group num>,RANGE,<elem start>,[elem end], [stride]

Both versions of this command allow the user to add elements to a cut. The first version of this command is intended for the user to enter a non-sequential list of elements from one group while the second command is intended for use with a continuous list of elements or a list with regular gaps between desired elements. This command will create the cut in memory if no cut has been defined for the cut num.

- cut num – number of the cut
- group num – number of the group where the elements will be added
- elem 1, elem N – element numbers to be added
- elem start – first element to be added
- elem end - last element to be added (default: start element number)
- stride - increment to the next element (default: 1)

### 9.10.4 CUTCLR Command

CUTCLR, <first cut>, [last cut],[step]

Clear a cut or list of cuts from memory. Once the cut is cleared, it no longer exists in memory. The user can redefine it by adding elements to the cut again but attempts to plot or use the cut for submodeling purposes will yield errors.

- first cut – number of the first cut to be deleted
- last cut – number of the last cut to be deleted (default = first cut)
- step – step between deleted cuts (default = 1)

### 9.10.5 CUTRMV Command

CutRMV,<cut num>,<group num>,<elem 1>, ... <elem N>

CutRMV,<cut num>,<group num>,RANGE,<elem start>,[elem end], [stride]

The two forms of this command remove elements from a cut data structure. The forms are similar to the forms CUTADD command;

- cut num – number of the cut
- group num – number of the group where the elements will be removed
- elem 1, elem N – element numbers to be removed
- elem start – first element to be removed
- elem end last element to be removed (default: element start)
- stride – increment of the next element to be removed (default: 1)

### 9.10.6 CUTVOL Command

CutVol,<cutnum>,[Xmin],[Xmax],[Ymin],[Ymax],[Zmin],[Zmax]

This command adds elements to the cut data structure defined by the user with the <cutnum> argument by volume. The volume is defined by the other arguments in the command. The user can leave any or all of the volume boundaries empty. By default any boundary argument left empty will be replaced by the appropriate minimum or maximum for the active model.

<cutnum> – Number of the destination cut

[Xmin] - Minimum x of the box

[Xmax] - Maximum x of the box

[Ymin] - Minimum y of the box

[Ymax] - Maximum y of the box

[Zmin] - Minimum z of the box

[Zmax] - Maximum z of the box

### 9.10.7 EXTRACTEXCAV Command

EXTRACTEXCAV, <Model>

This command will make a submodel of the excavation volume of the current model. All excavation elements must be explicitly defined for this command to work properly. (use ETYPEGEN) This command will not change the current active model it will only copy data into the destination model.

Model - model number of the excavation submodel

### 9.10.8 SLICE Command

SLICE, <cutnum>,<pointx>,<pointy>,<pointz>,<normalx>,<normaly>,<normalz>

---

This command will load a cut defined by cutnum with all of the elements that cross an infinite plane defined by the users point and normal entry. The Slice command will add all the new elements found by slice to the cut defined by cutnum already in memory. In earlier versions of the User Interface as the SUBMODELER module this command will replace the cut defined by cutnum. This change was made so the command functions like the other cut commands CUTADD and CUTVOL

- cutnum – number of the cut
- pointx – x of point on the plane
- pointy - y of point on the plane
- pointz - z of point on the plane
- normalx – x of the normal of the plane
- normaly – y of the normal of the plane
- normalz – z of the normal of the plane

### 9.10.9 SPLITGROUP Command

SPLITGROUP,<group>,<split>,[dir]

This command allows the user to split a group using the same algorithm that the PANELIZE command uses to split groups or a plane perpendicular one of global axes. This command only splits the group specified in the first argument. The group to be split can be any group in the model with any element type.

When using the dir argument the split argument will contain the floating-point location of the splitting plane in the global coordinate system. Otherwise, the split argument is group number of a shell group that will be used to define the splitting plane. The second group must be shell elements. The command was designed to be used with the WALLFLR command so the first shell in the group is used to define the splitting plane.

- group - The group number to be split.
- split - The group used as a reference for the split plane if the dir argument is not used. If using dir, the split argument defines location of the splitting plane in the global coordinate system.
- dir - Orientation of the splitting plane. (Added in User Interface 1.0.2)
  - X - Split the group with a plane perpendicular to the X direction of global coordinate system
  - Y - Split the group with a plane perpendicular to the Y direction of global coordinate system
  - Z - Split the group with a plane perpendicular to the Z direction of global coordinate system

### 9.10.10 TRANELEM Command

TRANELEM,<dest>,<group>,<begin>,<end>,<stride>

Transfers a list of elements from the active model directly to another model. These elements will be written/ overwritten in the destination model location. The group type in the destination will be modified and any existing elements destination model will be overwritten. All nodes that are referenced by the transferred element will also be overwritten in the destination model. The active model will be unchanged.

- dest – Number of the destination model
- group – Group number for element transfer
- begin – beginning element in the transfer list
- end – end number in the transfer list
- stride – element stride

### 9.10.11 TRANVOL Command

TRANVOL,<dest>,[Xmin],[Xmax],[Ymin],[Ymax],[Zmin],[Zmax]

Transfer all of the elements in a user defined volume into another directly model specified by destination reference number. The volume of the cube is specified by the minimum and maximum in global coordinates. The user can leave any of the boundary arguments blank this will make the code use the default of the model extent for that argument. The group types, elements, and nodes in the destination model will be changed.

- dest – Number of the destination model
- Xmin - Minimum x of the box
- Xmax - Maximum x of the box
- Ymin - Minimum y of the box
- Ymax - Maximum y of the box
- Zmin - Minimum z of the box
- Zmax - Maximum z of the box

## 9.11 File Conversion Commands

Command	Action	Description
ANSYS	Write model to ANSYS® input format	Section 9.11.1
ANSYSMODELTYPE	Change the model type for the Advanced ANSYS® option	Section 9.11.2
ANSYSREFORMAT	Reformat the beam groups to work better for ANSYS® write	Section 9.11.3

Command	Action	Description
CONVERT	Use one of the file converters to translate a file	Section 9.11.4
GENMATRIXDAMP	Convert the Stiffness in converted MATRIX27 elements into complex stiffness for SASSI general elements (Not Applicable to Current Version)	Section 9.11.5

### 9.11.1 ANSYS Command

ANSYS,[FileName],[Dir]

Writes a model in ANSYS® ADPL file format. The user can select a non-default location for this file by using the filename and dir arguments. The default uses the model information to determine the file name and location. Not all models are directly compatible between ACS SASSI and ANSYS®. It may be necessary to use ANSYSREFORMAT before using the ANSYS Command.

- FileName – The name of the file to be written
- Dir – The directory for the file to be written

### 9.11.2 ANSYSMODELTYPE Command

ANSYSMODELTYPE,<type>

This command sets the models operation mode for Advanced ANSYS® models.

<type> - Analysis type selection for advanced ANSYS®

- 1 - Changes operation mode to embedded
- 2 - Changes operation mode to surface

### 9.11.3 ANSYSREFORMAT Command

ANSYSREFORMAT,<Org>,<Map>

Due to the differences between ANSYS® and the ACS SASSI data format, some Beam element groups in ACS SASSI cannot be converted to ANSYS® Workbench™ correctly through direct translation. This command takes an ACS SASSI model and reformats the elements into different groups so the beams will be translated to ANSYS® ADPL format correctly. This is necessary due to a difference in the way beam end release conditions are handled between ACS SASSI and ANSYS®. ACS SASSI allows for beam end releases to be set per element, while ANSYS® handles these through element type key options. This command should be used with an empty active model.

<Org> – Model Number of the model to be reformatted

<Map> - A file name for the ACS SASSI User Interface to write a mapping file for the old and new beam groups

### 9.11.4 CONVERT Command

CONVERT,<ConSel>,<model>,<filename>,...

This command gives the user access to the file converters without using the menu command and without the need of the interactive GUI. The ANSYS® and the GT STRUDL® converters have additional arguments after the file name.

<ConSel>

SSI - for \*.hou to model conversion

ANSYS - for \*.cdb to model conversion

STRUDL - for GT STRUDL® to model conversion (not applicable in this version)

<model> - destination model number

<filename> - Full path name for the file to be converted.

#### **ANSYS® Converter Command**

CONVERT,ANSYS,...,<gravity>

<gravity> - gravity value for the model being converted

#### **GTSTRUDL® Converter Command (Not Applicable in This Version)**

CONVERT,STRUDL,...,<group>,<member>,<fematrib>,<prop>,<const>,<input>,<tie>

This converter has had very limited testing and should not be used for NQA applications. The joint file from the database should be placed in the filename name with its full path. the other files from the GT STRUDL® database should have the file names listed without the path. The converter will append the path to the rest of the files. All of the database files should reside in the same directory.

<group> - group database file

<member> - member database file

<fematrib> - FEM attribute file

<prop> - properties file

<const> - constraints file

<input> - mass input file

<tie> - spring properties

### 9.11.5 GENMATRIXDAMP Command (NOT USABLE IN THIS VERSION)

GENMATRIXDAMP,<begin>,[end],[stride],<damp>

ACS SASSI can convert the ANSYS® MATRIX27 element connections and the ANSYS® mass matrix directly from the \*.cdb. However, ACS SASSI uses a complex stiffness that can't be converted directly from the data in ANSYS® .cdb file. The GENMATRIXDAMP will take the stiffness matrix definition in ANSYS® and a user defined damping value to convert the stiffness matrix from ANSYS® into a complex stiffness matrix that ACS SASSI can use.

<begin> - the first matrix property to be converted

[end] - last matrix property to be converted. When using default (or left blank) the end value will be set equal to the begin value. (default = -1)

[stride] - increment for the next matrix property to be converted (default = 1)

<damp> - user defined damping value

## 9.12 Module Commands

Command	Action	Description
RUNANALYS	Run the ANALYS module in the ACS SASSI User Interface	Section 9.12.1
RUNCOMBIN	Run the COMBIN module in the ACS SASSI User Interface	Section 9.12.2
RUNEQUAKE	Run the EQUAKE module in the ACS SASSI User Interface	Section 9.12.3
RUNFORCE	Run the FORCE module in the ACS SASSI User Interface	Section 9.12.4
RUNHOUSE	Run the HOUSE module in the ACS SASSI User Interface	Section 9.12.5
RUNMOTION	Run the MOTION module in the ACS SASSI User Interface	Section 9.12.6
RUNPOINT	Run the POINT module in the ACS SASSI User Interface	Section 9.12.7
RUNRELDISP	Run the RELDISP module in the ACS SASSI User Interface	Section 9.12.8
RUNSOIL	Run the SOIL module in the ACS SASSI User Interface	Section 9.12.9
RUNSTRESS	Run the STRESS module in the ACS SASSI User Interface	Section 9.12.10

### 9.12.1 RUNANALYS Command

RUNANALYS,[model]

Run the ANALYS module on a model currently in ACS SASSI User Interface memory. A user must first use the AFWRITE command to generate the proper input file for the module. The models name and path need to be defined by the MDL or by using the model database. If the <model> argument default is used the ANALYS module will be run on the active model.

<model> - user selection of the model that will be used by the module (default = -1)

### **9.12.2 RUNCOMBIN Command**

RUNCOMBIN,[model]

Run the COMBIN module on a model currently in ACS SASSI User Interface memory. A user must first use the AFWRITE command to generate the proper input file for the module. The models name and path need to be defined by the MDL or by using the model database. If the <model> argument default is used the COMBIN module will be run on the active model.

<model> - user selection of the model that will be used by the module (default = -1)

### **9.12.3 RUNEQUAKE Command**

RUNEQUAKE,[model]

Run the EQUAKE module on a model currently in ACS SASSI User Interface memory. A user must first use the AFWRITE command to generate the proper input file for the module. The models name and path need to be defined by the MDL or by using the model database. If the <model> argument default is used the EQUAKE module will be run on the active model.

<model> - user selection of the model that will be used by the module (default = -1)

### **9.12.4 RUNFORCE Command**

RUNFORCE,[model]

Run the FORCE module on a model currently in ACS SASSI User Interface memory. A user must first use the AFWRITE command to generate the proper input file for the module. The models name and path need to be defined by the MDL or by using the model database. If the <model> argument default is used the FORCE module will be run on the active model.

<model> - user selection of the model that will be used by the module (default = -1)

### **9.12.5 RUNHOUSE Command**

RUNHOUSE,[model]

Run the HOUSE module on a model currently in ACS SASSI User Interface memory. A user must first use the AFWRITE command to generate the proper input file for the module. The models

name and path need to be defined by the MDL or by using the model database. If the <model> argument default is used the HOUSE module will be run on the active model.

<model> - user selection of the model that will be used by the module (default = -1)

### 9.12.6 RUNMOTION Command

RUNMOTION,[model]

Run the MOTION module on a model currently in ACS SASSI User Interface memory. A user must first use the AFWRITE command to generate the proper input file for the module. The models name and path need to be defined by the MDL or by using the model database. If the <model> argument default is used the MOTION module will be run on the active model.

<model> - user selection of the model that will be used by the module (default = -1)

### 9.12.7 RUNPOINT Command

RUNPOINT,[model]

Run the POINT module on a model currently in ACS SASSI User Interface memory. A user must first use the AFWRITE command to generate the proper input file for the module. The models name and path need to be defined by the MDL or by using the model database. If the <model> argument default is used the POINT module will be run on the active model.

<model> - user selection of the model that will be used by the module (default = -1)

### 9.12.8 RUNRELDISP Command

RUNRELDISP,[model]

Run the RELDISP module on a model currently in ACS SASSI User Interface memory. A user must first use the AFWRITE command to generate the proper input file for the module. The models name and path need to be defined by the MDL or by using the model database. If the <model> argument default is used the RELDISP module will be run on the active model.

<model> - user selection of the model that will be used by the module (default = -1)

### 9.12.9 RUNSOIL Command

RUNSOIL,[model]

Run the SOIL module on a model currently in ACS SASSI User Interface memory. A user must first use the AFWRITE command to generate the proper input file for the module. The models name and path need to be defined by the MDL or by using the model database. If the <model> argument default is used the SOIL module will be run on the active model.

<model> - user selection of the model that will be used by the module (default = -1)

### 9.12.10 RUNSTRESS Command

RUNSTRESS,[model]

Run the STRESS module on a model currently in ACS SASSI User Interface memory. A user must first use the AFWRITE command to generate the proper input file for the module. The models name and path need to be defined by the MDL or by using the model database. If the <model> argument default is used the STRESS module will be run on the active model.

<model> - user selection of the model that will be used by the module (default = -1)

## 9.13 Calculation Commands

Command	Action	Description
CALCC	Calculate the center of area for the model	Section 9.13.1
CALCSECTHIST	Calculate the stress history of a cross section based on multiple files	Section 9.13.2
CALCSECTHISTDB	Calculate the stress history of a cross section using the data from the binary stress database	Section 9.13.3
CALCM	Calculate the mass of the object	Section 9.13.5
CALCMOI	Calculate the moment of inertia	Section 9.13.6
CALCPAR	Calculate all six response quantities for a cross-section	Section 9.13.7
READSTR	Read the .ess stress file and apply to the model	Section 9.13.8
SECDATAOPT	Set the output request in stress for section cut data	Section 9.13.9
SHEAR	Calculate ultimate shear for a nonlinear panel	Section 9.13.9

### 9.13.1 CALCC Command

CALCC

Calculates the volume centroid for the active model in global coordinates. The calculation this command generates is shown included in the CALCPAR command

### 9.13.2 CALCSECTHIST Command

CALCSECTHIST,<infile>,<cutnum>,<pointx>,<pointy>,<pointz>,<normalx>,<normaly>,<normalz>,<rightx>,<righty>,<rightz>,<sysno>,[ts],<outfile>

Calculates forces for a cross-section based on a stress history file. This allows the user to batch the force calculation of a cut for every time step of the model. The results data will then be written to the outfile.

<infile> – Full path of the input file for the element center stress history. Similar format to the animation list file for PREP

<cutnum> – The cut to be used

<pointx> – x of point on the plane

<pointy> - y of point on the plane

<pointz> - z of point on the plane

<normalx> – x of the normal of the original cut plane in global coordinated

<normaly> – y of the normal of the original cut plane in global coordinated

<normalz> – z of the normal of the original cut plane in global coordinated

<rightx> – x of the local right in global coordinates

<righty> – y of the local right in global coordinates

<rightz> – z of the local right in global coordinates

<sysno> – the system number where the local system will be stored

[ts] - time step for each calculation. If ts = 0 the integer position in the infile will be used.

<outfile> – full path of the output file for the force history.

### 9.13.3 CALCSECTHISTDB Command

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

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

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

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

- cutnum - cut number to be used for the section history analysis

- px - x component of a point on the plane of the desired cross section
- py - y component of a point on the plane of the desired cross section
- pz - z component of a point on the plane of the desired cross section
- nx - x component of the normal of the desired cross section plane
- ny - y component of the normal of the desired cross section plane
- nz - z component of the normal of the desired cross section plane
- rx - x component of a vector defining the local systems positive X
- ry - y component of a vector defining the local systems positive X
- rz - z component of a vector defining the local systems positive X
- sys - system number to store the local cut system
- ts - time step of the stress
- start - starting step included in the table (default = 1)
- end - final step included in the table (default = last time step of the database)
- Stride - number of time steps advanced between each calculation reported in the table (default = 1)
- outfile - full path name of the results of the stress history calculations

#### 9.13.4 CALCM Command

CALCM

Calculates the total mass of the elements in the active model based on material properties and element volumes and calculates the total lumped masses of the current active model.

#### 9.13.5 CALCMOI Command

CALCMOI,<normalx>,<normaly>,<normalz>,<rightx>,<righty>,<rightz>,<sysno>

calculates the moment of inertia for the active model. It is intended to be used with the cut plane models.

<normalx> – x of the normal of the original cut plane in global coordinated

<normaly> – y of the normal of the original cut plane in global coordinated

<normalz> – z of the normal of the original cut plane in global coordinated

<rightx> – x of the local right in global coordinates

<righty> – y of the local right in global coordinates

<rightz> – z of the local right in global coordinates

<sysno> – the system number where the local system will be stored

#### 9.13.6 CALCPAR Command

CalcPar,<normalx>,<normaly>,<normalz>,<rightx>,<righty>,<rightz>,<sysno>,[verbose]

Gives the area centroid, Moment of inertia and current stress calculation for the active model. This command is intended to be used with model cross sections generated [CSECT](#) command.

The centroid is used as the origin of the local system for the Moment of inertia and current stress calculations. Stress data should be loaded onto the original model using the [READSTR](#) command before the cross section model is generated.

- normalx – x of the normal of the original cut plane in global coordinates
- normaly – y of the normal of the original cut plane in global coordinates
- normalz – z of the normal of the original cut plane in global coordinates
- rightx – x of the local right in global coordinates
- righty – y of the local right in global coordinates
- rightz – z of the local right in global coordinates
- sysno – the system number where the local system will be stored
- verbose – format flag for output.

The verbose output is a small table listing all of the model parameters with a “<parname> = ” string next to the number that was calculated. The output is verbose by default (verbose = -1). If the verbose flag is not equal to the default all of the calculated parameters will on a single line in the order they appear in the verbose table without the parameter name strings.

### 9.13.7 READSTR Command

READSTR, <Filename>, [Dir]

This command reads in the stress results for the current model from an .ess file. Once this data is loaded, the subsection command can be used and stresses can be calculated using the CALCSECTHIST command.

<Filename> – name of the .ess file to be loaded onto the file. Can be the full path  
[Dir] – The directory where the file is to be found

### 9.13.8 SECDATAOPT Command

SECDATAOPT,<flag>

This command sets the \*.ess file output request in the STRESS module input file.

<flag> - user output request for \*.ess files  
0 - no ess files written  
1 - save the .ess files for the entire time history

### 9.13.9 SHEAR Command

SHEAR, <panel>,[fc],[fy],[P],[Nu],[Fvw],[Fbe]

This command calculates the peak shear strength of a single panel or all wall panels. The SHEAR command uses four different peak shear equations, such as those provided by ACI 318-08, Wood, 1990, Barda et al., 1977 and Gulec-Whittaker, 2009 (please see Gulec and Whittaker, 2009 for details).

The lower bound value for Wood, 1990, and the upper bound value for Wood, 1990 and ACI 318-08 equations are also included. A total of six columns with computed peak shear strength are written for each panel. The columns of the result table are in order, the panel number, upper bound of ACI 318-08 and Wood, 1990, lower bound of Wood, 1990, Barda, 1977 and Gulec-Whittaker, 2009. For a single panel command, a title with names of equation is provided.

Barda et al., 1977:

The Barda equation (equation 2-7 or 4-7 in Gulec and Whittaker, 2009) is applicable to squat walls with heavily reinforced flanges (barbells). For typical shearwalls in nuclear facilities Barda equation could provide overly estimated shear strength values. Axial force effect is included.

$$V = \left( 8\sqrt{f'_c} - 2.5\sqrt{f'_c} \frac{h_w}{l_w} + \frac{N_u}{4l_w t_w} + \rho_v f_y \right) t_w (0.6l_w)$$

Wood, 1990:

The Wood equation appears (equation 2-8 in Gulec and Whittaker, 2009) close to be quite close to the median estimates for ultimate shear strength for various squat wall tests. Axial force is not included.

$$6\sqrt{f'_c} A_w \leq V = \frac{\rho_v A_w f_y}{4} \leq 10\sqrt{f'_c} A_w$$

ACI 318-08, 2008:

The ACI 318-08 Chapter 11 equation appears (equation 4-1 in Gulec and Whittaker, 2009) could provide overly estimated ultimate shear strengths. Axial force is not included.

$$V = (\alpha_c \sqrt{f'_c} + \rho_H f_y) A_w \leq 10\sqrt{f'_c} A_w$$

Gulec-Whittaker, 2009:

The Gulec-Whittaker equation appears (equation 6-9 in Gulec and Whittaker, 2009) to be also close close to the median estimates for the ultimate shear strength for various squat wall tests.

However, this Gulec-Whittaker equation is sensitive to the panel height/length aspect ratio. If this equation is applied to long panels the ultimate shear force goes up much closer to Barda, 1977 or ACI 318-08 shear force results, and even higher. Axial force is included.

$$V = \left( 1.5\sqrt{f'_c} A_w + 0.25F_{VW} + 0.20F_{BE} + 0.40N_U \right) / \sqrt{h_w / l_w}$$

The above  $F_{VW}$  force is the vertical reinforcement strength  $F_{VW} = \rho_v A_w f_y$  (kips or kN)

<panel> - panel number. If panel = 0 shear will be calculated for all panels defined by the user and reported in a space separated table with the shear equation results for each panel on a row.

<fc> -  $f'_c$ , the compressive strength (ksi or kN/m<sup>2</sup>)

<fy> - reinforcement yield strength (ksi or kN/m<sup>2</sup>)

<P > -  $\rho_v$  or  $\rho_H$ , the web reinforcement ratio for the vertical or horizontal direction

<Nu> - axial force (kips or kN)

<Abe> -  $A_{BE}$  used only for the Gulec-Whittaker equation. This parameter defines the boundary element vertical reinforcement area (ft<sup>2</sup> or m<sup>2</sup>) that is used to compute the boundary force

$$F_{BE} = A_{BE} f_{y, BE}$$

<Fybe> -  $f_{y, BE}$  used only for the Gulec-Whittaker equation. This parameter defines the boundary element vertical reinforcement yield stress (ksi or kN/m<sup>2</sup>) that is used to compute the boundary force

$$F_{BE} = A_{BE} f_{y, BE}$$

The application of this command to a large range of panels is possible only if the command input parameters related to the concrete material and reinforcement ratio parameters are identical (not applicable to Gulec-Whittaker that has more parameters).

The panel geometric data including the appropriate height, width and cross-sectional areas are automatically used by the SHEAR command for each panel.

The SHEAR command when computing the panel data uses the input gravity value to identify the FE model unit system, British (ft for the FE model geometry) or International (m for the FE model geometry). Any optional argument left blank will be set to 0 by default.

The user should provide the command parameters in the appropriate units specified below for British units, kips for forces and ksi for stresses, or International units as kN for forces and kN/m<sup>2</sup>

for stresses. The SHEAR command results for the peak shear forces is also reported in either British units as kips, or International units, as kN.

### 9.14 Plot Commands

**NOTE:** In the ACS SASSI UI, numerical data to be plotted with the graphing capabilities are stored in what are referred to here as "Line Objects". Line objects are loaded into the ACS SASSI UI using commands such as READSPEC and READTH, and are stored for that session only, and are cleared when the UI is closed. These line objects can be operated on using UI commands outlined in this section. For example, two line objects can be averaged by using the AVERAGE command in the ACS SASSI UI.

Command	Action	Description
ADDITION	Add line together and store it	Section 9.14.1
AVERAGE	Calculate the average of a group of line and store in a line	Section 9.14.2
AXES	Change the axes in a 2d model plot	Section 9.14.3
BROADEN	Broaden Selected line and save results to selected line	Section 9.14.4
BUBBLEPLOT	Plot Bubble (ZPA) for the model	Section 9.14.5
CAPTUREPLOT	Screen capture of the current plot	Section 9.14.6
CLOSEPLOT	Close the current plot	Section 9.14.7
CNGCENTER	Change the center of rotation for the 3d model plot	Section 9.14.8
CNGVIEW	Change the view of the current 3d model plot	Section 9.14.9
COLOR	Change the color in the color list	Section 9.14.10
CONTOURPLOT	Plot a 3d Contour plot	Section 9.14.11
CUTPLOT	Plot the 3d model with what elements are in the cut	Section 9.14.12
DEBUG	Display 3d debug text to the screen	Section 9.14.13
DEFORMPLOT	plot a 3d deformed shape plot	Section 9.14.14
ELECOLOR	Change element property coloration	Section 9.14.15
ELENUM	Show/Hide Element numbers on active plot	Section 9.14.16
GROUPNUM	Show/Hide Group numbers on active plot	Section 9.14.17
LBINCORS	Calculate the Lower Bound Incoherent Response Spectra	Section 9.14.18
LAYERPLOT	Plot a 2d cross section of the models soil layers	Section 9.14.19
LINECOMBIN	Linearly combine a set of lines with scalars	Section 9.14.20
LINENAME	Change the name of a line in memory	Section 9.14.21

Command	Action	Description
MARKERS	Add markers to a line	Section 9.14.22
MODEL PLOT	Plot a 3d model plot	Section 9.14.23
NODENUM	Show/Hide Node Number on active plot	Section 9.14.24
NODE PLOT	Plot a 3d node plot	Section 9.14.25
NODESEL	Select a node on a 3d plot	Section 9.14.26
PAUSE	Start or stop and animation	Section 9.14.27
PLOT RANGE	Change the range of the axes on a 2d plot	Section 9.14.28
PLOT TITLE	Change the title of a plot	Section 9.14.29
PROC FRAME	Process the frame animation files	Section 9.14.30
READSPEC	Read a spectrum file	Section 9.14.31
READTH	Read a File in the Time history format	Section 9.14.32
RSTCENTER	Reset the Center to Default	Section 9.14.33
RSTVIEW	Reset the View to Default	Section 9.14.34
SHADER OPTIONS	Change the shader options for the current plot	Section 9.14.35
SHOWDOF	Show the fixed degrees of freedom in some 3D plots	Section 9.14.36
SHOWMASS	Show the lumped masses in some 3D plots	Section 9.14.37
SHRINK	Shrink the elements in the Element Plot	Section 9.14.38
SOILPROP PLOT	Create a 2d soil property plot	Section 9.14.39
SPEC PLOT	Plot a spectrum plot	Section 9.14.40
SRSS	Combine lines using srss	Section 9.14.41
STIPPLE	Stipple line in a plot for Black and white printing	Section 9.14.42
SUBTRACTION	Subtract lines and store into another line	Section 9.14.43
TH PLOT	Create a 2d time history plot	Section 9.14.44
VECTOR PLOT	Plot a 3d vector plot with the current model	Section 9.14.45
WINDOW SETTINGS	Change the Windows Setting for the current plot	Section 9.14.46
WIREFRAME	Show the wireframe of the Element Plot	Section 9.14.47
WRITESPEC	Write a line in the spectrum file format	Section 9.14.48
WRITETH	Write a line in the time history file format	Section 9.14.49
XTITLE	Change the X axis title for a 2D graph	Section 9.14.50
YTITLE	Change the left side Y axis title for a 2D graph	Section 9.14.51
YTITLE2	Change the right side Y axis title for a 2D graph	Section 9.14.52

### 9.14.1 ADDITION Command

ADDITION, <dest>, <source1> ... <source 100>

This command allows the user to add up-to 100 line objects together and store the resultant line in the <dest> destination line object. This command is a version LINECOMBIN command where all the coefficients are equal to 1. The default name of the line object created in <dest> will be “Linear Combin.”.

- Dest – Integer number for the destination line object
- source1 ... 100 – Integer number for input line objects

### 9.14.2 AVERAGE Command

AVERAGE,<dest>,<source1> ... <source 100>

This command averages up to 100 lines objects and stores the resultant line object in the <dest> number. The resultant line object will have the X values of all the source line objects. Linear Interpolation is used when finding values for line objects between points defined by the data. Extrapolation will use the y-value first or last defined point of the line object. The default name of the line object created in <Dest> will be “Average Line”.

- dest – Integer number for the destination line object
- source1 ... 100 – Integer number for input line objects

### 9.14.3 AXES Command

AXES,<MaxTickX>,<MaxTickY>,<MinTickX>,<MinTickY>,<LogX>,<LogY>

this command give the user control of the axis on the current 2d plot.

<MaxTickX>

- 0 - turn off thick grid line on X axis
- 1 - turn on thick grid line on X axis

<MaxTickY>

- 0 - turn off thick grid line on Y axis
- 1 - turn on thick grid line on Y axis

<MinTickX>

- 0 - turn off thin grid line on X axis
- 1 - turn on thin grid line on X axis

<MinTickY>

- 0 - turn off thin grid line on Y axis
- 1 - turn on thin grid line on Y axis

<LogX>

- 0 - X axis standard
- 1 - X axis logarithmic

<LogY>

- 0 - Y axis standard
- 1 - Y axis logarithmic

#### 9.14.4 BROADEN Command

BROADEN,<Dest>,<Smooth1>,<Smooth2>,<source1> ... <source 100>

This command applies the broadening algorithm to up to 100 line objects and stores the resultant in the <dest> line object. Linear Interpolation is used when finding values for line objects between points defined by the data. Extrapolation will use the y-value first or last defined point of the line object. The default name of the line object created in <Dest> will be “Envelope”.

- Dest – Integer number for the destination line
- Smooth1 – Peak bridging percentage
- Smooth2 – Peak broadening percentage
- source1 ... 100 – Integer number for input lines

#### 9.14.5 BUBBLEPLOT Command

BUBBLEPLOT,<BufferDir>,<MnF>,<MxF>,<ST>,<MnR>,<MxR>,<Col>

Creates a Bubble plot for the active model. If no arguments are given to the command a “Load Frame Data” popup window will appear take information from the user. Partial argument entry may cause inconsistent results.

- BufferDir - Processed frame storage directory
- MnF – Starting Frame Number
- MxF – End frame number
- ST – Stride
- MnR – Minimum data value
- MxR – Maximum data value
- Col – Data column selection

#### 9.14.6 CAPTUREPLOT Command

CAPTUREPLOT,<FileName>

Capture a plot image to a file via command line. This command only works for plots. A screen image cannot be captured with this command if no plot is active. This command will save the plot to a PNG file format if the (.png) extension is contained somewhere in the FileName. If the .png extension is not found the file will be saved in the Bitmap format.

- FileName - Name of the file where the image will be saved

### 9.14.7 CLOSEPLOT Command

CLOSEPLOT

close the active plot window

### 9.14.8 CNGCENTER Command

CNGCENTER, <X>, <Y>, <Z>

Changes the center of rotation for a 3d plot.

<X> - Change X center location

<Y> - Change Y center location

<Z> - Change Z center location

### 9.14.9 CNGVIEW Command

CNGVIEW,<rX>,<rY>,<rZ>,<px>,<py>,<zoom>

Changes the view of a 3d plot by the six arguments in the command. The argument's values can be seen in the plot window when debug mode is enabled. This is useful for viewing two different plots from exactly the same viewpoint. This command's functionality is identical to using the change view button on the plot toolbar.

<rX> - rotation in the X axis

<rY> -rotation in the Y axis

<rZ> - rotation in the Z axis

<px> - Screen pan horizontal

<py> - screen pan vertical

<zoom> - Zoom constant

### 9.14.10 COLOR Command

COLOR,<Palette>,<Num>,<R>,<G>,<B>

Changes the various color palettes. This command provides the same functionality as the color menu option.

<Palette> – Name of the color palette

<Num> – Color Number of the selected color palette

<R> – Red component value between 0 - 255

<G> – Green component value between 0 - 255

<B> – Blue component value between 0 - 255

### 9.14.11 CONTOURPLOT Command

CONTOURPLOT,<BufferDir>,<MnF>,<MxF>,<ST>,<MnR>,<MxR>,<Col>

Creates a contour plot for the active model. If no arguments are given to the command a “Load Frame Data” popup window will appear take information from the user. Partial argument entry may cause inconsistent results.

- BufferDir - Buffer Directory
- MnF – Starting frame number
- MxF – End frame number
- ST – Stride
- MnR – Minimum data value
- MxR – Maximum data value
- Col – Data column selection

### 9.14.12 CUTPLOT Command

CUTPLOT,[Cut],[Model]

Creates a Cut Plot. If no cur and model number are specified, a GUI window for the cut plot will appear on the screen to input the cut and model numbers.

Cut – Number of the Cut to be plotted

Model – Number of the Model to be plotted

### 9.14.13 DEBUG Command

DEBUG,[switch]

Turns on and off the Debug menu for the current plot. The debug menu will show view angle and animation information. This information can be useful in conjunction with the CNGVIEW command.

- switch - Option to show or hide the debug menu
  - 0 Turn off debug mode
  - 1 Turn on debug mode
  - 2 (default) Switch debug mode on/off

### 9.14.14 DEFORMPLOT Command

DEFORMPLOT,<BufferDir>,<MnF>,<MxF>,<ST>,<Scale>

---

Creates a deformed shape plot for the active model. If no arguments are given to the command a "Load Frame Data" popup window will appear take information from the user. Partial argument entry may cause inconsistent results.

- BufferDir - Processed frame storage directory
- MnF – Starting frame number
- MxF – End frame number
- ST – Stride
- Scale – Scalar to increase deformation for animation

#### 9.14.15 ELECOLOR Command

ELECOLOR,<val>

Changes the element property color scheme in the active element model plot. The elements can be colored by group number, material number or Beam/Spring property number. For elements that don't have a material or property number specified, a default color is used. This command's functionality is the same as the plot toolbar buttons for "Group Colors", "Material Colors", and "Property Colors".

<val> - user choice for the color differentiation option

- 1 = Group
- 2 = Material
- 3 = Property

#### 9.14.16 ELENUM Command

ELENUM,[opt]

Show or Hide the element numbers on the active plot if the plot capable of displaying those numbers. Using the default input, the command will toggle the element number on and off. The numbers from ELENUM and the GROUPNUM command are mutually exclusive. Using this command when displaying group numbers will result in them being hidden and the element numbers will be displayed.

[opt] - user input option

- 1 = toggle element numbers (default)
- 0 = turn off element numbers
- 1 = turn on element numbers

**9.14.17 GROUPNUM Command**

GROUPNUM,[opt]

Show or Hide the group numbers on the active plot if the plot capable of displaying those numbers. Using the default input, the command will toggle the group number on and off. The numbers from GROUPNUM and the ELENUM command are mutually exclusive. Using this command when displaying element numbers will result in them being hidden and the group numbers will be displayed.

[opt] - user input option

-1 = toggle group numbers (default)

0 = turn off group numbers

1 = turn on group numbers

**9.14.18 LBINCORS Command**

LBINCORS,&lt;out&gt;,&lt;in&gt;

<out> - Reference number the output line is to be stored

<in> - Reference number of the input line data

**9.14.19 LAYERPLOT Command**

LAYERPLOT

Create a soil layer plot for the active model.

**9.14.20 LINECOMBIN Command**

LINECOMBIN,&lt;Dest&gt;,&lt;Line1&gt;,&lt;Coeff1&gt; ... [Line100],[Coeff100]

Creates a linear combination of up to 100 line objects, and stores the resultant line object in the <dest>. Each line objects y values are scaled by the associated coefficient then all the scaled line objects are added together. The resultant line object will have the X values of all the line objects used in the combination. Linear Interpolation is used when finding values for line objects between points defined by the data. Extrapolation will use the y-value first or last defined point of the line object. The default name of the line object created in <Dest> will be "Linear Combin." The user can combine any number of lines objects, but a scalar coefficient must be defined for each line object being combined.

- Dest – Number of the destination line object
- Line1...100 – Number of the line objects to be combined
- Coeff1...100 – Coefficient used to scale the line object

**9.14.21 LINENAME Command**

LINENAME,<Num>,<Name>

Changes the name of a line object. This name is a global line property, therefore changing the name will change the name of the line object in all graphs, not just the active graph

<Num> – Number of the line object to be changed

<Name> – The new name of the line object

**9.14.22 MARKERS Command**

MARKERS,<Mark>,<Ln1> ... <Ln50>

Turn markers on or off for a line object. This is a global line object property, therefore changing the marker option will change marker display of the line object in all graphs, not just the active graph.

<Mark> – Turn Line Markers on or off

0 - Off

1 - On

<Ln1 ... Ln50> – Line Numbers to Change Markers on

**9.14.23 MODELPLOT Command**

MODELPLOT

Creates an element plot of the active model.

**9.14.24 NODENUM Command**

NODENUM,[opt]

Show or hide the node numbers on the active plot if the plot capable of displaying those numbers. Using the default input, the command will toggle the node numbers on and off.

<opt> - user input option

-1 = toggle element numbers (default)

0 = turn off element numbers

1 = turn on element numbers

**9.14.25 NODEPLOT Command**

NODEPLOT

Creates a node plot of the active model.

**9.14.26 NODESEL Command**

NODESEL,<N1>, ... <N20>

Selects up to 20 nodes that will be highlighted on a 3D plot. This command highlights the nodes in the model on all current and future plots. Each plot type has its own different display rules on how these nodes are highlighted. If more than 20 nodes are to be selected, multiple instances of this command can be used, as nodes are not deselected between commands. The nodes can be deselected by using NODESEL command again with the node number to be deselected.

<N1 ... N20> – Node number to select on the plot

**9.14.27 PAUSE Command**

PAUSE,[pz]

This command allows the user to start or stop the animation of an animated plot.

<pz> - start or stop the animation  
-1 – Toggle start/stop (default)  
0 – start the animation  
1 – stop the animation

**9.14.28 PLOTRANGE Command**

PLOTRANGE,<Xmin>,<Xmax>,<Ymin>,<Ymax>

Changes the range of the active 2D plot

<Xmin> – Minimum value of x axis  
<Xmax> – Maximum value of x axis  
<Ymin> - Minimum value of y axis  
<Ymax> - maximum value of y axis

**9.14.29 PLOTTITLE Command**

PLOTTITLE,<Title>

Changes the title of the active plot. This command work with both 2D line and all 3D plots. The command is ignored by the soil layer plot.

- Title – The Plot Title

**9.14.30 PROCFRAME Command**

PROCFRAME,[AniFile],[BufferDIR],[Data],[anitype]

Process frames for animation without using the graphical interface. If the user leaves the AniFile or BufferDIR argument blank the process frame popup window from the Process Animation Frame List menu option will appear. Processing frames can take a long time for some animations, so this command will open a progress bar at the bottom of the screen. The user will be prevented from inputting data into the User Interface until the animation processing is complete.

- AniFile - Animation Frame list file full path name
- BufferDIR - Processed frame storage directory
- Data – Label for the animation that is stored in the database
- Anitype – Integer tag for the type of animation data that was processed. This tag is for sorting the database only. All animation data frame data is processed the same way regardless of the value of this tag.
  - 0 - Bubble (default)
  - 1 - Vector
  - 2 - Contour
  - 3 - Time History

#### 9.14.31 READSPEC Command

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

Reads a spectrum file into a line object for plotting

<SpecFile> – Full path of the spectrum file to be read

<numLines> – number of lines to be read from the file

<Line> - Line Reference Number for each line to be read

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

#### 9.14.32 READTH Command

READTH,<THFile>,<Pair>,<Num>

Reads a time history file into a line object for plotting

<THFile> – Full path of the Time History file to be read

<Pair> – option to read time history in 1 or 2 column format

0 - read one column time history (ACS SASSI output format)

1 - read two column time history with time/accel. pairs

<Num> – reference number to store the line object

#### 9.14.33 RSTCENTER Command

RSTCENTER

Resets the center of rotation of the plot to the default. The default center is located at the center of bounding box that is built by finding the models maximums and minimums all the nodes used by elements in the global coordinate system.

#### 9.14.34 RSTVIEW Command

RSTVIEW

Reset the view of a 3d plot to the default. The default view is the top down view of the model when the plot is first opened. The default zoom and view location is influenced by bounding box that is built by finding maximums and minimums all the nodes connected to elements in the model.

#### 9.14.35 SHADEROPTIONS Command

SHADEROPTIONS,[<points>],[<linewidth>],[<shrink>],[<scale>]

Changes shader options or opens Shader Options window. All of the argument in this command are optional. The command will only change the value for filled in arguments. Entering no arguments will bring up the Shader Options window.

<points> - adjust the max point size in the node/bubble plot

<linewidth> - adjust the outline width of solid elements

<shrink> - adjust amount of shrink in the element plot

<scale> - adjust the scale factor for animations

#### 9.14.36 SHOWDOF Command

SHOWDOF,[<label1>],...,[<label6>]

Show the fixed degrees of freedom that the user selects if the active plot is capable of displaying the information. The user selects which directions to show either by using the labels for the proper direction or using the check boxes which appears if the enters the command without any labels. Graphs capable of displaying the DOF information will show a marker over every node that has a degree of freedom fixed in a direction that a user requested.

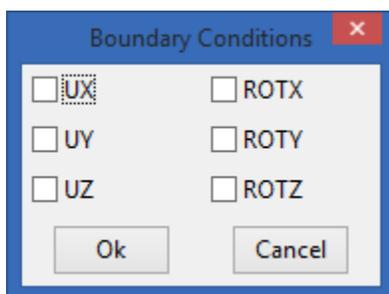


Figure 9.7: SHOWDOF window

<label> - degree of freedom to show

X = Show fixed in X  
Y = Show fixed in Y  
Z = Show fixed in Z  
XX = Show fixed in XX  
YY = Show fixed in YY  
ZZ = Show fixed in ZZ  
DISP = Show fixed in X, Y and Z  
ROT = Show fixed in XX,YY and ZZ  
ALL = Show fixed in All DOF

#### 9.14.37 SHOWMASS Command

SHOWMASS,[opt]

Show or Hide the lumped mass on the active plot if the plot capable of displaying those markers. If the user uses the default input the command will toggle the markers on and off. The model element plot markers denote the direction of masses. If the node marker has a red section part of the lumped mass is in the x direction, green denotes y direction, and blue if for the z direction. Other plots that display mass markers only show the presence of a mass at a node.

[opt] - user input option

-1 = toggle mass markers(default)  
0 = turn off mass markers  
1 = turn on mass markers

#### 9.14.38 SHRINK Command

SHRINK,[switch]

Shrinks the elements on a model plot. (Only valid on Element Plot)

[Switch]

1 Turn on shrink mode  
0 Turn off shrink mode  
-1(default) Switch shrink mode on/off

#### 9.14.39 SOILPROPLOT Command

SOILPROPLOT, <PropName>

Creates a Soil Property plot by command line. If a <Propname> is not enter a “Select Dynamic Soil Property” popup window will appear to take information from the user. The <PropName> is case sensitive and must match a Soil Property Name current loaded into memory for a plot to appear.

---

<PropName> - The name of the dynamic soil property curves

#### 9.14.40 SPECPLOT Command

SPECPLOT,<Line1>, ... ,<Line50>

Creates a spectrum plot by command line. This command uses the line object minimums and maximums to define the default plot extent. Reference numbers that are not associated with line objects before this command is entered will be ignored. The value of -1 is used to recognize the end of line entry all valid line object reference numbers after a <LineN> = -1 will be ignored.

- Line – The reference number of the line object to be plotted

#### 9.14.41 SRSS Command

SRSS,<dest>,<source1> ... [source 100]

Applies SRSS to up to 100 line objects. The result will be stored in the <dest> line object. The resultant line object will have the X values of all the source line objects. Linear Interpolation is used when finding values for line objects between points defined by the data. Extrapolation will use the y-value first or last defined point of the line object. The default name of the line object created in <Dest> will be “SRSS Line”.

- Dest – Integer number for the destination line object
- source1 ... 100 – Integer number for source line object numbers

#### 9.14.42 STIPPLE Command

STIPPLE,<switch>

Turn on or off stippling for a plot. Stippling breaks up the continuous line on a plot so it is easier to differentiate lines on a non-color printout.

<switch>

- 0 Turn off Stipple mode
- 1 Turn on Stipple mode
- 1 (default) Switch Stipple mode on/off

#### 9.14.43 SUBTRACTION Command

SUBTRACTION,<dest>,<source1> ... <source 100>

Subtracts up to 99 line objects from the initial source line object via command. This command stores the result in the line object specified by the <Dest> argument of this command. This

---

command is a version LINECOMBIN command where the first coefficients is 1 and all rest the coefficients are equal to -1. The default name of the line object created in <dest> will be “Linear Combin.”.

- Dest – Integer number for the destination line object
- source1 ... 100 – Integer number for input line objects

#### 9.14.44 THPLOT Command

THPLOT,<Line1>, ... ,<Line50>

Creates a time history plot by command line. This command uses the line object minimums and maximums to define the default plot extent. Reference numbers that are not associated with line objects before this command is entered will be ignored. The value of -1 is used to recognize the end of line entry all valid line object reference numbers after a <LineN> = -1 will be ignored.

- Line – The reference number of the line objects to be plotted

#### 9.14.45 VECTORPLOT Command

VECTORPLOT,<BufferDir>,<MnF>,<MxF>,<ST>,<Scale>

Create a Vector plot for the active model by command line. If no arguments are given to the command a “Load Frame Data” popup window will appear take information from the user. Partial argument entry may cause inconsistent results.

- BufferDir - Processed frame storage directory
- MnF – Starting Frame Number
- MxF – End Frame Number
- ST – Stride
- Scale – Scalar to Increase Deformation for animation

#### 9.14.46 WINDOWSETTINGS Command

WINDOWSETTINGS

Opens the window setting window so the user can change the settings of the active plot. This command will bring up the pop up depending on if this is for a 2D or 3D plot. Future versions of this command should have argument to change the setting from the command entry.

#### 9.14.47 WIREFRAME Command

WIREFRAME,<Switch>

---

turn on or off the wireframe mode in the Model element plot.

<Switch>

- 0 Turn off wire frame mode
- 1 Turn on wire frame mode
- 1 (default) Switch wireframe mode on/off

#### 9.14.48 WRITESPEC Command

WRITESPEC, <SpecFile>, <Num1>, ... , [Num50]

Writes line object(s) data into a Spectrum file format. Up to 50 lines can be written to a single file using this command. Each line stored in the file will have points generated for x-values from all of the line objects. Linear Interpolation is used when finding values for line objects between points defined by the data. Extrapolation will use the y-value first or last defined point of the line object.

- SpecFile – The full path name of the file that will be saved.
- Num – The reference number of the line object data to be written

#### 9.14.49 WRITETH Command

WRITETH, <THFile>, <Num>

Writes line object data into a time history format file. This is a single column file format, therefore the line object data must have a constant time interval. The time interval will be calculated by finding the difference between x-values of the first two points in the line object.

- THFile – The full path name of the file that will be saved.
- Num – The reference number of the line to be written

#### 9.14.50 XTITLE Command

XTITLE, <label>

Add or change the text of the X-axis title of the current graph.

<label> - the text that will appear on the graph

#### 9.14.51 YTITLE Command

YTITLE, <label>

Add or change the text of the left side Y-axis title of the current graph.

<label> - the text that will appear on the graph

**9.14.52 YTITLE2 Command**

YTITLE2,&lt;label&gt;

Add or change the text of the right side Y-axis title of the current graph.

<label> - the text to be shown on the graph

**9.15 Programming Commands**

Command	Action	Description
ADDRND	Append random numbers to the end of a variable	Section 9.15.1
CD	Change Current working directory	Section 9.15.2
FOREACH	Loop on a variable	Section 9.15.3
LOADMACRO	Load a Macro pre file	Section 9.15.4
LOADVAR	Load a Variable from a file	Section 9.15.5
MACRO	Use a Macro	Section 9.15.6
MACROLIST	Show all macro names currently in ACS SASSI User Interface memory	Section 9.15.7
MKDIR	Make a new directory	Section 9.15.8
REDUCESET	Sort variable contents and remove duplicate entries	Section 9.15.9
RND	Fills a variable with a random number	Section 9.15.10
RNDSEED	Seeds or reseeds random number generator	Section 9.15.11
SETVAR	Set Variables	Section 9.15.12
SHOWVAR	Show contents of a variable	Section 9.15.13
VAR	Create a variable	Section 9.15.14
VARLIST	Show all variable and variable contents currently in ACS SASSI User Interface memory	Section 9.15.15

**9.15.1 ADDRND Command**

ADDRND,&lt;var&gt;,&lt;numsamples&gt;,&lt;dist&gt;,&lt;param1&gt;,...,&lt;paramN&gt;

This command will append random numbers to the end of a variable. This command shares the same algorithms as the RND command but does not delete the data currently in the variable list.

- var - variable name
- numsamples - number of values the variable will have after this command
- dist Random distribution type
  - UNI - uniform (double floating point)
  - UNIINT - uniform (integer)
  - NORM - Normal
  - LOGNORM - Lognormal

- POISSON - poisson
- param - parameters for the random distribution type
  - UNI, UNINT
    - param1 - minimum of distribution range
    - param2 - maximum of distribution range
  - NORM, LOGNORM
    - param1 - mean of distribution
    - param2 - standard deviation of distribution
  - POSSION
    - param1 - mean of distribution

### 9.15.2 CD Command

CD,<dir>

Change the current working directory that the User Interface is using. The new directory path name can be absolute or relative to the current working directory. The new directory path must already exist or this command will produce an error.

- dir - directory path name.

### 9.15.3 FOREACH Command

FOREACH,<Var>,<Command>

For each allow the user to loop a single command using variables. Loops can be nested and macros can be used as the command in the loop. See Variables and Loops for a description on how to use variables and the FOREACH command.

- Var – Variable name to be looped on
- command - the command to be used in the loop

### 9.15.4 LOADMACRO Command

LOADMACRO,<Name>,<MACROFILE>

Associates a Macro Name with a file. See Macro example for explanation on how to define and use macros

<Name> – Name of the Macro

<MACROFILE> – The file full path name containing the macro definition

### 9.15.5 LOADVAR Command

LOADVAR,<filename>

This command will load a variable from a file. The file should be in an ASCII format and each item should be on a separate line.

<filename> - full path name of the file containing the variable data.

### 9.15.6 MACRO Command

MACRO,<Name>,<1>, <2>, ... ,<N>

The macro command allows the user to run a previously loaded macro. See Macro for explanation on how to define and use macros

<Name> - Name of the Macro

<1>, <2>, ... <N> – The Data that replaces the macro variable of the same number

### 9.15.7 MACROLIST Command

MACROLIST

MACROLIST

List of all the macro names currently defined and available for uses in ACS-SASSI User Interface memory. The command will also list the macro file names associated with the macros in memory.

### 9.15.8 MKDIR Command

MKDIR,<dir>

Create a new directory with an either absolute or relative path name. The relative path is based on the current working directory, which always defaults to the user interface install directory. The user can change working directory with CD or MDL commands. This command will show an error to the command history if a new directory was not created. This command will give a warning if the directory already exist.

- dir - name of the directory to be created

### 9.15.9 REDUCESET Command

REDUCESET,<var>,[sorttype] (Not usable in this

This command will sort all of the elements in the variable list and remove duplicate values in the variable list. All values in the variable list are stored as character strings. The order of numbers

when sorted lexicographically is not the same as the numeric order of the same set. If a numerically sorted list is needed, the `sorttype` argument will convert the elements of the variable list before the set reduction is implemented. The numerically sorted variable list will then be stored as character strings after the operation.

- `var` - variable name that the command will work on
- `sorttype` - Specify the way each element in the variable list will be sorted and reduced
  - `STRING` - Sort based on lexicographic order (default)
  - `INT` - Sort based on integer order
  - `FLOAT` - Sort based on double precision floating point ordering

**Note:** Using the `INT` and `FLOAT` `sorttype` for variable list that have non-numeric data elements may cause errors or inconsistent sorting behavior.

### 9.15.10 RND Command

`RND,<var>,<numsamples>,<dist>,<param1>,...,<paramX>`

This command fills a variable with a random number list. This command will overwrite any data that is currently in the variable.

- `var` - variable name
- `numsamples` - number of values the variable will have after this command
- `dist` Random distribution type
  - `UNI` - uniform (double floating point)
  - `UNIINT` - uniform (integer)
  - `NORM` - Normal
  - `LOGNORM` - Lognormal
  - `POISSON` - poisson
- `param` - parameters for the distribution random
  - `UNI, UNINT`
    - `param1` - minimum of distribution range
    - `param2` - maximum of distribution range
  - `NORM, LOGNORM`
    - `param1` - mean of distribution
    - `param2` - standard deviation of distribution
  - `POSSION`
    - `param1` - mean of distribution

### 9.15.11 RNDSEED Command

RNDSEED,<seed>

Seeds or reseeds the random number generator for the RND and the ADDRND commands. The <seed> argument must be a positive integer.

- seed - seed number for the random number generator.

### 9.15.12 SETVAR Command

SETVAR, ...

This is a placeholder it has no effect on the model or any data. This command allows the user to change a variable as described in Variables and Loops.

### 9.15.13 SHOWVAR Command

SHOWVAR,<varname>

This command will show the contents of the variable specified by varname. If the variable doesn't exist the command will show a variable not defined error message.

<varname> - Name of the variable to be shown

### 9.15.14 VAR Command

VAR,<Name>,<X1>...,<Xn>

Define a variable to be used in the ACS SASSI User Interface. See Variables and Loops for a description on how to use variables and the VAR command. Currently there is no way to change a variable other than redefining it and overwriting its contents by using the VAR command again with the same variable name.

<Name> – The name of the variable

<X1>,...<Xn> – Values to be stored In the variable

### 9.15.15 VARLIST Command

VARLIST

This command will list the variable names in ACS-SASSI User Interface memory followed by the list of variable's contents. If the variable contains a single value that value will be shown or else the number of elements in the variable will be shown. To see the contents of a variable that contain multiple values use the SHOWVAR command.

## 9.16 Water Modeling Commands

Command	Action	Description
FILLPOOL	Fill a shell or solid pool with water solids	Section 9.16.1
REFINEMODEL	Subdivide all Shell and solid elements in a model	Section 9.16.2
LISTPOOLINTER	List the interface nodes of a pool in the original model	Section 9.16.3

### 9.16.1 FILLPOOL Command

**FILLPOOL**,<Stiff>,<Sensitivity>,<EmptyLevels>,<ShellArea>,<offset>,<stiff2>

This command fills a pool with solid water elements, a spring pool water/wall interface and the surface areas of the interface using shells. The pool is filled using an algorithm used in the EXCAV command where the floor of the pool and the wall Z-levels are used to create group 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 user input. Finally a shell group is added that approximates the areas of the interface which may be useful in further processing of the pool in ANSYS®.

The user should create a sub-model of the pool to be filled before using this command. The pool 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  $10^6$ )

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

<EmptyLevels> - Number of Z-levels, starting at the highest level, not to be filled by water (Default 0)

ShellArea - Flag to produce shell areas. (Default -1)

ShellArea = 1 - Create the shell areas in the pool model

ShellArea  $\neq$  1 - Do not create the shell areas in the pool model

<offset> – node number to start numbering of pool elements. This number should be greater than or equal to last node number from the original 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 perpendicular to the normal. perpendicular stiffness used to model non fluid materials that may fill a pool (Default 0)

### 9.16.2 REFINEMODEL Command

## REFINEMODEL

This Command will subdivide all of the quad shells and non-prism solids in a model. Each quad shell will be subdivided into 4 shells using the midpoints of the edges to split the element. Non-prism solids will be split into 8 elements using the midpoints of the edges to split the elements. All triangular elements and prism elements will be unaffected by this command.

**9.16.3 LISTPOOLINTER Command**

LISTPOOLINTER,<Pool>

This command lists all of the elements in the original model that are also in the pool submodel. The original model and the pool model must be loaded in to the ACS SASSI UI. The original model should be set as the active model using the ACTM command. Once the original model is activated, the LISTPOOLINTER command can be used with the Pool model number in the argument. The command will then compare the nodes in the pool interface with nodes in the original model, and will list the nodes that share the same position and node number.

This command will only work with a pool filled with the FILLPOOL command.

<Pool> - The Model number of the pool submodel

**9.17 Nonlinear & Panel Commands**

Command	Action	Description
B	Nonlinear Beam Definition	Section 9.17.1
BBC	Add a backbone curve from a file	Section 9.17.2
BBCGEN	Generate BBC curves for defined panels	Section 9.17.3
BBCI	Sets BBC information	Section 9.17.4
BBCP	Defines single point on BBC	Section 9.17.5
BBCX	Backbone curve definition command	Section 9.17.6
BBCY	Backbone curve definition command	Section 9.17.7
BEAMPILE	Separates piles from model, and reconnects them with a spring at pile interface	Section 9.17.8
DCOUPLEBEAM	Add Springs at beam intersections	Section 9.17.9
DELBBC	Delete Backbone Curves	Section 9.17.10
DELBM	Delete Nonlinear Beam	Section 9.17.11
DELNLS	Delete Nonlinear Soil Layers	Section 9.17.12
DELSR	Delete Non Linear Spring	Section 9.17.13
DGRDFLR	Modify Young's modulus of floor panel	Section 9.17.14
EDGE	Split panel based on edges	Section 9.17.15
EDGEMODEL	Applies the EDGE command to all of the wall groups	Section 9.17.16

Command	Action	Description
	in the model	
EQL	Nonlinear analysis options	Section 9.17.17
MERGE PANEL	Merge a Panel Model to the Solids and beams of the original model	Section 9.17.18
NLSLAYER	Add a nonlinear soil layer definition	Section 9.17.19
NLSOIL	Set Parameters for the nonlinear soil option	Section 9.17.20
NONLINBAT	Create a Generic Nonlinear Batch Run	Section 9.17.21
NONLINMOTDISP	Add panel corner nodes to output request list	Section 9.17.22
P	Add a panel to the active model	Section 9.17.23
PANELIZE	Separate Shells in the model into Panels	Section 9.17.24
PDEL	Delete panel(s) from the active model	Section 9.17.25
PLIST	List or check panel(s) in the active model	Section 9.17.26
PNLGEN	Create panel definitions for shell groups	Section 9.17.27
S	Nonlinear Spring definition command	Section 9.17.28
SOILREDEF	Redefine excavation volume layers based on a 2D soil model	Section 9.17.29
SOLIDPILE	Separates piles from model, and reconnects them with a spring at pile interface	Section 9.17.30
UNIPNL	Create unique group for each element of a group	Section 9.17.31
WALLFLR	Separate Shells Walls and Floors into Separate Groups	Section 9.17.32

### 9.17.1 B Command (Not usable in this version)

B,<num>,<group>,<spgroup>,<bbc>,<force>,<end1>,<end2>

Defines a nonlinear beam and its properties for the NONLINEAR module. This command associates already defined beam element(s) with a backbone curve. This command does not make any new elements, change material, or any other properties of the beam(s) defined by other commands.

- num - Beam Number
- group - beam group number for the nonlinear beam
- spgroup - spring group number that is connected to the end of the beam
- bbc - backbone curve used for the beam definition.
- force - force option
- end1 - element at one end of the beam.
- end2 - element at other end of the beam.

### 9.17.2 BBC Command

BBC,<num>,<type>,<points>,<yield>,<file>

Load a backbone curve from a file. The file format is a list defining the points in X Y pairs. The command will attempt to load the number of points in the points argument, so the user must the number of points in the file before attempting to load.

<num> - number of the backbone curve

<type> - type of backbone curve

<points> - number of points in the curve

<yield> - yield point for the curve

<file> - file name containing the backbone curve points

### 9.17.3 BBCGEN Command

BBCGEN,<Panel>,<ShearModel>,[fc],[fy],[Pn],[Nu],[bre],[bys],[CrackingForceLevel]

The BBCGEN command automatically generate the backbone curve(s) for a single panel or all panels using the computed ultimate shear strength values that are identical with those obtained using the SHEAR command. The user can run first the SHEAR command and get the computed shear strength estimates based on ACI 318-08, Wood, Barda and Gulec-Whittaker empirical equations (see the SHEAR command for details). Before using the BBCGEN command, the user needs to decide which of the four shear strength models described for the SHEAR command would like to use for the nonlinear SSI analysis via Option NON.

The BBCGEN command always generates a 22 point BBC curves, the first point being the cracking point of the BBC curve. The next 20 points will be equidistantly spaced along the strain axis until the yield point is reached. The final shear failure point for all BBC curves will be defined by default at (shear strain = 2% and shear force = 1.02 x ultimate shear force value).

All of the shear strength model parameters specified by the user in the BBCGEN command are optional. If the user leaves a command parameter blank, the command will use 0 for that variable. This command assumes the panel references a non-mpty group with a uniform material across all elements. It user responsibility to check that all elements within a panel have same thickness and elastic material properties.

- Panel - Panel number. If Panel = 0, the BBC curves will be generated for all panels defined by the user assuming the same command parameters. The Panel = 0 option, it can be used in conjunction of submodels, to define properties of panel subsets.
- ShearModel – Number of the shear model type that will be used to generate the backbone curve. The shear model equations can be found in the description of the SHEAR command. The upper or lower bound will be used for the BBC if the calculated shear for

ACI 318-08 or Wood 1990 fall outside the bounds specified in the model equations. The ShearModel values can be (see the SHEAR command for details):

- 1 - ACI 318-08
- 2 - Wood 1990
- 3 - Barda 1977
- 4 - Gulec-Whittaker 2009
- $f_c$  – concrete compressive strength (ksi or kN/m<sup>2</sup>)
- $f_y$  - reinforcement yield strength (ksi or kN/m<sup>2</sup>)
- $P_n$  - web reinforcement ratio; it could be for horizontal or vertical direction depending on the selected ShearModel
- $N_u$  - axial force (kips or kN)
- $F_{vw}$  – only for Gulec-Whittaker model, see SHEAR command description (kips or kN)
- $F_{be}$  - only for Gulec-Whittaker model, see SHEAR command description (kips or kN)
- CrackForceLevel – If not zero, it defines the ratio between the cracking shear force and ultimate shear force to determine the cracking point of the BBC curve. The nonzero value of this parameter can be between 0.10 and 0.50. Other values are not permitted.
  - 0 – Default option for building BBC curves - Uses the ASCE 4-17 standard recommendation in Section C.3.3.2 for defining the concrete cracking stress level by the value of  $3\sqrt{f'_c}$
  - [0.1-0.5] – Uses the cracking shear/ultimate shear ratio to build the BBC curves.

#### 9.17.4 BBCI Command

BBCI,<num>,<yield>,<type>

Command that set the BBC curve information.

- num - number of the backbone curve;
- yield - yield point of the curve
- type - type of the curve

#### 9.17.5 BBCP Command

BBCP,<num>,<point>,<X>,<Y>

Defines a single point in a BBC curve

- num - number of the backbone curve
- point - numerical order of the point on the curve
- X - X component of the point
- Y - Y component of the point

### 9.17.6 BBCX Command

BBCX,<num>,<points>,<yield>,<X1>, ... <Xn>

Defines the x-axis points of the backbone curve

<num> - number of the backbone curve

<points> - number of points in the curve

<yield> - yield point for the curve

<X1> ... <Xn> - X component of the points on the curve

### 9.17.7 BBCY Command

BBCY,<num>,<points>,<yield>,<Y1>, ... <Yn>

Defines the y-axis points of the backbone curve

<num> - number of the backbone curve

<points> - number of points in the curve

<yield> - yield point for the curve

<Y1> ... <Yn> - Y component of the points on the curve

### 9.17.8 BEAMPILE Command (Not applicable in this version)

BEAMPILE

This command separates piles from the rest of model, then reconnects piles to the model by adding springs along the pile interface. The command begins by duplicating the beam nodes and reconnects the element to the duplicate node. Then springs are added between the original node and the duplicate connected to the pile. A spring groups will be added for each time this command is issued. The one spring groups connect beam to the rest of the model with transnational stiffness of  $10^7$ . All springs created have a 4% damping. Beam piles are identified by setting the ETYPE of the beam to 2. This command only works with identified beam elements. The SOLIDPILE command has been created for models that use piles with solid pile elements. *(Not applicable in this version)*

### 9.17.9 DCOUPLEBEAM Command (Not usable in this version)

DCOUPLEBEAM

This command will add springs to the ends of beams and at beam intersection that are perpendicular to the z-axis to simulate nonlinear structures. The ACS SASSI UI finds all of the beams elements and determines the ends and intersections based connectivity with other elements. The command then adds a new node that the beam end and a spring with  $10^7$  stiffness in all directions connect the new original beam end node to the new end node. The new springs

---

are placed in a new group and each spring has its own spring constant so that each spring can be modified independently. *(Not applicable in this version)*

### 9.17.10 DELBBC Command

DELBBC,<start>,<end>,<stride>

Delete backbone curves that are in ACS SASSI User Interface memory

<start> - the starting backbone curve to delete

<end> - the last backbone curve to delete

<stride> - stride increment for deleting backbone curves

### 9.17.11 DELBM Command

DELBM,<start>,[end],[stride]

Delete nonlinear beams created by the [B](#) command. This will not remove any elements from the model. The beams specified in this command will no longer be flagged as nonlinear.

- start - starting number of the beams to delete
- end - final number of beam to delete (default = start)
- stride - stride increment for delete (default = 1). The command will warn the user that the default stride was used if the argument is left blank or the user gives a stride value less than 1.

### 9.17.12 DELNLS Command

DELBBC,<start>,[end],[stride]

Delete backbone curves in the active model.

- start - the starting backbone curve to delete
- end - the last backbone curve to delete (default = start)
- stride - stride increment for deleting backbone curves (default = 1) The command will warn the user that the default stride was used if the argument is left blank or the user gives a stride value less than 1.

### 9.17.13 DELSPR Command

DELNLS,<start>,[end],[stride]

---

Delete nonlinear soil layer definition created by the NLSLAYER command. This does not modify any information that has been defined by L, SOIL or any other command used to create the linear soil model.

- start - starting number of the soil data to delete
- end - final number of soil data to delete (default = start)
- stride - stride increment for delete (default = 1) The command will warn the user that the default stride was used if the argument is left blank or the user gives a stride value less than 1.

#### 9.17.14 DGRDFLR Command

DGRDFLR,<scale>

This Option NON command modifies the Young's modulus of the floor panel materials in a panel model. The command builds a list of materials based on Z parallel floor panels from a panel model. The user enters a scale factor for the Young's modulus for all materials that are to be scaled. This command assumes that the user has created panel groups using the WALLFLR command and the GROUPEMAT command before using this command.

- scale - scale factor for the materials young's modulus

#### 9.17.15 EDGE Command

EDGE,<panel>,[X],[Y],[Z]

This Option NON command will split a panel groups based on the results of an edge detection algorithm. This command is to be used after the WALLFLR command has created a panel model. The user will select a group with the <panel> argument of the command. The command will identify the external panel edges as well as the edges of any openings in the panel group and use the edges to split the panel group. All new group will be added to the end of the model. The user can modify the split of the panel ignore all edges found that are parallel to an axis of the global coordinate system.

- panel - group number of the panel to be split
- X - Flag for edges parallel to the global X coordinate system
  - 0 Use edges (default)
  - 1 Ignore edges
- Y - Flag for edges parallel to the global Y coordinate system
  - 0 Use edges (default)
  - 1 Ignore edges
- Z - Flag for edges parallel to the global Z coordinate system
  - 0 Use edges (default)
  - Ignore edges

#### 9.17.16 EDGEMODEL Command

EDGEMODEL,[x],[y],[z]

The Option NON command applies the EDGE command to all of the wall groups in the model.

- x - ignore edges parallel to the global x axis
  - 0 Use edges (default)
  - Ignore edges
- y - ignore edges parallel to the global y axis
  - 0 Use edges (default)
  - Ignore edges
- z - ignore edges parallel to the global z axis
  - 0 Use edges (default)
  - Ignore edges

#### 9.17.17 EQL Command

EQL,<disp>,<NonLinOpts>,<dampCutoff>,<dampScale>,<ElasticD>

Set the options for the non-linear simulation. This command is similar to HOUSE,ANALYS or STRESS commands in that it sets header information for the nonlinear module input (\*.eq) file.

This information can also be set interactively using the NONLINEAR analysis options tab, found in the Options→Analysis menu selection.

- disp - displacement factor
- NonLinOpts - non-linear options
- dampCutoff - damping cutoff value
- dampScale - damping scaling factor
- ElasticD - Include elastic damping flag
  - 0 - Don't include
  - 1 - Include

### 9.17.18 MERGEPANEL Command

MERGEPANEL,<Panel>

This command is a special merge command to combine panel model with the non shell elements of the original model. The user should set the active model to the original model using ACTM. The user should delete the shell group(s) from the original model because these shells will be replaced by the panel model. The user should then use the MERGEPANEL command with the model number for the panel model as the argument. The result will be the combination of the original model with the panel model groups and materials added to the end of the group and material list of the original model.

<Panel> - the model number of the panel model

### 9.17.19 NLSLAYER Command

NLSLAYER,<Num>,[curvefit],[B],[S],[refStrain],[Vis]

Define a nonlinear soil layer data set for the soil module. User can curve fit and not define any other properties for the curve. Otherwise, the user must define all of the curve properties. The default value for all of the arguments in this command is 0.

The argument for this command are based on the following equation used for the hyperbolic stress-strain model for soils

$$\tau = \frac{G_o \gamma}{1 + \beta \left( \frac{\gamma}{\gamma_r} \right)^s} + \eta \dot{\gamma}$$

Where  $G_o$  is the elastic shear modulus,  $\tau$  is the shear stress,  $\gamma$  is the shear strain,  $\gamma_r$  is the reference shear strain (typically around 0.03%),  $\beta$  is a constant (typically around 1-1.4), and  $s$  is a constant (typically around 0.7-0.9),  $\eta$  is soil viscosity, and  $\dot{\gamma}$  is shear strain rate.

---

<Num> - soil layer number  
<curvefit> - auto fit the curve.  
    0 = not using curve fit  
    1 = using curve fit  
<B> - Beta  
<S> - S exponent  
<refStrain> - reference strain  
<Vis> - soil viscosity

### 9.17.20 NLSOIL Command

NLSOIL,<Opt>,<NSTimeSunInc>,<DispConv>,<ForceConv>,<Equallt>,<BedInt>,<NLDampType>,<MMmult>,<SMmult>

This command sets the global nonlinear [SOIL](#) options for the soil module. This command does not change any information for the linear soil model.

<Opt> - flag to use nonlinear soil  
    0 - do not use nonlinear  
    1 - use nonlinear  
  
<NSTimeSunInc> - Sub-increments per timestep  
  
<DispConv> - Displacement convergence error  
  
<ForceConv> - Force convergence error  
  
<Equallt> - Equilibrium iterations  
  
<BedInt> - Bedrock interface  
    0 - Rigid  
    1 - Viscoelastic (Currently Disabled)  
  
<NLDampType> - Damping Type  
  
<MMmult> - Mass Matrix Multiplier  
  
<SMmult> - Stiff Matrix Multiplier

### 9.17.21 NONLINBAT Command

NONLINBAT,<Sel>

---

Generate a generic batch run file for running the nonlinear SSI analysis via Option NON. This batch file will contain information about the active model and the module locations that ACS SASSI User Interface is currently pointing. The user can use this file to run a nonlinear analysis or as a starting point for a custom nonlinear analysis batch file.

<Sel> - File Selection

0 - single direction

1 - three direction

### 9.17.22 NONLINMOTDISP Command

NONLINMOTDISP

This Option NON command finds the corner nodes of all panels defined by the P command. The corner nodes are then added to the output request list of the MOTION and RELDISP modules. This command uses a node connection counting algorithm to find the edges of the panels. This algorithm may not find all edges for panels that use triangular meshes or concave polygon panels. The user has to check visually the SSI model panel before finding the panel corners, to avoid ill geometries. As a guidance, the user should define the wall panels with rectangular shapes, or as close as possible to the rectangular shapes, and a low-ratio aspects for the panels deforming predominantly due to shear forces.

### 9.17.23 P Command

P,<num>,<group>,<bbc>,<disp>,<force>

This Option NON command defines wall panels for nonlinear structure SSI analysis. This command associates the SSI model data with a finite shell element group to create the wall panel. This command does not define any new groups or elements and no linear SSI model information is changed by this command. All shells in a panel group should be coplanar. Coplanar shell groups can be created by using the WALLFLR command.

- num - panel number
- group - group number
- bbc - back bone curve number
- disp - displacement type
- force - force option

### 9.17.24 PANELIZE Command

PANELIZE

---

This Option NON command is to be used only on a panel model created by the WALLFLR command. The command will further split the walls and floors based on where the wall and floor groups intersect each other. First the intersections of the floors and wall is determined then the groups will be subdivided along the intersection. This command may produce more groups than the user needs. In this case, panels can be combined using the MERGEGROUP command. If the user needs to split a group using a wall that is not intersecting the group, the SPLITGROUP command can be used.

### 9.17.25 PDEL Command

PDEL,<start>,[end],[stride]

Delete nonlinear panels that are in ACS-SASSI User Interface memory. This command will only remove nonlinear model data created by the P command This will not remove any elements, linear model data, or other nonlinear data from the model.

- start - the starting backbone curve to delete
- end - the last backbone curve to delete (default = start)
- stride - stride increment for deleting backbone curves (default = 1) The command will warn the user that the default stride was used if the argument is left blank or the user gives a stride value less than 1.

### 9.17.26 PLIST Command

PLIST,[start],[end],[stride]

Command that list the panel data in ACS SASSI User Interface memory. The user can select the panel information for display by using the arguments or all the panel data in ACS SASSI User Interface memory will be listed if the defaults are used.

<start> - starting number of the panels to be listed (default = 1)

<end> - end number of the panels to be listed (default = largest panel number in memory)

<stride> - increment to the next panel listed (default = 1)

### 9.17.27 PNLGEN Command

PNLGEN

This Option NON command generates panel data for each shell group that is found to be in the vertical plane. This command will associate a panel numbers in groups in an ascending order based on group number. Each panel will have BBC number equal to the panel number and the force option and displacement type equal to 1. The user must fill in the backbone curve information for every panel using the BBCI and BBCP commands or loading BBC information from an external

---

file using the BBC command. This command uses the first element of group to determine the panel orientations. This command is intended to be used on a nonlinear panel model. The shells in the model need to have been regrouped using at least the WALLFLR command.

### 9.17.28 S Command

S,<num>,<group>,<elem>,<bbc>,<disp>,<force>

This command adds nonlinear properties to springs. This command does not add any elements to the model. This command adds nonlinear model information to a spring defined with [E](#) command.

- num - nonlinear spring number
- group - group number
- elem - element number
- bbc - backbone curve number
- disp - displacement type
- force - force option

### 9.17.29 SOILREDEF Command (NOT VALIDATED in this version)

SOILREDEF,<soil>,<dir>

This command will redefine the excavation volume layering for a 3D model based on a 2D soil model. Both models should be in ACS SASSI User Interface memory before using this command. The user should select the 3D model to be changed by using the ACTM before using this command. Only the layer list and the excavation volume layer information of the 3D model will be changed by this command.

The 2D soil model must be defined on the global X-Z plane. If the layering modifications is to be applied to the Y horizontal of the 3D model the dir argument can be used to transform the soil model form the X-Z plane to the Y-Z plane.

---

All excavation volume elements for the 3D model must be explicitly defined by ETYPE or ETYPEGEN before using this command. The soil layering redefinition can only be applied along the global coordinate system, but the user can use ROTATE command to adjust the orientation of the 3D model before using this command.

<soil> - the 2D soil model number

<dir> - horizontal direction

0 - X axis

1 - Y axis

### 9.17.30 SOLIDPILE Command

SOLIDPILE,<group>,[stiff],[soft],[stiff2]

This command separates piles from the rest of model, then reconnects piles to the model by adding springs along the pile interface. The command begins by duplicating the nodes along the groups interface and reconnects the element to the duplicate node. Then springs are added between the original node and the duplicate connected to the pile. 4 spring groups will be added for each time this command is issued. The first three spring groups connect sidewalls to the rest of the model with 1d stiffness in each direction. The fourth group connects the bottom of the piles with springs that have stiffness in the Z direction. All springs created have a 4% damping. Pile group for this command must be made of solid elements.

- group - pile group number
- stiff - XY stiffness of sidewall springs (default =  $10^7$ )
- soft - Z stiffness of sidewall springs (default = 10)
- stiff2 - Z stiffness of bottom springs (default  $10^7$ )

### 9.17.31 UNIPNL Command

UNIPNL,<group>

This command will take all of the elements in one group and create a unique group for each element in the group. This command is intended to used when creating panel models for nonlinear analysis. This command only changes groups and element numbers. No element connection, element data will be changed by this command. The user determines which group will be split by the first argument to the command.

- group - group number that will be split into unique element groups.

**9.17.32 WALLFLR Command****WALLFLR**

The WALLFLR command will take the current active model and delete all of the non shell elements in the model. Then the command will attempt to separate all of the shells into different wall and floors groups based on a coplanarity test of the shell elements. If 5 or more elements are found to be coplanar, then these elements will be put into a new group. All of the elements that were not put into wall or floor groups because there were not enough coplanar shells to form a new wall or floor will be found in group 1.

Each of the walls and floors will be labeled using the group title GTIT. The titles are based on if the group is the parallel to a dimension of the global coordinate system or if the wall is oblique of the global coordinate system.

Once this Command is complete the model is considered to be a panel model. The user may use this panel model for Nonlinear Modeling, or the user can further split the model using the PANELIZE command.

### 9.18 Binary Commands

Command	Action	Description
ACCDBANI	Creates an animation from the acceleration binary database in memory	Section 9.17.29
BINFRAMEOUT	Write frame files from a database	Section 9.17.29
BINOUT	output flags for binary files	Section 9.17.29
BINSTRTBL	Output selected stresses from a group in an ACSII CSV format	Section 9.17.29
COMBACCDB	Combine three databases acc binary format	Section 9.17.29
COMBDISPDB	Combine three databases int the disp/acc binary format	Section 9.17.29
COMBDISPDIR	Combine three displacement component binary files into a complete displacement database	Section 9.17.29
COMBTHSDB	combine stress binary database from file	Section 9.17.29
DELDB	Delete a database from User Interface Memory	Section 9.17.29
DISPDBANI	Creates a animation from the displacement binary database in memory	Section 9.17.29
LOADACCDB	Load a acceleration binary database from file	Section 9.17.29
LOADDISPDB	Load a displacement binary database from file	Section 9.17.29
LOADTHSDB	Load a stress binary database from file	Section 9.17.29
MAXDBFRAME	create a frame with the global maximum for all components	Section 9.17.29
THSDBANI	Creates a animation from the stress binary database in memory	Section 9.17.29

### 9.18.1 ACCANIDB Command

ACCDBANI,<dir>,[label]

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

- dir - Buffer directory for the animation files.
- label - Description label of the animation data. This label is stored in the animation database, and is used to identify the selected animation file when the animation data is loaded/reloaded

### 9.18.2 BINFRAMEOUT Command

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

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

If using time value, the user must provide the proper time step. The translation and the rotation will be written into separate files if the split argument is set to 1 for the acceleration or/and displacement databases. For Stress database the split command will be ignored.

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

### 9.18.3 BINOUT Command

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

---

This command sets the binary options for the inputs of the SSI modules. If an argument is left blank the value of the associated flag will remain unchanged.

- mot - Motion binary database
  - 0 - Do not write database
  - 1 - Write binary database
- str - Stress binary database
  - 0 - Do not write database
  - 1 - Write binary database
- reldisp - Relative displacement binary database
  - 0 - Do not write database
  - 1 - Write binary TFD database (Not used in this version)
  - 2 - Write binary THD Database

#### 9.18.4 BINSTRTBL Command

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

Create a CSV table format for selected stresses. The first line of the file will be column labels for each column. Each subsequent line will be the Group and element number then each component stress for that element.

If the step argument is left blank or set to -1 the maximum of each component will be written to the table. This Maximum is the absolute max for the component, but this number will have its sign maintained written to the table. Otherwise, the user must provide an integer number index of the time step to be reported. If this number is higher than the maximum time step an error will be posted to the user in the command window.

The variable specified by the user in EVar must be loaded with the element number before running this command. All element number stored in this variable should be positive integer numbers. If the user request an element that is not in the database all components reported for that element will be shown as 0 in the table.

- group - Group number
- EVar - Variable name of the element list to be written to the file
- step - Time step of the stresses to be written to the file. If time step value is set -1 the maximum of stresses for each component will be printed in the file. (default = -1)
- file - File name for the output file. This filename should contain the full directory path.

#### 9.18.5 COMBACCDB Command

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

---

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

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

### 9.18.6 COMBDISPDB Command

COMBDISPDB,<Xfile>,<Yfile>,<Zfile>,<Comb>

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

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

### 9.18.7 COMBDISPDIR Command

COMBDISPDIR,<Xfile>,<Yfile>,<Zfile>,<Comb>

Specific to the RELDISP relative displacement databases. Combine the three directional displacement component history binary databases for the principal components (as X-X or Y-Y or Z-Z) with the coupling responses (as Y-X, Z-X or X-Y, Z-Y or X-Z, Y-Z) into a single input direction displacement database. It should be used before COMBDISPDB command

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

### 9.18.8 COMBTHSDB Command

COMBTHSDB,<Xfile>,<Yfile>,<Zfile>,<Comb>

Combine three input element stress/force/moment history binary databases for X, Y and Z seismic input into a single database.

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

### 9.18.9 DELDB Command

DELDDB,[sel]

Delete selected binary database(s) from the UI memory during interactive session

- sel - selection for which database to delete.
  - ALL - remove all databases from memory (default)
  - ACC - Acceleration database
  - THS - Stress database
  - DISP - Relative displacement database

### 9.18.10 DISPDBANI Command

DISPDBANI,<dir>,[label]

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

- dir - Buffer directory for the animation files.
- label - Description label of the animation data. This label is stored in the animation database, and is used to identify the selected animation file when the animation data is loaded/reloaded

### 9.18.11 LOADACCDBANI Command

LOADACCDB,<file>

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

- file - full path name of binary acceleration database

### 9.18.12 LOADDISPDB Command

LOADDISPDB,<file>

---

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

- file - full path name of binary displacement database

#### 9.18.13 LOADTHSDB Command

LOADTHSDB,<file>,[sel]

Load the STRESS binary element stress/force/moment history database into the UI memory. Only a single STRESS binary database can be loaded into the UI memory at a time.

- file - full path name of the binary stress database
- sel - format of the binary database
  - 0 - load ACS-SASSI format stress database (default)
  - 1 - load ANSYS® format stress database

#### 9.18.14 MAXDBFRAME Command

MAXDBFRAME,<Type>,[dir]

This command will find the maximum value of a database currently loaded into User Interface memory. The maximum value data will be loaded into animation database as a single frame animation. The data will be written in a binary file located in either the working directory or a user defined directory.

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

#### 9.18.15 THSDBANI Command

THSDBANI,<dir>,[label]

Create an SSI element contour plot animation from the STRESS element output history binary database that is loaded in the UI memory.

- dir - Buffer directory for the animation files.
- desc - Description label of the animation data. This label is stored in the animation database, and is used to identify the selected animation file when the animation data is loaded/reloaded

## 9.19 Thick Shell Commands

Command	Action	Description
THSHLSMH	Average/smooth of the TSHELL element transverse forces	Section 9.17.29
THSHLSTR	Face Strain/Stress for TSHELLS	Section 9.17.29

### 9.19.1 THSHLSMH Command (NON VALIDATED in this version)

THSHLSMH,<passes>,[type],[workdir]

Smear function to be applied to the thick shell shear element output components such as the local transverse shear forces. This function attempts to smooth the adjacent shear forces using either the average (a constant window) or a weighted average based on data windowing

- passes - number of adjacent element layers
- type - type for smoothing
  - 0 - Weighted Average (default); uses a Parzen blending window
  - 1 - Average
- workdir - directory for the output frame (default = working directory)

### 9.19.2 THSHLSTR Command

THSHLSTR,<flag>

Flag for the output of face stress/strain for thick shell elements. This command only effects the values computed by the STRESS module.

- flag - output flag for thick shells face stress/strain
  - 0 - Do not write stress/strain (default). Only write the 8 basic element components
  - 1 - write stress/strain. Write stress/strain for faces based on the maximum values computed for the basic element components.

# 10 Check Errors and Warnings

## 10.1 Overview

After running CHECK, a list of errors and warnings may be displayed. The CHECK in the ACS SASSI User Interface should produce similar output to PREP however the error file now has headers that show which module the errors and warning are associated with. The AFWRITE instruction will first execute the CHECK instruction. Any module that does not pass CHECK will not have the associated module input written.

## 10.2 Errors

### Error 1: Illegal Acceleration of Gravity

- The acceleration of gravity has a negative or zero value.
- Set the analysis options by selecting the Options / Analysis command.

### Error 2: Illegal Node for Symmetry Plane/Line <i>

- The symmetry plane / line <i> has a node either with illegal number or the node was not defined.
- Redefine the node for the symmetry plane/line.

### Error 3: Illegal Number of Nodes for Symmetry Plane/Line <i>

- The symmetry plane / line <i> has only one node.
- Redefine the nodes for the symmetry plane/line .

### Error 4: Nodes from Symmetry Line <i> Are Equal

- The nodes from symmetry line <i> are the same or have the same coordinates.
- Redefine the nodes for the symmetry plane/line.

### Error 5: Nodes from Symmetry Plane <i> Are Collinear

- The nodes from symmetry plane <i> are collinear.
- Redefine the nodes for the symmetry plane/line.

### Error 6: Undefined Element <e>, Group <g>

- Your model has a gap in group <g>, element <e>.
- Define element <e> from group <g> or compress the elements by using the ECOMPR instruction.

**Error 7: Element <e> from Group <g> Has Too Few Nodes**

- Element <e> from group <g> from your model has too few nodes compared with the nodes number of the group type.
- Define all nodes or change the group type.

See also: GROUP instruction.

**Error 8: Element <e>, Group <g> Has 0 Length**

- This error occurs for two-node elements which nodes are the same or have the same coordinates.
- Check both node numbers, and the coordinates of both nodes.

**Error 9: Nodes from Element <e>, Group <g> Are Collinear**

- This error occurs for three-node or four-node elements in which the nodes are the same, have the same coordinates, or are collinear.
- Check all node numbers and the coordinates of all nodes for the specified element.

**Error 10: Element <e>, Group <g> Has Improper Release Code**

- This error occurs when a component of I end release code is set to 1 and the corresponding component of the J end release code is also set to 1.
- Check I and J end release codes and modify one of them with the KI or KJ instructions.

**Error 11: Zero Area for Element <e>, Group <g>**

- This error occurs for four-node elements when some of the nodes are collinear or the same.
- Check all four nodes and their coordinates.

**Error 12: Warped Element <e>, Group <g>**

- This error occurs for four-node elements when not all four nodes are in the same plane.
- Check all four nodes and their coordinates.

**Error 13: Material <m> Is not Defined**

- One of the model's elements uses material <m> which is not defined.
- Check the elements for material / soil layer indices, check element type, or define material <m>.

**Error 14: Elasticity Modulus from Material <m> Is Illegal**

- The elasticity modulus from material <m> has a negative or zero value.
- Correct the elasticity modulus from the specified material.

**Error 15: Poisson Coefficient from Material <m> Is Illegal**

- The Poisson coefficient from material <m> has a negative or zero value.
- Correct the Poisson coefficient from the specified material.

**Error 16: Specific Weight from Material <m> Is Illegal**

- The specific weight from material <m> has a negative value.
- Correct the specific weight from the specified material.

**Error 17: P-Wave Damping Ratio from Material <m> Is Illegal**

- The P-wave damping ratio from material <m> has a negative value.
- Correct the P-wave damping ratio from the specified material.

**Error 18: S-Wave Damping Ratio from Material <m> Is Illegal**

- The S-wave damping ratio from material <m> has a negative value.
- Correct the S-wave damping ratio from the specified material.

**Error 19: Soil Layer <l> Is not Defined**

- One of the model's elements uses soil layer <l> which is not defined.
- Check the elements for material / soil layer indices, check element type, or define soil layer <l>

**Error 20: Thickness from Soil Layer <l> Is Illegal**

- The thickness from soil layer <l> has a negative or zero value.
- Correct the thickness from the specified soil layer.

**Error 21: Specific Weight from Soil Layer <l> Is Illegal**

- The specific weight from soil layer <l> has a negative value.
- Correct the specific weight from the specified soil layer.

**Error 22: P-Wave Velocity from Soil Layer <l> Is Illegal**

- The P-wave velocity from soil layer <l> has a negative value.
- Correct the P-wave velocity from the specified soil layer.

**Error 23: S-Wave Velocity from Soil Layer <l> Is Illegal**

- The S-wave velocity from soil layer <l> has a negative value.
- Correct the S-wave velocity from the specified soil layer.

**Error 24: P-Wave Damping Ratio from Soil Layer <l> Is Illegal**

- The P-wave damping ratio from soil layer <l> has a negative value.
- Correct the P-wave damping ratio from the specified soil layer.

**Error 25: S-Wave Damping Ratio from Soil Layer <l> Is Illegal**

- The S-wave damping ratio from soil layer <l> has a negative value.
- Correct the S-wave damping ratio from the specified soil layer.

**Error 26: Property <p> Is not Defined**

- One of the model's BEAMS elements uses property <p> which is not defined.
- Check the elements for property indices or define property <p>.

**Error 27: Axial Area from Property <p> Is Illegal**

- The axial area from property <p> is less than or equal to 0.
- Correct the area from the specified property.

**Error 28: Shear Area 2 from Property <p> Is Illegal**

- The shear area 2 from property <p> has a negative value.
- Correct the shear area 2 from the specified property.

**Error 29: Shear Area 3 from Property <p> Is Illegal**

- The shear area 3 from property <p> has a negative value.
- Correct the shear area 3 from the specified property.

**Error 30: Torsion Inertia Moment from Property <p> Is Illegal**

- The torsion inertia moment from property <p> has a negative or zero value.
- Correct the torsion inertia moment from the specified property.

**Error 31: Flexural Inertia Moment 2 from Property <p> Is Illegal**

- The flexural inertia moment 2 from property <p> has a negative value.
- Correct the flexural inertia moment 2 from the specified property.

**Error 32: Flexural Inertia Moment 3 from Property <p> Is Illegal**

- The flexural inertia moment 3 from property <p> has a negative value.
- Correct the flexural inertia moment 3 from the specified property.

**Error 33: Spring Property <p> Is not Defined**

- One of the model's SPRING elements uses spring property <p> which is not defined.
- Check the elements for property indices or define spring property <p>. (instruction SC)

**Error 34: Spring Constant X from Spring Property <p> Is Illegal**

- The spring constant x from spring property <p> has a negative value.
- Correct spring constant x from the specified spring property .

**Error 35: Spring Constant Y from Spring Property <p> Is Illegal**

- The spring constant y from spring property <p> has a negative value.
- Correct spring constant y from the specified spring property .

**Error 36: Spring Constant Z from Spring Property <p> Is Illegal**

- The spring constant z from spring property <p> has a negative value.
- Correct spring constant z from the specified spring property .

**Error 37: Spring Constant XX from Spring Property <p> Is Illegal**

- The spring constant xx from spring property <p> has a negative value.
- Correct spring constant xx from the specified spring property .

**Error 38: Spring Constant YY from Spring Property <p> Is Illegal**

- The spring constant yy from spring property <p> has a negative value.
- Correct spring constant yy from the specified spring property.

**Error 39: Spring Constant ZZ from Spring Property <p> Is Illegal**

- The spring constant zz from spring property <p> has a negative value.
- Correct spring constant zz from the specified spring property.

**Error 40: Mass out of Defined Nodes Range**

- Check found masses that exceed the defined nodes range.
- Define more nodes or delete the masses. (instructions MTDEL and MRDEL)

**Error 41: Node <n> Is Not Defined - Group <g> Element <e>**

- Element <e> from group <g> uses node <n> which is not defined.
- Check the element for node errors or define node <n>.

**Error 42: No Nodes Defined**

- The model has no nodes.
- This is a fatal error. Check has stopped after this message. You cannot check a model without nodes.

**Error 43: No Groups Defined**

- Your model has no groups.
- This is a fatal error. Check has stopped after this message. You cannot check a model without groups.

**Error 44: Frequency Set <s> Is Not Defined**

- The frequency set specified in the analysis options is not defined.
- Check the analysis options or define the frequency set.

**Error 45: Mode 1 and Mode 2 Are Both Deselected**

- The mode 1 and mode 2 options for the SITE module are both deselected.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 46: No Top Layers**

- The top layers list for the SITE module is empty.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 47: Illegal Number of Layers for Halfspace Simulation**

- The number of layers for halfspace simulation must either be zero or lie between 4 and 20.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 48: Illegal Frequency Step**

- The frequency step has a negative value.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 49: Illegal Time Step of Control Motion**

- The time step has a negative value.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 50: Illegal Number of Values for Fourier Transform**

- The number of values for Fourier Transform has a negative value.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 51: All Wave Fields Are Deselected**

- All wave field options are deselected.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 52: Illegal Incident Angle of Wave <w>**

- The incident angle of wave <w> is not in the range (0,360) degrees.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 53: Illegal Value for Frequency <i>**

- The value for frequency <i> of ratio curve is less than or equal to zero.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 54: Illegal Value for Wave <w> Ratio at Frequency <i>**

- The value for the wave ratio at frequency <i> is less than or equal to zero or is greater than 1.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 55: Illegal Sum of Wave Ratios at Frequency <i>**

- The sum of the wave ratios for selected wave types at frequency <i> is not 1.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 56: Illegal Last Layer Number in Near Field Zone**

- The last layer number has a negative value.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 57: Illegal Radius of Central Zone**

- The radius of the central zone has a zero or negative value.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 58: Illegal Coherence Parameter**

- The coherence parameter must be greater than or equal to 0.1.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 59: Illegal Mean Soil Shear Wave Velocity**

- The mean soil shear wave velocity is less than or equal to zero. Set the analysis parameters by selecting the Options / Analysis command.

**Error 60: Illegal Number of Mesh Points / Embedment Level**

- The number of mesh points is negative or zero.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 61: No Forces Defined**

- Your model has no forces or moments for the FORCE module.
- Define a set of forces or moments using the F or MM instructions.

**Error 62: Node for Force / Moment <i> is not defined**

- Your model has a force / moment on an undefined node.
- Define the node using the N instruction or delete the force / moment using the FDEL or MMDEL instructions.

**Error 63: Illegal Coordinate Transformation Angle**

- The coordinate transformation angle is not in the range (0,360) degrees.
- Set the analysis parameters by selecting the Options / Analysis command.

**Error 64: No Nodal Output Request**

- There are no node lists for output defined.
- Add some node lists by selecting the Options / Analysis command.

**Error 65: Illegal Nodal Output Request: <n>**

- One of the defined node lists for output requests contains the illegal node number <n>, explicitly or contained in a range.
- Correct the node list by selecting the Options / Analysis command.

**Error 66: Nodal Output Request Defined More Than Once: <n>**

- The defined node lists for output requests contain the node number <n> more than once, explicitly or contained in a range.
- Correct the node lists by selecting the Options / Analysis command.

**Error 67: Illegal Output Time History Step**

- The output time history step has a negative value.
- Correct output time history step by selecting the Options / Analysis command.

**Error 68: Illegal Total Duration To Be Plotted**

- The total duration to be plotted has a negative value.
- Correct total duration to be plotted by selecting the Options / Analysis command.

**Error 69: Illegal First Frequency for RS Analysis**

- The first frequency for RS analysis has a negative value.
- Correct first frequency for RS analysis by selecting the Options / Analysis command.

**Error 70: Illegal Last Frequency for RS Analysis**

- The last frequency for RS analysis has a negative value.
- Correct last frequency for RS analysis by selecting the Options / Analysis command.

**Error 71: Illegal Number of Frequency Steps For RS Analysis**

- The number of frequency steps for RS analysis has a negative value.
- Correct number of frequency steps for RS analysis by selecting the Options / Analysis command.

**Error 72: Illegal Damping Ratio For RS Analysis**

- One of the damping ratios from the damping ratio list has a value that is not between 0 and 1.
- Correct damping ratio for RS analysis by selecting the Options / Analysis command.

**Error 73: Acceleration Time History File Does Not Exist**

- The acceleration time history file does not exist or is misspelled.
- Correct the acceleration time history file by selecting the Options / Analysis command.

**Error 74: Illegal First Record Number**

- The first record number for the acceleration time history file has a negative value or is greater than the number of records in the file.
- Correct the first record number by selecting the Options / Analysis command.

**Error 75: Illegal Last Record Number**

- The last record number for the acceleration time history file has a negative value.
- Correct the last record number by selecting the Options / Analysis command.

**Error 76: First Record Number Larger Than Last Record Number**

- The first record number for the acceleration time history file is larger than the last record number.
- Correct the record numbers by selecting the Options / Analysis command.

**Error 77: Multiplication Factor and Maximum Value Of Time History Are Both Zero**

- The multiplication factor and maximum value used to scale the acceleration time history file are both zero.
- Correct the multiplication factor or maximum value by selecting the Options / Analysis command.

**Error 78: Multiplication Factor and Maximum Value Of Time History Are Both Non-Zero**

- The multiplication factor and maximum value used to scale the acceleration time history file are both non-zero.
- Correct the multiplication factor or maximum value by selecting the Options / Analysis command.

**Error 79: No Element Output Request**

- There are no element lists for output defined.
- Add some element lists by selecting the Options / Analysis command.

**Error 80: Illegal Group For Output Request: <g>**

- One of the defined element lists for output requests are set to the illegal group number <g>.
- Correct the element list by selecting the Options / Analysis command.

**Error 81: Illegal Element Output Request: <e>, Group <g>**

- One of the defined element lists for output requests set to group number <g> contains the illegal element number <e>, explicitly or contained in a range.
- Correct the element list by selecting the Options / Analysis command.

**Error 82: Element Output Request Defined More Than Once: <e>, Group <g>**

- The defined element lists for output requests contain the element number <e> of group <g> more than once, explicitly or contained in a range.
- Correct the element lists by selecting the Options / Analysis command.

**Error 83: Matrix Property <p> Is not Defined**

- One of the model's GENERAL elements uses matrix property <p> which is not defined.
- Check the elements for property indices or define matrix property <p>. (see instructions MXR, MXI, and MXM)

**Error 84: No RS Input Files Specified**

- All three spectrum input files for ACS SASSI EARTH module are left blank.
- Set at least one spectrum input file by selecting the Options / Analysis command.

**Error 85: RS Input File <i> Does Not Exist**

- The spectrum input file <i> for ACS SASSI EQuAKE module does not exist.
- Set the spectrum input file <i> by selecting the Options / Analysis command.

**Error 86: Invalid RS Output File <i>**

- The spectrum output file <i> for ACS SASSI EQuAKE module is left blank.
- Set the spectrum output file <i> by selecting the Options / Analysis command.

**Error 87: Invalid Acceleration Output File <i>**

- The acceleration output file <i> for ACS SASSI EQuAKE module is left blank.
- Set the acceleration output file <i> by selecting the Options / Analysis command.

**Error 88: Invalid Acceleration Input File <i>**

- The acceleration input file <i> for ACS SASSI EQuAKE module is left blank.
- Set the acceleration output file <i> or deselect the acceleration input option by selecting the Options / Analysis command.

**Error 89: Number of Frequencies Does Not Match RS Input File <i>**

- The number of records from the spectrum input file <i> for ACS SASSI EQuAKE module is not equal to the number of frequencies.
- Change the number of frequencies or change the spectrum input file <i> by selecting the Options / Analysis command.

**Error 90: Illegal Initial Random Number**

- The initial random number for ACS SASSI EQuAKE module is less or equal to zero.
- Correct the initial random number by selecting the Options / Analysis command.

**Error 91: Illegal Number of Frequencies**

- The number of frequencies for ACS SASSI EQuAKE module is less or equal to zero.
- Correct the number of frequencies by selecting the Options / Analysis command.

**Error 92: Illegal Duration**

- The total duration for ACS SASSI EQuAKE module is less or equal to zero.
- Correct the total duration by selecting the Options / Analysis command.

**Error 93: No Correlation Factors Defined**

- The correlation factors for ACS SASSI EQUAKE module are not defined.
- Define the correlation factors or deselect the correlation option by selecting the Options / Analysis command.

**Error 94: Illegal Correlation Factor**

- One of the correlation factors for ACS SASSI EQUAKE module is greater than 1.
- Correct the correlation factor by selecting the Options / Analysis command.

**Error 95: No Dynamic Soil Properties Assigned**

- No dynamic soil properties were assigned to sublayers for ACS SASSI SOIL module.
- Set the dynamic soil properties by selecting the Options / Analysis command.

**Error 96: Too Many Dynamic Soil Properties**

- The sublayers for ACS SASSI SOIL module use more than 15 dynamic soil properties.
- Change the dynamic soil properties by selecting the Options / Analysis command.

**Error 97: Dynamic property <p> has no shear modulus curve**

- The dynamic soil property <p> has no data defining the shear modulus - shear strain curve.
- Define the shear modulus - shear strain curve for the dynamic soil property by selecting the Options / Analysis command.

**Error 98: Dynamic property <p> has illegal shear modulus values**

- One of the shear modulus values defining the dynamic soil property <p> is less than zero or greater than 1.
- Correct the shear modulus for the dynamic soil property by selecting the Options / Analysis command.

**Error 99: Dynamic property <p> has no damping curve**

- The dynamic soil property <p> has no data defining the damping ratio - shear strain curve.
- Define the damping ratio - shear strain curve for the dynamic soil property by selecting the Options / Analysis command.

**Error 100: Number of Acceleration Values Is Illegal**

- The number of acceleration values for ACS SASSI SOIL module is less than or equal to zero.
- Correct the number of acceleration values by selecting the Options / Analysis command.

**Error 101: Cut-Off Frequency Is Illegal**

- The cut-off frequency for ACS SASSI SOIL module is less than zero.
- Correct the cut-off frequency by selecting the Options / Analysis command.

**Error 102: Illegal Reading Format**

- The reading format for ACS SASSI SOIL module is illegal.
- Correct the reading format by selecting the Options / Analysis command.

**Error 103: Illegal Number of Header Lines**

- The number of header lines for ACS SASSI SOIL module is less than zero.
- Correct the number of header lines by selecting the Options / Analysis command.

**Error 104: Illegal Control Layer Number**

- The control layer number for ACS SASSI SOIL module is illegal.
- Correct the control layer number by selecting the Options / Analysis command.

**Error 105: Illegal Number of Iterations**

- The number of iterations for ACS SASSI SOIL module is less than zero.
- Correct the number of iterations by selecting the Options / Analysis command.

**Error 106: Illegal Strain Ratio**

- The strain ratio for ACS SASSI SOIL module is not between 0 and 1.
- Correct the strain ratio by selecting the Options / Analysis command.

**Error 107: No Damping Ratios Defined**

- There are no defined damping ratios for ACS SASSI SOIL.
- Define the damping ratios by selecting the Options / Analysis command.

**Error 108: Illegal Multiplier for Acceleration of Gravity**

- The multiplier for the acceleration of gravity for ACS SASSI SOIL module is less than or equal to 0.
- Correct the multiplier for the acceleration of gravity by selecting the Options / Analysis command.

**Error 109: Illegal Second Layer Number for Layer <i>**

- The second layer number for layer <i> for ACS SASSI SOIL module is illegal.
- Correct the second layer number by selecting the Options / Analysis command.

**Error 110: Illegal Frequency Step for Layer <i>**

- The frequency step for layer <i> for ACS SASSI SOIL module is less than or equal to zero.
- Correct the frequency step by selecting the Options / Analysis command.

**Error 111: Illegal Number of Smoothings for Layer <i>**

- The number of times the spectrum is to be smoothed for layer <i> for ACS SASSI SOIL module is less than zero.
- Correct the number of smoothings by selecting the Options / Analysis command.

**Error 112: Illegal Number of Values to Be Saved for Layer <i>**

- The number of values to be saved for layer <i> for ACS SASSI SOIL module is less than zero.
- Correct the number of values to be saved by selecting the Options / Analysis command.

**Error 113: Illegal Apparent Velocity for Line D**

- The apparent velocity for line D is less than or equal to zero.
- Correct the apparent velocity for line D by selecting the Options / Analysis command.

**Error 114: Illegal Directional Coherence Factor**

- The directional coherence factor is less than zero.
- Correct the directional coherence factor by selecting the Options / Analysis command.

**Error 115: No Multiple Excitation Data Defined**

- The multiple excitation option is selected, but no data is defined.
- Define multiple excitation data or deselect the option by selecting the Options / Analysis command.

**Error 116: Illegal First Node Number for Motion <i>**

- The first node number for motion <i> is illegal.
- Correct the first node number by selecting the Options / Analysis command.

**Error 117: Illegal Last Node Number for Motion <i>**

- The last node number for motion <i> is illegal.
- Correct the last node number by selecting the Options / Analysis command.

**Error 118: Illegal Spectral Amplification Ratio for Motion <i>**

- One of the spectral amplification factors is not between 0 and 10.
- Correct the spectral amplification factor by selecting the Options / Analysis command.

**Error 119: Spectral Amplification Ratios for Motion <i> Do Not Match Frequencies**

- The number of spectral amplification ratios must be equal to the number of frequencies from the selected frequency set.
- Correct the spectral amplification ratios or the frequency set by selecting the Options / Analysis command.

**Error 120: Frequency set <i> is Empty. Change the active frequency set in the SITE tab of the Analysis Options**

- The Frequency set for relative displacement is empty the user can change the set by going to the site menu in analysis option
- The user can add frequencies to the set by using the FREQ command

**Error 121: No Panels Specified**

- The user attempted to use the panel module without defining any panels
- The user can use the P command to defined a panel

**Error 122: Material <i> referenced in panel <k> does not exist**

- A material in a panel definition does not exist.
- The user can change the panel to use a material that does exist by using the P command.
- The user can add a material that uses reference number <i> using the M command.

**Error 123: Group <i> referenced in panel <k> does not exist**

- A group in a panel definition does not exist.
- The user can change the panel to use a group that does exist by using the P command.
- The user can add a group that uses reference number <i> using the GROUP command.

**Error 124: Node <i> is a fixed interaction node**

- Node <i> is fully fixed and an interaction node.
- The user can change the interaction status of the node <i> using the INT command.
- The user can change the fixed status of the node <i> using the D command.

**Error 125: Not enough nonlinear soil properties have been defined for the Nonlinear soil model.**

- Nonlinear soil layers are less than number of soil layers

**Error 126: Force Option <i> not supported for panel <k> in this version**

- Force option not supported for panel in this version of the user interface
- Set the force option value = 1

**Error 127: Force Option <i> not supported for spring <k>**

- Force option not supported for springs in this version of the user interface
- Set the force option value = 4

**Error 128: Displacement Option <i> not supported for panel <k>**

- Displacement option not supported for panel in this version of the user interface
- Set the displacement option value = 1

### 10.3 Warnings

**Warning 1: Gap Found at Node <n>**

Check found a gap at node <n>. You can define node <n> with fixed DOF, or AFWRITE will generate it automatically (only in the analysis files, not in the model) with fixed DOF.

**Warning 2: Distorted Element <e>, Group <g>, Face <f>**

This message occurs for SHELL, PLANE elements with four or three nodes, when the smallest angle is too small in comparison with the greatest angle. All faces are checked for SOLID elements. This is not an error, but you can get bad results from the analysis program.

**Warning 3: Warped Element <e>, Group <g>**

This message occurs for four-nodes SHELL, PLANE elements when the four nodes are not quite on the same plane.

**Warning 4: Unused Node <n>**

Check found node <n> that is not used in any element (except K-nodes from BEAMS elements). You can set the node to fixed DOF, or AFWRITE will set the fixed DOF automatically in the analysis files, not in the model.

**Warning 5: Translational Mass in Node <n> Is on Fixed DOF**

A translational mass was defined on node <n>, which has fixed DOFs. This is not an error, but the analysis program will not take that mass into account.

**Warning 6: Rotational Mass in Node <n> Is on Fixed DOF**

A rotational mass was defined on node <n>, which has fixed DOFs. This is not an error, but the analysis program will not take that mass into account.

**Warning 7: Group <g> Has no Elements**

Check found group <g> with no elements. This is not an error, AFWRITE will skip this group when writing the analysis file.

**Warning 8: Too Many Top Layers**

The top layer list from module SITE has more than 100 soil layers. Only the first 100 are written.

**Warning 9: Number of Values for Fourier Transform Is Not Power of 2**

The number of values for Fourier transform must be a power of 2. ACS SASSI PREP writes the nearest power of 2 to the analysis files.

**Warning 10: Force in Node <n> Is on Fixed DOF**

A force was defined on node <n>, which has fixed DOFs. This is not an error, but the analysis program will not take that force into account.

**Warning 11: Moment in Node <n> Is on Fixed DOF**

A moment was defined on node <n>, which has fixed DOFs. This is not an error, but the analysis program will not take that moment into account.

# 11 References

1. Abrahamson, N. (1993), "Spatial Variation of Multiple Support Inputs", the 1st US Seminar on Seismic Evaluation and Retrofit of Steel Bridges, *University of California at Berkeley*, San Francisco, October 18, 1993.
2. Abrahamson, N. (2005). "Spatial Coherency for Soil-Structure Interaction", *Electric Power Research Institute Report 1012968*, Palo Alto, CA. December.
3. Abrahamson, N. (2006). Program on Technology Innovation: "Spatial Coherency for Soil-Structure Interaction", *Electric Power Research Institute, Palo Alto, CA*, and US Department of Energy, Germantown, MD, Report 1014101, December.
4. Abrahamson, N. (2007). "Hard Rock Coherency Functions Based on the Pinyon Flat Data", *Electric Power Research Institute, Palo Alto, CA*, and US Department of Energy, Germantown, MD, Report 1015110, December.
5. American Concrete Institute, "Code Requirements for Nuclear Safety-Related Concrete Structures", *ACI 349-06*, 2006.
6. American Society of Civil Engineers, "Seismic Design Criteria for Structures, Systems, and Components in Nuclear Facilities", *ASCE 43-05*, 2005.
7. American Society of Civil Engineers, "Seismic Analysis for Safety-related Nuclear Structures and Commentary", *ASCE 4-17*, 2017 (to be published in March, 2017).
8. Applied Technology Council, *Modeling and Acceptance Criteria for Seismic Design and Analysis of Tall Buildings*, PEER/ATC 72-1, October 2010.
9. Barda, F., J.M. Hanson, and G.W. Corley, *Shear Strength of Low-Rise Walls with Boundary Elements*, ACI Special Publication SP 53-8, 1977.
10. Gergely, P. (1984). "Seismic Fragility of Reinforced Concrete Structures and Components for Application in Nuclear Facilities", Prepared for USNRC, Published as NUREG/CR-4123,
11. Ghiocel D., M. (2015a) "SASSI Flexible Volume Substructuring Methods for Deeply Embedded Structures; Selection of Excavated Soil Interaction Nodes and Element Meshing" *SMIRT22 Proceedings*, Division V, Manchester, UK, August 10-14
12. Ghiocel D., M. (2015b) "Seismic Motion Incoherency Effects of Soil-Structure Interaction (SSI) and Structure-Soil-Structure Interaction (SSSI) of Nuclear Structures for Different Soil Site Conditions", *SMIRT22 Proceedings*, Division V, Manchester, UK, August 10-14
13. Ghiocel D., M. (2015c) "Fast Nonlinear Seismic SSI Analysis Using A Hybrid Time-Complex Frequency Approach For Low-Rise Nuclear Concrete Shearwall Buildings", *SMIRT22 Proceedings*, Division V, Manchester, UK, August 10-14
14. Ghiocel D, M. (2014a). "Effects of Seismic Motion Incoherency on SSI and SSSI Responses of Nuclear Structures for Different Soil Site Conditions", *U.S. Department of Energy Natural Phenomena Hazards Meeting*, Session on "Soil-Structure Interaction", Germantown, MD,

USA, October 21-22

15. Ghiocel D, M. (2014b). SASSI Methodology-Based Sensitivity Studies for Deeply Embedded Structures Such As Small Modular Reactors (SMRs)", *U.S. Department of Energy Natural Phenomena Hazards Meeting*, Session on "Soil-Structure Interaction", Germantown, MD, USA, October 21-22
16. Ghiocel, D.M., Yue, D., Fuyama, H., Kitani, T. and McKenna, M. (2013a), "Validation of Modified Subtraction Method for Seismic SSI Analysis of Large-Size Embedded Nuclear Islands", *SMiRT22 Proceedings*, Division V, San Francisco, California, August 18-23
17. Ghiocel, D.M., "Comparative Studies on Seismic Incoherent SSI Analysis Methodologies" (2013b), *SMiRT22 Proceedings*, Division V, San Francisco, California, August 18-23
18. Ghiocel, D.M., Todorovski, L., Fuyama, H. and Mitsuzawa, D., "Seismic Incoherent Soil-Structure Analysis of A Reactor Building Complex on A Rock Site" (2011a), *SMiRT21 Proceedings*, Division V, Paper 825, New Delhi, India, November 6-11
19. Ghiocel, D.M., Lee, I. (2011b), "Seismic Soil-Structure Interaction (SSI) Effects for Large-Size Surface and Embedded Nuclear Facility Structures", *2011 World Congress on Advances in Structural Engineering and Mechanics*, ASEM11, Seoul, Korea, September 18-22
20. Ghiocel, D.M. (2010a) "Some Insights and Brief Guidance for Application of Subtraction/Flexible Interface Method to Seismic SSI Analysis of Embedded Nuclear Facilities", GP Tech Technical Investigation Report, GP Technologies, Rochester, NY, GPT-TIR-01-0930-2010, September 30
21. Ghiocel, D.M., Todorovski, L., Fuyama, H. (2010b), "Seismic SSI Response of Reactor Building Structures", *Proceedings of the OECD NEA/IAEA SSI Workshop*, Ottawa, October 6-8, 2010
22. Ghiocel, D.M., Short, S. and Hardy, G. (2010c) "Seismic Motion Incoherency Effects for Nuclear Complex Structures On Different Soil Site Conditions", *Proceedings of the OECD NEA/IAEA SSI Workshop*, Ottawa, October 6-8
23. Ghiocel, D.M, Li, D., Brown, N. and Zhang, J.J. (2010d) "EPRI AP1000 NI Model Studies on Seismic Structure-Soil-Structure Interaction (SSSI) Effects", *Proceedings of the OECD NEA SSI Workshop*, Ottawa, October 6-8
24. Ghiocel, D.M, Stoyanov, G., Adhikari, S. and Aziz, T.(2010e) , "Seismic Motion Incoherency Effects for CANDU Reactor Building Structure", *Proceedings of the OECD NEA/IAEA SSI Workshop*, Ottawa, October 6-8
25. Ghiocel, D.M., Li, D., Tunon-Sanjur, L. (2009a). "Seismic Motion Incoherency Effects for AP1000 Nuclear Island Complex", *The 20th Structure Mechanics in Reactor Technology*, *Proceedings of SMiRT20 Conference*, Paper 1852, Helsinki, August 14-18
26. Ghiocel, D.M., Short, S. and Hardy, G. (2009b). "Seismic Motion Incoherency Effects on SSI Response of Nuclear Islands with Significant Mass Eccentricities and Different Embedment Levels" *The 20th Structure Mechanics in Reactor Technology*, *Proceedings of SMiRT20*

Conference, Paper 1853, Helsinki, August 14-18

27. Ghiocel, D.M. (2007a). "Stochastic and Deterministic Approaches for Incoherent Seismic SSI Analysis as Implemented in ACS SASSI", Appendix C, Electric Power Research Institute, Palo Alto, CA and US Department of Energy, Germantown, MD, EPRI Report TR-1015110, November
28. Ghiocel, D.M. And Ostadan, F. (2007b). "Seismic Ground Motion Incoherency Effects on Soil-Structure Interaction Response", The 19th Structure Mechanics in Reactor Technology, Proceedings of SMiRT19 Conference, Paper K05/4, Toronto, August
29. Ghiocel, D.M. (2004) "Stochastic Simulation in Engineering Predictions", Chapter 20, the CRC Press Handbook on "Engineering Design Reliability", Eds. Nikolaidis, Ghiocel and Singhal, CRC Press, December
30. Ghiocel, D.M. and Wang, L. (2004) "Seismic Motion Incoherency Effects on Structures", Proceedings 3<sup>rd</sup> UJNR Workshop on Soil-Structure Interaction, March 29-30, 2004, Menlo Park, California, March 30-31
31. Ghiocel, D.M. and Ghanem, R. (2002), "Stochastic Finite-Element Analysis of Seismic Soil-Structure Interaction", ASCE, Journal of Engineering Mechanics, Vol. 128, No. 1, January
32. Ghiocel, D.M., (1998), "Uncertainties of Seismic Soil-Structure Interaction Analysis: Significance, Modeling and Examples", Proceedings of the US-Japan SSI Workshop, USGS, Menlo Park, California, September 22-23
33. Ghiocel, D.M et. al. (1996a) "Seismic Motion Incoherency Effects on Dynamic Response", 7th ASCE Speciality Conference on Probabilistic Mechanics and Structural Reliability, Worcester
34. Ghiocel, D.M. et al. (1996b), "On Soil-Structure Interaction (SSI) Issues for Deep Foundation Structures", 6<sup>th</sup> Symposium on NPP Structures, Equipment and Piping, Raleigh, North Carolina, December
35. Ghiocel, D.M. et al. (1996c), "Seismic Motion Incoherency Effects on Dynamic Response", 7th ASCE EMD/STD Joint Speciality Conference on Probabilistic Mechanics and Structural Reliability, Worcester, MA, August
36. Ghiocel, D.M. et al. (1996d), "Probabilistic Seismic Analysis Including Soil-Structure Interaction", 7th ASCE EMD/STD Joint Speciality Conference on Probabilistic Mechanics and Structural Reliability, Worcester, MA, August,
37. Ghiocel, D.M. et al. (1996e). "Effects of Random Field Modeling of Seismic Ground Motion on Structural Dynamic Response", 37th AIAA/ASME/ASCE Conference on Structures, Structural Dynamics and Materials, Section on Probabilistic Applications, Salt Lake City, April, 1996
38. Ghiocel, D.M.,et al. (1995), "Seismic Soil-Structure Interaction Effects on Probabilistic Floor Response Spectra", Proceedings of the 1995 Annual ASME PVP Conference, Honolulu, July

39. Ghiocel, D. M. et al. (1991). "Evaluation of Seismic Soil-Structure Interaction by Different Approaches", 2<sup>nd</sup> International Conference on Recent Advances in Geotechnical Earthquake Engineering and Soil Dynamics, Vol. 3, St. Louis, Missouri-Rolla, March
40. Ghiocel, D. M. et al. (1990a). "Seismic Soil-Structure Interaction Effects for Shearwall Buildings on Soft Clays", 10<sup>th</sup> European Conference on Earthquake Engineering, ECEE, Moscow, September
41. Ghiocel, D.M., et al. (1990b). "Evaluation of SSI and SSSI Effects on Seismic Response of Nuclear Heavy Buildings by Different Approaches", the 10th European Conference on Earthquake Engineering, ECEE, Moscow, September
42. Ghiocel, D. M. (1986a). "Probabilistic Seismic Soil-Structure Interaction Analysis", 8th European Conference on Earthquake Engineering, ECEE, Lisbon, September
43. Ghiocel, D. M et al. (1986b). "Effects of Spatial Character of Seismic Random Excitations on Structures", 8th European Conference on Earthquake Engineering, ECEE, Lisbon, September
44. Ghiocel, D. M. (1986c). "PRELAMOS - Computer Program for Stochastic Parameter Estimation of Accelerograms", 5<sup>th</sup> National Symposium on Informatics in Civil Engineering, Sibiu, September
45. Ghiocel, D.M. (1985a). "Probabilistic Seismic Analysis of the Containment Structure of a Nuclear Reactor Building", Journal of Civil Engineering, vol. 11, Bucharest
46. Ghiocel, D.M., et al. (1985b). "Seismic Risk Evaluation for Buildings Including Soil-Structure Interaction", Scientific Bulletin of Civil Engineering, ICB, Bucharest
47. Ghiocel, D.M., et al. (1983a). "Actual Tendencies in Seismic Analysis of Nuclear Power Plants Structures", Scientific Bulletin of Civil Engineering, vol.2, ICB, Bucharest
48. Ghiocel, D.M.,et al., (1983b). "Advanced Computational Methods in Seismic Analysis of Nuclear Power Plants, Scientific Bulletin of Civil Engineering, vol. 1, ICB, Bucharest
49. Ghiocel, D. M., et al. (1983c). "Structural Reliability Analysis of Nuclear Power Plant Subjected to Seismic Load using Advanced Numerical Methods", National Symposium on Cybernetic Applications in Industry, Bucharest, June
50. Gulec, C.K. and A.S. Whittaker, *Performance-Based Assessment and Design of Squat Reinforced Concrete Shear Walls*, Technical Report, MCEER, No. 09-0010, Sept. 2009.
51. Hardin, B. O. and Drnevich, V. P. (1972): "Shear Modulus and Damping in Soils: Design Equations and Curves", Journal of the Soil Mechanics and Foundations Division, ASCE, Vol. 98, No., SM7, 667 – 692.
52. Johnson, J.J., Short, S.A., and Hardy, G.S. (2007). Modeling Seismic Incoherence Effects on NPP Structures: Unifying CLASSI and SASSI Approaches, Paper K05/5, SMiRT 19, Toronto, Canada, August 12-17, 2007.
53. Reed J.,W. and Kennedy R.P (1994). Prepared for EPRI, Published in EPRI TR# 103959

entitled "Methodology for Developing Seismic Fragilities".

54. Luco, J.E. and A. Mita (1987). Response of Circular Foundation to Spatially Random Ground Motion, *Journal of Engineering Mechanics*, American Society of Civil Engineers, Vol. 113, No. 1, January. 1987, pp. 1-15.
55. Luco, J.E. and H.L. Wong (1986). Response of a Rigid Foundation Subjected to a Spatially Random Ground Motion, *Earthquake Engineering and Structural Dynamics*, Vol. 14, pp. 891-908.
56. Lysmer, J. and Kuhlemeyer, R. L. (1969): " Finite Dynamic Model for Infinite Media", *Journal of Engineering Mechanics Division*, ASCE, Vol. 95, No. EM4, 859 – 877.
57. Lysmer, J., Udaka, T., Tsai, C - F. and Seed, H. B. (1975): "FLUSH - A Computer Program for Approximate 3-D Analysis of Soil Structure Interaction Problems", Report No. ERC 75 - 30, Earthquake Engineering Research Center, UCB, November.
58. Lysmer, J. (1978): Analytical Procedures in Soil Dynamics, "Report No. EERC 78/29, Earthquake Engineering Research Center, University of California, Berkeley, December.
59. Lysmer, J., Tabatabaie - Raissi, M., Tajirian, F., Vahdani, S., and Ostadan, F (1981); "SASSI - A System for Analysis of Soil - Structure Interaction", Report No. UCB 81 - 02, Geotechnical Engineering, University of California, Berkeley, April.
60. Nie, J. Braverman, J., and Costantino, M. (2013). "Seismic Soil-Structure Interaction Analyses of a Deeply Embedded Model Reactor – SASSI Analyses", US Department of Energy, Brookhaven National Laboratory, BNL 102434-2013, New York
61. Ostadan, F., Tseng, Wen S., Lilhanand, K. (1987). "Application of Flexible Volume Method to Soil-Structure Interaction Analysis of Flexible and Embedded Foundation", 9th SMIRT Conference, Lausanne, Switzerland.
62. Seed, H. B. and Idriss, I. M. (1969): "The Influence of Soil Conditions on Ground Motion during Earthquake", *Journal of Soil Mechanics and Foundations Division*, ASCE, Vol. 94, No., SM1, 99 - 137, December.
63. Seed, H. B. and Idriss, I. M. (1970): "Soil Moduli and Damping Factors for Dynamic Response Analysis", Report EERC 70 - 10, Earthquake Engineering Research Center, UCB, December.
64. Short, S.A., G.S. Hardy, K.L. Merz, and J.J. Johnson (2006). Program on Technology Innovation: Effect of Seismic Wave Incoherence on Foundation and Building Response, Electric Power Research Institute, Palo Alto, CA and US Department of Energy, Germantown, MD, Report No. EPRI TR-1013504, December.
65. Short, S.A., G.S. Hardy, K.L. Merz, and J.J. Johnson (2007). Validation of CLASSI and SASSI to Treat Seismic Wave Incoherence in SSI Analysis of NPP Structures, Electric Power Research Institute, Palo Alto, CA and US Department of Energy, Germantown, MD, EPRI Report TR-1015110, November
66. Tabatabaie - Raissi, M. (1982): "Flexible Volume Method for Dynamic Soil - Structure

- Interaction Analysis”, Ph. D. Dissertation, University of California, Berkeley.
67. Tajirian. F. (1981): “Impedance Matrices and Interpolation Techniques for 3-D Interaction Analysis by the Flexible Volume Method”, Ph. D. Dissertation, University of California, Berkeley.
  68. Tseng and Lilhanand. (1997) Soil-Structure Interaction Analysis Incorporating Spatial Incoherence of Ground Motions, Electric Power Research Institute, Palo Alto, CA, Report No. TR-102631 2225.
  69. US NRC ISG Report (2008), “Seismic Issues Associated with High Frequency Ground Motion in Design Certification and Combined License Applications”, ISG-01, US Nuclear Regulatory Commission, Washington, DC, May, <http://www.nrc.gov/reading-rm/doc-collections/isg/col-app-design-cert.html>
  70. Vahdani, S. (1984) “Impedance Matrices for Soil- Structure Interaction Analysis by the Flexible Volume Method”, Ph. D. Dissertation, University of California, Berkely.
  71. Wass, G. (1972) “Earth Vibration Effects and Abatement for Military Facilities- Analysis Method for Footing Vibrations Through Layered Media”, Technical Report S - 71 - 14, U. S. Army Engineer Waterways Experimental Station, Vicksburg, Mississippi, September.
  72. Wood, S., “Shear Strength of Low-Rise Reinforced Concrete Walls,” ACI Structural Journal, Vol. 87, No. 12, pp. 99-107, 1990.