Transactions, SMiRT-24
BEXCO, Busan, Korea - August 20-25, 2017
Division VII, Paper 468

ACCURATE LINEAR AND NONLINEAR SEISMIC SSI ANALYSIS BASED ON ANSYS FE MODELING USING EXTENDED SASSI METHODOLOGY

Dan M. Ghiocel[1], Mike Saremi[2]

[1] Chief of Engineering and President, Ghiocel Predictive Technologies, Inc., New York, USA
[2] Principal Consultant and seismic Technical Lead, AMEC Foster Wheeler, Knutsford, UK

INTRODUCTION

Herein we describe a new capability for seismic SSI analysis based on integrating two FEA codes, ACS SASSI (2016) and ANSYS (2014). This capability permits to apply the SASSI methodology directly to refined ANSYS FE models. This ACS SASSI-ANSYS integration capability is called the ACS SASSI Option A-AA (ANSYS for A and Advanced ANSYS interfacing for AA). The main benefit is that the FE modelling of the NPP structures can be done only in ANSYS using a single refined FE model, without the need to create and validate a separate FE model in ACS SASSI for performing SSI analysis. This reduces the analyst effort tremendously. In many of the past projects, the seismic SSI analysis is performed on a simplified dynamic SASSI model and the stress analysis is performed on a more detailed FE model in either ANSYS, NASTRAN or other general purpose FE package. This in effect doubles the analyst's efforts as he would need to build and validate two different FE models with different refinement level and ensure that they are dynamically tuned. This is no longer the case now, since the recent ACS SASSI versions can run efficiently FE model sizes up to 300,000 nodes or 1.5 million dofs on a single MS Windows workstation platform with sufficient RAM. This capability enables the analyst to use the large ANSYS FE element library which includes refined element formulation such as SOLID185, SHELL181, BEAM188, PIPE188, COMBIN14, MPC184 Rigid Link and/or Rigid Beam, and even fluid elements, such as FLUID80 and super elements such as MATRIX50. The analyst can also use for ANSYS FE modelling couple nodes (CP command) and constraint equations (CE commands) as he wishes.

The ACS SASSI Options AA-A capability provides a refined two-step SSI approach in which the 1st step (Option AA) uses directly the linearized ANSYS structural model for performing the dynamic SSI analysis via SASSI methodology, and the 2nd step (Option A) automatically exports the time-varying SSI structural responses, nodal acceleration and/or displacements, as consistent set of input boundary conditions to the ANSYS FE model for subsequent stress analysis. The SSI responses can be exported to ANSYS for all time steps, or only critical time steps.

It should be noted that in the 2nd step (Option A), the analyst can include more refined FE types, such as the FLUID30 elements useful for including the seismic fluid-structure dynamic interaction effects, and eventually local nonlinear material and/or geometric aspects within the structure or at the foundation-soil interface by including contact surfaces to simulate the soil separation effects. To save time for the analyst, the ACS SASSI User-Interface (UI) is capable of automatically generating contact surfaces at the foundation-soil interface for the ANSYS model in the ADPL input file format, just ready to be used by the analyst.

The integrated Option AA-A SSI analysis capability is based on a "cascaded approach" which implies that there is no feedback effect due to the local structural and foundation nonlinearities on the SSI soil motions at the foundation-soil interface. This assumption appears to be rational for many practical applications, except for some particular situations for which nonlinear aspects are quite large. For such cases, the SSI dynamic analysis has to be performed by nonlinear time integration in ANSYS assuming that the foundation SSI response motions are applied below the contact surfaces at the foundation-soil interface.

Currently the integrated Option AA-A capability is applied in a number of international commercial in nuclear projects. In this paper, a brief overview of the Option AA-A capability and few useful case studies are included. The benefit in terms of the accuracy of the SSI analysis results is obvious as indicated by the case studies included.

BRIEF DESCRIPTION OF OPTION AA-A SSI METHODOLOGY

The integrated Option AA-A capability is described in Figure 1. It is shown the Option AA is for performing SSI analysis (1st step), while Option A is for performing stress analysis (2nd step).

The *Option AA capability* enables the use of an ANSYS structural FE model for SSI analysis directly, without the need for converting the structural model to ACS SASSI. The ANSYS structural stiffness, mass and damping matrices are directly used for SSI analysis. The SSI relative displacements, absolute accelerations and response spectra for the ANSYS structural model are fully computed within the ACS SASSI framework.

To extract the ANSYS model structural stiffness, mass and damping matrices, the user has to run in ANSYS, a simple ADPL macro called *gen_kmc.mac.* This ANSYS macro that is automatically installed by the ACS SASSI software and should be run with a parameter of 0 value for the structure FE model and 1 value for the excavated soil FE model, *gen_kmc,'.',0,'.'* and *gen_kmc,'.',1,'.'*, respectively. Then, the ANSYS model .cdb files are loaded into the ACS SASSI UI in two separate FE active models which are merged into a single SSI FE model. The entire UI command script for reading and merging the ANSYS FE models is simple, as described below:

** Read ANSYS Structure Struct.cdb in Model 1, and define structure elements of type 1 for structure*
Actm,1
Convert,ansys, Struct.cdb,32.2
Etypegen,1
** Read ANSYS Excavated Soil Soil.cdb in Model 2, and define soil elements of type 2 for excavated soil*
Actm,2
Convert, ansys, Soil.cdb,32.2
Etypegen,2
** Create SSI model by combining Models 1 and 2 in Model 3, and add information on ground surface and*
Actm,3
MergeSoil,1,2,1,,,,mappingfile.txt

In the above UI command sequence, the lines starting with * are comment lines. After creating the SSI model, the ground surface and the interaction nodes has to be defined by the user using GROUNELEV and INTGEN commands. After this, the ANSYS SSI model is ready to be saved and run as a standard ACS SASSI SSI model input.

The Option A capability is used to transfer the SSI response motions for all time steps or selected critical steps as input boundary conditions for the subsequent ANSYS quasi-static stress analysis. Option A is a “theoretically exact” solution for the linearized model stress analysis. However, in Option A step, some local nonlinear effects, such as nonlinear material and geometry aspects, including sliding and foundation-soil separation, can be included in the ANSYS stress model.

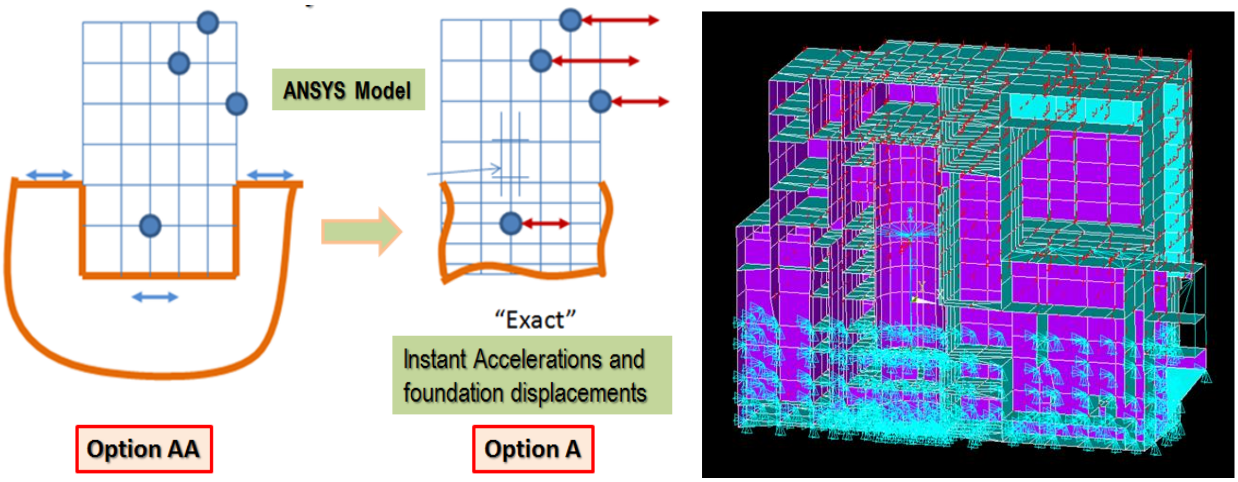

Figure 1 Integrated ACS SASSI-ANSYS SSI Analysis Using Option AA (1st Step) and Option A (2nd Step) Using ANSYS FE Structural Model (left plot)

Option A can also be used for evaluating the seismic pressures on foundation walls including the soil separation effects. This is achieved by performing the ANSYS nonlinear contact analysis for few selected critical time steps when the largest sliding forces and overturning moments occurred. This application of Option A is shown in Figure 2.

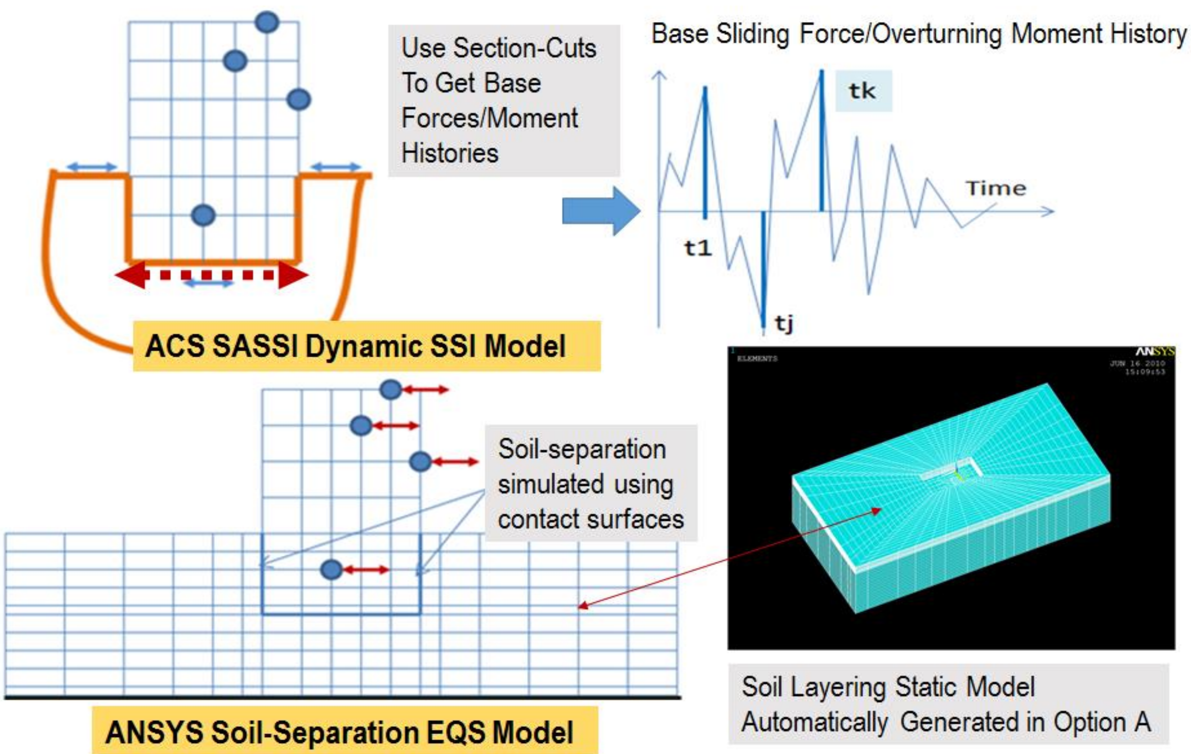

Figure 2 Application of Option A (2nd Step) for the ANSYS Nonlinear Analysis to Evaluate Foundation-Soil Separation Effects Including Surrounding Soil FE Model (right plot)

The Options AA and A are designed to be highly user-friendly, efficient and safe for the analyst. UI commands and simple window dialogs are used. As an illustration of the UI dialog simplicity, Figure 3 shows two captured screens of the ACS SASSI UI window dialogs for preparing i) the SSI analysis using the Option AA for an ANSYS model including a MATRIX50 super element called “sldbox”, and ii) the ANSYS stress analysis using the Option A by the SSI responses containing the nodal structure accelerations and the foundation displacements exporting as static load steps applied to the ANSYS model.

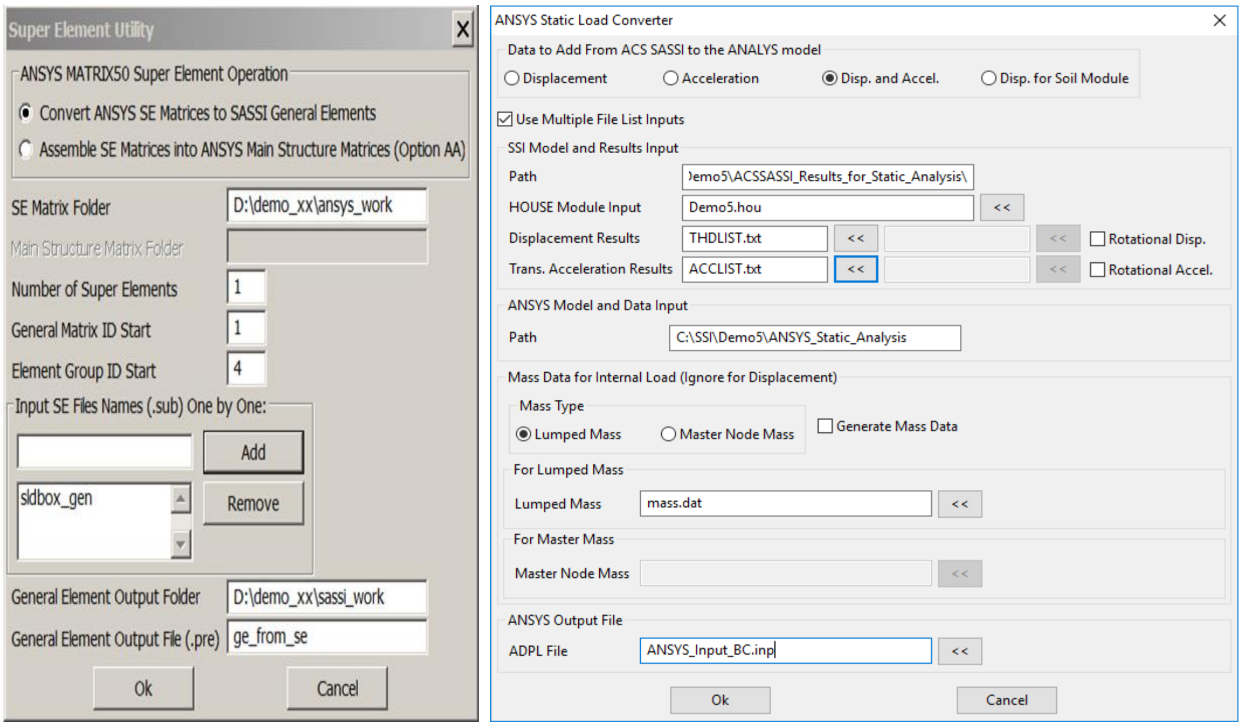

Figure 3 Option AA UI Input for SSI Using An ANSYS Model with Super Elements (left) and Option A UI Input for Stress Analysis Using Accelerations and Displacements as Input to ANSYS Model (right)

ILLUSTRATIVE CASE STUDIES

In this section a number of selected case studies are shown. The case studies illustrate the application of both Option AA and Option A capabilities.

The Option AA-A SSI analysis case studies include: i) Option AA for the evaluation of the water sloshing in a 80ft x 50ft size concrete pool using the ANSYS FLUID80 element to model the water fluid, ii) Option AA-A for the comparison of the SSI responses of the R/B complex using the ANSYS model vs. the standard ACS SASSI model, and iii) Option A for the evaluation of the seismic soil pressures for a deeply embedded pool structure including soil-foundation separation effects.

The water pool case study is described in Figure 4. The concrete pool shell model was filled with water using the ACS SASSI UI FILLPOOL command to fill the pool with solid-type elements that are converted in ANSYS into the FLUID80 elements (upper left plot). The ANSYS coupled fluid-structure model matrices were extracted from the ANSYS water-pool model using the ADPL *gen_kmc.mac* macro and the model .cdb was input in ACS SASSI UI for extracting required model topology information.

Then, running ACS SASSI in the Option AA mode, the water-pool SSI acceleration response was obtained as shown in the Figure 4 right plots. The upper plot shows the water fluid low-frequency large sloshing motion at the surface center, while the lower plot shows the high-frequency wall vibration.

The overall motion of the water fluid in the pool is shown in the lower right plot. It should be noted that the water and the pool meshes should have different nodes, being connected only by contact spring elements normal to the wall or using coupled nodes.

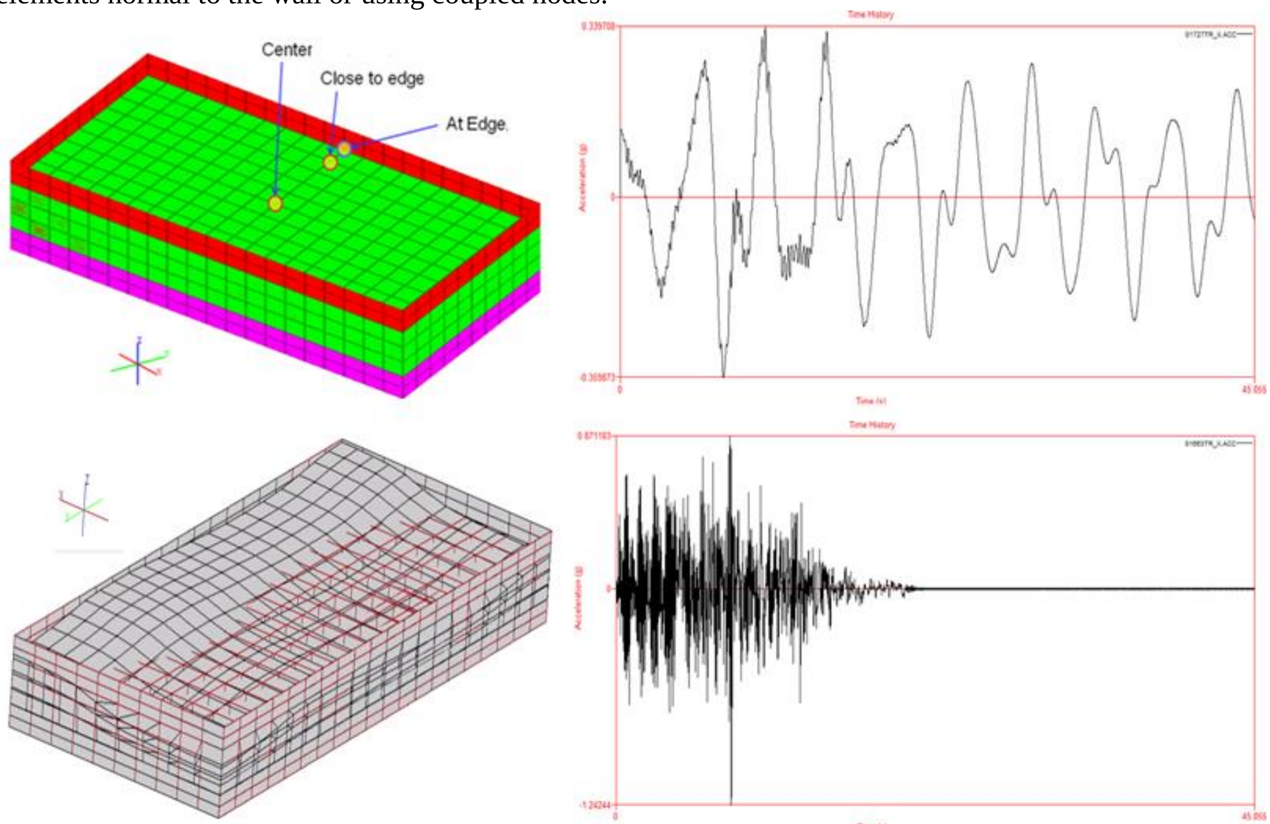

Figure 4 Water Concrete Pool Option AA Case Study Results

The R/B complex Option AA-A SSI case study is shown in Figures 5 through 9. The SSI analysis inputs were defined by the RG1.60 seismic spectrum anchored at 0.30g and stiff soil conditions. For the SSI analysis both the standard ACS SASSI model and the ANSYS model were considered and their results were compared. The ANSYS model included the state-of-the-art BEAM188, SOLID185 and SHELL181 elements, while standard ACS SASSI model included the basic SOLID and SHELL elements that correspond to the ANSYS BEAM4/44, SOLID45 and SHELL63 elements that do not include the element shear flexibility effects which could be significant for the thick walls and floors of the nuclear building. Since most of the current FE models of nuclear structure are shell models, the use of the refined SHELL181 elements is relevant. It should be noted that the ANSYS SHELL181 is a highly refined shell finite element produce by the MIT research work applicable to both thin and thick shells. The ANSYS SHELL181 element outperforms in terms of accuracy and spurious mode stability other thick shell elements implemented in other FE codes.

Figure 5 shows the locations considered at the top of Internal Structure for computing the in-structure response spectra (ISRS) and the wall forces and moments.

Figure 6 shows a comparison of the ISRS computed for the standard ACS SASSI model vs. the ANSYS model at Node 69135 that is located at the top floor of the Internal Structure as indicated in Figure 3. The differences between the ISRS are negligible for the X-direction (longitudinal for R/B) and up to 10-15% for Y-direction (transverse for R/B). This remark is relatively valid for many locations.

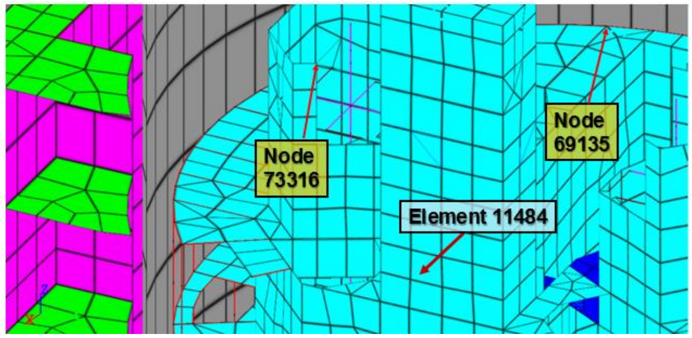

Figure 5 Selected Locations for SSI Responses of Internal Structure (IS)

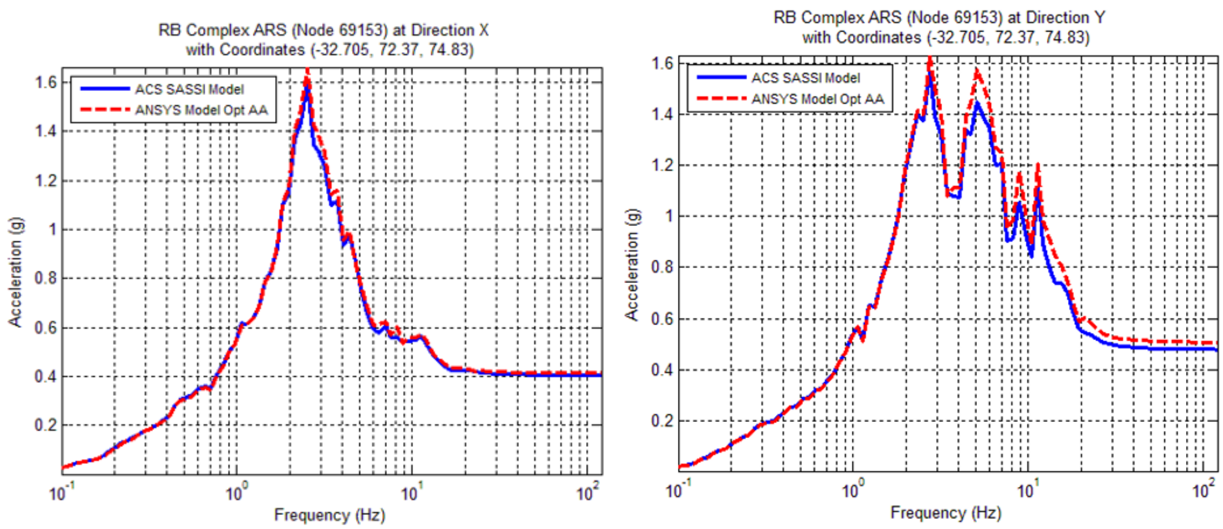

Figure 6 Comparative ACS SASSI (red) vs. ANSYS (blue) ISRS at Node 69153 at Top Floor of IS; Horizontal (left) and Vertical (right)

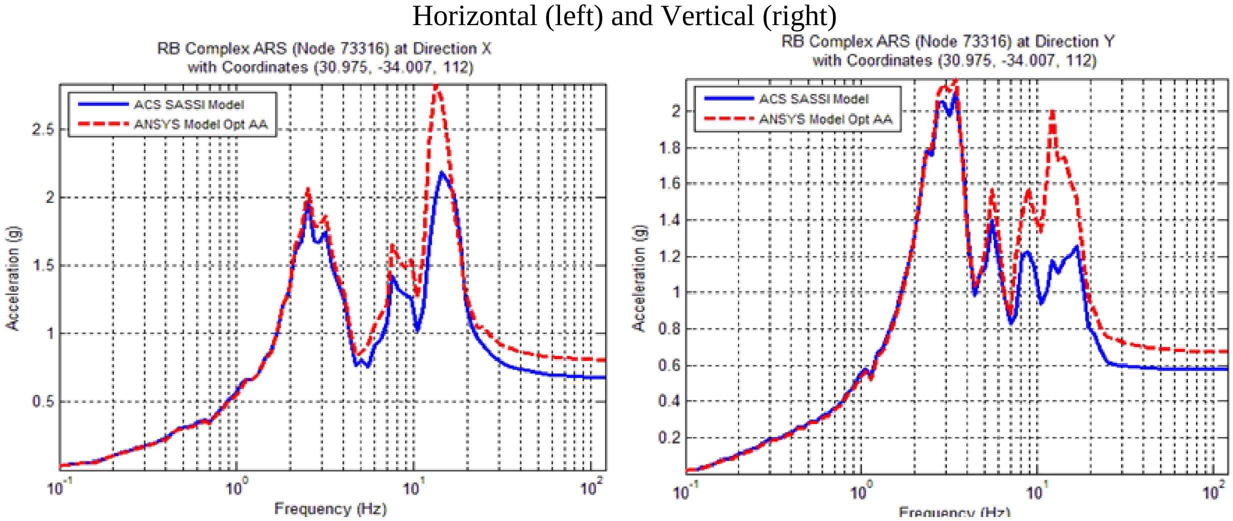

Figure 7 Comparative ACS SASSI (Red) vs. ANSYS (Blue) ISRS at Node 73316 at Free Top Wall of IS; Horizontal (left) and Vertical (right)

Figure 7 shows a comparison of the ISRS computed for the standard ACS SASSI model vs. the ANSYS model at Node 73316 that is located at the free top of the Internal Structure wall as indicated in Figure 3. The ISRS differences are quite large for the 8-18 Hz frequency range. This large difference is due the different SHELL element types in the two FE models. It should be noted that always the ANSYS model ISRS responses have significant larger spectral peaks in the higher frequencies than the ACS SASSI model ISRS for shell elements that have free edges or are close to large openings in the structure walls. These results illustrate the benefit of using the refined ANSYS SHELL181 for the FE modelling.

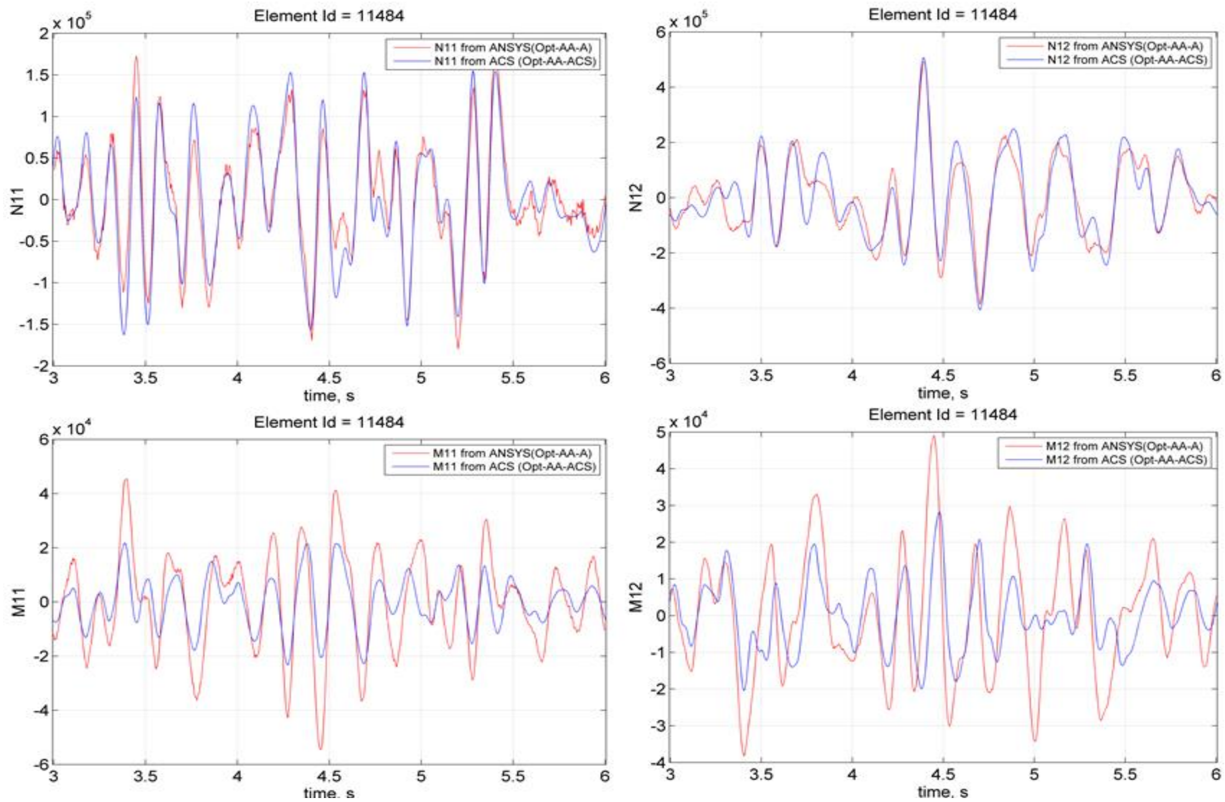

Figure 8 Comparative Element In-plane Forces (upper) and Out-of-plane Moments in R/B Complex Internal Structure Wall between Standard ACS SASSI Model (Blue) and ANSYS Model (Red)

Figure 8 compares the in-plane (membrane) axial and shear forces, N11 and N12, and the out-of-plane bending and torsional moments per unit length, M11 and M12, for the Internal Structure wall shell element as indicated in Figure 3. The differences between the ANSYS and the standard ACS SASSI element in-plane forces is minimal, while the differences between the out-of-plane moments, bending and torsional moments, are quite significant. Moments computed using ANSYS SHELL181 are larger than moments computed using ACS SASSI.

Figure 9 shows a comparison between the in-plane axial stress SYY and the out-of-plane torsional moment per unit length MXX for a floor of the building (at the center of the floor). The comparative plots show the same trend as in Figures 8 and 9 indicating good matching for the in-plane forces and not good matching for the out-of-plane moments, this time the ANSYS SHELL181 results being only slightly larger than the ACS SASSI results.

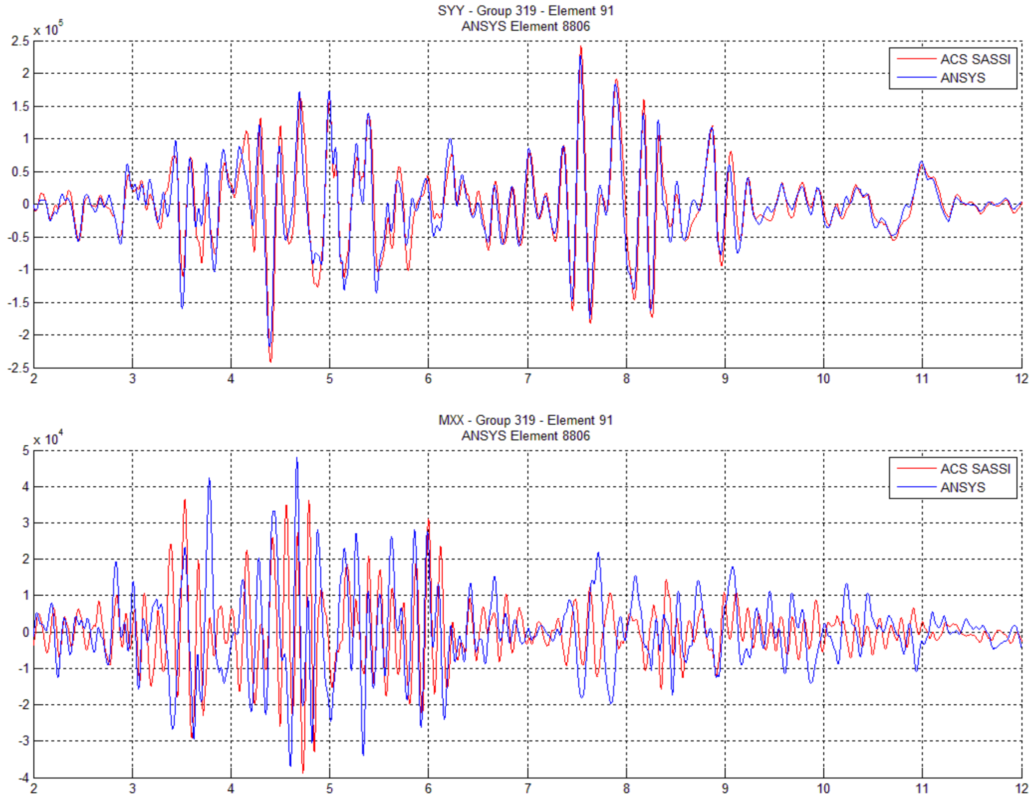

Figure 9 Comparative In-Plane Axial Stress and Out-of-Plane Bending Moment for A Floor Center Shell Element for Standard ACS SASSI Model (Red) and ANSYS Model (Blue)

Occasionally, the out-of-plane moments can be also lower in ANSYS than in ACS SASSI, depending on the structure local wall interactions and load transfer between walls and slabs. Around openings and close to free edges the discrepancies between ANSYS Option AA-A and standard ACS SASSI SSI analysis results are larger.

The third case study addresses using Option A the effects of soil-foundation separation on the seismic soil pressure for a deeply embedded box type concrete structure as shown in Figure 10. The sizes of the structure is 80 ft x 50 ft in the horizontal plane. The embedment is 30 ft. The soil Vs was considered 1000 fps. Seismic input was defined by the El Centro NS acceleration component scaled to 1g and applied in the transverse direction of the pool structure.

The envelope of the seismic soil pressures computed in the ACS SASSI adjacent soil elements along the longitudinal wall are shown in the Figure 10 upper left plot. No soil separation is included. The maximum soil pressure goes up to about 11 ksf. Using Option A for a selected critical time step when the overturning moment reaches its maximum, all the SSI response nodal forces on the structure were exported automatically as equivalent-static seismic loads to the ANSYS nonlinear model including the surrounding soil layering with contact-surfaces at the foundation-soil interface (automatically generated by the UI SOILEMESH command). The results of the ANSYS nonlinear contact analysis are shown in the Figure 10 upper right plot

In the same plot the contour stresses are shown for the normal stresses perpendicular to the embedded wall surface or the seismic soil pressures. The contour plot of the soil pressures indicates that the soil pressures on the leeward face are zero due to the soil separation from the embedded wall. Also, on the compression side of the foundation wall the soil pressures go up to about 25-30 ksf (the local normal stress values at the wall corners are too high since the soil nonlinear material behaviour was not considered here).

The structure behaviour including the soil separation effects is illustrated in the lower left plot. For the severe 1g seismic motion, the soil-separation occurs for both the side walls and the baseslab. The visible large loss of contact with the surrounding soil suggest that there is a large increase of the soil pressures on the compression side of the foundation wall.

The soil pressure results are provided in the lower right plot that compares the absolute values of the soil pressures for the assumptions of “no soil separation” (blue) and “with soil separation” (red), respectively. The line plots are the soil pressures on the FE model element faces from bottom to top of the embedment part. The element numbering on the horizontal axis of the plot is from the bottom to the top of structure (the first 120 elements are in the baseslab, while the last 40 elements are at the top of structure). The two maximum soil pressure values for the “no soil separation” case correspond to the mid-span locations along the two longitudinal walls on the top of these walls.

For “no soil separation” (blue line), these two peaks correspond to the absolute values of maximum tension and maximum compression, respectively that are basically identical. For the “with soil separation” case (red line) the first soil pressure peak that corresponds to the maximum tension at the wall mid-span for the “no soil separation” case goes down to the zero value for “with soil separation” case and the second pressure peak that corresponds to the maximum compression at the wall mid-span for the “no soil separation” moves at the wall two top corners and increases its amplitude to about 25-30 ksf.

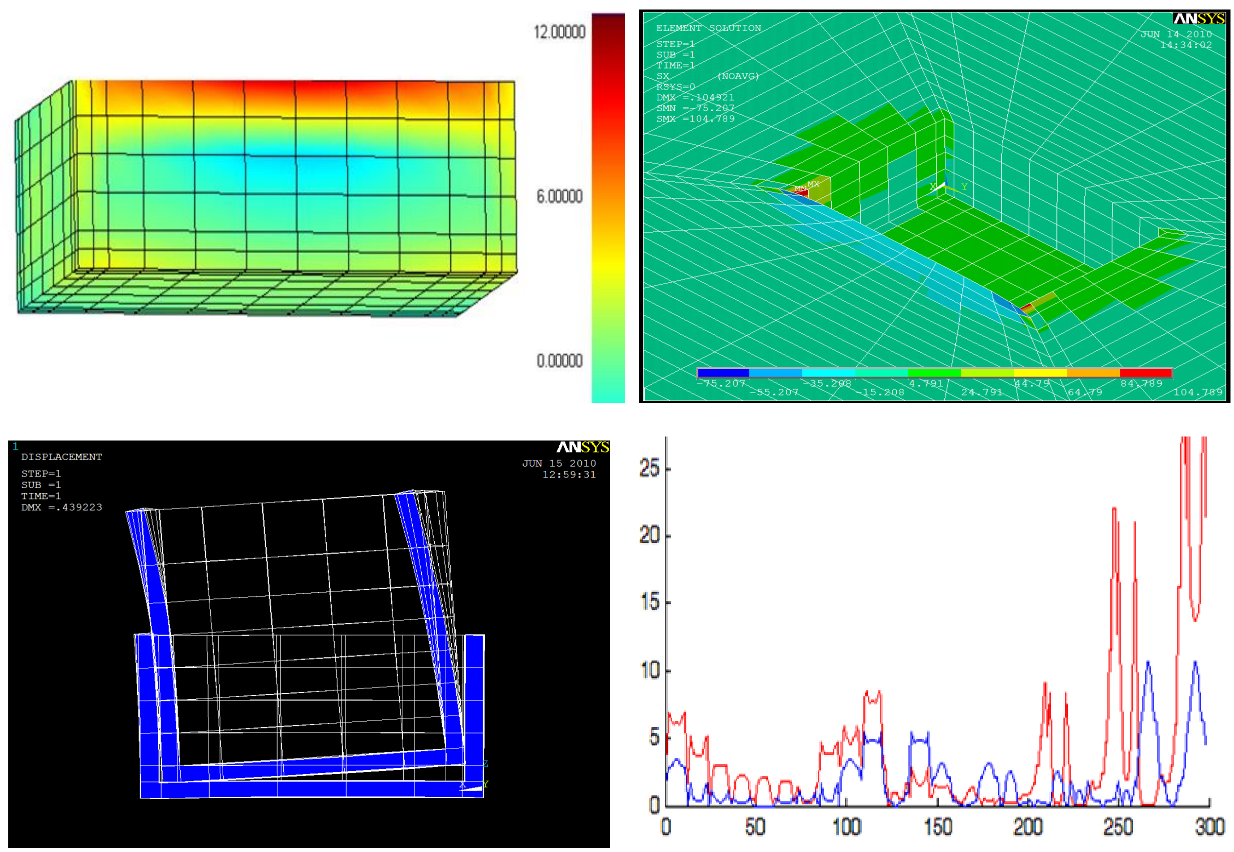

Figure 10 Using Option A to Evaluate Foundation-Soil Separation Effects on Soil Pressures

It should be noted that this large amplification of about 2.5-3.0 times of the maximum soil pressure computed using the Option A ANSYS contact analysis is due to the two main influential factors which are the foundation size-embedment depth ratio (foundation width is 50 ft and embedment is 30 ft) and the severe seismic forces on the structure for the 1g maximum ground acceleration seismic input. For large-size foundations with shallow embedments, or moderate seismic inputs, the soil pressure increases due to the soil separation effects are expected to be more modest than those shown in Figure 10.

CONCLUSIONS

The paper introduces several new capabilities for seismic SSI analysis based on the ACS SASSI-ANSYS software integration within the ACS SASSI Options A-AA capability. The FE modelling can be made directly in ANSYS using a sufficiently refined FE model without the need to create and validate separate FE models for the SSI and the stress analysis, as it was frequently done in the past. .

The paper shows that the use of the ANSYS refined finite elements within the ACS SASSI Options A-AA analysis could improve significantly the accuracy of the SSI analysis results. For the computed ISRS, the differences between the ANSYS model results and standard ACS SASSI model results are not significant, except for few locations close to large openings and free-edges. For structural forces and moments, the differences are usually larger for the out-of-plane moments than the in-plane forces, most likely with the ANSYS results being larger than the ACS SASSI results.

Further the ACS SASSI Option A capability can be used for performing efficient ANSYS nonlinear contact analysis to include nonlinear material and geometric aspects, such as soil-separation effects. It is shown that for the investigated deeply embedded structure, the soil separation effects affected severely the seismic soil pressures on the foundation walls.

REFERENCES

American Society of Civil Engineers (2005) “Seismic Design Criteria for Structures, Systems, and Components in Nuclear Facilities”, ASCE 43-05 Standard.

American Society of Civil Engineers (2017), “Seismic Analysis for Safety-related Nuclear Structures and Commentary” ASCE 4-16 Standard

ANSYS Inc. (2016) ANSYS Version 15, Mechanical Version, Pittsburgh, PA

Ghiocel D.,M. and Saremi, M. (2017) “Automatic Computation of Strain-Dependent Concrete Cracking Pattern for Nuclear Structures for Site-Specific Applications”, SMiRT22 Conference Proceedings, Division V, Busan, Korea, August 20-25

Ghiocel Predictive Technologies, Inc. (2016). “ACS SASSI - An Advanced Computational Software for 3D Dynamic Analyses Including SSI Effects”, ACS SASSI Version 3 User Manuals, December, Rochester, NY, USA

Ghiocel D.,M. (2015) “Fast Nonlinear Seismic SSI Analysis Using A Hybrid Time-Complex Frequency Approach For Low-Rise Nuclear Concrete Shearwall Buildings”, SMiRT22 Conference Proceedings, Division V, Manchester, UK, August 10-14