Transactions, **SMiRT-26**
Berlin/Potsdam, Germany, July 10-15, 2022
Division III

QUASI ONE-STEP SOIL STRUCTURE INTERACTION SEISMIC ANALYSIS FOR UK HPR1000

Zhao Ming[1], Ming Tan[2], Zhang Tao[3], Stewart Gallocher[4], Tong Congqing[5]

[1] Senior Engineer, State Key Laboratory of Nuclear Power Safety Monitoring Technology and Equipment, China Nuclear Power Engineering Co. Ltd, China (zhao_ming@cgnpc.com.cn)
[2] Technical Principal, Mott Macdonald, UK (ming.tan@mottmac.com)
[3] Principal Engineer, State Key Laboratory of Nuclear Power Safety Monitoring Technology and Equipment, China Nuclear Power Engineering Co. Ltd, China (zt_csn@cgnpc.com.cn)
[4] Consultant, Mott Macdonald, UK (stewart.gallocher@steelbricks.com)
[5] Senior Engineer, State Key Laboratory of Nuclear Power Safety Monitoring Technology and Equipment, China Nuclear Power Engineering Co. Ltd, China (tongcongqing @cgnpc.com.cn)

ABSTRACT

Using the work undertaken for the Generic Design Assessment (GDA) in the UK for UK HPR1000 [GNSL, 2020], this paper provided an overview of a pragmatic quasi one-step soil structure interaction seismic analysis approach using ANSYS and ACS-SASSI. The approach presented here removed many limitations of the classical multistep approach, but is considerably less time and process demanding than generally more sophisticated one-step approach which can be impractical to do so in a large-scale nuclear project.

Unlike most other “two-step or multi-step” method, the analytical seismic models for the first and second steps are identical and that the analytical results are transferred directly on a node by node basis. Seismic demand analysed in ACS-SASSI are transferred directly into ANSYS using a mapped displacement approach in time domain on a node by node basis.

INTRODUCTION

In the design of nuclear power plants, seismic demand can often be one of the governing load cases and one of the most challenging to correctly analyse and predict the responses of the structures. Traditionally for a large-scale nuclear power plant, the seismic analysis is often performed using a “multi-step or a two-step method” of which a dynamic response analysis is performed as a first step to establish system-level responses. Calculations of demands on components of the structures are performed in subsequent steps.

As noted in ASCE/SEI 4-16 [ASCE, 2017], one of the limitations of the multistep method is that the subsequent analysis, static or dynamic, will generally use a mathematical model with a fixed base, which will not enable the calculation of seismic pressures acting on the underside of the raft. Another major limitation is that it is often frequency based and the time domain information is lost, and this could be significant from design saving perspective [Tan, 2017]. Alternatively, adopting a generally more sophisticated one-step approach is often overly time demanding and impractical to do so in a large-scale nuclear project.

Using the work undertaken for the Generic Design Assessment (GDA) in the UK for UK HPR1000 [GNSL, 2020], this paper focus on proposing an alternative approach, attempting to combine the benefits of both approaches, which is based on a quasi one-step method.

UK HPR1000

The UK HPR1000 is a Pressurised Water Reactor using the Chinese Hualong technology with electric output of approximately 1180MW. The UK HPR1000 has evolved from a sequence of reactors that have been constructed and operated in China since the late 80s, including the M310 design used at Daya Bay and Ling'ao (Units 1 and 2), the CPR1000, the CPR1000+ and the more recent ACPR1000. The first two units of CGN's HPR1000, Fangchenggang NPP Units 3 and 4, are under construction in China. Fangchenggang NPP Unit 3 is the reference plant for the UK HPR1000.

With the intention to be deployed to the Bradwell 'B' site in the UK, the UK HPR1000 was put forward for GDA in January 2017, to be assessed jointly by the regulators - Office for Nuclear Regulation (ONR) and the Environment Agency. The regulators provided independent scrutiny to ensure that the reactor design is applicable to UK regulatory standards of safety, security and environmental protection. The GDA for the UK HPR1000 was successfully completed in February 2022, with the issuing of a Design Acceptance Confirmation (DAC) from the ONR and a Statement of Design Acceptability (SoDA) from the Environment Agency [ONR, 2022].

OVERALL SEISMIC ANALYSIS APPROACH

The seismic analysis methodology for a typical Safety Class 1 structure within the UK HPR1000 GDA design is based upon a quasi one-step method, in which the SSI analysis is undertaken in the ACS SASSI environment and the seismic structural stresses are recovered through ANSYS program.

Unlike most other "two-step or multi-step" method, the analytical seismic models for the first and second steps are identical and that the analytical results are transferred directly on a node by node basis. Hence, this is considered a "quasi one-step" and not a "two-step" method. Figure below shows an overall seismic analysis approach adopted for the UK HPR1000. A summary of each individual phase for the overall seismic approach is provided as follows.

STEP 0 – MODEL CONSTRUCTION PHASE

Three-dimensional Finite Element (FE) Model using ANSYS was constructed with details of coarse mesh (around 1.5m grid) model for seismic (termed as ANSYS Model 1), fine mesh (around 0.75m grid) model for static (termed as ANSYS Model 2). The seismic coarse model contains the same amount of details as per the static fine model, which includes the raft, all primary walls and slabs, containment, equipment, etc.

For the SSI analysis in the ACS SASSI environment, the SASSI Model was transferred from the ANSYS Model 1 directly using the built-in interfacing tool developed by ACS SASSI. The SASSI Model is geometrically identical to the ANSYS Model 1.

The development of analysis models taking into account the requirements as per ASCE/SEI 4-16 Chapter 3 requires careful considerations. For example:

- Selection of Finite Element Type and Mesh Size
- Structural Material Properties
- Modelling of Stiffness
- Modelling of mass

- Modelling of hydrodynamic effects

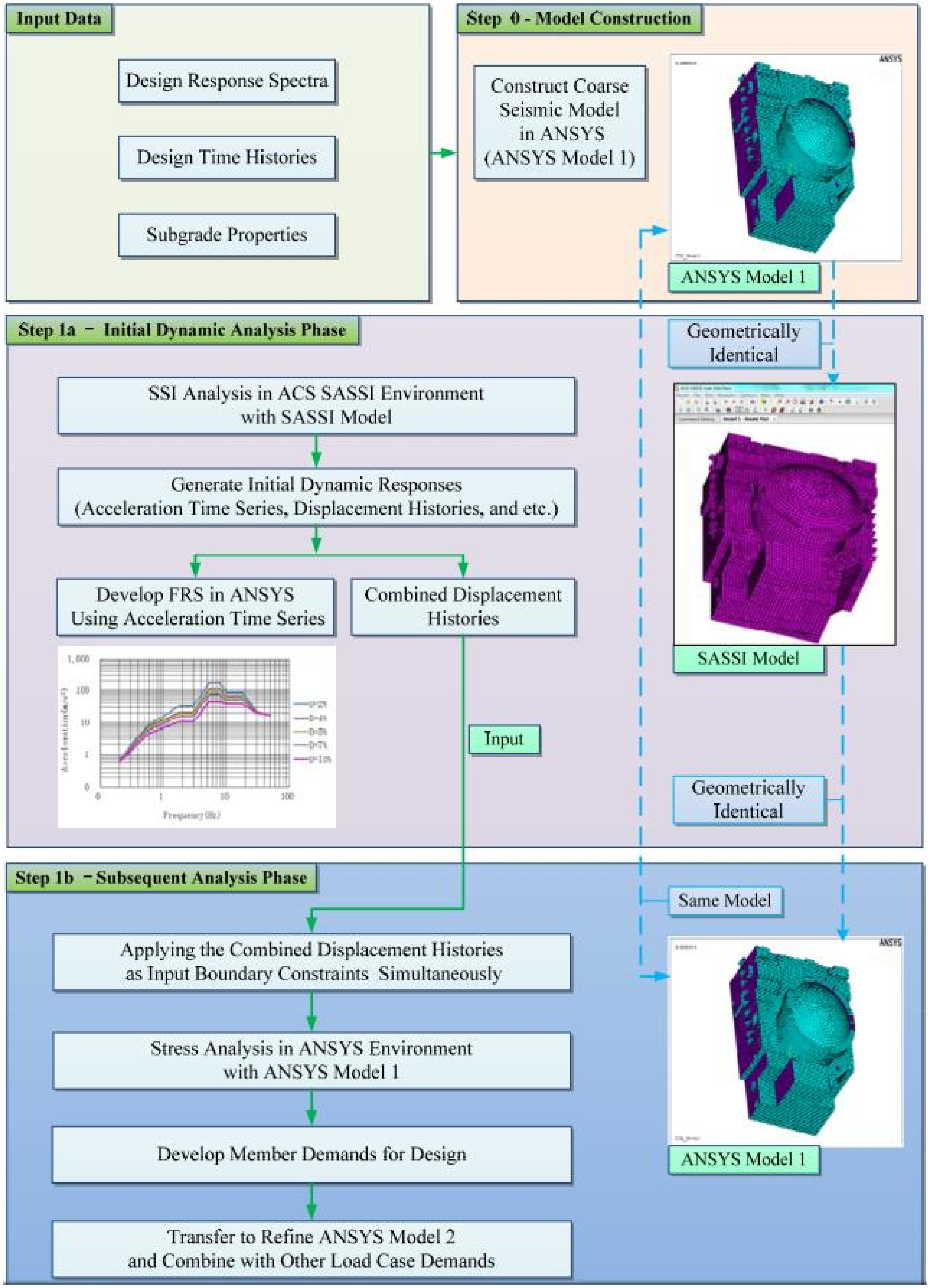

Figure 1. Overall Seismic Analysis Approach

STEP 1a – INITIAL DYNAMIC ANALYSIS PHASE

Following the model construction phase, the initial dynamic analysis step of which the SSI analyses are performed. This was performed using the computer program ACS SASSI with the SASSI Model.

As mandated by Chapter 5 of ASCE/SEI 4-16 [ASCE, 2017], the seismic analysis of safety class 1 structures is performed considering the effects of SSI with supporting subgrade, which is captured in the initial dynamic analysis phase. This is performed using the computer program ACS SASSI with the SASSI Model. Soil elements and parameters were added to the SASSI Model in the ACS SASSI environment.

The SSI accounts for the effects of coupling between a structure and its supporting medium during an earthquake. If the structure is light and founded on hard rock, the motion at the structure foundation level is approximately the same as the ground motion in the free field. Conversely, if the structure is massive or founded on soft soil, the foundation motion may differ significantly from that in the free field. Hence, the SSI effects are considered for the seismic analysis of structures.

Three-dimensional SASSI models are adopted to perform the SSI analysis, capturing the three-dimensional dynamic response of the structure during seismic excitation. It is based on substructuring method that partitions the SSI model into free field soil and structure plus foundation minus the excavated soil. The site model of the free field soil is represented by horizontally infinite layers that can capture the frequency dependent response of the site. The SASSI Model calculates the overall seismic response of structures, including floor acceleration time series and displacement histories for different generic subgrade conditions.

ACS SASSI calculates the response of the SSI system by solving the equations of motion in frequency domain for selected set of frequencies. The solution is then interpolated for the range of frequencies of interest. Fast Fourier Transformation and inverse Fast Fourier Transformation technique are used to transform the input motion and the nodal responses of the system between frequency and time domains.

The flexibility of raft foundations of the structure is considered in the SASSI analysis when developing the overall SSI seismic responses. The SSI analysis considers two orthogonal horizontal and one vertical earthquake components. The SSI analyses for each of the three directional earthquake components are performed separately.

Three-dimensional finite element model using ANSYS will be constructed with details of coarse mesh (around 1.5m grid) model for seismic (termed as ANSYS Model 1), fine mesh (around 0.75m grid) model for static (termed as ANSYS Model 2). The seismic coarse model will contain the same amount of details as per the static fine model, which includes the raft, all primary walls and slabs, containment, equipment, etc.

For the design of SSE1 structures, a set of subgrade profiles of various shear wave velocities are selected for use in the SSI analysis. This is agreed with the Regulator beforehand to ensure a reasonable representation of the candidate sites. The GDA considers three baseline generic site conditions, Super Soft (150m/s), Soft (500m/s) and Medium (1100m/s), which are representative of the site conditions as per the generic candidate sites.

There are a number of uncertainties and possible variations in the determination of SSI responses. It is important that the uncertainties are well understood and have been carefully considered. The uncertainties include:

- Variation of input motion

- Damping level for FRS generations
- Concrete cracking effect
- Embedment effect
- Structure-soil-structure interaction
- Uncertainties in subgrade properties
- Effect of groundwater
- Effect of incoherence
- Effect of long period ground motion

A series of SSI analysis cases were analysed either as the principal SSI analysis cases, or as sensitivity cases to account for various uncertainties.

In addition, the acceleration time series from the SASSI Model were also used to generate FRS for all subgrade and analysis case. The FRS is generated by enveloping the response spectra of the reference locations in the floor. The reference locations are selected such that the FRS can represent the dynamic behaviour of the structures appropriately. The response spectra are calculated based on the acceleration time series which are obtained from the SSI analysis.

For all soil conditions, the acceleration time series are calculated using the complex frequency-domain analysis method with computer program ACS SASSI. The two horizontal and one vertical time histories are applied individually, and the resultant time series are obtained by the algebraic summation of the co-directional time series from the three individual analyses in accordance with Section 6.2.1.1(b) of ASCE/SEI 4-16 [ASCE, 2017]. The resulting time series are used to compute the corresponding response spectra. To account the uncertainties in the dynamic behaviour of the soil and structures, the FRS are broadened by the approach of peak broadening. The broadening is ±15% at each peak frequency in the amplified region.

Combination of spatial components are undertaken using algebraic sum. For example, x components from X-direction, Y-direction and Z-direction SSI analyses were summated algebraically for each time step.

STEP 1b – SUBSEQUENT ANALYSIS PHASE

Subsequent analysis has been undertaken in the ANSYS environment with the ANSYS Model 1 to obtain the seismic member demands. Nodal displacement histories were extracted from the Step 1a work and applied as input boundary constraints in the time domain to the seismic model (ANSYS Model 1). Before applying to the ANSYS Model 1, co-directional responses to each of the three earthquake components were first combined using algebraic sums at each time step according to ASCE/SEI 4-16 Section 4.2.2 (c) pt.2[ASCE, 2017]. Again, as the SASSI model is geometrically identical to the ANSYS Model 1, this was undertaken seamlessly. This process was undertaken for different generic subgrade conditions.

The main purpose of this phase of work is to generate seismic member demands based on the initial SSI analysis work and to include any seismic effect that has not been captured in the initial SSI work. The output from this step is the enveloped seismic demand for various baseline SSI analysis cases to be transferred into the static model (ANSYS Model 2) to combine with the rest of the static combinations. The flowchart summary for the data transfer is shown in the figure below:

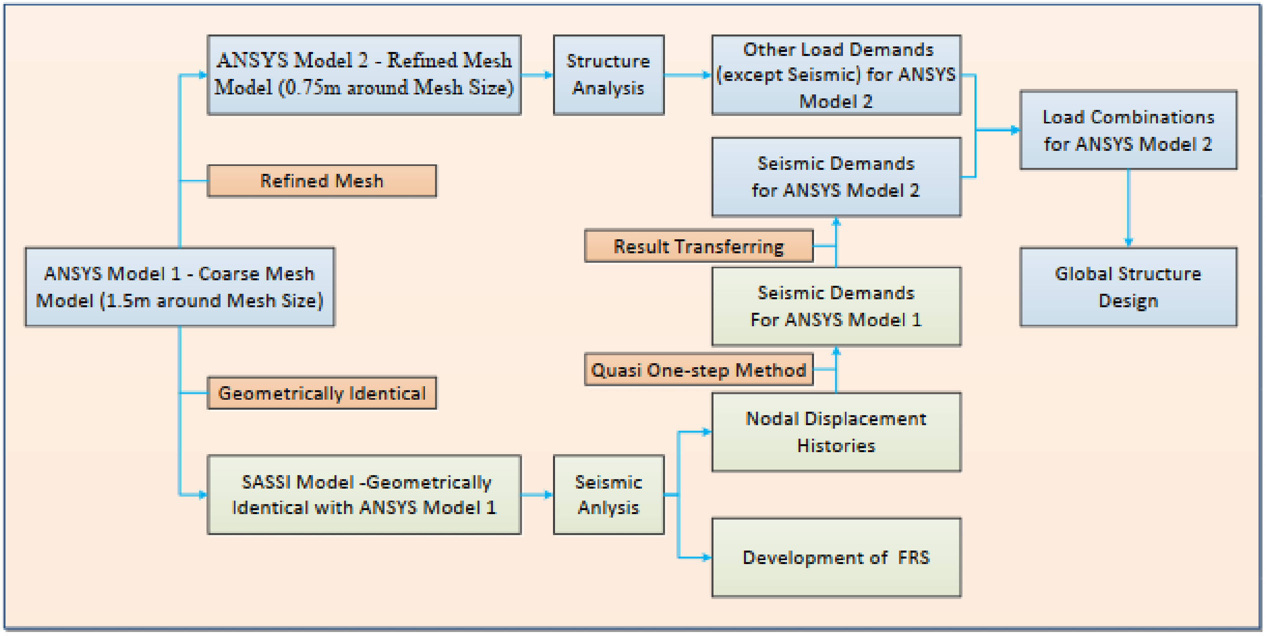

Figure 2. Data transfer between various models

This phase of analysis was carried out in the ANSYS environment with ANSYS Model 1. Nodal displacements histories for each of the three directional earthquake components will be extracted from ACS SASSI and co-directional responses to each of the three earthquake components are combined using algebraic sums at each time step in the time domain according to ASCE/SEI 4-16 Section 4.2.2 (c) pt.2, [ASCE, 2017]. The combined nodal displacement histories for all degrees of freedom are applying simultaneously as input boundary constraints in the time domain to the seismic model (ANSYS Model 1) to perform the subsequent analysis time step by time step and the maximum response will be recorded for design. As the SASSI Model is geometrically identical to the ANSYS Model 1, this will be processed seamlessly. This process will be done for different generic subgrade conditions. The flowchart for the subsequent analysis is shown in figure 3.

A number of additional seismic effects have not been captured in the initial dynamic analysis phase, including

- Effect of accidental torsion
- Dynamic soil pressure
- Hydrodyamic effects

These are added in this phase and are briefly described below.

The effect of accidental torsion is calculated at each floor level by static analysis assuming a torsional moment equal to the product of the story shear and 5% of the plan dimension perpendicular to the direction of motion of the structure at that level, according to Section 3.1 (i) of ASCE/SEI 4-16, [ASCE, 2017].

The story shear at each time step were obtained from the subsequent analysis and the torsional moment is applied to the structure on at each time step, and the response added to the main seismic analysis SSI output.

In the GDA phase, the dynamic soil pressure has simply been calculated in accordance with the simplified method described in Section 8.2.2 of ASCE/SEI 4-16, [ASCE, 2017]. The corresponding resultant force and overturning moment will be obtained and applied to the corresponding structural components.

The supporting walls, floors and roofs of ponds and tanks are designed to withstand the hydrodynamic loads generated by horizontal and vertical accelerations. The approach for UK HPR1000 to capture this demand accounts for the requirements of ASCE/SEI 4-16 Section 3.1(f) [ASCE, 2017]. The impulsive component has been applied as hydrodynamic mass to the walls and floors nodes up to level of the water surface in the finite element model. Equivalent masses will be distributed across the surface of the walls and floors and ensure that the location of the centre of mass is retained. Considering that the convective portion is likely insignificant and that it will be out-of-phase of the impulsive portion, the effect has therefore not be explicitly captured. In addition, for the GDA, the additional hydrodynamic forces are applied using equivalent lateral forces as per ACI 350.3-06 [ACI, 2016], which is based upon Housner's original work.

Once the seismic effect that has not been captured in the initial SSI work been included, the enveloped seismic demand for various SSI analysis cases were then transferred into the static model (ANSYS Model 2) to combine with the rest of the static combinations for design.

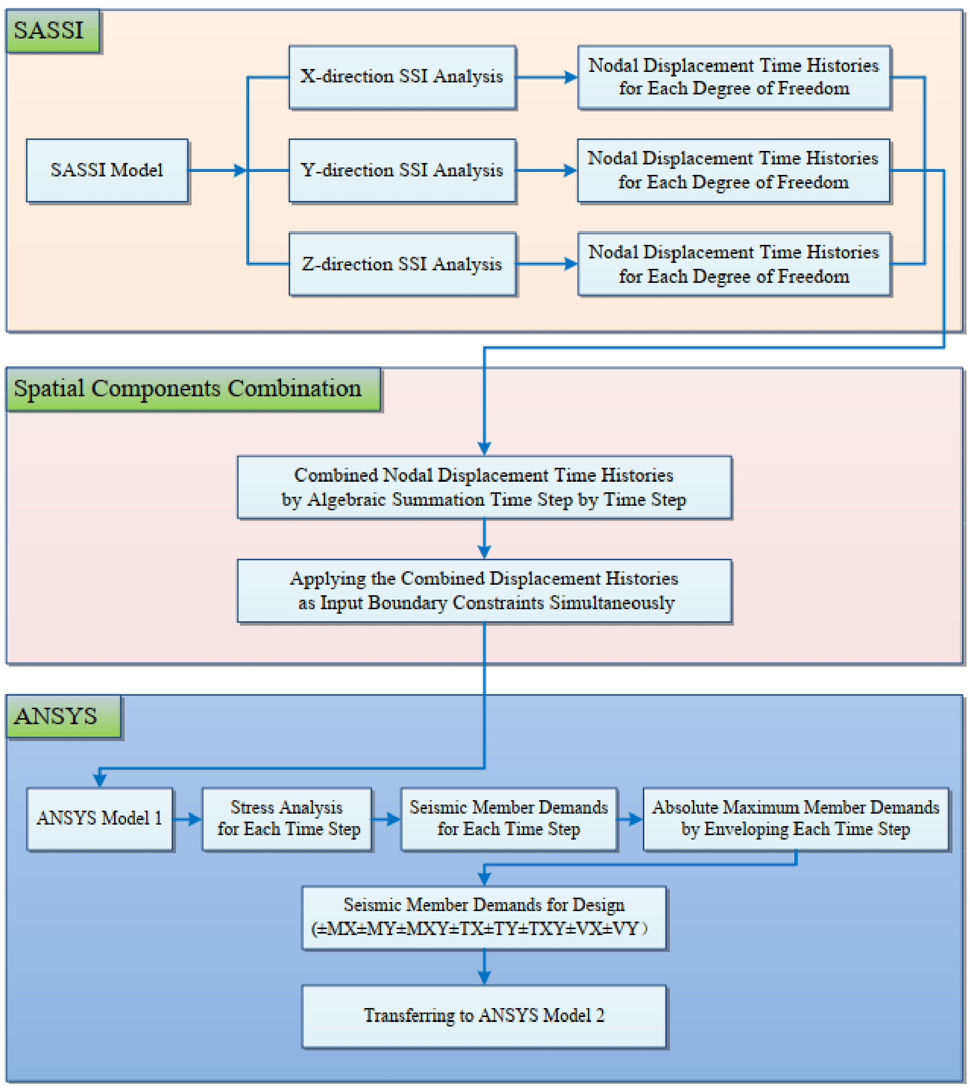

Figure 3. Subsequent analysis

CONCLUSION

Using the work undertaken for the Generic Design Assessment (GDA) in the UK for UK HPR1000 [GNSL, 2020], this paper provided an overview of a pragmatic quasi one-step soil structure interaction seismic analysis approach using ANSYS and ACS-SASSI. The approach presented here removed many limitations of the classical multistep approach, but is considerably less time and process demanding than generally more sophisticated one-step approach which can be impractical to do so in a large-scale nuclear project.

REFERENCES

GNSL (2020), Pre-Construction Safety Report, HPR/GDA/PCSR/0016, Rev 001.

Tan M, Hadfield J, Gumley R and Nitta Y (2017). "Design Optimisation of Stack Structure Using Time Domain Seismic Analysis" *SMiRT-24, Busan, South Korea.*

ASCE/SEI 4-16 (2017) (2017), ASCE/SEI4-16 Seismic Analysis of Safety-Related Nuclear Structures, American Society of Civil Engineering.

ONR (2022), https://www.onr.org.uk/new-reactors/uk-hpr1000/dac-soda.htm

ACI 350.03-06 (2016), Seismic Design of Liquid-Containing Concrete Structures and Commentary, American Concrete Institute.