Transactions, SMiRT-23
Manchester, United Kingdom - August 10-14, 2015
Division V, Paper ID 703

SOIL-STRUCTURE INTERACTION UNDER STATIC LOADS ACCOUNTING FOR THE EFFECT OF ADJACENT BUILDINGS: DEFINITION AND USE OF A FLEXIBILITY MATRIX FOR THE SOIL MODELLING

Pierre-Louis Regazzoni[1], Sébastien Reynaud[2], Van-Ngan Nguyen[3], Dao Pham-Thi-Anh[4], Silvano Erlicher[5], Dominique Allagnat[6], Roberto Pinto-Ward[7]

[1] PhD, Engineer, Egis Structures Environnement – Géotechnique, France.
[2] Senior Engineer, Egis Structures Environnement – Géotechnique, France.
[3] Engineer, Egis Industries, France.
[4] Senior Engineer, Egis Industries, France.
[5] PhD, Technical Director, Egis Industries, France.
[6] Technical Director, Egis Structures Environnement – Géotechnique, France.
[7] Engineer, EDF, France.

ABSTRACT

During the design phase of nuclear buildings, the soil-structure interaction under static loads is often modelled using Winkler soil-springs. In the simplest cases, the stiffness of these independent springs is defined by neglecting the effect of soil and buildings outside the vertical projection of the studied foundation. Conversely, when these effects have to be accounted for, the stiffness can be obtained after several iterations involving both the structural finite element models (providing at each iteration the reactions at the structure foundation nodes) and the soil model (giving at each iteration the soil settlement). The reaction forces to soil settlement ratio (node-by-node) provide the stiffness estimation at each node for each iteration. When the soil settlements computed at two subsequent iterations are almost identical, iteration is stopped. The obtained stiffness values depend on the load case. Hence, a different modelling approach, based e.g. on the soil flexibility matrix is preferable.

A three-step procedure for the flexibility matrix calculation is presented in this paper: (i) a complete geomechanical model of the site with geological layers, faults…is designed with the software GDM; (ii) this model is then implemented within the software FLAC 3D; (iii) using the FLAC3D model, a unit load is applied on each point of the soil surface under the analysed buildings: the displacements computed for each unit load on all points become one column of the flexibility matrix.
This matrix is finally integrated as a "superelement" within the software ANSYS for the structural computation. The effectiveness and accuracy of this procedure is discussed with reference to the case study of a nuclear power plant.

Keywords: Soil-structure interaction, flexibility matrix, substructuring method, superelement

INTRODUCTION

The purpose of this article is to present the methodology used to define the soil-structure interaction under static loads and accounting for the effect of adjacent buildings of a nuclear power plant. Nuclear facilities are characterized by their geometrical complexity and by the large size of the buildings. This has induced the need for robust numerical tools and the need to develop a specific three-step methodology.
The first phase was the implementation of the soil geometry, taking into account the different states of alteration, the fault network, the accurate lithology and the exact geometry of building foundations. This

was achieved using specific tools like GDM and AutoCAD. The second analysis step was the implementation of the soil model (lithology and mechanical parameters) in the software FLAC3D used for the soil mechanical computations and soil-structure interaction (SSI). The third phase was the SSI conducted to compute the soil flexibility matrix. This matrix is then used to represent the soil in the building FE model used for the design calculations.

Actually, in structural calculations the soil is usually represented by linear springs defined at each node of the model on the underside of the raft. For the definition of their stiffness, an iterative process is used, a first guess is made on spring stiffness in the building model; this gives the nodal reactions under the foundation, which are applied on the soil model. Then, the loading is applied and a first couple force/displacement on each node of the model is calculated. This allows defining a new set of stiffness which is implemented for the structural computation. A new set of force applied to the soil is calculated and the previous process is renewed. After several iterations, a convergence criterion is reached.

To simplify this process, and under condition of linearity of the soil behavior, it is proposed to compute the soil flexibility matrix. The main advantage is to facilitate the integration of the soil stiffness in the software used for the structural calculations.

THE GEOLOGICAL 3D MODEL

Geological context of the studied site

The layers of soil encountered are the following, from the most recent to the older deposits:

- "Made ground" (anthropic deposits): clay and sandy clay (thickness: 1 - 2m);
- Over burden deposits: clay less or more gravely (thickness: ~1m);
- Blue Lias formation: alternation of mudstones and limestones (thickness : 75m;
- Lilstock formation : limestones and mudstones (thickness : 1,3 - 3,4m);
- Westbury formation : black shales with limestones beds (thickness : 7 - 11,5m);
- Blue Anchor formation : grey to green mudstones and siltstones, with anhydrite-gypsum levels (thickness : 26 - 36m);
- Red Mudstones formation: red and green mudstones and siltstones with nodules and levels of anhydrite -gypsum.

The foundation raft level is mainly in the lower part of the Blue Lias formation (mudstones and sandstones).

Three different water levels are taken into account for design computations: i) the natural water level, ii) the level for the construction period, 21m below the natural water level (after drawdown), iii) the level for the exploitation period, at 2m below the natural water level (permanent water level drawdown).

Numerical tool and model construction method

The software used to build the geological model is the geo-modeler GDM. It allows building a geological model based on the interpolation of data between boreholes. A computer-aided design (CAD) with AutoCAD software allowed supplementing and correcting strata calculated with the software "GDM".

In order to build a geometrical model of the geological layers (3D geo-modeler), the synthesis of the factual data (boreholes data) and the interpreted data (geological map) is required.

All factual data and geological rules are captured in the GDM software. Some complementary assumptions are done in order to complete the data-set. These assumptions concerned mainly the layer thickness. Moreover, some conditions are imposed on the model. These conditions arise from geological processes applied by the software on the geometry of the strata. Data complements are done essentially in the South-West and Northern part of the model.

For interpolating the data, a grid of 2mx2m is used. This grid size allows a sufficient detail in the numerical model of the soil for a reasonable computation times. For the interpolation computations of the DTM (Digital Terrain Model) of the site topography, we use the ordinary kriging (or linear kriging without derivative). For the computations of the geological surfaces, we use the universal kriging (or kriging with trend). This choice is justified by the dip of the layers in the Northern part of the model.
Due to numerical difficulties (spatial scattering of the geological data), it was not possible to build the fault model with GDM. Consequently, the GDM model has been computed without faults. Based on the analysis of a geologist, the faults are represented into surface object with software AutoCAD and Covadis.
The GDM model and the faults are then integrated into AutoCAD. This ensures the continuity between the lithological model and the faults system. To fit the geometrical model to our geological analysis, some redrawn work has been done with AutoCAD.
The AutoCAD files include all the geometrical information concerning the geological layers and the fault system of the site. An example of cross-section is given in Figure 1 on the right. On the left of Figure 1 also presents an extract of the geological map at the natural ground.

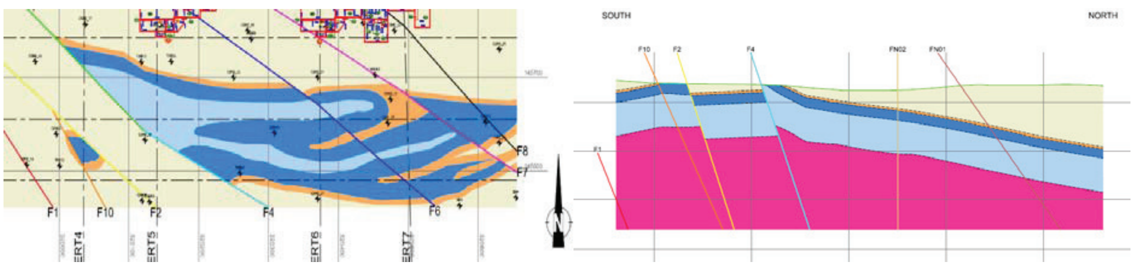


Figure 1 : Example of cross-section of the numerical geological model (right) and extract of the geological map from the numerical geological model (left)

The geological model (essentially lithology and faults geometry) does not include all the information on the soil behavior. Thus, several additions have been made on the geological model to prepare the geotechnical model (essentially mechanical behavior, and additional layers) :

- The layers of earth-fills and the surface soil layers, their base are definied as geometrical surface. This two layers have specific mechanical properties.
- The states of alteration of the rock that apply from the surface until deep, regardless of the nature of the rock: the parameters are defined according to the depth.

Geotechnically, the mechanical parameters are given by the point position compared to the geological layer and the alteration degrees.
In summary, in the numerical model, it is set 7 geotechnical layers: 5 rocky layers and 2 soil layers. The rocky layer is divided in to 4 sub-layers according to degree of alteration. Model size is about 1 000 000 soil volume element (800m x 500m x 100m thick).

THE SOIL MECHANICAL 3D MODEL

Aim of soil mechanical model and numerical tool

The soil mechanical model must allow representing the correct behavior of soil under the structure static loads accounting for the interaction effects due to neighboring buildings. This model should allow an accurate representation of the soil structure interaction (SSI) which is considered as an input data for the structural models.
The lithology and the geo-mechanical parameters are implemented in FLAC 3D code. This code allows the modeling of faults system and the use of a wide choice of constitutive laws, particularly the anisotropy

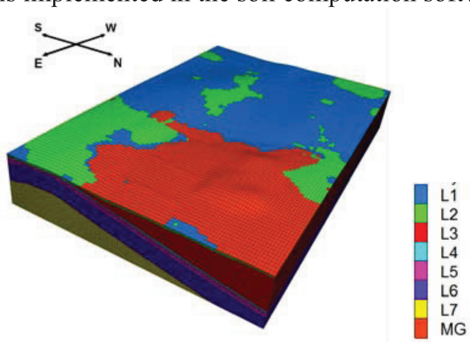
of the rock formations. This model takes into account the phasing of construction and the corresponding water levels.

Soil constitutive laws and parameters

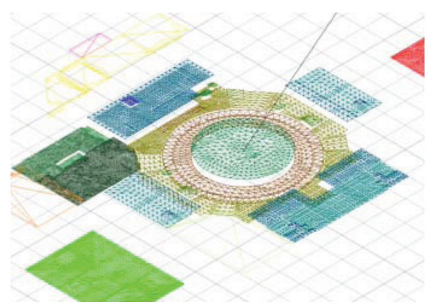
As mentioned above, two types of mechanical behavior are possible: rock behavior or loose soil behavior. The mechanical parameters are defined for the different layers. In order to analyse the impact of non-linearity on the global ground behavior, two set of parameters are defined: one set defines the behaviour without failure (linear elastic constitutive laws), another set defines the failure criteria. The non-linearity comes from the failure model. Consequently, for each type of mechanical behavior, two set of parameters are used:

1- Soil:
   a. 1st set: **elasto-plastic behaviour** with isotropic behaviour (2 parameters) and a Mohr-Coulomb criterion (3 parameters)
   b. 2nd set: **elastic behaviour**, isotropic behaviour;
2- Rock:
   a. 1st set: **elasto-plastic behaviour** with isotropic elasticity and a Hoek-Brown criterion (3 parameters)
   b. 2nd set: **elastic behaviour**, transversely isotropic model (5 parameters).

The value of elastic modulus ranges from roughly 10 MPa for the soils to 1.000 – 10.000 MPa for the rocks. The mechanical parameters (modulus, Poisson's ratio, friction angle) were adjusted according to the in-situ tests (dilatometer, pressuremeter) and laboratory data (triaxial test, oedometer tests).
Concerning the fault system, the geometry of each principal fault is represented accurately. The mechanical law considered for faults is an elastic law, tangent and normal to the surface, and a Mohr-Coulomb criterion.
The "base" of the foundation rafts is modelled with a CAD software in three dimensions. Then, the model is implemented in the soil computation software.



(a)



(b)

Figure 2 – (a) View of the initial soil computational model (FLAC 3D) with the different geotechnical layers without alteration layer and faults; (b) View of the slab geometry with the CAD software

Assessment of non-linearity effects

The soil mechanical 3D model was used to assess the overall behaviour of ground under the dead weight of all buildings (long term behaviour). Several tests were performed to check the model robustness. In particular, the comparison between the displacements calculated with FLAC 3D and those calculated with another simple computation method showed good compliance. This model enables, firstly, to know the details of the soil behaviour under each building and, secondly, to calculate the soil structure interaction comprehensively (stress, strain, displacement…).

The effect of non-linearity is assessed by analysing the occurrence of plasticity zones and by comparing the displacement obtained by the elasto-plastic model, with that obtained by the elastic model. Plastic zones appear at the last load step; nevertheless the movement obtained with both models (elastic and elasto-plastic model) are almost identical. It is then concluded that the soil behaves linearly under the operating loads. Thus the computation of the flexibility matrix can be performed. The following section explains the methodology.

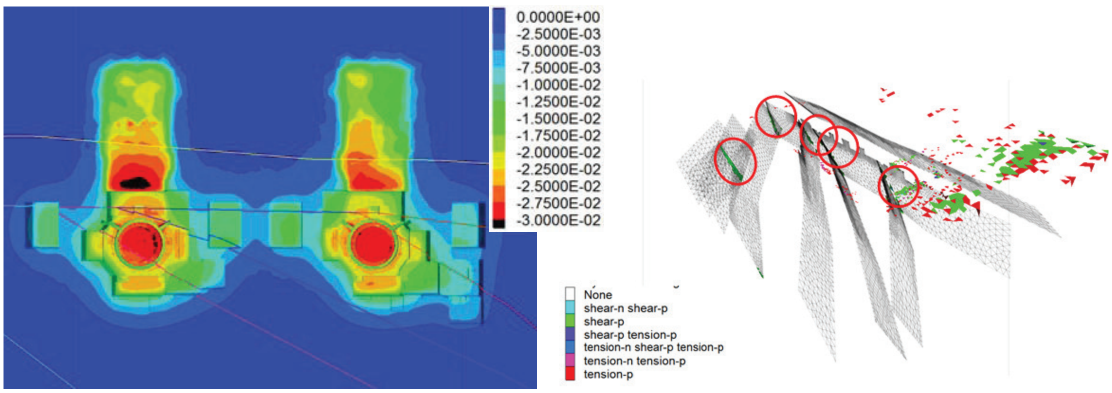


Figure 3 – On the left, map of Z-displacement of the soil surface under the deadweight (color-scale is in meter); on the right, view of plastic volumes of soil and representation of the fault system (color-scale shows the occurrence of plasticity)

COMPUTATION OF THE FLEXIBILITY MATRIX FROM FLAC 3D SOIL MODEL

Since the soil behavior is supposed linear, the force-displacement relationship is described by the following equation:

$$\{\delta\} = [S] \times \{F\} \tag{1}$$

where:

- $\{\delta\}$: displacement vector (model of the soil surface under the buildings)
- $\{F\}$: force vector
- $[S]$: flexibility matrix of soil model

The displacement vector $\{\delta\}$ at each point is obtained by the multiplication of the force vector $\{F\}$ by a flexibility matrix $[S]$.

In order to compute the flexibility matrix from the FLAC 3D model, a unitary force is applied at each point under the buildings according to one spatial direction (X, Y or Z). During the process, the three spatial directions are swept. The result of one computation is the displacement field under the buildings for one unit load. This vector is recorded as a structured text file which is easily readable by computer language.

So, based on the text file, the flexibility matrix may be built and formatted to be read by another computer program, like the ANSYS software.

The main advantages of this method with respect to more commonly used on independent Winkler springs, are that the computed solution is accurate (taking into account the 3D effect), the effect of loads of neighbouring buildings are taken into account at each point of the model and the answer takes into account the terms of coupling xz and yz (non-diagonal terms).

The flexibility matrix [S] generated by FLAC 3D software is a square matrix of size “3nx3n”, where n is the number of nodes of solid soil model in FLAC 3D at the interface with the foundation raft of the structure (see equation (2)). The matrix is composed by 9 sub-matrices:

- [Sxx], [Sxy],[Sxz]: corresponding to the displacement of each node in the respective direction X, Y, Z under the unit force of X direction,
- [Syx], [Syy],[Syz]: idem under the unit force of Y direction,
- [Sxz], [Syz],[Szz]: idem under the unit force of Z direction.

$$\begin{Bmatrix} \delta_{x,1}\\ \vdots \\ \delta_{x,n}\\ \delta_{y,1}\\ \vdots\\ \delta_{y,n}\\ \delta_{z,1}\\ \vdots\\ \delta_{z,n}\end{Bmatrix} = \begin{bmatrix} \begin{matrix} s_{xx,1,1} & \cdots & s_{xx,1,n}\\ \vdots & \ddots & \vdots \\ s_{xx,n,1} & \cdots & s_{xx,n,n}\end{matrix} & [S_{xy}] & [S_{xz}] \\ \begin{matrix} s_{yx,1,1} & \cdots & s_{yx,1,n}\\ \vdots & \ddots & \vdots \\ s_{yx,n,1} & \cdots & s_{yx,n,n}\end{matrix} & [S_{yy}] & [S_{yz}] \\ [S_{zx}] & [S_{zy}] & [S_{zz}] \end{bmatrix} \begin{Bmatrix} F_{x,1}\\ \vdots \\ F_{x,n}\\ F_{y,1}\\ \vdots\\ F_{y,n}\\ F_{z,1}\\ \vdots\\ F_{z,n}\end{Bmatrix} \qquad (2)$$

MODELLING THE SSI BASED ON THE SUBSTRUCTURING METHOD

The substructuring method for static analysis consists on condensing a group of finite elements into one reduced stiffness matrix called *superelement* (reduced model). The following section presents the construction of the reduced stiffness matrix representing the soil.

Reduced stiffness matrix, case 1: all buildings are modelled

Consider the finite element model G constitutes by the two substructures: A (structures model from 1 to n) and B (solid soil model) as shown in the figure below. The finite element model G (A+B) can be subdivided in the following Degrees of Freedom (DOF’s) sets:

- a: internal DOF’s of structures model (A),
- b: internal DOF’s of soil model (B),
- i: interface DOF’s between soil and structures.

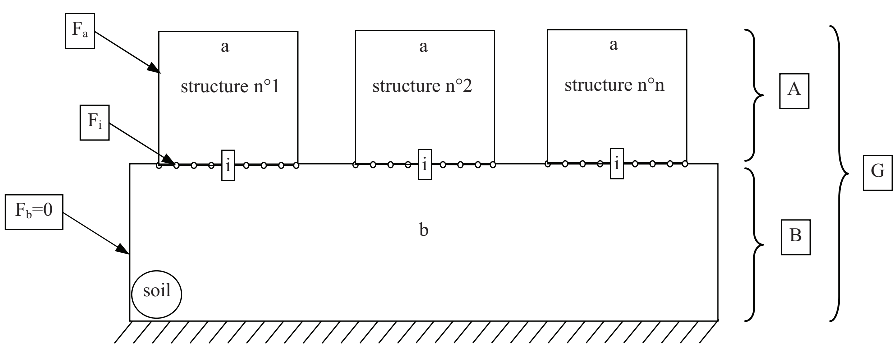


Figure 4: Structures from 1 to n and soil models

The construction of the reduced stiffness matrix of the soil model (B) at the interface DOF’s “i” is totally independent from the structural model (A) (see technical paper of AJA about substructuring method). Thus, this task can be performed using a separate soil model.

Consider the finite element solid soil model as shown in figure above. After applying constraint conditions, static analysis equations become in flexibility submatrices form as follows:

$$\begin{bmatrix} S_{ii} & S_{ib} \\ S_{bi} & S_{bb} \end{bmatrix} \begin{Bmatrix} F_i \\ F_b \end{Bmatrix} = \begin{Bmatrix} \delta_i \\ \delta_b \end{Bmatrix} \tag{3}$$

In soil model, only interface surface (i) of soil with the structure is the loaded area ($F_i \neq 0$). All the other parts of soil model are considered as unloaded ($F_b = 0$). Thus, the equation (3) can be simplified:

$$[S_{ii}]\{F_i\} = \{\delta_i\} \tag{4}$$

$$\Leftrightarrow \{F_i\} = [S_{ii}]^{-1}\{\delta_i\} \tag{5}$$

Thus, the reduced stiffness matrix is the inverse of flexibility matrix at interfaces DOF's:

$$[K_{ii}^{red}] = [S_{ii}]^{-1} \tag{6}$$

- $[S_{ii}]$ is computed as explained in the previous section

To resume, in order to perform a static analysis of structures by substructuring method, the following items are needed (see Figure 5):

- finite element model of structures (A),
- reduced stiffness matrix of soil to interface DOF's calculated as the inverse of the flexibility matrix at the interface DOF's between structure and soil.

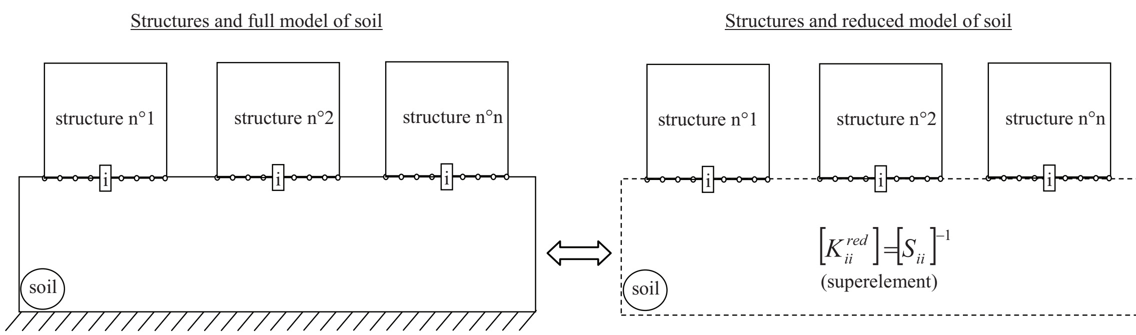


Figure 5: Substructure method applied for SSI – multiple structures founded on the soil

Reduced stiffness matrix, case 2: only one building is modelled

In practice, a separate structure is studied with its own load cases which are independent from the other structures.

For instance, we consider the model of structure 1 only. Then, the structures from n°2 to n are replaced by load fields correspond to the deadweight load at interface DOF's of these structures with soil model.

The interface DOF's between all structures from n°1 to n and soil is subdivided in two sets:

- 1: interface DOF's between structure n°1 and soil,
- 2: interface DOF's between structures n°2 to n and soil.

The equation (4) is written by submatrices of two sets 1 and 2 DOF's as below:

$$\begin{bmatrix} S_{11}^i & S_{12}^i \\ S_{21}^i & S_{22}^i \end{bmatrix} \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \begin{Bmatrix} \delta_1 \\ \delta_2 \end{Bmatrix} \tag{7}$$

- $\left[S_{11}^i\right]$: soil flexibility submatrix of set 1 of interface DOF's (structure n°1)
- $\left[S_{22}^i\right]$: soil flexibility submatrix of set 2 of interface DOF's (structures n°2 to n)
- $\left[S_{12}^i\right]$, $\left[S_{21}^i\right]$: coupling flexibility submatrices between set 1 and 2 of interface DOF's,
- $\{F_1\}$: load path field at set 1 of interface DOF's under studied load case on structure n°1,
- $\{F_2\}$: load path field at set 2 of interface DOF's under studied load case on structures n°2 to n.

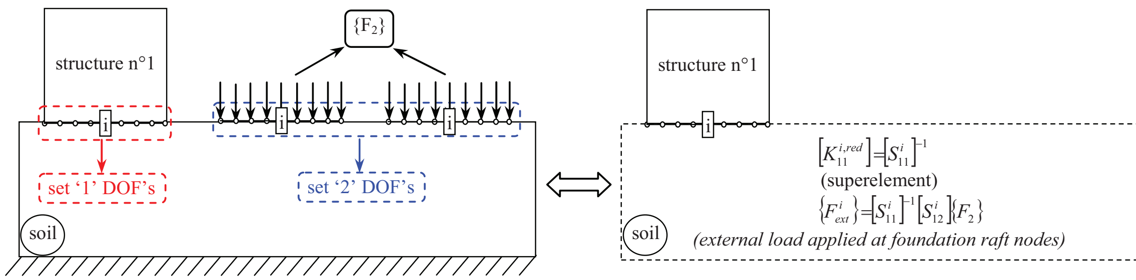


Figure 6: Substructure method applied for SSI–interaction between structures

The equation (7) results in following equations:

$$\left[S_{11}^i\right]\{F_1\} + \left[S_{12}^i\right]\{F_2\} = \{\delta_1\} \tag{8}$$

$$\rightarrow \left[S_{11}^i\right]\{F_1\} + \left[S_{12}^i\right]\{F_2\} = \{\delta_1\} \tag{9}$$

$$\rightarrow \{F_1\} + \left[S_{11}^i\right]^{-1}\left[S_{12}^i\right]\{F_2\} = \left[S_{11}^i\right]^{-1}\{\delta_1\} \tag{10}$$

$$\rightarrow \left[K_{11}^{i,red}\right]\{\delta_1\} = \{F_1\} + \{F_{ext}\} \tag{11}$$

$$\rightarrow \left[K_{11}^{i,red}\right] = \left[S_{11}^i\right]^{-1} \quad \& \quad \{F_{ext}\} = \left[S_{11}^i\right]^{-1}\left[S_{12}^i\right]\{F_2\}$$

- $\left[K_{11}^{i,red}\right]$: reduced stiffness matrix of interface DOF's between structure n°1 and soil,
- $\{F_{ext}\}$: load field at interface DOF's between structure n°1 and soil due to the deadweight of the other buildings.

To resume, in order to perform a static analysis of one structure taking into account the effect of adjacent buildings by substructuring method, the following items are needed (see Figure 6):

- finite element model in ANSYS of considered structure (structure n°1),
- reduced stiffness matrix $\left[K_{11}^{i,red}\right]$, that is the inverse of flexibility matrix of soil at interface DOF's between structure n°1 and soil, integrated in ANSYS code as a superelement matrix,
- external force $\{F_{ext}\}$ at foundation raft nodes of considered structure (n°1), calculated from other structures n°2 to n, considered as a separated load case.

CASE STUDY: UK EPR NUCLEAR ISLAND

In this section, the results of SSI study for NI2 building are shown (see figure below). The reduced stiffness matrix $\left[K_{11}^{i,red}\right]$ at interface DOF's between soil and NI2 building is calculated and then integrated in ANSYS model as a superelement. This matrix is applied for all static loads.

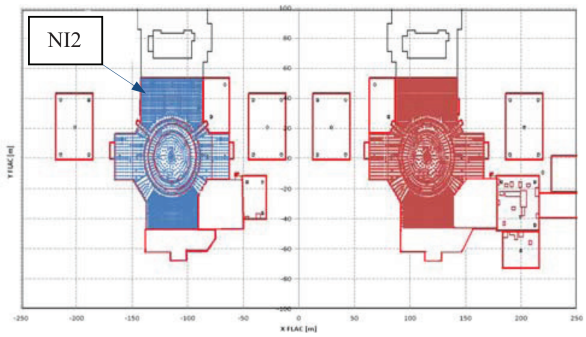


Figure 7: Localisation of NI2 building in site

The Figure 8 presents the settlement of foundation raft of NI2 building under deadweight loads of neighbour buildings (without deadweight of NI2 building). The settlement observed on the right side of foundation raft, where many other buildings are located, is more significant than the one on the left side, where only one building is located.

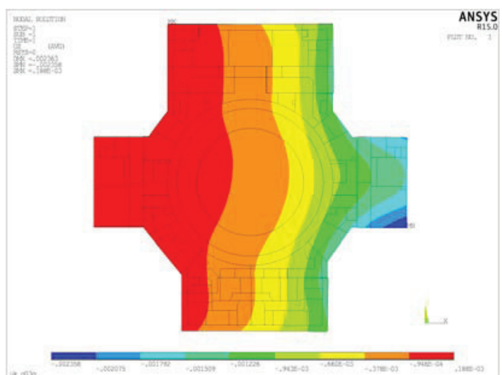


Figure 8: Settlement field of foundation raft under deadweight of neighbour buildings

The following figure presents the settlement of foundation raft of NI2 building under deadweight of NI2 building (without influence of deadweight of neighbour buildings):

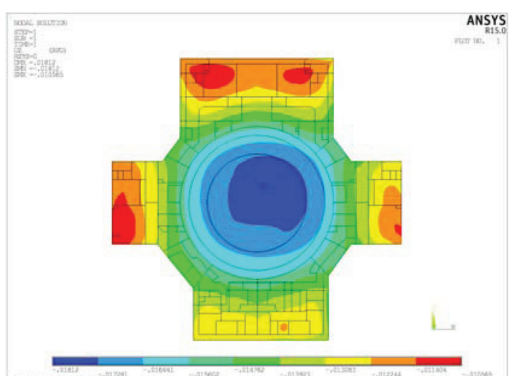


Figure 9: Settlement field of foundation raft under self-weight load of NI2 buildings

CONCLUSION

In this paper, a methodology has been presented for the accurate representation of the soil by a flexibility matrix into a finite element model for static structural calculations. The main assumption is that the soil behavior is linear under the considered loads.

In detail, for the analysed case study, the soil mechanical 3D model was built with several numerical tools: GDM and Autocad for the geological model, which has been implemented in FLAC3D with the suitable mechanical laws of soil. This model was used to assess the overall behaviour of ground under the effect of the full structure deadweight. The effect of non-linearity is assessed by analysing the occurrence of plasticity zones and it was concluded that the soil behaves linearly. Thus the computation of the soil flexibility matrix has been performed.

We recall that the soil flexibility matrix is an intrinsic representation of the soil, which is totally independent from the stiffness, loading, geometry of structures. Then, any modification of structures does not require the reconsideration of the boundary condition (soil) and the results are the same than the ones obtained with the complete model. This considerably improves the usual methodology based on the use of independent soil springs whose load dependent stiffness is obtained by an expensive iterative process.

REFERENCES

GDM version 7.0, Bureau des recherches géologiques et minières (BRGM), France.

AutoDesk, AutoCAD 2011, http://www.autodesk.fr/products/autodesk-autocad/overview.

ITASCA, Finite Lagrangian Approach of Continua (F.L.A.C.3D) version 5.0, http://www.itascacg.com/software/flac.

P.G. Kelly (October 1998). "Development and interaction of segmented fault systems", University of Southampton, UK.

Bowyer and P.G. Kelly (1995). "Strain and scaling relationship of faults and veins at Kilve, Somerset", *Annual Conference of the Ussher Society,* M. O'N.

Prudden, H.C. (2005). "Strike-slip faulting in Somerset and adjacent areas", The Ussher Society.

Engelder, T. and Peacock David C.P (June 2000). "Joint development normal to regional compression during flexural-flow folding : the Lilstock buttress anticline, Somerset, England" *Journal of Structural Geology*.

Kim, Y.S. , Peacock, David C.P. and Sanderson, David J. (November 2002). "Fault damage zones," *Journal of Structural Geology.*

Rives, T. (October 1992). "Mécanismes de formation des diaclases dans les roches sédimentaires, approche expérimentale et comparaison avec quelques exemples naturels," *Université de Montpellier II,* Montpellier, France.

AJA, A. M. (2000). "Sub-modelling techniques for static analysis", *MSC Software First South European Technology Conference*, Principauté de Monaco, 7-9 June 2000.

ANSYS software: Reduction Technique, Part 2 : Substructuring – Applicability and Examples