23rd ABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

DYNAMIC INTERACTION BETWEEN STRUCTURES MODELED BY TWO-DIMENSIONAL FINITE ELEMENT AND THE SOIL-FOUNDATION SYSTEM

Ronaldo Carrion
Department of Mining and Petroleum Engineering, Polithecnic School, University of São Paulo – USP – Brazil
rcarrion@usp.br

Luiz Carlos Facundo Sanchez
Department of Telecommunications Engineering, São João da Boa Vista Faculty, Paulista State University – UNESP – Brazil
lsanches@sjbv.unesp.br

Phelipe Gabriel Varoni
Department of Mechanical Engineering, São Carlos School of Engineering, University of São Paulo – USP – Brazil
phelipe.varoni@usp.br

Abstract.
In the present article a direct version of the Boundary Element Method (BEM) is applied to synthesize the 3D dynamic compliance matrix of a rigid and massless foundation interacting with unbounded soil profiles. The foundation compliance matrix is coupled to a 2D structure, modeled by Finite Element Method (FEM) leading to the dynamic response of a coupled structure-foundation-soil system.
The direct version of the 3D Boundary Element Method (DBEM) is built based on the stationary fundamental solution of an elastic full-space where viscoelastic effects are incorporated by means of the elastic-viscoelastic correspondence principle.
The article applies a methodology to couple rigid bodies with the BEM mesh. The result of the soil-foundation interaction is given in terms of a dynamic compliance matrix for a rigid and massless foundation.
The analysis of 2D structure is based on the FEM where linear triangular finite elements are coupled to soil-foundation system. This strategy allows performing frequency domain analysis of a structure-foundation-soil system.

Keywords: *2D Finite Element Method, Dynamic Soil-structure Interaction, Boundary Element Method*

1. INTRODUCTION

In the classical modal analysis of structures the ordinary dynamic informations are related to the natural frequencies of the system (eigenvalues) and to the modes of vibration (eigenvectors). This analysis allows uncoupling the system of equations and the dynamic solution becomes the weighted superposition of the uncoupled solutions. However, there are some pre-requisites to perform it. The system of equations that describes the structure must be linear and has constant coefficients.

A typical structure with constant coefficients can be exemplified by a couple of masses supported by a fixed foundation bonded to the soil. Most common model states that the foundation has a zero displacement, regardless the effort acting on it. Under many circumstances this assumption is reasonable but for many other conditions this is not a good description of reality.

It is important to notice that the soil interacts with the foundation, which has inertia and stiffness and, therefore, also deforms or moves. If soil deforms or moves under the action of efforts, the foundation bonded to it also moves. Based on this last comment, the hypothesis formulated above, which describes a fixed foundation, is no longer reasonable. It means that one cannot perform the classical modal analysis of dynamical systems where the soil is included.

It is possible to evaluate the dynamic soil-structure interaction in the frequency domain (Hall and Olivetto, 2003). However, for this complete analysis, it is necessary to solve the whole system of equations for each frequency.

This work shows a methodology to deal with a two-dimensional dynamic soil-structure interaction analysis coupling Finite Element Method (FEM) and Boundary Element Method (BEM).

2. STATIC ANALYSIS

To present the first step of methodology, static analysis is evaluated. In this case, a structure is submitted to an external load and since it is not reasonable to obtain the displacements of some points by classic methods of solid mechanics, it can be achieved by using FEM (Kim and Sankar, 2009). The two-dimensional structures are discretized into three-nodes *Constant Strain Triangular* (*CST*) elements. The nodes displacements can be calculated using the linear system below:

$$[K]*\{u\}=\{F\} \tag{1}$$

The $[K]$ matrix is known as stiffness matrix, $\{u\}$ is the displacement vector and $\{F\}$ the load vector. This global stiffness matrix is composed from the stiffness matrices of each element (Hughes, 1987).

The elementary stiffness matrix is obtained by the following equation:

$$[k]=h\,A[B]^T[C_\sigma][B] \tag{2}$$

Where h is the element thickness, A its area of element, B a coefficient matrix relative to the local nodes coordinates and $[C_\sigma]$ a coefficient matrix related to the Young's modulus and Poisson's ratio. The elementary stiffness matrix is 6x6, since each element has three nodes with two degrees of freedom per node.

Boundary conditions must be applied to solve the system which implies in removing rows and columns related to constrained degrees of freedom of the stiffness matrix. Once the system was solved (for displacements) the entire system can be solved again to obtain reactions on the boundary.

Displacements inside the element can be interpolated in terms of its nodal displacement, using the shape functions. They are shown below for an element, where x and y are the point coordinates where displacement is aimed, A the area of the element and f, b and c coefficients obtained from the coordinates of local nodes. The interpolated displacement varies linearly in each coordinate direction.

$$\begin{aligned}
N_1(x,y)&=\frac{1}{2A}(f_1+b_1x+c_1y)\\
N_2(x,y)&=\frac{1}{2A}(f_2+b_2x+c_2y)\\
N_3(x,y)&=\frac{1}{2A}(f_3+b_3x+c_3y)
\end{aligned} \tag{3}$$

With the shape functions, the displacements interpolation u and v can be done as:

$$\begin{aligned}
u(x,y)&=[N_1 \quad N_2 \quad N_3]\begin{Bmatrix}u_1\\u_2\\u_3\end{Bmatrix}\\
v(x,y)&=[N_1 \quad N_2 \quad N_3]\begin{Bmatrix}v_1\\v_2\\v_3\end{Bmatrix}
\end{aligned} \tag{4}$$

Numerical indexes represent local nodes.

An algorithm was built in order to standardize the points with interpolated displacements on each element, ending with 36 points for an element. Below, in Figure 1, is outlined how points are determined.

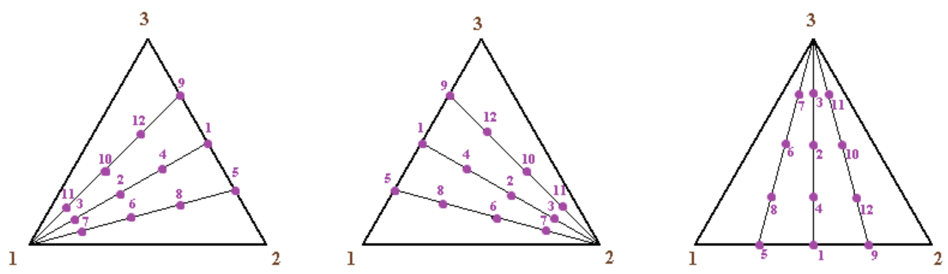

Figure 1. Internal points of triangular elements.

Example

To validate FEM code the following example, showed in Figure 2 was simulated. This Figure brings a structure with three triangular finite elements with nodes coordinates, two constrains and an external load. All elements have the same properties: $E = 210GPa$, $\nu = 0.3$ and $h = 5*10^{-5} m$.

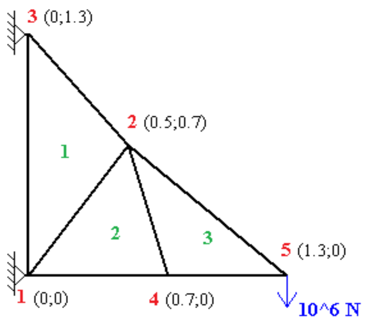

Figure 2. 2D structure with respective mesh.

Displacements of all degree of freedom (nodes) are listed bellow and in agreement with literature.

$$u = \{0 \quad 0 \quad 0.0861 \quad -0.1509 \quad 0 \quad 0 \quad -0.2566 \quad -0.4471 \quad -0.2210 \quad -1.2129\}^T$$

Since static results are correct, a new code with dynamic analysis can be implemented.

3. DYNAMIC ANALYSIS

The goal in this dynamic analysis is to obtain the nodal displacements for a harmonic dynamic load, in a range of excitation frequency. In this work the analysis is performed in frequency domain rather than time one, so the results are plotted as *Frequency Response Function* (FRF). According to theories of mechanical vibration (Rao, 2011), the equation that represents the behavior of a structure under a dynamic loading, without damping, in the frequency domain is:

$$[K - \omega^2 M]\{u\} = [F] \tag{5}$$

$[K]$ matrix and $\{u\}$ and $\{F\}$vectors are already known from static analysis. $[M]$ is known as mass matrix and ω is the variable frequency. The first term of this equation is known as compliance matrix $[S]$, so Eq. (5) can be represented as:

$$[S]\{u\} = [F] \tag{6}$$

Global mass matrix is obtained from the basic mass matrix as used to obtain the global stiffness matrix. Elementary mass matrices are presented below (Vaz, 2011).

$$[m] = \frac{\rho A h}{3}\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \tag{7}$$

In dynamic analysis by FEM there are two types of mass matrix: distributed or *consistent* and *lumped*. In the first type the mass of an element is distributed on it; in the second one a simplification is made considering the element mass is concentrated only in its nodes. In this work only the lumped matrix is used.

As in elementary stiffness matrix, *A* represents the area of the element and *h* its thickness. The new term ρ represents the mass density of the element.

Compliance matrix is calculated for each value of frequency, the nodal displacements are obtained and the linear system is solved. FRF of a given node displacements is plotted and the values on which these displacements have peaks tending to positive infinity can be observed, characterizing the natural vibration frequencies, eigenvalues.

In fact, all structures presents damping and a widely known one is the proportional damping $[C]$, which is directly proportional to the linear combination of stiffness and mass matrices.

$$[C]=\alpha[K]+\beta[M] \tag{8}$$

where α and β are predefined constants.

In this case Eq. (5) can be written as:

$$\left[K-\omega^2 M+(\omega C)i\right]\{u\}=[F] \tag{9}$$

It is important to observe that the variables become complex since the damping tends to dissipate energy from the system. The first term is also called compliance matrix, although it is different from the previous equation.

Another way to dissipate energy from the structure can be done by adding the soil effect. It behaves as a dynamic absorber (dashpot) in the structure, minimizing its amplitude in the resonance condition. One or more degrees of freedom are attached to the soil, which leads to a similar effect to the proportional damping, but with specific parameters.

Soil has compliance values for each direction that vary according to the excitation frequency. In this case, complex variables appear due to the presence of damping. Soil modifies the system stiffness matrix according to which nodes are connected. These terms are calculated as below, and *S* is the impedance value for a given direction.

$$S_{soil}=a\,G\,S \tag{10}$$

where *G* is the Shear Modulus and *a* is the foundation semi-width.

The new stiffness matrix obtained by the addition of the soil effect is now called $[K_{soil}]$. New displacements can be calculated from Eq. (11) and since $[K_{soil}]$ has complex values, *real* refers to its real part and *imag* to its imaginary part (Carrion, 2002).

$$\left[real(K_{soil})-\omega^2 M+(\omega C+imag(K_{soil}))i\right]\{u\}=[F] \tag{11}$$

With the addition of the soil effect the frequency responsible for the first peak in the FRF is known as rigid body natural frequency, which can be evaluated as:

$$\omega_{rb}=\sqrt{\frac{k}{m}} \tag{12}$$

where *k* is the static stiffness and *m* the mass of the structure.

All the three situations pointed above (with and without damping and with soil effect incorporation) for the dynamic behavior of a structure are exemplified below.

Example

To check the dynamic analysis it was created an example based on the following mesh showed in Figure 3. It aims to simulate a free-clamped bar whose results can be compared with those obtained by a one-dimensional analysis. The red dot indicates the plotted displacements node and blue arrows indicate the loads, each one with 5000 N. The horizontal dimension of the structure is 10m and the vertical is 1m.

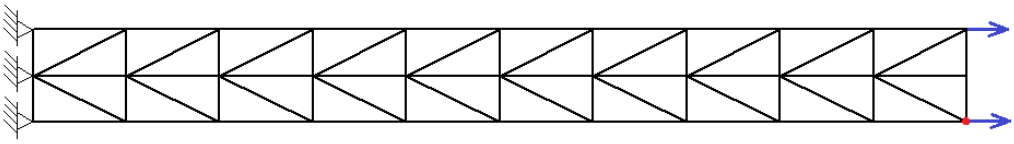

Figure 3. “2D” free-clamped bar mesh.

For this example the elements have the following properties: $E = 210GPa$, $\nu = 0.3$, $h = 5*10^{-5}m$ and $\rho = 7860 kg/m^3$. After calculating mass and stiffness matrices the damping matrix is obtained with constants $\alpha = 0.0001$ and $\beta = 0$. The soil compliance, obtained from another code based on BEM, is an input data read from an input file with $G = 144GPa$ and $a = 0.01m$.

The code runs analysis for each frequency value, obtaining the structural nodal displacement. The resulting FRF for three cases can be compared in Figures 4 and 5.

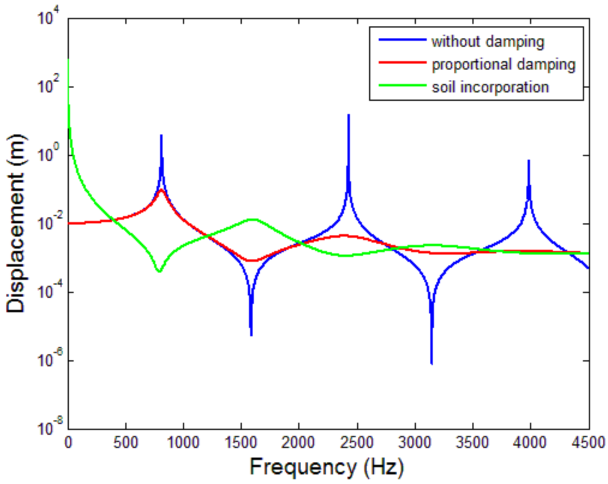

Figure 4. Horizontal displacement - FRF.

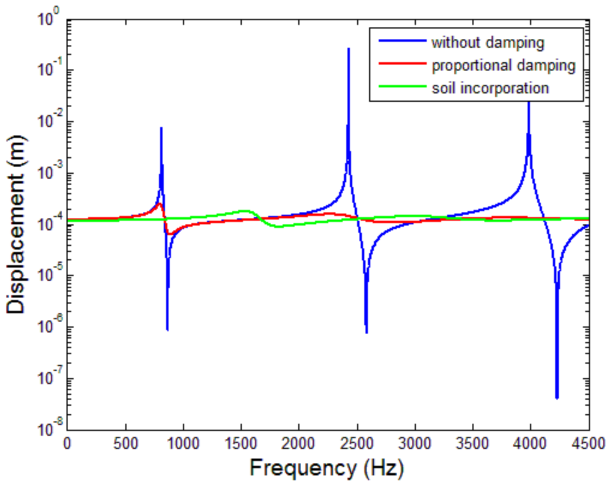

Figure 5. Vertical displacement - FRF.

For code validation purpose only the first three eigenfrequencies are shown.

It is worth noting that while proportional damping decreases the displacement amplitude, the addition of the soil effect shifts the peaks to the left. Another observation is that the vertical displacement is around 100 times smaller than the horizontal one, since the structure is being loaded in the last direction.

For horizontal displacement, Figure 4, the rigid body natural frequency was analytically calculated in 458.6515 Hz, which is slightly different from the numerical one, due to approximations inherent to the methodology.

4. CONCLUSIONS

It could be noticed in this work the enormous usefulness of the finite element method. In the first step, the implemented code based on triangular elements mesh allowed to calculate static displacements all over the element (nodes and internal points). In the second step, the code simulates the dynamic behavior of structures, providing its natural vibration frequencies (eigenvalues) and FRF's with and without damping besides the soil effect incorporation.

This methodology showed to be an efficient alternative to evaluate the influence of soil on the dynamic response of 2D structures modeled by FEM with relative low computational cost.

Clearly this type of code becomes a very powerful tool in projects or other engineering problems since it provides dynamic behavior of a two-dimensional structure under the soil effect.

A more detailed analysis of the system response must be performed furthermore; however current results are very promising.

5. ACKNOWLEDGMENTS

The researches involved in this article have received support from SANTANDER. This is great fully acknowledged.

6. REFERENCES

Carrion, R., 2002, Boundary Element Method for Steady State 3D Domains. PhD Thesis, State University of Campinas - UNICAMP, (in portuguese).

Hall, W. S. and Olivetto, G., 2003, Boundary Element Methods for Soil-Structure Interaction, Kluwer Academic Publishers, Dordrecht.

Hughes, T. J. R., 1987, The Finite Element Method: Linear Static and Dynamic Finite Element Analysis, Prentice-Hall.

Kim, N. H., Sankar, B. V., 2009, Introduction to Finite Element Analysis and Design, John Wiley & Sons.

Rao, S., 2011, Mechanical Vibrations, Prentice-Hall.

Vaz, L. E., 2011, Finite Element Method in Structural Analysis, Elsevier, (in portuguese).

7. RESPONSIBILITY NOTICE