SoftwareX 20 (2022) 101265

Contents lists available at ScienceDirect

SoftwareX

journal homepage: www.elsevier.com/locate/softx

Original software publication

MultiFEBE: A multi-domain finite element–boundary element solver for linear mixed-dimensional mechanical problems

Jacob D.R. Bordón *, Guillermo M. Álamo, Luis A. Padrón, Juan J. Aznárez, Orlando Maeso

Instituto Universitario de Sistemas Inteligentes y Aplicaciones Numéricas en Ingeniería, Universidad de Las Palmas de Gran Canaria (ULPGC), Las Palmas de Gran Canaria 35017, Spain

ARTICLE INFO

Article history:
Received 3 August 2022
Received in revised form 13 October 2022
Accepted 11 November 2022

Keywords:
Boundary elements
Finite elements
Structural analysis
Mixed-dimensional coupling

ABSTRACT

MultiFEBE integrates multiple finite element and boundary element models for solving linear static and time harmonic multi-domain interaction problems within the field of computational mechanics. It allows to couple bounded or unbounded two- and three-dimensional continuum inviscid fluids, elastic solids or poroelastic media with beam and shell structural elements. This paper summarizes the models and numerical methods implemented in the code, and illustrates its use and capabilities through the computation of the dynamic response of the support structure of an offshore wind turbine fixed to the seabed through a jacket structure and three suction caissons.

Code metadata

Current code version	v2.0.0
Permanent link to code/repository used for this code version	https://github.com/ElsevierSoftwareX/SOFTX-D-22-00224
Code Ocean compute capsule	none
Legal Code License	GPL-2.0
Code versioning system used	git
Software code languages, tools, and services used	Fortran 2003, Fortran preprocessor, OpenMP, GNU Fortran, GNU Make and CMake.
Compilation requirements, operating environments & dependencies	OpenBLAS library, GNU/Linux and Windows (MSYS2 for compilation).
If available Link to developer documentation/manual	https://github.com/mmc-siani-es/MultiFEBE
Support email for questions	jacobdavid.rodriguezbordon@ulpgc.es

1. Motivation and significance

The Boundary Element Method (BEM) is a numerical technique especially suited to solve problems that involve domains that can be considered unbounded such as, the open air around a noise source, or the soil beneath a structure's foundation. On the other hand, the Finite Element Method (FEM) is a widely adopted numerical technique due to its versatility and adaptability. Some codes recently published involving these methodologies are, for instance, those in [1–6]. There exist many kinds of problems whose analysis can benefit from coupling both methods. In such cases, coupling can even be performed between models of different dimensionality. For instance, mixed-dimensional coupling between BEM and FEM are particularly interesting for the dynamic analysis of structures interacting with the surrounding media because it combines the best of both approaches: BEM intrinsic fulfillment of the Sommerfeld radiation condition (for unbounded domains), and FEM's ability to model structural elements. In this case, a three-dimensional BEM model of the ground could be coupled, for instance, to a three- (solid), two- (shell) or one-dimensional (beam) model of a foundation.

Many different methods and models have been developed over the years with these ideas in mind. Thus, the code presented in this paper gathers and integrates different contributions that have been developed in this field within the research group over the years involving BEM–BEM [7–13] and BEM–FEM [14–18] models, including most of the latests developments for coupling 3D BEM media with 1D FEM structural elements and 2D FEM

* Corresponding author.
E-mail address: jacobdavid.rodriguezbordon@ulpgc.es (Jacob D.R. Bordón).

https://doi.org/10.1016/j.softx.2022.101265

structural elements, and 2D BEM media with 1D FEM structural elements, where BEM regions can be of fluid, viscoelastic and poroelastic nature. These latter coupling strategies [19–21] require the use of hypersingular and dual-BEM formulations for each of the different types of media (fluid, elastic solid, poroelastic medium), which are both analytically and numerically challenging [22].

The code presented here is especially suited to tackle problems in the field of computational mechanics. For this reason, an application example to the problem of the dynamic response of the support structure of an offshore wind turbine is given at the end of the paper, after the underlying governing equations and numerical techniques, and the description of the software arquitecture, are briefly presented.

2. Model description

2.1. Governing equations

Within the context of computational mechanics, different models can be used to describe the mechanical response of continuum media. Three different models are considered: inviscid fluid, elastic solid, and poroelastic medium. All these models are subjected to the following hypotheses: (1) material properties are isotropic and homogeneous within each region, (2) loads are small enough to consider a linear elastic response (small displacements and linear stress–strain law hypotheses). The governing equations of such models are:

- **Inviscid fluid**. Under the mentioned hypotheses, the fluid is at rest and it suffers small vibrations around the resting position, i.e. it conveys acoustic wave propagation. The governing equation is the Helmholtz equation [23,24]. This model is available for regions modeled by the BEM in time harmonic analyses.
- **Elastic solid**. It corresponds to the usual linear viscoelastic solid whose response is governed by Navier equations [23]. This model is available for regions modeled both by the FEM and the BEM in static and in time harmonic analyses.
- **Poroelastic medium**. The considered porous medium consists of an elastic solid frame saturated by a compressible viscous fluid, whose response in the frequency domain can be represented by Biot's poroelasticity equations [25]. This model is available for regions modeled by the BEM in time harmonic analyses.

The three-dimensional governing equations of these models are general, and can be applied to any domain shape. However, they can be simplified via degeneration (under proper hypotheses) or via a direct Strength of Materials approach when the geometry has certain simplifying regularities. This leads to the usual two-dimensional continuum models (plane strain, plane stress and axisymmetric), and structural models (bar, beam/arch, plate/shell). These simplified models are advantageous because they are simple to define, computationally more efficient, and their results are easy to analyze. Thus, MultiFEBE also includes the following types of structural elements (see, e.g., Bathe [26] and Oñate [27,28]):

- Discrete translational and translational-rotational springs and dashpots,
- Straight bars, Euler–Bernoulli beams and Timoshenko beams,
- Curved Timoshenko beams based on the degeneration of the solid, and
- Reissner–Mindlin shell elements based on the degeneration of the solid.

2.2. Numerical methods

The static and dynamic response of any given problem that can be represented using the above models and a set of loads and boundary conditions are solved herein using the Boundary Element Method and the Finite Element Method. In both cases, the governing equations are transformed, via different strategies, into a set of algebraic equations that can be solved to find the solution:

The Boundary Element Method The BEM requires the computation of the different terms involved in the corresponding Boundary Integral Equations (BIEs), that can be obtained from a weighted residual formulation of governing equations or directly from reciprocity relationships. They relate the value of a field variable (displacements, stresses, etc.) at a given point (collocation point $\mathbf{x}^{\mathrm{i}}$) with the value of primary and secondary field variables along the boundary Γ of the domain Ω. If body loads are present, domain integrals also appear in them. For example, for an elastic solid domain, the BIE can be written using indicial notation as:

$$u_l^{\mathrm{i}} + \int_\Gamma t_{lk}^* u_k \,\mathrm{d}\Gamma = \int_\Gamma u_{lk}^* t_k \,\mathrm{d}\Gamma + \int_\Omega u_{lk}^* b_k \,\mathrm{d}\Omega \qquad (1)$$

where u_l^{i} and t_l^{i} are displacements and tractions at a given collocation point $\mathbf{x}^{\mathrm{i}} \in \Omega$, and $\square^*$ denotes a term of the fundamental solution or Green's function considered.

Most of the advantages and disadvantages of this method lie on the requirement of the knowledge of the fundamental solution or a Green's function. On the one hand, it allows to reduce by one the dimensionality of the problem (thus requiring the discretization of only the boundary, and not the domain, in general) and it also intrinsically fulfill Sommerfeld's radiation condition, handling unbounded regions very efficiently, which allows to deal, rigorously and elegantly, with wave propagation phenomena through soil, water and air. On the other hand, it reduces the versatility and flexibility of the method because it makes more difficult to address non-linear aspects, and it produces fully populated matrices in the final system of equations. This system of equations is built by writing the BIEs for each node of the boundary element mesh, and reorganizing the resulting set of algebraic equations after applying loads and boundary conditions. A complete description of the BEM can be found for example in [29,30]. Hypersingular and dual-BEM formulations for each of the different types of media (fluid, elastic solid, poroelastic medium) are also implemented in the present work [19–21,31–33].

The Finite Element Method In the case of the FEM, the transformation is performed through a weighted residual approach or via the use of the principle of virtual work, and partitioning the domain into a set of subdomains (finite elements), where it is assumed that field variables vary according to some function (typically polynomials). On an element level, algebraic equations appear in the form of an equilibrium equation:

$$\left(\mathbf{K}^{(e)} - \omega^2 \mathbf{M}^{(e)}\right) \cdot \mathbf{a}^{(e)} - \mathbf{Q}^{(e)} \cdot \mathbf{f}^{(e)} = \mathbf{q}^{(e)} \qquad (2)$$

where $\mathbf{K}^{(e)}$ and $\mathbf{M}^{(e)}$ respectively are the element stiffness and mass matrices, $\mathbf{a}^{(e)}$ is the vector of element degrees of freedom (displacements, rotations, etc., of each element node), $\mathbf{Q}^{(e)}$ is the matrix which transforms distributed forces $\mathbf{f}^{(e)}$ into equivalent nodal loads, and $\mathbf{q}^{(e)}$ is the element equilibrating load vector. All element equilibrium

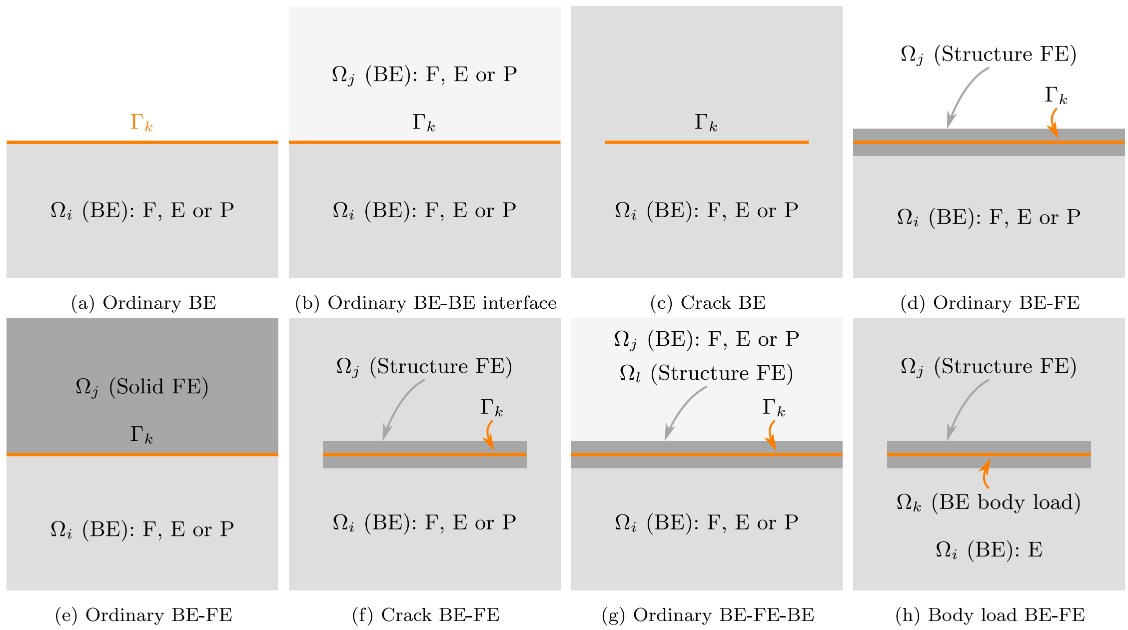


Fig. 1. Type of Boundary Element (BE) and Finite Element (FE) situations, including coupling between them. BE region material models are denoted as: inviscid fluid (F), elastic solid (E), poroelastic medium (P).

equations are assembled using compatibility and equilibrium conditions to form the global equilibrium equation, which together with the boundary conditions allow building a global algebraic system of equations. A complete description of the FEM can be found, for instance, in [26,27,34].

2.3. BEM-BEM and BEM-FEM coupling

One of the main distinctive features of the presented software is the integration of many types of coupling between BEM regions (BEM–BEM coupling), and BEM and FEM regions (BEM–FEM coupling). A summary of these are shown in Fig. 1.

BEM-BEM coupling. When two BEM regions $\Omega_i - \Omega_j$ of different material models: inviscid fluid (F), elastic solid (E) or poroelastic medium (P); are interacting each other through an interface Γ_k, up to six different interfaces are possible: F–F, F–E, F–P, E–E, E–P and P–P. A complete description of such interface conditions are described in Aznárez et al. [12].

BEM-FEM coupling. The following cases have been implemented:

- Beam finite elements embedded in 3D viscoelastic regions modeled by BEM, as described for instance in [14].
- Beam finite elements embedded in 2D fluid regions modeled by BEM, and using crack boundary elements [19] for acoustic analyses.
- Beam finite elements embedded in 2D viscoelastic, poroelastic regions modeled by BEM, using crack boundary elements [20].
- Shell finite elements embedded in 3D fluid, viscoelastic or poroelastic regions modeled by BEM, using crack boundary elements [21].
- Shell finite elements embedded in 3D viscoelastic regions modeled by BEM, using body surface load elements.
- Other types of coupling such as the coupling one side of shell finite elements with ordinary boundary elements, or the coupling of solid finite elements with ordinary boundary elements, are more widespread in the literature.

Fig. 1 presents an overview the possible situations. In all cases, the coupling is performed on a node-by-node basis, i.e. the so-called engineering direct approach, so finite element mesh and boundary element mesh must be conforming.

3. Software description

3.1. Software architecture

Fig. 2 presents an overview of the overall arquitecture of MultiFEBE. The programming paradigm is mainly procedural, though some object-oriented features are also used (Fortran modules and derived types). Intrinsic modules are used as much as possible (e.g. `iso_fortran_env`). Implicit variables are not allowed, i.e. `implicit none` must be used everywhere, and the following naming convention is used for identifiers in variables, derived types, modules and functions:

- Identifiers must reflect their function to improve code readability without requiring comments.
- Use short identifiers where possible.
- Identifiers with several words must be delimited by underscore, e.g. `write_file`.

MultiFEBE has a two-layer design:

Application layer This layer contains the main program (see Fig. 2), a module with the case data, and the high-level subroutines which read command line arguments and input files, perform case main computations (allocation, assembling, solving), and write output files.

Computational layer This layer mainly contains a library of interdependent modules called FBEM, which implements the core derived types (data structures) and computational subroutines. Fig. 3 depicts the functionality and relationship between modules.

The resulting BEM–FEM models lead to a partially dense (due to BEM equations) and partially sparse (due to FEM equations) linear system of equations. To take full advantage of the current

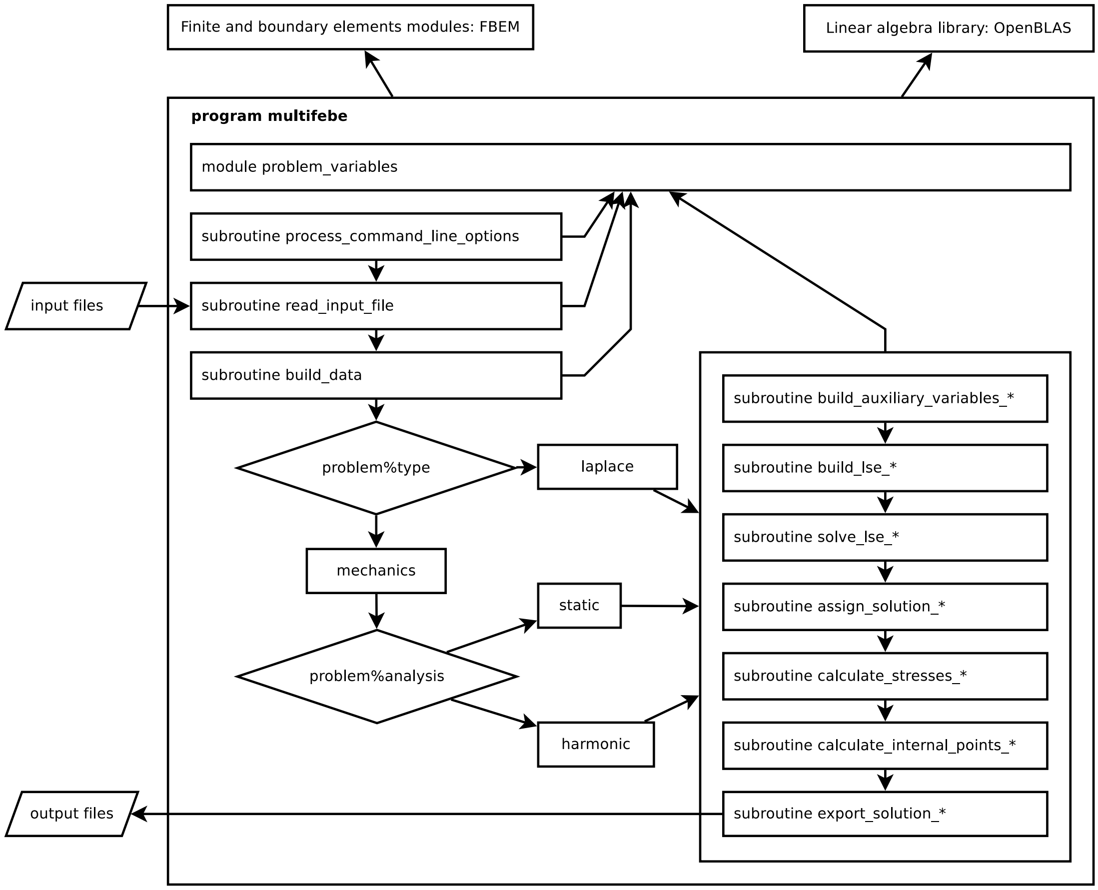


Fig. 2. Overview of the general arquitecture of the MultiFEBE code. Application layer.

shared memory multiprocessor systems, parallelization is used at the two main computational stages: (a) When building the linear system of equations, for-loops over elements are parallelized using OpenMP with dynamic scheduling; and (b) For solving the linear system of equations, OpenBLAS implementation of LAPACK APIs is used.

3.2. Software functionalities

MultiFEBE is a solver for performing linear elastic static and time harmonic analyses of continuum and structural mechanics problems comprising multiple interacting regions. The regions and structural models available are those described in Section 2.1. Two fundamental solutions are implemented in MultiFEBE: the full-space point load fundamental solution is available in all cases; and the half-space point load Green's function is also available for elastostatics and inviscid fluids. These half-space Green's functions do not require the discretization of the free-surface, which greatly reduces computational time. For further reading, see Brebbia and Dominguez [29,30], and Bordón [35].

MultiFEBE's input and output is provided via plain text files. Meshes can be defined in different formats, including Gmsh format, and it also produces post-processing files for Gmsh's pre- and post-processor [36]. It also accepts GiD mesh files through a template file. Tables 1 and 2 summarize the finite and boundary element types implemented in MultiFEBE.

The material models, numerical methods and coupling strategies available are all briefly described in Section 2. Finite elements can be coupled to ordinary boundary, crack boundary and body load elements in order to study, for instance, soil–structure, fluid–structure and soil–fluid–structure interaction and wave propagation problems. Vertical and inclined incident seismic planar wave fields, together with the usual loading and boundary conditions, have also been implemented. More details are provided in the documentation of the code, that can be found in the repository.

4. Illustrative example: Dynamic response of an OWT

In this application example, the dynamic response of an offshore wind turbine fixed to the seabed through a jacket structure and three suction caissons is analyzed. To this end, a seismic input (vertically incident S-wave) is considered. Fig. 4 shows an exploded view of the mesh, where each part is indicated.

DTU 10 MW Reference Wind Turbine [37] is taken as the base wind turbine. Some simplifications are made in order to capture the basic physics. The Rotor–Nacelle Assembly (RNA) is modeled simply as an equivalent mass matrix at the tower top. The tower is modeled using beam finite elements (Timoshenko theory). Other details about it can be found at [37]. The jacket is a structure of tubular members modeled with beam finite elements. The jacket design comes from a preliminary design based on basic requirements for a mean sea level of 20 m. The tower and the jacket are connected through the so-called transition piece, which consists of a base plate and diagonal members connected to the legs. The foundation consists of three suction caissons (tripod configuration). Each suction caisson has a lid (plate), whose center is connected to a leg, and a cylindrical skirt (shell), and their length to diameter ratio is 1. All the structure is made of steel.

The soil is considered to be an elastic solid half-space with shear wave velocity of 180 m/s. The soil region is modeled using the BEM with the elastodynamic full-space Green's function (fundamental solution), so the free-surface and the soil-lid require

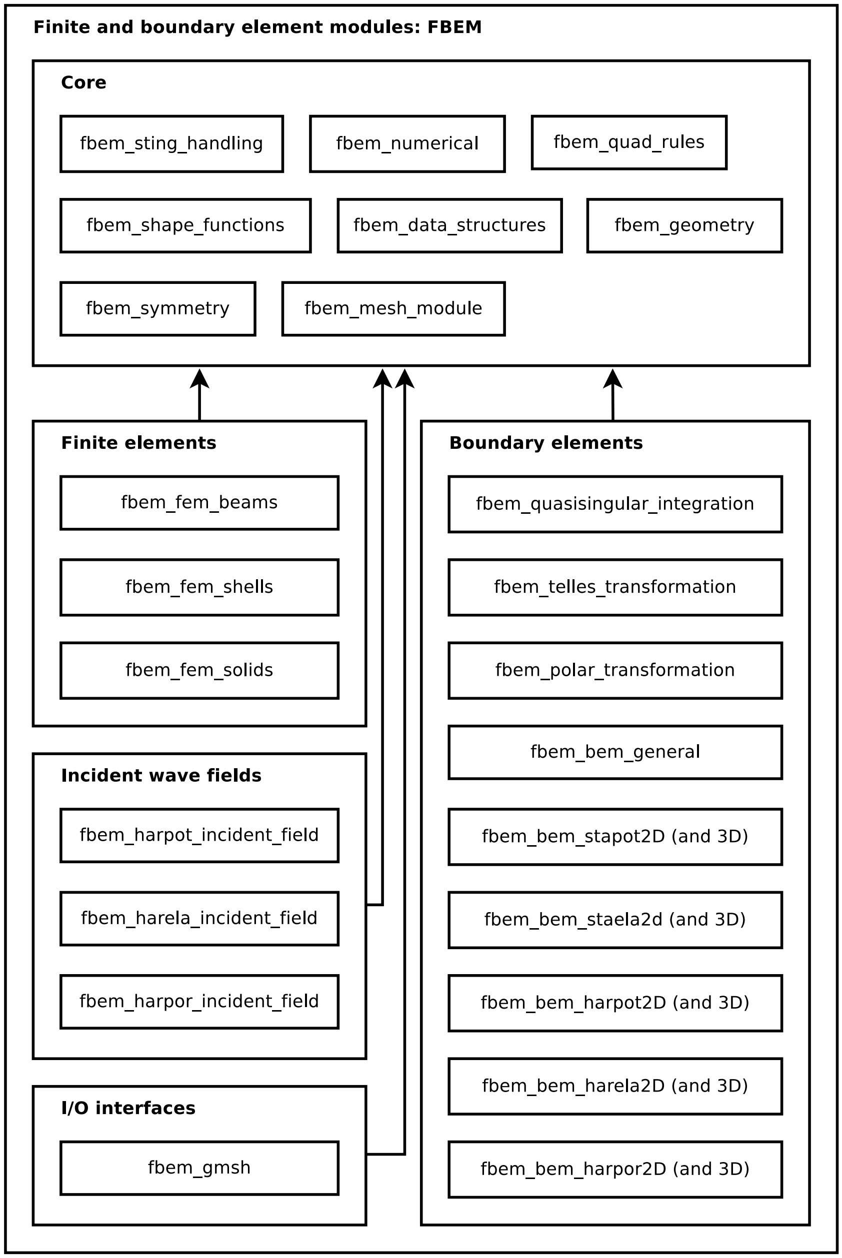


Fig. 3. FBEM library of modules corresponding to the computational layer of MultiFEBE.

boundary elements, since these are the domain boundaries. The soil–skirt interface is modeled with body surface load elements. Note that the soil-lid and soil–skirt meshes are conforming to the corresponding parts of the structure modeled with finite elements.

Fig. 5 shows the horizontal displacement response at the tower top (RNA) normalized by the horizontal displacement of the S-wave at the seabed. Two different results are superimposed: first the flexible base model (the coupled model including soil–structure interaction), and second the rigid base model (the finite element mesh with fixed legs at the seabed). This response function allows to indirectly observe the vibration modes. The peaks indicate some of the natural frequencies (those which moves the RNA significantly). Fig. 6 shows the deformed shapes at the previously mentioned peak frequencies. The first peak is clearly related to the fundamental mode, which is the first mode of the tower (basically a cantilever beam fixed at the transition piece base). The second peak is also related to the second mode of the tower, although there is some extra flexibility introduced by the jacket. Third to fifth peaks correspond to coupled modes of tower and jacket bracing.

From the comparison between flexible base and rigid base, beneficial soil–structure effects are observed. First, peak frequencies decrease in value, indicating the obvious loss of stiffness introduced by the foundation. Secondly, a much more relevant increase of damping is observed. This reduces by an important amount the displacement magnitude (note that displacement axis is in logarithmic scale).

5. Conclusions

This paper presents MultiFEBE, a code aimed at the efficient analysis of continuum mechanics problems using the finite element method, the boundary element method and, especially,

Table 1
Finite elements implemented in MultiFEBE.

Solid (or continuum) finite elements
3 or 6 nodes triangular elements for two-dimensional plane strain problems 4, 8 or 9 nodes quadrilateral elements for two-dimensional plane strain problems
0D Structural elements
Discrete mass and inertia elements
1D Structural finite elements (bars and beams)
2 nodes bar elements 2 or 3 nodes Euler–Bernoulli or Timoshenko straight beam elements (doubly symmetric cross-section) 3 or 4 nodes curved element based on the degeneration from solid (Timoshenko theory, doubly symmetric cross-section)
Spring and dashpots finite elements
Translational and translational-rotational spring/dashpot 2 nodes elements.
2D Structural finite elements (shells). Reissner–Mindlin theory
3 or 6 nodes triangular and 4, 8 or 6 nodes quadrilateral shell element based on the degeneration from solid (full/selective/reduced integration) 9 nodes quadrilateral MITC element (locking-free)

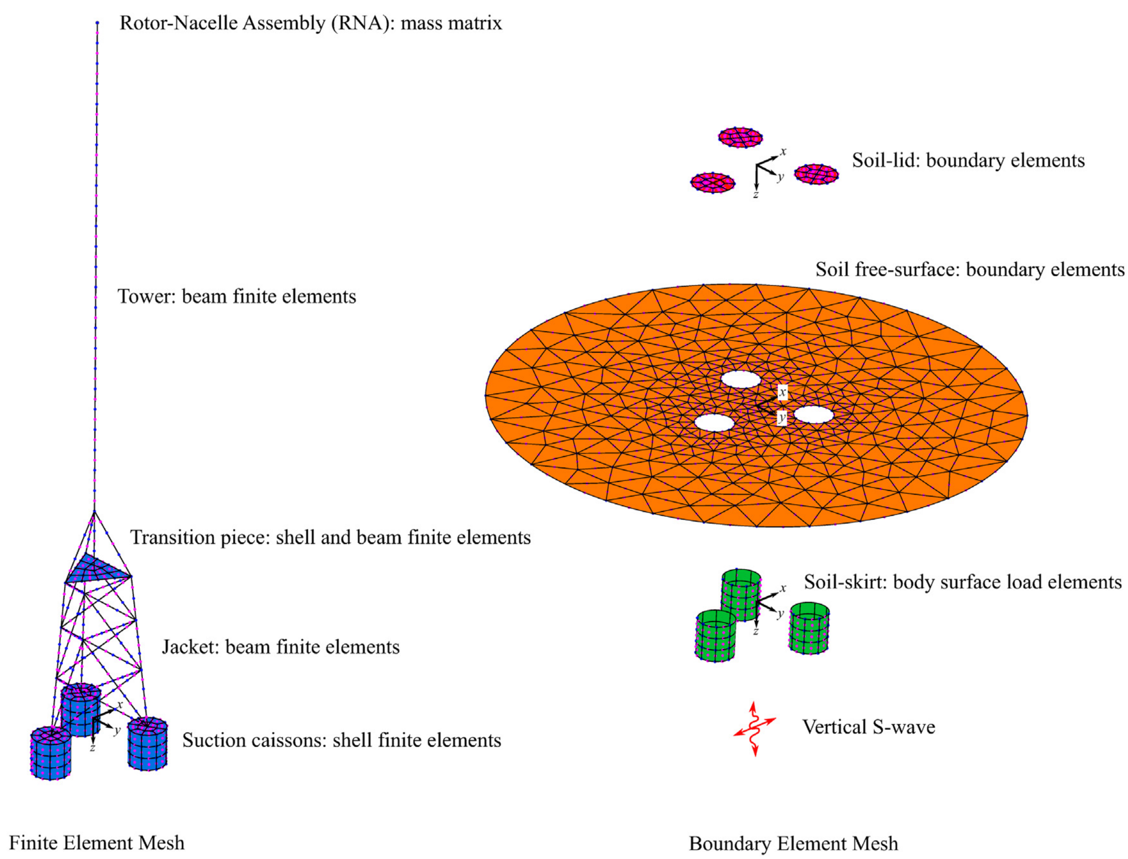


Fig. 4. Exploded view of a coupled model of finite elements and boundary elements for the dynamic analysis of an offshore wind turbine.

different types of couplings between them, including mixed-dimensional situations in which structural beams and shells are embedded in continuum regions. Inviscid fluid, viscoelastic and poroelastic regions can be modeled, together with bars, Euler–Bernoulli beams, Timoshenko beams and Reissner–Mindlin shells. Crack and stress concentration problems can also be addressed.

The code is especially suited to soil–structure, soil–fluid–structure and acoustic fluid–structure problems, and includes coupling features that are not available in common commercial software due to the advantages of the singular, hypersingular and dual boundary element methodologies in combination with the different coupling options with structural finite elements. The

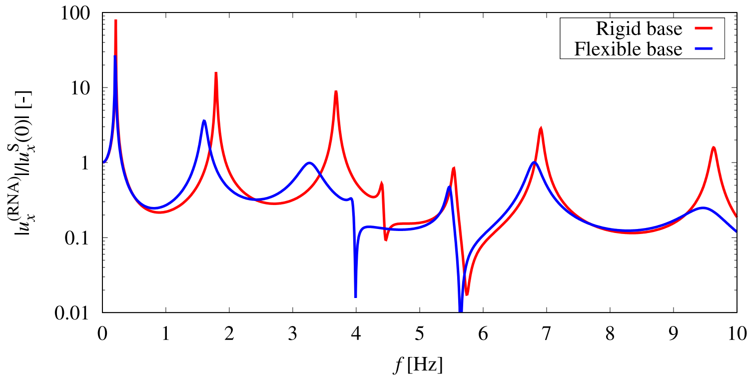
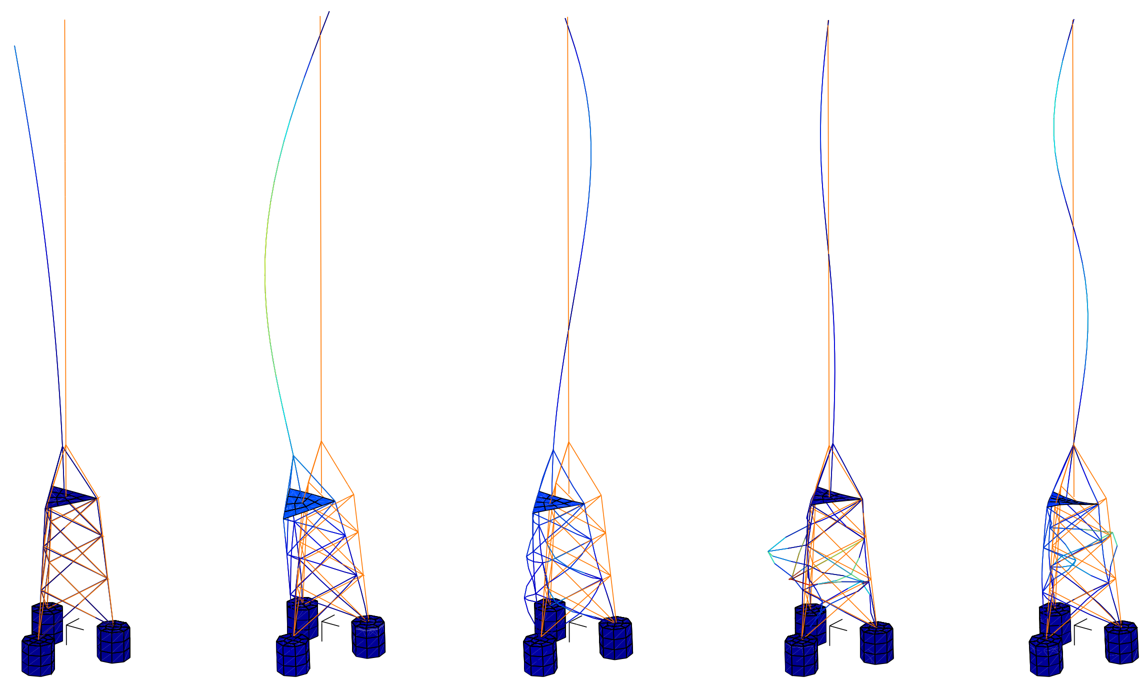


Fig. 5. Normalized horizontal displacement response at the tower top.



(a) $f = 0.2$ Hz (b) $f = 1.6$ Hz (c) $f = 3.26$ Hz (d) $f = 5.47$ Hz (e) $f = 6.81$ Hz

Fig. 6. Deformed shapes of the offshore wind turbine structure at different frequencies.

Table 2
Boundary elements implemented in MultiFEBE.

Two-dimensional plane strain problems
2, 3 or 4 line ordinary or crack boundary elements.
2, 3 or 4 line body load elements.
Three-dimensional problems
3 or 6 triangular and 4, 8 or 9 quadrilateral ordinary or crack boundary elements.
2, 3 or 4 line body load elements (only in elastic solid BE regions).
3 or 6 triangular and 4, 8 or 9 quadrilateral body load elements.

analysis of problems related to the dynamic response of caisson foundations or pipe piles in viscoelastic and poroelastic soils are some of the most relevant possible applications.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Data availability

I have shared the link to the code repository.

Acknowledgments

This work has been developed with the support of research projects: PID2020-120102RB-I00, funded by the Agencial Estatal de Investigación of Spain, MCIN/AEI/10.13039/ 501100011033; ProID2020010025, funded by Consejería de Economía, Conocimiento y Empleo (Agencia Canaria de la Investigación, Innovación y Sociedad de la Información) of the Gobierno de Canarias and FEDER; and BIA2017-88770-R, funded by Subdirección General de Proyectos de Investigación of the Ministerio de Economía y Competitividad (MINECO) of Spain and FEDER.

We would like to thank Ángel Gabriel Vega Rodríguez, who is giving momentum to the project by working on different parts of it.

References

[1] Galvín P, Romero A. A MATLAB toolbox for soil–structure interaction analysis with finite and boundary elements. Soil Dyn Earthq Eng 2014;57:10–4.
[2] Moore JL, Morgan NR, Horstemeyer MF. ELEMENTS: A high-order finite element library en C++. SoftwareX 2019;10(100257).
[3] Dölz J, Harbrecht H, Kurz S, Multerer M, Schöps S, Wolf F. Bembel: The fast isogeometric boundary element C++ library for Laplace, Helmholtz and electric wave equation. SoftwareX 2020;11(100476).
[4] François S, Schevenels M, Dooms D, Jansen M, Wambacq J, Lombaert G, et al. Stabil: An educational matlab toolbox for static and dynamic structural analysis. Comput Appl Eng Ed 2021;29(5):1372–89. http://dx.doi.org/10.1002/cae.22391.
[5] Kammer DS, Albertini G, Ke C-Y. UGUCA: A spectral-boundary-integral method for modeling fracture and friction. SoftwareX 2021;15(100785).
[6] Rahman MM, Nahar TT, Kim D. FeView: Finite element model (FEM) visualization and post-processing tool for OpenSees. SoftwareX 2021;15(100751).
[7] Chirino F, Maeso O, Aznárez JJ. Un modelo numérico para la estimación de la eficiencia de pantallas acústicas. Anales de IngenierÍa MecÁnica 1998;4:33–9.
[8] Maeso O, Domínguez J. Earthquake analysis of arch dams. I: Dam-foundation interaction. J Eng Mech 1993;119:496–512. http://dx.doi.org/10.1061/(ASCE)0733-9399(1993)119:3(496).
[9] Domínguez J, Maeso O. Earthquake analysis of arch dams. II: Dam-water-foundation interaction. J Eng Mech 1993;119:513–30. http://dx.doi.org/10.1061/(ASCE)0733-9399(1993)119:3(513).
[10] Maeso O, Aznárez JJ, Domínguez J. Three-dimensional models of reservoir sediment and effects on seismic response of arch dams. Earthq Eng Struct Dyn 2004;33:1103–23. http://dx.doi.org/10.1002/eqe.392.
[11] Maeso O, Aznárez JJ, García F. Dynamic impedances of piles and groups of piles on saturated soils. Int J Comput Struct 2005;83:769–82. http://dx.doi.org/10.1016/j.compstruc.2004.10.015.
[12] Aznárez JJ, Maeso O, Domínguez J. BE analysis of bottom sediments in dynamic fluid-structure interaction problems. Eng Anal Bound Elem 2006;30:124–36. http://dx.doi.org/10.1016/j.enganabound.2005.10.002.
[13] Toledo R, Aznárez JJ, Maeso O, Greiner D. Optimization of thin noise barrier designs using evolutionary algorithms and a dual BEM formulation. J Sound Vib 2014;334:219–38. http://dx.doi.org/10.1016/j.jsv.2014.08.032.
[14] Padrón LA, Aznárez JJ, Maeso O. BEM-FEM coupling model for the dynamic analysis of piles and pile groups. Eng Anal Bound Elem 2007;31:473–84. http://dx.doi.org/10.1016/j.enganabound.2006.11.001.
[15] Padrón LA, Aznárez JJ, Maeso O. Dynamic structure-soil-structure interaction between nearby piled buildings under seismic excitation by BEM-FEM model. Soil Dyn Earthq Eng 2009;29:1084–96. http://dx.doi.org/10.1016/j.soildyn.2009.01.001.
[16] Padrón LA, Aznárez JJ, Maeso O. 3-D boundary element - finite element method for the dynamic analysis of piled buildings. Eng Anal Bound Elem 2011;35:465–77. http://dx.doi.org/10.1016/j.enganabound.2010.09.006.
[17] Álamo GM, Martínez-Castro AE, Padrón LA, Aznárez JJ, Gallego R, Maeso O. Efficient numerical model for the computation of impedance functions of inclined pile groups in layered soils. Eng Struct 2016;126:379–90. http://dx.doi.org/10.1016/j.engstruct.2016.07.047.
[18] Álamo GM, Padrón LA, Aznárez JJ, Maeso O. Numerical model for the dynamic and seismic analysis of pile-supported structures with a meshless integral representation of the layered soil. Bull Earthq Eng 2022;20:3215–38. http://dx.doi.org/10.1007/s10518-021-01287-7.
[19] Bordón JDR, Aznárez JJ, Maeso O. A 2D BEM-fem approach for time harmonic fluid-structure interaction analysis of thin elastic bodies. Eng Anal Bound Elem 2014;43:19–29. http://dx.doi.org/10.1016/j.enganabound.2014.03.004.
[20] Bordón JDR, Aznárez JJ, Maeso O. Two-dimensional numerical approach for the vibration isolation analysis of thin walled wave barriers in poroelastic soils. Comput Geotech 2016;71:168–79. http://dx.doi.org/10.1016/j.compgeo.2015.08.007.
[21] Bordón JDR, Aznárez JJ, Maeso O. Dynamic model of open shell structures buried in poroelastic soils. Comput Mech 2017;60:269–88. http://dx.doi.org/10.1007/s00466-017-1406-3.
[22] Bordón JDR, Aznárez JJ, Maeso O. Numerical integration scheme for singular integrals based on polar coordinates free from angular quasi–singularities. Eng Anal Bound Elem 2019;103:126–36. http://dx.doi.org/10.1016/j.enganabound.2019.03.010.
[23] Achenbach JD. Wave propagation in elastic solids. North-Holland; 1973.
[24] Morse PM. Vibration and sound. The American Institute of Physics; 1983.
[25] Biot MA. Theory of propagation of elastic waves in a fluid-saturated porous solid. I. Low-frequency range. J Acoust Soc Am 1956;28:168–78. http://dx.doi.org/10.1121/1.1908239.
[26] Bathe KJ. Finite Element Procedures. Prentice Hall; 1996.
[27] Oñate E. Structural analysis with the finite element method. linear statics. Volume 1. basis and solids. Springer-CIMNE; 2012.
[28] Oñate E. Structural analysis with the finite element method. linear statics. Volume 2. beams, plates and shells. Springer-CIMNE; 2013.
[29] Brebbia CA, Domínguez J. Boundary elements: an introductory course. WIT Press - Computational Mechanics Publications; 1989.
[30] Domínguez J. Boundary elements in dynamics. WIT Press; 1993.
[31] Chirino F, Domínguez J. Dynamic analysis of cracks using the boundary element method. Eng Fract Mech 1989;34:1051–61. http://dx.doi.org/10.1016/0013-7944(89)90266-X.
[32] Sáez A, Gallego R, Domínguez J. Hypersingular quarter-point boundary elements for crack problems. Internat J Numer Methods Engrg 1995;38:1681–701. http://dx.doi.org/10.1002/nme.1620381006.
[33] Domínguez J, Ariza MP, Gallego R. Flux and traction boundary elements without hypersingular or strongly singular integrals. Internat J Numer Methods Engrg 2000;48:111–35. http://dx.doi.org/10.1002/(SICI)1097-0207(20000510)48:13.3.CO;2-P.
[34] Zienkiewicz OC, Taylor RL, Zhu JZ. The finite element method: its basis and fundamentals. Elsevier; 2013.
[35] Bordón JDR. Coupled model of finite elements and boundary elements for the dynamic analysis of buried shell structures (Ph.D. thesis), Universidad de Las Palmas de Gran Canaria; 2017, URL http://hdl.handle.net/10553/41789.
[36] Geuzaine C, Remacle J-F. Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities. Internat J Numer Methods Engrg 2009;79:1309–31. http://dx.doi.org/10.1002/nme.2579.
[37] Bak C, Zahle F, Bitsche R, Kim T, Yde A, Henriksen LC, et al. Description of the DTU 10 MW reference wind turbine. techreport DTU Wind Energy Report-I-0092, DTU Wind Energy; 2013.