MultiFEBE

Tutorial 12: harmonic analysis of an arch dam. Compliant base model for a 112 m water depth

J.C. Galván and L.A. Padrón

May 2023

|  |  |
| --- | --- |
|  |  |
| (a) | |
|  |  |
| (b) | |
| Figure 1: 3D mesh used for the BEM analysis. Dam wall embedded in a prismatic canyon with an extension of the free-field discretization of 240 m and a 112 m water depth. (a) Fluid region modeled as half space domain. (b) Fluid region modeled as regular full space domain. | |

**1 Problem description**

In this twelfth tutorial, a harmonic analysis of an arch dam is performed using the Boundary Element Method (BEM) where the dam wall is embedded in a prismatic canyon with an extension of 240 m and 112 m water depth behind the dam (Fig. 1). Here, two cases will be presented; in the first one, the fluid region is modeled as half space domain (Fig. 1.a), while in the second one, the fluid region is modeled as regular full space domain (Fig. 1.b). The models were impinged by seismic time-harmonic plane waves. For this analysis, it was assumed that the incident wave field consists solely of plane SH waves propagating vertically with a horizontal direction of the *y*-axis free-field ground surface motion (Fig. 2.a) in order to calculate the frequency response function (FRF) at a node located at the midpoint of the dam crest (Fig. 2.b).

|  |
| --- |
| Direction of the  *y*-axis ground motion  Plane SH waves  (a)  Study node    (b) |
| Figure 2: (a) Excitation along the *y*-axis direction. (b) Study node |

The concrete dam wall and the foundation rock material are assumed to be viscoelastic while the water domain is represented as an inviscid fluid under small amplitude motions. Table 1 presents the material properties considered for each one of the regions [1, 2].

|  |  |  |  |
| --- | --- | --- | --- |
| **Property** | **Dam concrete** | **Foundation rock** | **Water** |
| Young’s modulus (elastic modulus), *E*(MPa) | 19599.92 | 28999.99 | - |
| Mass density, *ρ* (kg/m3) | 2300 | 2143 | 1000 |
| Poisson’s ratio, *υ* | 0.2 | 0.2 | - |
| Internal damping ratio, *ξ* | 0.01 | 0.01 | - |
| Pressure wave velocity (m/s) | - | - | 1438 |

Table 1: Material properties

**2. Pre-processing**

In the same way as in tenth and eleventh tutorials, the mesh is generated from the GiD program and the BE meshes consist of nine-node quadratic quadrilateral elements and six-node quadratic triangular elements.

**2.1 Mesh generation with GiD**

**2.1.1. Fluid region modeled as half space domain**

In this first case, twelve layers (or boundaires) are defined, one per contour (Fig. 3, layer name in red box).

|  |  |
| --- | --- |
| Crest    Left\_abutment  Downstream\_face  Right\_abutment | Upstream\_face    Upstream\_water |
| Foundation\_rock  Foundation  Right\_abutment  Foundation  Left\_abutment  Downstream | |
| Foundation\_water  Left\_water\_face  Right\_water\_face  water\_face\_boundary\_condition | |
| Figure 3: Layers (or boundaires) used for the BEM analysis. Fluid region modeled as half space domain | |

**2.1.2. Fluid region modeled as regular full space domain**

In the second case, thirteen layers (or boundaires) are defined, one per contour (Fig. 4, layer name in red box).

|  |  |
| --- | --- |
| Crest    Downstream\_face  Right\_abutment | Upstream\_face    Upstream\_water |
| Foundation  Left\_abutment  Foundation\_rock  Right\_abutment  Foundation  Left\_abutment  Downstream | |
| Upstream\_water  Right\_water\_face  top\_water | Left\_water\_face  water\_face\_boundary\_condition    Foundation\_water |
| Figure 4: Layers (or boundaires) used for the BEM analysis. Fluid region modeled as regular full space domain | |

It is worth noting that in both case studies, the nodes common to different contours will be duplicated and they will have different numbering.

Once the mesh is generated, it is written in GiD native MultiFEBE format using the implemented template file \*.bas shown in the MultiFEBE Reference Manual (Appendix C). Finally, the file generated contains nodes, elements and parts of the mesh which, in this example, will be copied and pasted in the input data file.

* 1. **Input data file**

**2.2.1. Fluid region modeled as half space domain**

As mentioned in another tutorials, the first part is the problem definition in the section [problem]. This example is a 3D harmonic mechanical problem.

[problem]

n = 3D

type = mechanics

analysis = harmonic

Then, a list of frequencies is generated by specifying the number of frequencies. It has been defined an analysis of 154 frequencies, from 0.01 Hz to 6.9 Hz.

[frequencies]

Hz

list

154

0.01

0.1

0.2

.

.

.

6.7

6.8

6.9

The mesh is going to be read from the same input file, so sections [nodes], [elements] and [parts] must be written in the script.

[nodes]

8998

1 6.3999999801429453e+01 -2.1691687371202244e+02 1.2000000000000000e+02

2 6.2291666476369890e+01 -2.1703672055522847e+02 1.1499999999999999e+02.

.

.

.

8997 -8.7797583250000002e+01 2.0085721969122281e+02 1.1800000000000000e+02

8998 -8.8844999999999999e+01 2.0085721958829708e+02 1.2000000000000000e+02

[elements]

2151

288 tri6 1 1 2634 2551 2606 2584 2573 2621

290 tri6 1 1 2508 2484 2568 2498 2519 2541

.

.

.

2113 quad9 1 7 1804 2074 2078 1815 1924 2077 1942 1810 1930

2114 quad9 1 7 1796 2075 2074 1804 1919 2071 1924 1799 1923

[parts]

12

1 Downstream\_face

2 Upstream\_face

3 Left\_abutment

4 Right\_abutment

5 Foundation

6 Crest

7 Foundation\_rock

8 water\_face\_boundary\_condition

9 Foundation\_water

10 Left\_water\_face

11 Right\_water\_face

12 Upstream\_water

In this example the nodal solutions will be exported. For it, the section [export] takes the form

[export]

complex\_notation = cartesian

nso\_nodes = 1 3107

where the complex notation is set as cartesian and the result for specific node is taking by specifying the number of node (1) and the identifier of the node (3107).

As the problem has three materials, the section [materials] will need four lines: a first line for the number of materials in the model and a line per material with their properties such as tag, type, *E*, *ρ*, *ν*, *ξ* and *c*.

[materials]

3

1 elastic\_solid E 19599921600. rho 2300. nu 0.2 xi 0.01

2 elastic\_solid E 28999999999.92 rho 2142.85 nu 0.2 xi 0.01

3 fluid c 1438.6 rho 1000.

In the next section [boundaries], it is necessary to specify the number of boundaries in the first line, and a line per boundary by indicating the boundary identifier, the identifier of the part that discretize it, and finally the boundary class. In this example there are 12 boundaries: boundary 1 is the part 1 of the mesh, boundary 2 the part 2, boundary 3 the part 3, boundary 4 the part 4, and so on.

[boundaries]

12

1 1 ordinary

2 2 ordinary

3 3 ordinary

4 4 ordinary

5 5 ordinary

6 6 ordinary

7 7 ordinary

8 8 ordinary

9 9 ordinary

10 10 ordinary

11 11 ordinary

12 12 ordinary

In this example, in the section [bem formulation over boundaries], the collocation strategy in five boundaries will be nodal, while in boundaries 5, 7, 8, 9, 10, 11 and 12 a non-nodal collocation strategy is prefered for all the nodes along its boundaries, with a displacement towards inside each element of 1% the width of the element, this is:

[bem formulation over boundaries]

boundary 1: sbie

boundary 2: sbie

boundary 3: sbie

boundary 4: sbie

boundary 5: sbie\_boundary\_mca 0.01

boundary 6: sbie

boundary 7: sbie\_boundary\_mca 0.01

boundary 8: sbie\_boundary\_mca 0.01

boundary 9: sbie\_boundary\_mca 0.01

boundary 10: sbie\_boundary\_mca 0.01

boundary 11: sbie\_boundary\_mca 0.01

boundary 12: sbie\_boundary\_mca 0.01

The format of the [regions] section consists of a first line indicating the number of regions, 3 in this case. For each region there is a block of data consisting of several lines of data. The first line of each block is the region identifier and the region class, it is, 1 be and 2 be, for the first and second region, respectively; for the third region, it takes de form: 3 be half-space 3 112 0 where half-space means that the fluid region is modeled as half space domain while 112 m reffers to the water depth. As the regions are a BE regions, the second line indicates the number of boundaries and a list of boundaries, with their orientation signs (7 1 2 3 4 5 6 12, for the first region; 7 -3 -4 -5 7 -9 -10 -11, for the second region, and 5 8 9 10 11 -12 for the third region). The third line of each block defines the material while the fourth line defines the number and a list of BE body loads (0 in this case). Finally, the fifth line defines the number and a list of incident fields (0 for the first block; 1 1 for the second block). The format of the section is:

[regions]

3

1 be

7 1 2 3 4 5 6 12

material 1

0

0

2 be

7 -3 -4 -5 7 -9 -10 -11

material 2

0

1 1

3 be half-space 3 112 0

5 8 9 10 11 -12

material 3

0

0

Now, in the section [incident waves] the incoming waves are defined. The general format has a first line for the number of waves (1), a second line for the wave identifier (1), a third line for the wave class (plane), a fourth line for the space (half-space with *np* = 3 (3*D* case), *xp* = 120.0 (foundation rock height), *bc* = 1 (3*D* case)), a fifth line for the variable (0 for displacement), the amplitude (1.,0.), the reference point (*x*0(1) = 0., *x*0(2) = 0., *x*0(3) = 0.) and the angles (*varphi* = -90., *theta* = 90.), a sixth line for symmetry options (*xs*(1) = 0., *xs*(2) = 0., *xs*(3) = 0., *symconf*(1) = 0., *symconf*(2) = 0., *symconf*(3) = 0.) and a seventh line for the region type (viscoelastic) and the wave type (sh). So, the format of the section takes the form:

[incident waves]

1

1

plane

half-space 3 120.0 1

0 (1.,0.) 0. 0. 0. -90. 90.

0. 0. 0. 0. 0. 0.

viscoelastic sh

In these kind of problems, the far-field reservoir is usually simplified by specific boundary conditions set at a truncated boundary placed far enough from the points of interest in the study (Galván et al. [3]). Sommerfeld radiation condition (Galván et al. [3]) will be applied at the truncated boundary (eight boundary, called water\_face\_boundary\_condition). This is an uncoupled equoation for each node of the truncated boundary which is incorporated in the MultiFEBE code. So section [conditions over be boundaries] takes the form:

[conditions over be boundaries]

boundary 8: 2

**2.2.2. Fluid region modeled as regular full space domain**

As mentioned, the first part is the problem definition in the section [problem]. This example is a 3D harmonic mechanical problem.

[problem]

n = 3D

type = mechanics

analysis = harmonic

Then, a list of frequencies is generated by specifying the number of frequencies. It has been defined an analysis of 154 frequencies, from 0.01 Hz to 6.9 Hz.

[frequencies]

Hz

list

154

0.01

0.1

0.2

.

.

.

6.7

6.8

6.9

The mesh is going to be read from the same input file, so sections [nodes], [elements] and [parts] must be written in the script.

[nodes]

9348

1 6.3999999801429453e+01 -2.1691687371202244e+02 1.2000000000000000e+02

2 6.2291666476369890e+01 -2.1703672055522847e+02 1.1499999999999999e+02.

.

.

.

9347 -8.7797583250000002e+01 2.0085721969122281e+02 1.1800000000000000e+02

9348 -8.8844999999999999e+01 2.0085721958829708e+02 1.2000000000000000e+02

[elements]

2237

288 tri6 1 1 2665 2581 2638 2614 2603 2652

290 tri6 1 1 2537 2514 2599 2527 2549 2571

.

.

.

2234 quad9 1 13 8341 8396 8138 8077 8376 8268 8109 8208 8243

2235 quad9 1 13 8642 8738 8396 8341 8689 8523 8376 8477 8505

[parts]

13

1 Downstream\_face

2 Upstream\_face

3 Left\_abutment

4 Right\_abutment

5 Foundation

6 Crest

7 Foundation\_rock

8 water\_face\_boundary\_condition

9 Foundation\_water

10 Left\_water\_face

11 Right\_water\_face

12 Upstream\_water

13 top\_water

As mentioned before, the nodal solutions will be exported. For it, the section [export] takes the form

[export]

complex\_notation = cartesian

nso\_nodes = 1 3146

where the complex notation is set as cartesian and the result for specific node is taking by specifying the number of node (1) and the identifier of the node (3146).

As the problem has three materials, the section [materials] will need four lines: a first line for the number of materials in the model and a line per material with their properties such as tag, type, *E*, *ρ*, *ν*, *ξ* and *c*.

[materials]

3

1 elastic\_solid E 19599921600. rho 2300. nu 0.2 xi 0.01

2 elastic\_solid E 28999999999.92 rho 2142.85 nu 0.2 xi 0.01

3 fluid c 1438.6 rho 1000.

In the next section [boundaries], it is necessary to specify the number of boundaries in the first line, and a line per boundary by indicating the boundary identifier, the identifier of the part that discretize it, and finally the boundary class. In this example there are 13 boundaries: boundary 1 is the part 1 of the mesh, boundary 2 the part 2, boundary 3 the part 3, boundary 4 the part 4, and so on.

[boundaries]

13

1 1 ordinary

2 2 ordinary

3 3 ordinary

4 4 ordinary

5 5 ordinary

6 6 ordinary

7 7 ordinary

8 8 ordinary

9 9 ordinary

10 10 ordinary

11 11 ordinary

12 12 ordinary

13 13 ordinary

In this example, in the section [bem formulation over boundaries], the collocation strategy in five boundaries will be nodal, while in boundaries 5, 7, 8, 9, 10, 11 and 12 a non-nodal collocation strategy is prefered for all the nodes along its boundaries, with a displacement towards inside each element of 1% the width of the element, this is:

[bem formulation over boundaries]

boundary 1: sbie

boundary 2: sbie

boundary 3: sbie

boundary 4: sbie

boundary 5: sbie\_boundary\_mca 0.01

boundary 6: sbie

boundary 7: sbie\_boundary\_mca 0.01

boundary 8: sbie\_boundary\_mca 0.01

boundary 9: sbie\_boundary\_mca 0.01

boundary 10: sbie\_boundary\_mca 0.01

boundary 11: sbie\_boundary\_mca 0.01

boundary 12: sbie\_boundary\_mca 0.01

boundary 13: sbie

The format of the [regions] section takes of the another case format. The second line indicates the number of boundaries and a list of boundaries, with their orientation signs being for the first region: 7 1 2 3 4 5 6 12; for the second region: 7 -3 -4 -5 7 -9 -10 -11, and for the third region: 5 8 9 10 11 -12 13.

[regions]

3

1 be

7 1 2 3 4 5 6 12

material 1

0

0

2 be

7 -3 -4 -5 7 -9 -10 -11

material 2

0

1 1

3 be

6 8 9 10 11 -12 13

material 3

0

0

Then, in the section [incident waves], the format of this section takes, also, of the another case format.

[incident waves]

1

1

plane

half-space 3 120.0 1

0 (1.,0.) 0. 0. 0. -90. 90.

0. 0. 0. 0. 0. 0.

viscoelastic sh

As mentioned before, Sommerfeld radiation condition will be applied at the truncated boundary (eight boundary, called water\_face\_boundary\_condition). This is an uncoupled equoation for each node of the truncated boundary and it is incorporated in the MultiFEBE code. By other hand, the pressure of the nodes belonging to the thirteen boundary (top\_water) is equal to zero and it must be specified in the input file, so section [conditions over be boundaries] takes the form:

[conditions over be boundaries]

boundary 8: 2

boundary 13: 0 (0.,0.)

1. **Results and discusión**

**3.1 Nodal solutions file (\*.nso)**

The FRFs of the study node computed with the harmonic analysis using two different softwares (Aznárez et al. [4] and MultiFEBE) are plotted in figure 5. As mentioned, the system excitation consists of the *y*-axis free-field ground surface motion (Fig. 2 (a)).

Firstly, in figure 5 it can be seen a very good agreement between the softwares employed. By other hand, figure 6 shows the real (a) and imaginary (b) parts of the FRFs of the study node where it can be also observed a very good agreement between the softwares employed.

|  |
| --- |
|  |
| Figure 5: FRFs. Transversal response of the midpoint of the dam crest |

|  |  |
| --- | --- |
| (a) | (b) |
| Figure 6: Real (a) and imaginary (b) parts of the FRFs of the study node | |

**3.2 Gmsh results file (\*.pos)**

In this section, the dam wall second and third mode shapes of vibration, this is, the first two symmetrical mode shapes of vibration will be plotted using the Gmsh software with the output file obtained from MultiFEBE code following the steps showed in tenth tutorial. It should be noted that the frequencies corresponding to the mode shapes are obtained from the maximum peaks of the imaginary part of FRF (Fig. 6.b)

Ç

|  |  |
| --- | --- |
| (a) | (b) |
| Figure 8: Second mode shape of vibration (3.39 Hz). (a) Fluid region modeled as half space domain, (b) Fluid region modeled as regular full space domain. | |

|  |  |
| --- | --- |
| (a) | (b) |
| Figure 9: Third mode shape of vibration (5.25 Hz). (a) Fluid region modeled as half space domain, (b) Fluid region modeled as regular full space domain. | |

**References**

[1] Documento XYZT. Presa de Soria. Tech. rep., Dirección General de Obras Hidráulicas. Ministerio de Obras Públicas y Transportes; 1991.

[2] Infraestructura de datos espaciales de canarias. Modelo de terreno LIDAR. 2017, <http://www.idecanarias.es>.

[3] J. Galván, L. Padrón, J. Aznárez, O. Maeso, Boundary element model for the analysis of the dynamic response of the Soria arch dam and experimental validation from ambient vibration tests. Engineering Analysis with Boundary Elements, 2022, 144: 67–80

[4] J. J. Aznárez, O. Maeso, and J. Domínguez. BE analysis of bottom sediments in dynamic fluid-structure interaction problems. Engineering Analysis with Boundary Elements, 2006, 30:124–136