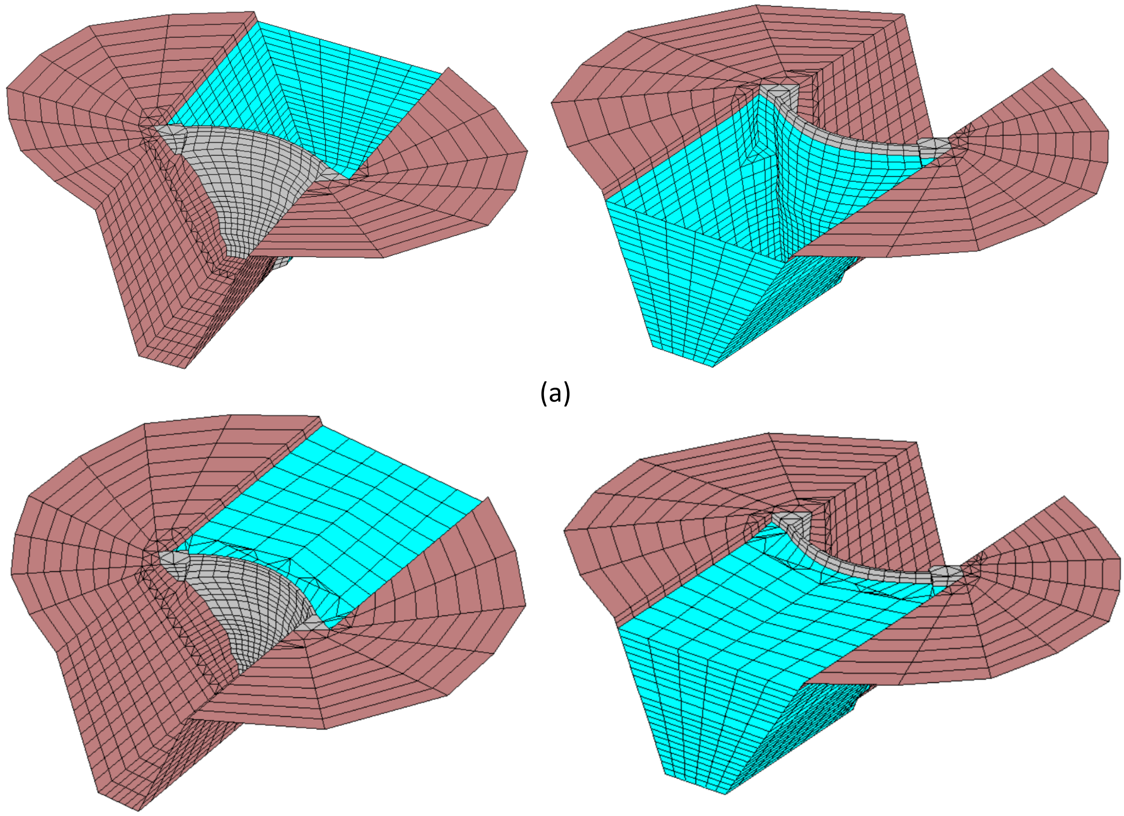
MultiFEBE
ME-TH-CO-006 [TUTORIAL]
Soria arch dam: compliant base model with a 112 m water depth

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).

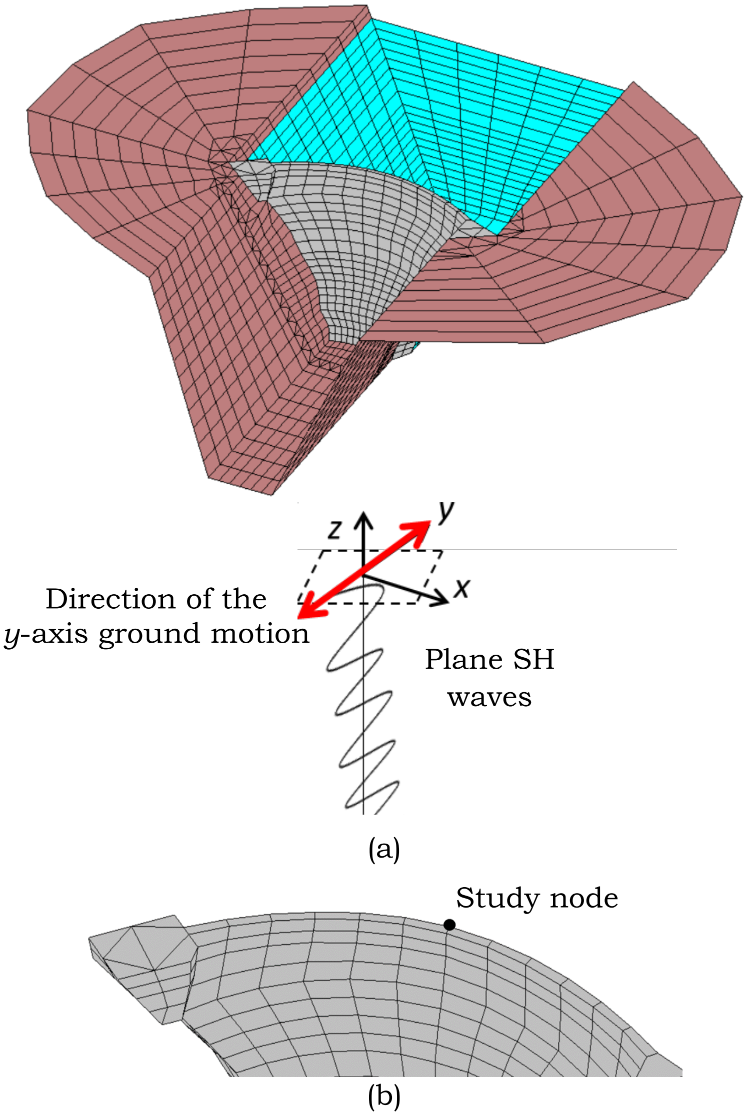


Figure 2: (a) Excitation along the y-axis direction. (b) Study node

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/m^3)	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).

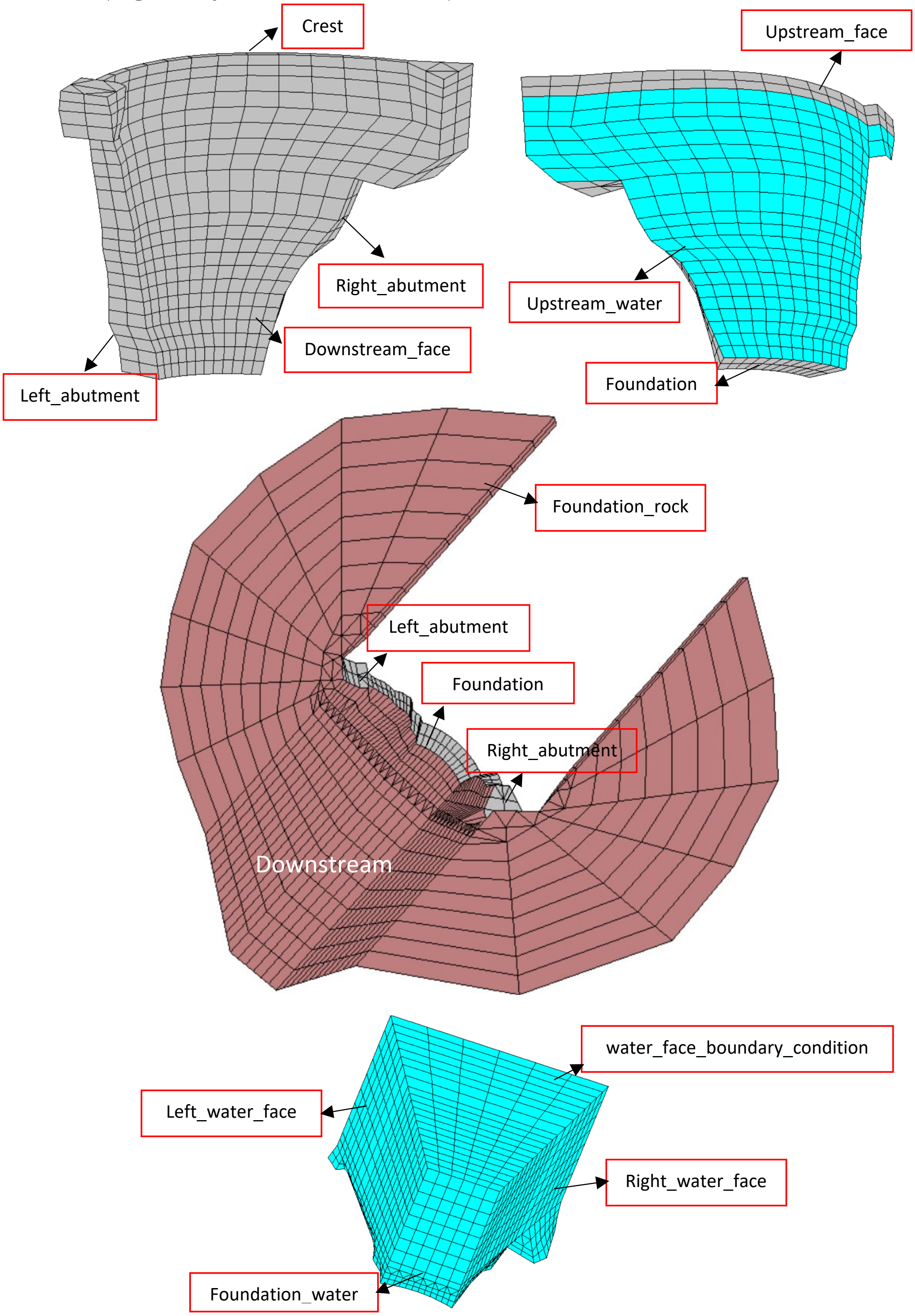


Figure 3: Layers (or boundaires) used for the BEM analysis. Fluid region modeled as half space domain

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).

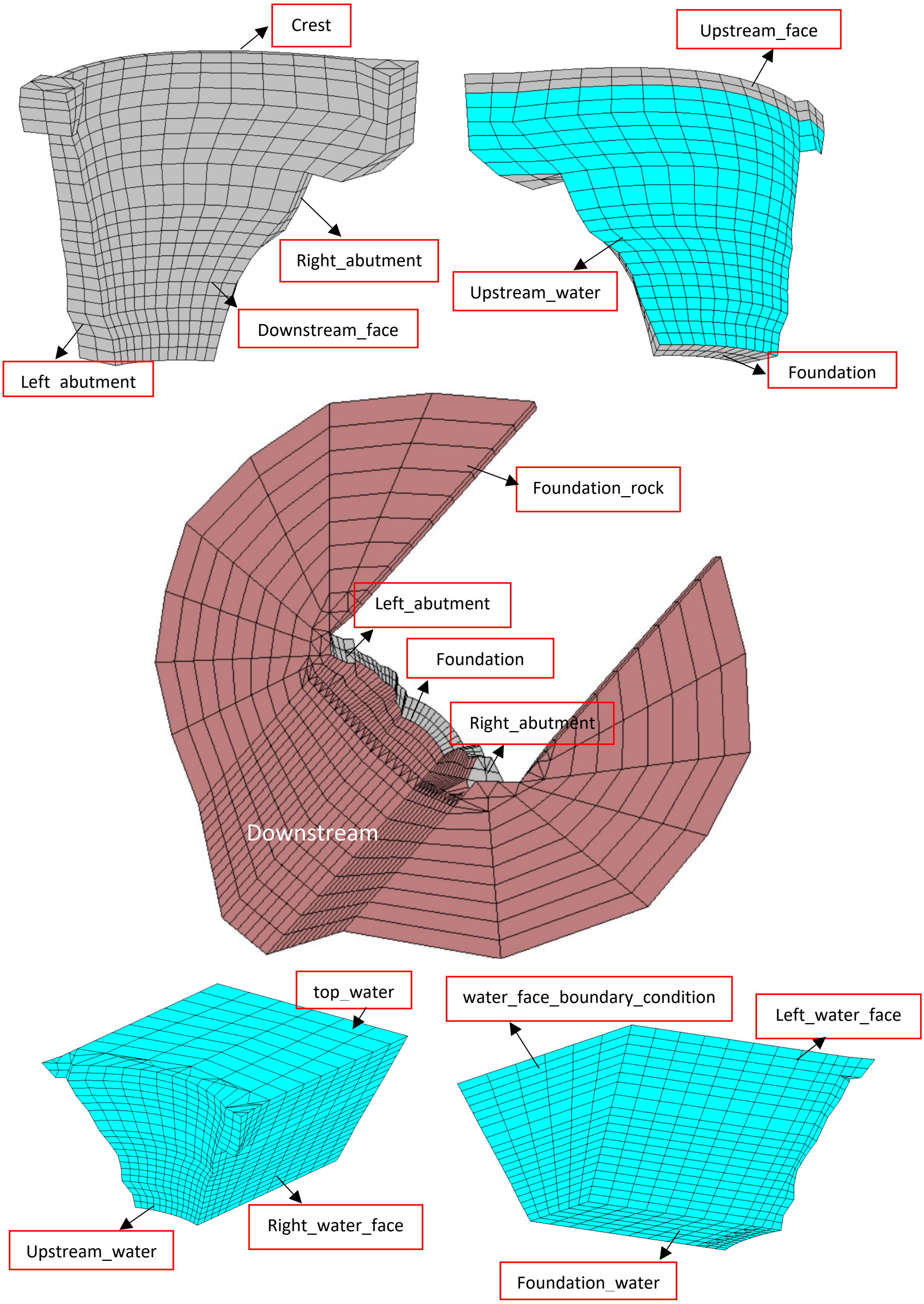


Figure 4: Layers (or boundaires) used for the BEM analysis. Fluid region modeled as regular full space domain

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.

2.2 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 ($3D$ case), xp = 120.0 (foundation rock height), bc = 1 ($3D$ case)), a fifth line for the variable (0 for displacement), the amplitude (1.,0.), the reference point ($x0(1)$ = 0., $x0(2)$ = 0., $x0(3)$ = 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, υ, 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.)
```

3 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.

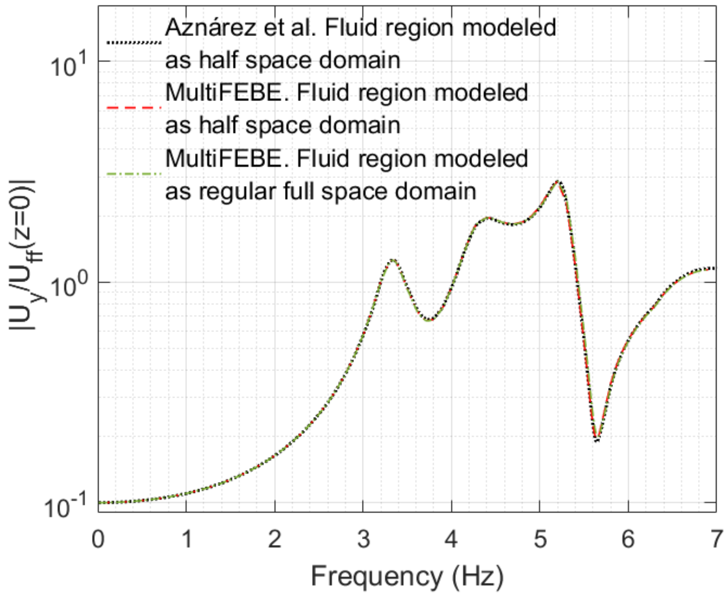


Figure 5: FRFs. Transversal response of the midpoint of the dam crest

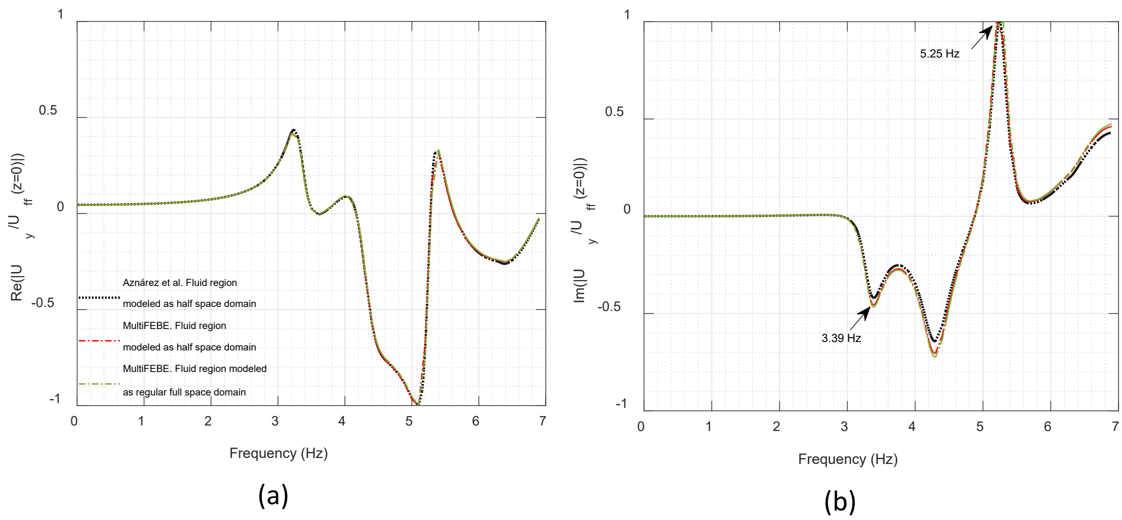


Figure 6: Real (a) and imaginary (b) parts of the FRFs of the study node

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)

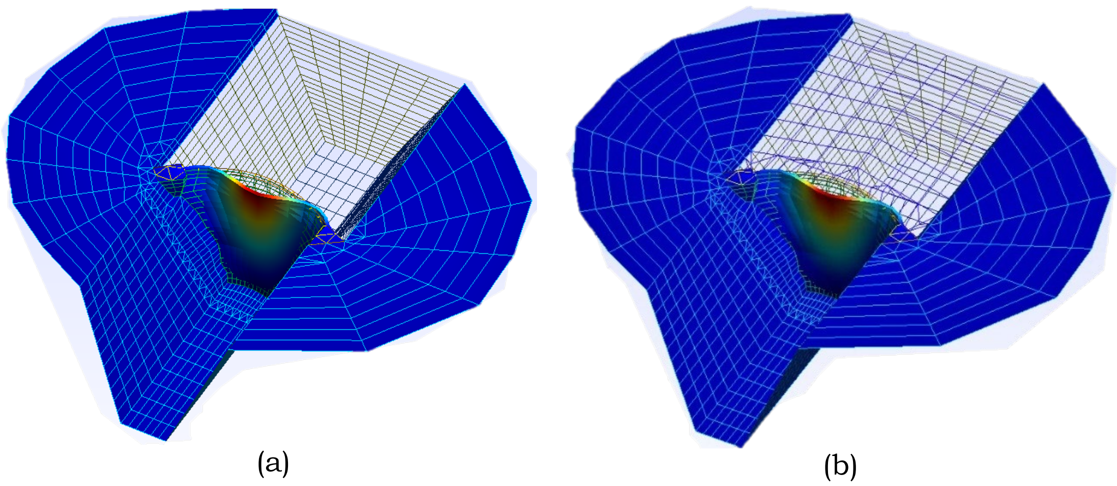


Figure 8: Second mode shape of vibration (3.39 Hz). (a) Fluid region modeled as half space domain, (b) Fluid region modeled as regular full space domain.

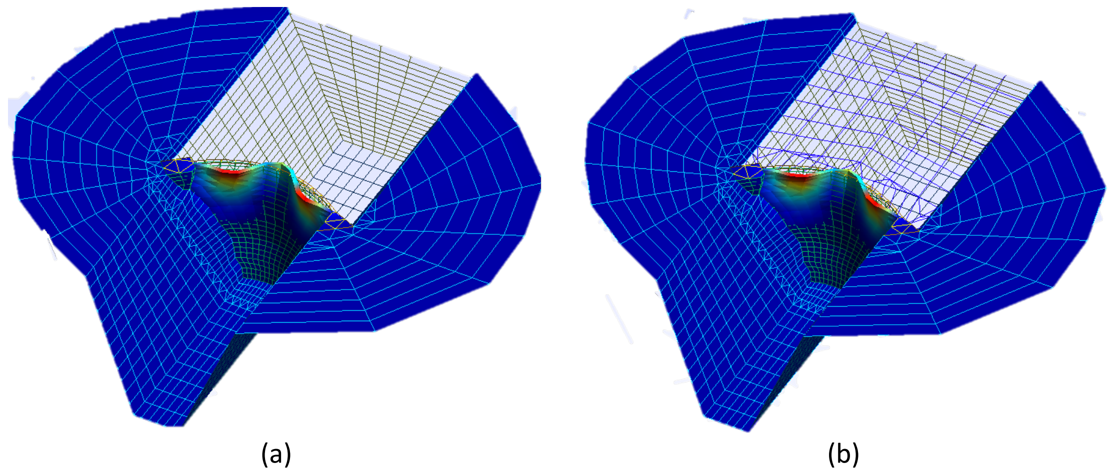


Figure 9: Third mode shape of vibration (5.25 Hz). (a) Fluid region modeled as half space domain, (b) Fluid region modeled as regular full space domain.

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