MultiFEBE

ME-TH-EL-002 [TUTORIAL]
Cantilever wall (BEM model)

J.C. Galván, L.A. Padrón and Jacob D. R. Bordón

July 2023

1 Problem description

In this BEM model tutorial, a harmonic analysis of an elastic cantilever wall is performed using the Boundary Element Method (BEM). Figure 1 shows the boundary element mesh used for this analysis. The model is subjected to a unit harmonic transversal displacement along the "y" direction at the base of the cantilever wall in order to calculate the frequency response function (FRF) at a node located at the midpoint of the top face (Fig. 1).

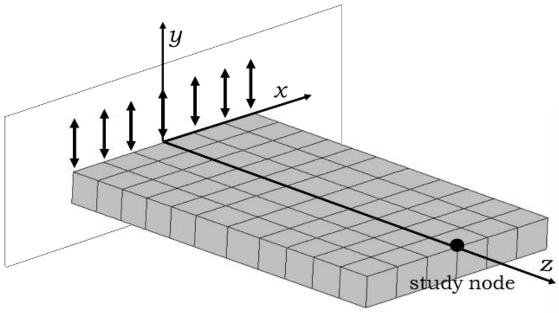


Figure 1: Model of a cantilever wall subjected to harmonic vibration at the base. Study node

Figure 2.a shows the cantilever wall geometry. The concrete cantilever wall material is assumed to be viscoelastic with the properties shown in Table 1.

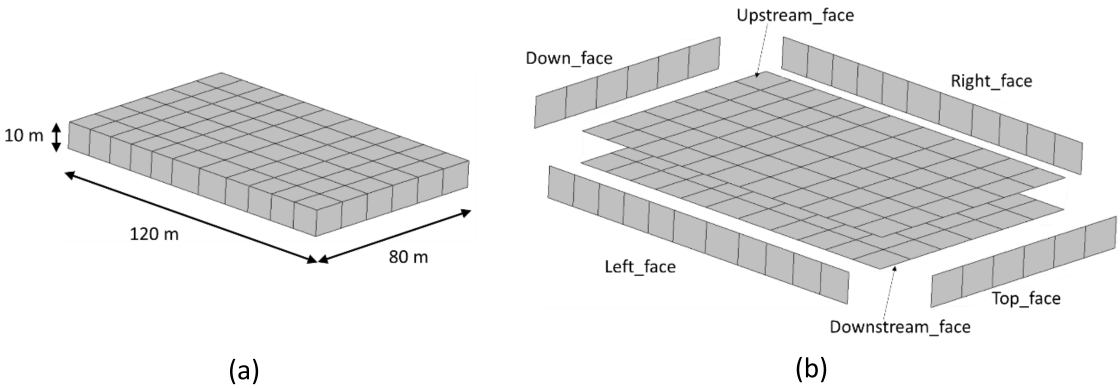


Figure 2: Cantilever wall geometry (a) and layers (or boundaires) (b)

Property	Dam concrete
Young's modulus (elastic modulus), E (MPa)	19599.92
Mass density, ρ (kg/m^3)	2300
Poisson's ratio, υ	0.2
Internal damping ratio, ξ	0.01

Table 1: Material properties

On the other hand, the first three analytical natural frequencies to this problem are obtained by means of the following equations [1]:

$$f_1 = \frac{1.875^2}{2\pi l^2}\sqrt{\frac{E\cdot I}{\rho\cdot A}} \quad (Hz)$$
$$f_2 = \frac{4.694^2}{2\pi l^2}\sqrt{\frac{E\cdot I}{\rho\cdot A}} \quad (Hz) \qquad (1)$$
$$f_3 = \frac{7.855^2}{2\pi l^2}\sqrt{\frac{E\cdot I}{\rho\cdot A}} \quad (Hz)$$

where I is the cross section moment of inertia, l is the cantilever wall lenght and A is the cross section. So, according to the geometry (Fig. 2.a) and material properties (table 1), the first three analytical natural frequencies of the cantilever wall are:

$$f_1 = 0.327\ Hz$$
$$f_2 = 2.050\ Hz \qquad (2)$$
$$f_3 = 5.750\ Hz$$

2. Pre-processing

In this case, the mesh is generated from the GiD program.

2.1 Mesh generation with GiD

GiD is a pre and post processor for numerical simulations. This software allows to create geometries going from the most basic elements (points) to the highest order elements (volumes) and to define parameters to use later in the MultiFEBE script. In the ME-TH-EL-003 [TUTORIAL], it will be explained, step by step, the way to obtain the mesh of a complex structure.

In this example, the generated mesh from the GiD program will be copied and pasted in the input data file.

2.2 Input data file

First, in section [problem], the type of problem and the type of analysis to be performed is defined. This example is a 3D mechanical problema under time harmonic analysis:

```
[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 103 frequencies, from 0.01 Hz to 6.9 Hz.

```
[frequencies]
Hz
list
103
0.01
0.1
0.2
0.25
.
.
.
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]
  814
 1  -4.0000000000000000e+01  -1.0000000000000000e+01   1.2000000000000000e+02
 2  -4.0000000000000000e+01  -1.0000000000000000e+01   1.2000000000000000e+02
                                   .
                                   .
                                   .
 813   4.0000000000000000e+01   0.0000000000000000e+00   0.0000000000000000e+00
 814   4.0000000000000000e+01   0.0000000000000000e+00   0.0000000000000000e+00

[elements]
  166
  1 quad9 1   1   29   13   35   42   18   20   38   31   24
  2 quad9 1   1   58   29   42   70   40   31   57   61   45
                                   .
                                   .
                                   .
  165 quad9 1   6   767   769   796   792   768   784   794   781   782
  166 quad9 1   6   792   796   812   807   794   805   810   802   804

[parts]
   6
   1 Upstream_face
   2 Downstream_face
   3 Left_face
   4 Right_face
   5 Top_face
   6 Down_face
```

In the section [export], several export and notation settings are defined. In this example, the nodal solutions will be exported. The complex notation is set as cartesian and the results for specific nodes are taking by specifying the number of nodes (1) and the identifiers of the nodes (131).

```
[export]
complex_notation = cartesian
nso_nodes = 1 131
```

As the problem has just one material, the section [materials] will need two lines: a first line for the number of materials in the model and a second line for the properties such as tag, type, E, ρ, ν and ξ.

```
[materials]
1
1 elastic_solid E 19599921600. rho 2300. nu 0.2 xi 0.01
```

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 6 boundaries (Fig. 2.b): boundary 1 is the part 1 of the mesh, boundary 2 the part 2, boundary 3 the part 3, boundary 4 the part 4, boundary 5 the part 5 and boundary 6 the part 6 and all of them are ordinary boundaries.

```
[boundaries]
6
1 1 ordinary
2 2 ordinary
3 3 ordinary
4 4 ordinary
5 5 ordinary
6 6 ordinary
```

The format of the [regions] section consists of a first line indicating the number of regions, 1 in this case. For each region there must be a block of data consisting of several lines of data. The second one is the region identifier and the region class (1 be). As the region is a BE region, the third line indicates the number of boundaries and a list of boundaries, with their orientation signs (6 1 2 3 4 5 6). The fourth line defines the material. The fifth line defines the number and a list of BE body loads (0) and the sixth line defines the number and a list of incident fields (0), beeing the format of the section:

```
[regions]
1
1 be
6 1 2 3 4 5 6
material 1
0
0
```

Now, in the section [conditions over be boundaries] all boundary conditions will be specify in global coordinates because they are planar and their normal vectors are parallel to one of the global axes. As a 3D problem, there are three lines for every boundary: a first line for the ”*x*” direction, the second one for the ”*y*” direction and the third one for the ”*z*”, where the first number of every line indicates the type of condition (here 0 for displacement and 1 for traction) and the second one its value in complex number because it is a harmonic analysis. In this case, a unit harmonic horizontal displacement along the ”*y*” direction was given at the down face of the cantilever wall. So, we have:

- At the left and right faces, boundary 3 and 4, respectively, displacements are equal to zero in the ”*x*” direction and tractions are equal to zero in the ”*y*” and ”*z*” directions.

- At the top face, boundary 5, tractions are equal to zero.

- As mentioned before, at the down face, a unit harmonic horizontal displacement along the ”*y*” direction was given, beeing the displacements along the ”*x*” and “*z*” directions equal to zero.

```
[conditions over be boundaries]

boundary 3:   0 (0.,0.)
              1 (0.,0.)
              1 (0.,0.)
boundary 4:   0 (0.,0.)
              1 (0.,0.)
              1 (0.,0.)
boundary 5:   1 (0.,0.)
              1 (0.,0.)
              1 (0.,0.)
boundary 6:   0 (0.,0.)
              0 (1.,0.)
              0 (0.,0.)
```

3 Results and discusión

3.1 Nodal solutions file (*.nso)

The FRFs of the study node can be obtained analytically from the equations above. This analytical solution, whose .m file called “cantilever_beam.m” is attached, it is compared against the results obtained from MultiFEBE (Fig. 3 and 4). As mentioned, the system excitation is defined as a harmonic uniform unitary displacement field along the down face at the ”*y*” direction (Fig. 1).

In figure 3, it can be seen, in the first two peaks, a very good agreement between the numerical and analytical solution. These two peaks correspond to the first two natural frequencies of the cantilever wall.

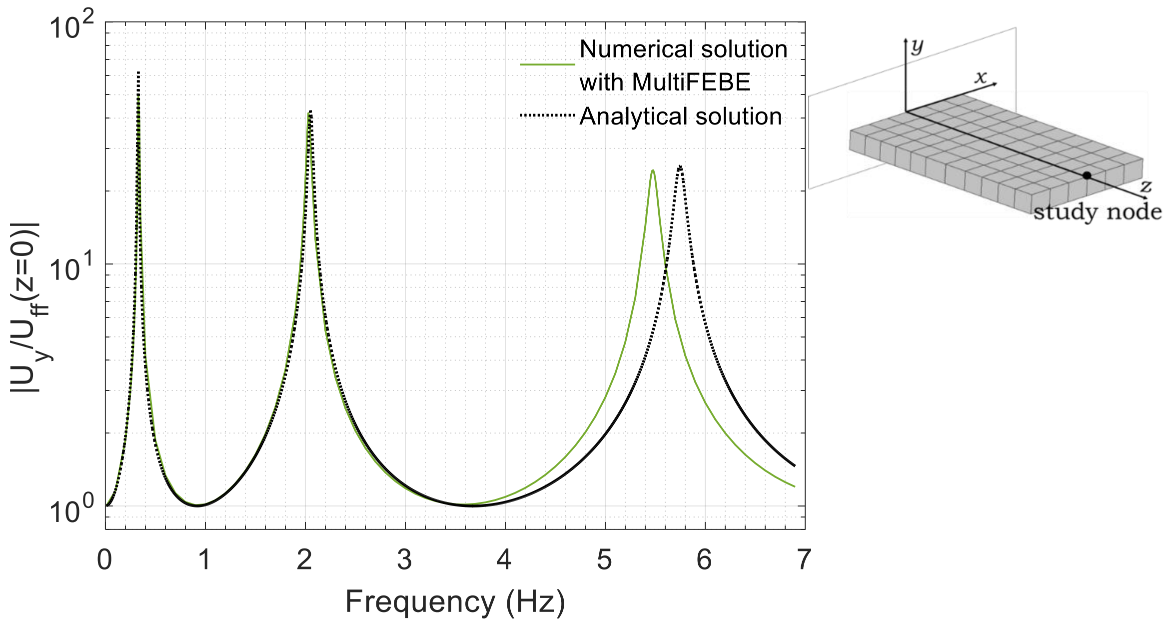


Figure 3: FRFs. Transversal response of the study node of the cantilever wall

By other hand, figure 4 shows the real (a) and imaginary (b) parts of the FRFs of the study node with the numerical and analytical solution. It can be also observed a very good agreement in the first two peaks.

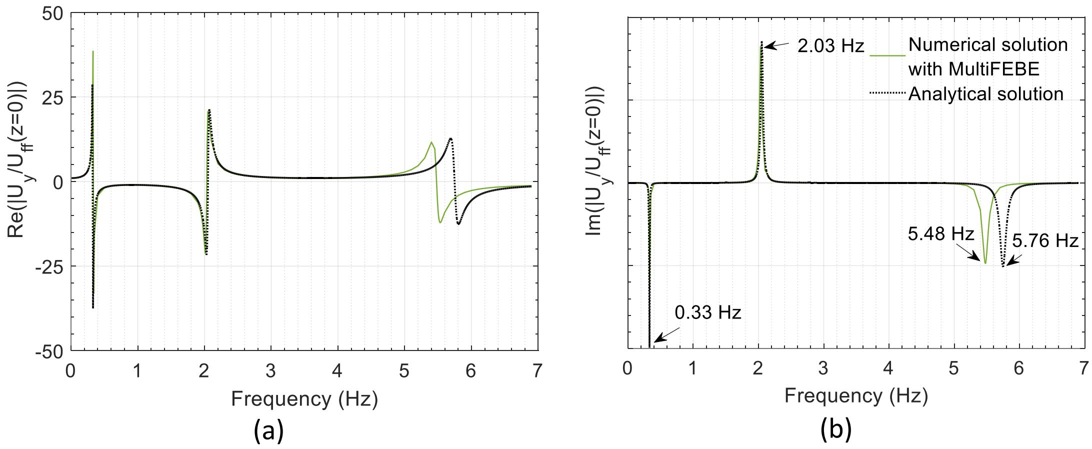


Figure 4: Real (a) and imaginary (b) parts of the FRFs of the study node

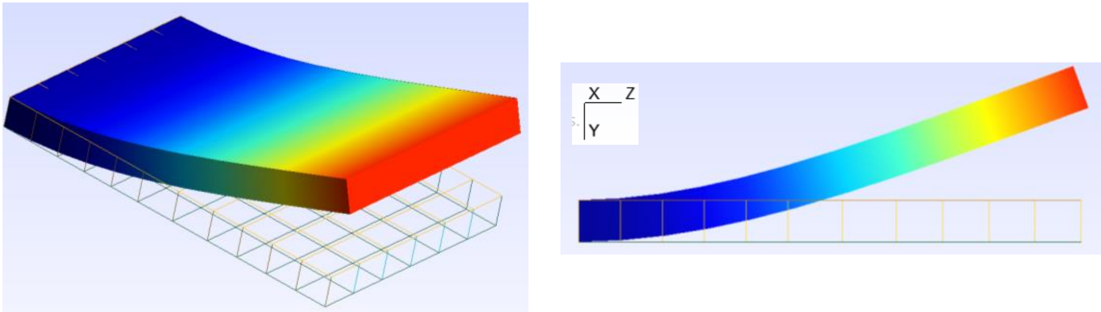
Table 2 presents the comparison between the analytical and numerical frequencies of the system. A very good agreement is observed between the two sets of results except for the third natural frequency. It should be noted that, the first two analytical frecuencies coincide with the first two numerical frecuencies because the relationship between l/h (120/10)>10, but after that, while the frequency increases, the analytical frequency is further away from the numerical frequency. It should be highlighted that the analytical model is the Euler-Bernoulli beam model.

	Numerical frequencies (Hz)	Analytical frequencies (Hz)
f_1	0.33	0.327
f_2	2.03	2.050
f_3	5.48	5.750

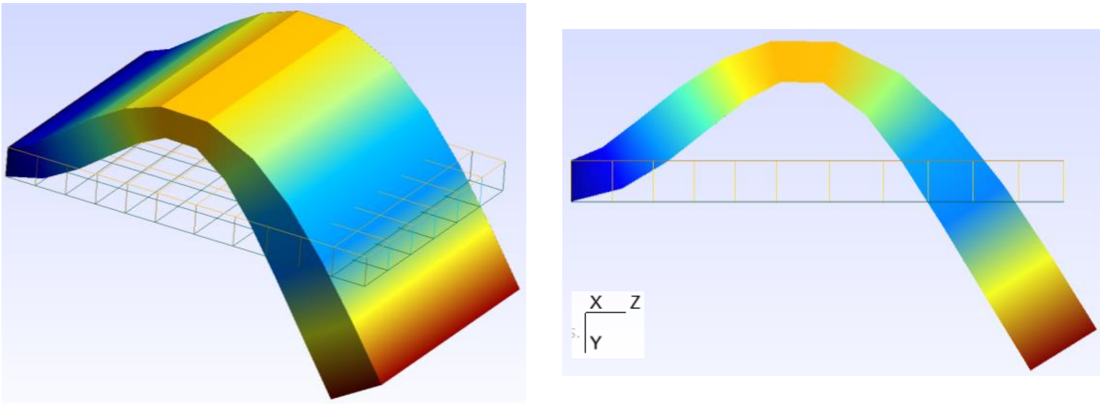
Table 2: Comparison between the analytical and numerical frequencies

3.2 Gmsh results file (*.pos)

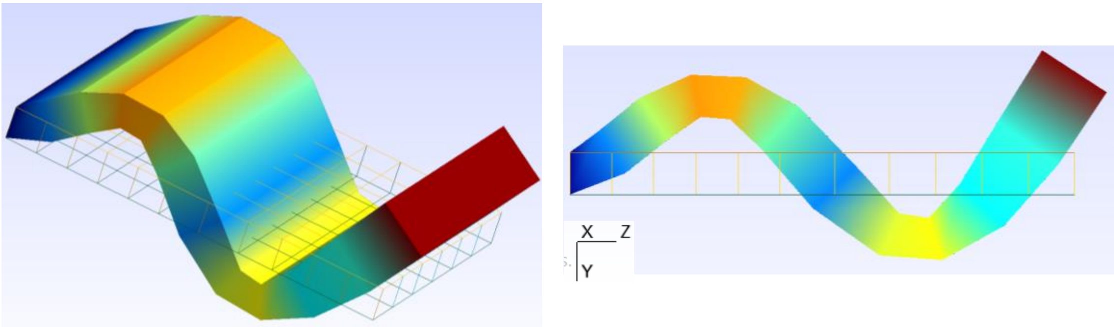
In this section, the three first mode shapes of vibration of the cantilever wall will be plotted using the Gmsh software with the output file obtained from MultiFEBE code. It should be noted that the frequencies corresponding to the mode shapes are obtained from the maximum peaks of the imaginary part of FRF (Fig. 4.b).



First mode shape (0.33 Hz)



Second mode shape (2.03 Hz)



Third mode shape (5.48 Hz)

Figure 7: Mode shapes of vibration

In the ME-TH-EL-003 [TUTORIAL], it will be explained, step by step, how to obtain the mode shapes of vibration of a structure using the Gmsh software.

References

[1] R. W. Clough and J. Penzien. “Dynamics of structures.” Computers & Structures. Inc., (2003)