Supplementary material for:

A general purpose element-based approach to compute dispersion relations in periodic materials with existing finite element codes

Camilo Valencia, Juan Gomez and Nicolás Guarín-Zapata

This supplementary material presents the details required to conduct Bloch analysis of a periodic 2D homogeneous material using the finite element code FEAP. The mesh can be viewed in GMSH from the `homoge.msh` file provided. In particular we show how to create the input file in relation to the schematic cell shown in figure 2 in the paper. All the notation used in this document corresponds to that of the finite element code. The supplied material also contains a compiled version of FEAPpv including a user element subroutine (UEL) to perform the dispersion analysis. This is a compilation for Debian-64b linux systems. The binary file `feappv` included in the set of files, contains the user subroutine identified as `User element 05` to compute the dispersion relations of any general FE mesh of a periodic material. This UEL uses 4-noded quad elements with a classic linear interpolation scheme and a 2D elastodynamic kinematic model under plane strain assumption. To run the subroutine for the provided example problem use the classic syntax to run a FEAP analysis: ./feappv -iIFeap. We also coded commands `IKX` and `IKY`, which increase the value of K_x and K_y respectively by $\Delta K_i = \pi/63$ to cover the first Brillouin zone. The eigenvalues are output in the `OFeap` file.

1 UEL details

Figure 2 in the paper shows a schematic representation of the mesh for a unit cell in a periodic material. The nodes in the mesh are divided into three groups:

- Group 1: Nodes located on one border of the unit cell
- Group 2: Nodes located on the opposite border of the unit cell and equivalent to nodes in group 1.
- Group *int*: Nodes that do not belong neither to group 1 nor 2. These are the interior nodes in the mesh.

If a particular element of the mesh contains at least one node of group 2, it needs to be defined in a different way in the FEAP input file.

FEAP input file

To use the UEL, the set of materials must be defined in the FEAP input file as:

```
FEAP * * Start record and title
...
```

```
MATErial,ID
 USER 05
 ELAS,ISOT   E  nu
 DENS mass   rho
!Blank
...
END mesh
...
...
STOP
```

When assigning materials to the elements of the mesh, the material `ID` has to be defined in the following manner:

- `ID` ≥ 10 for elements containing at least one node of group 2.
- $1 \leq$ `ID` ≤ 9 for the rest of elements.

Additionally, C_m^a and C_m^c have to be defined for the elements in which `ID` ≥ 10, i.e., elements in one of the borders of the unit cell. C_m^a is defined with the built-in `ELEM` command of FEAP. To define C_m^a we created a command named `BELE`. The syntax of this new command is:

```
FEAP * * Start record and title
...
BELE,neb
N 1 (ND_i, i=1,NEN)
...
END mesh
...
...
STOP
```

where `neb` is the number of elements for which C_m^c have to be defined. `N` is the element number and `ND-i` is the set of nodes defining element `N`. Finally, every degree of freedom of nodes in group 2 has to be fixed, so that they will be removed from the global arrays, yielding to the reduced eigenvalue system required to compute the dispersion relations.

Next we present a schematic example to define a simple mesh in the input file. Figure 4(a) in the paper shows the proposed mesh. The division of nodes for this case is:

- Group 1: Nodes 1, 2 and 4.
- Group 2: Nodes 3, 6, 7, 8 and 9.
- Group *int*: Node 5.

The mesh definition in the FEAP input file for this simple mesh reads:

```
FEAP * * Start record and title
...
COOR
...
```

```
ELEM
1 0 1 1 2 5 4
2 0 10 2 1 4 5
3 0 10 4 5 2 1
4 0 10 5 4 1 7
!
BELE,3
2 1 2 3 6 5
3 1 4 5 8 7
4 1 5 6 9 8
!
BOUN
3 0 1 1
6 0 1 1
7 0 1 1
8 0 1 1
9 0 1 1
!
END mesh
...
...
STOP
```