

***dmb_fem2*: dynamic analysis of a 2D structure**

user manual

1. Starting the script, loading of a FEM model (file .inp) and graphical utilities

To start the script from the Matlab's Command Window, select the folder where the *dmb_fem2* software is stored, run it typing *dmb_fem2* in the Matlab's Command Window and press *Return*.

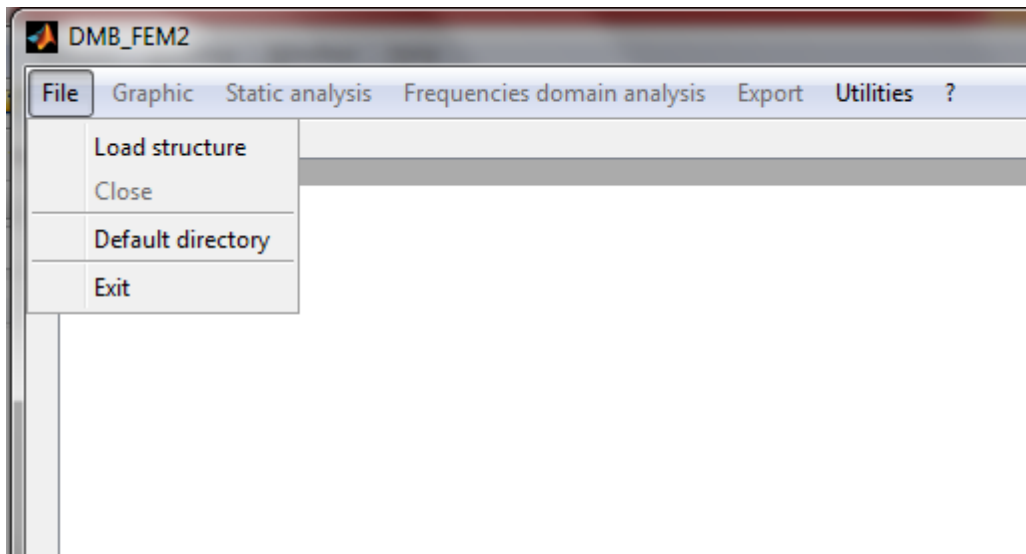


Figure 1.1

Figure 1.1 shows the *dmb_fem2* graphical user interface at the start up when all of the menus are disabled except for the *File* and the *Utilities* menus.

The *Default directory* option allows to select the folder where the input data of the structure are stored; the *Load structure* option allows to load the FEM model of a mechanical system / structure, in the form of an ascii file with extension *.inp (see section 6).

In the following, the data of the example structure *Beam1.inp* are loaded: it is a horizontal beam divided in 8 beams elements; the left extremity is fixed to the ground, while the right extremity is free (Figure 1.2)

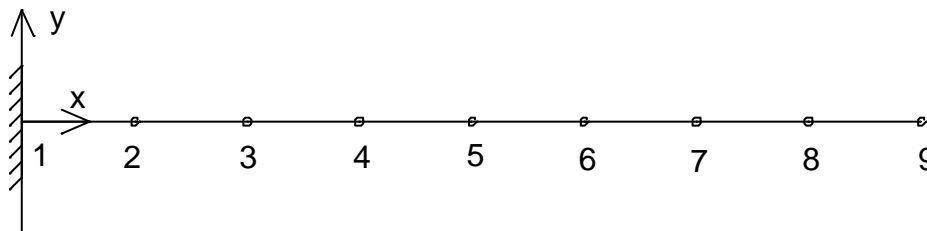


Figure 1.2

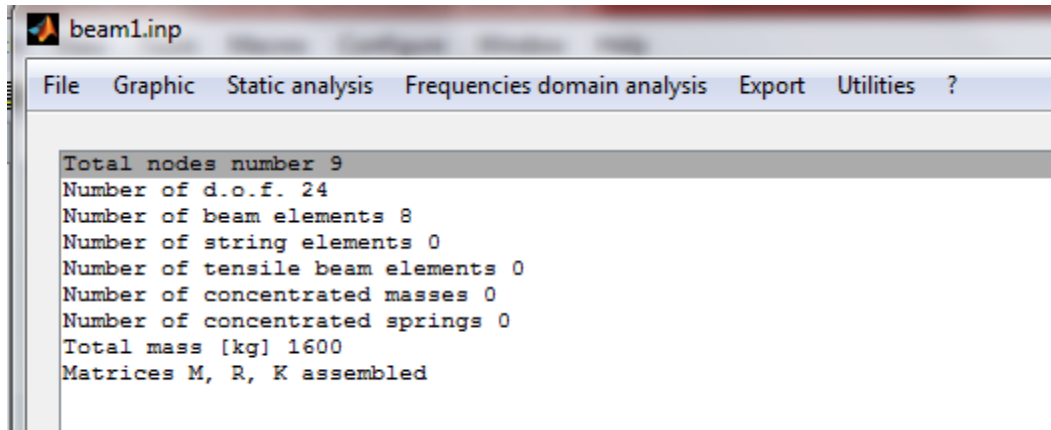


Figure 1.3

Once the input data have been loaded, the software prints the main information about the structure (Figure 1.3). If the file has been correctly loaded, the software assembles the structural matrices, prints the correspondent message and enables the other menus.

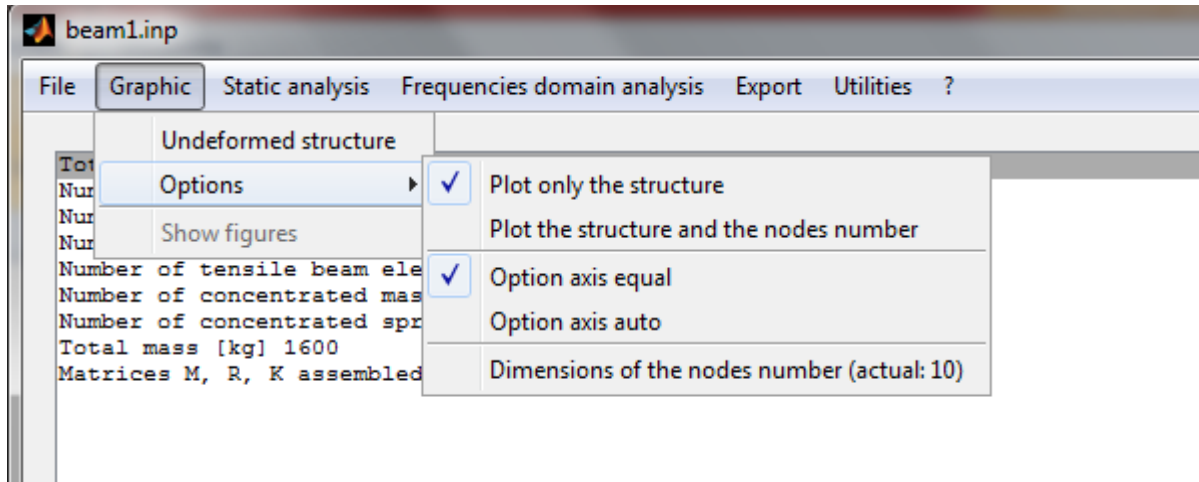


Figure 1.4

The menu *Graphic* allows to display the undeformed structure. There are some options in order to modify the default graphical style. As an example, Figure 1.6 shows the *Beam1* structure (the *Plot the structure and the nodes number* option has been selected before this plot). Note that constraints are not displayed in this visualization.

The sign conventions of the software are shown in the figure below:

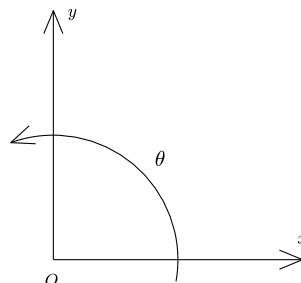


Figure 1.5

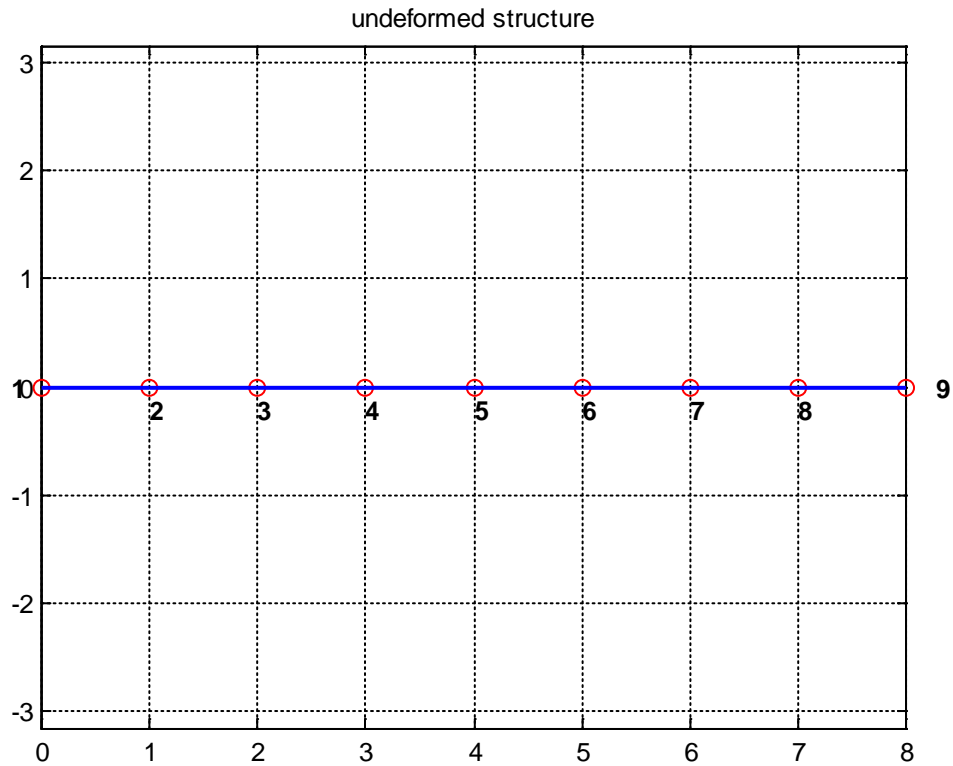


Figure 1.6

2. Static analysis

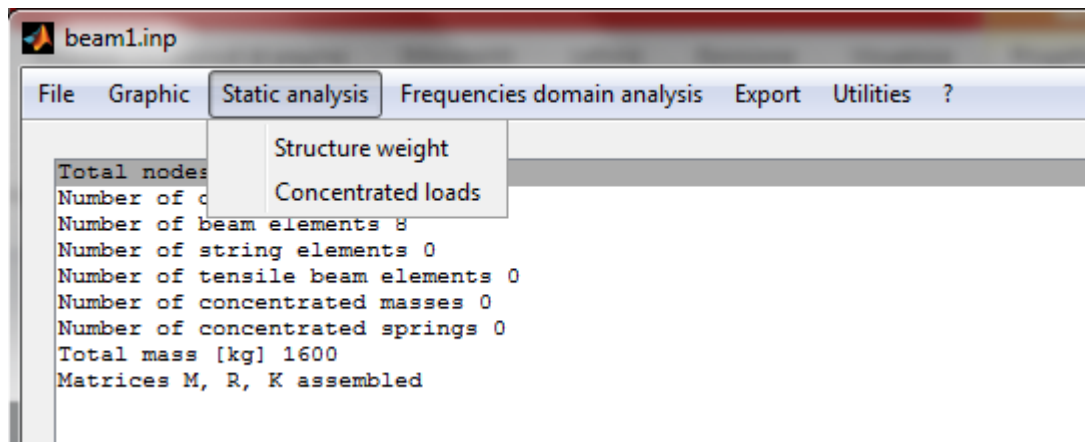


Figure 2.1

The *Static analysis* menu (Figure 2.1) allows to perform a static calculation of the structure's displacement under either its own weight or a set of static forces and moments. The outputs returned are the nodal generalized displacements, i.e. displacements and rotations of the nodal sections. Figure 2.2 shows the static deformation

(solid line) under the structure's own weight; the deformed shape is amplified by a scale factor reported in the title. Note that a value of the scale factor is automatically computed so that the structure's displacement is magnified in a way that makes it visible considering that the actual displacements are small compared to the actual size of the structure. The scale factor can be changed using the button *Change scale factor* of the local menu. If you don't want to modify the scale factor, select the *Exit* button in order to continue the analysis.

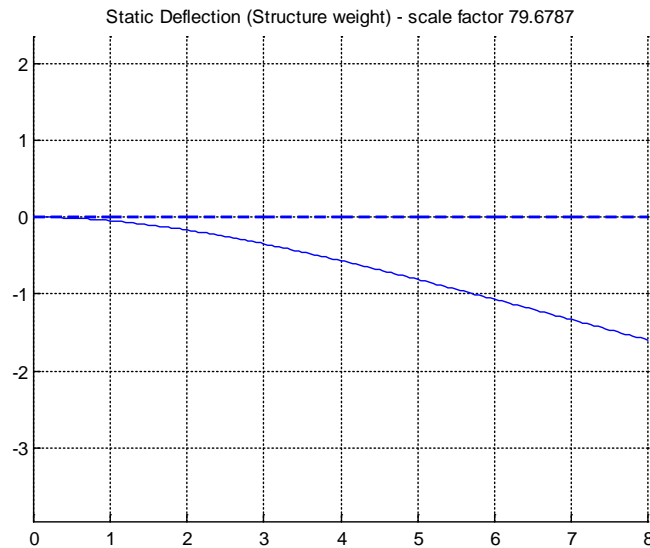


Figure 2.2

The x and y nodes displacements, together with the section rotations, are printed in the Matlab Command Window as well (in this case the scale factor is not applied). The units are meter and radians (Figure 2.3).

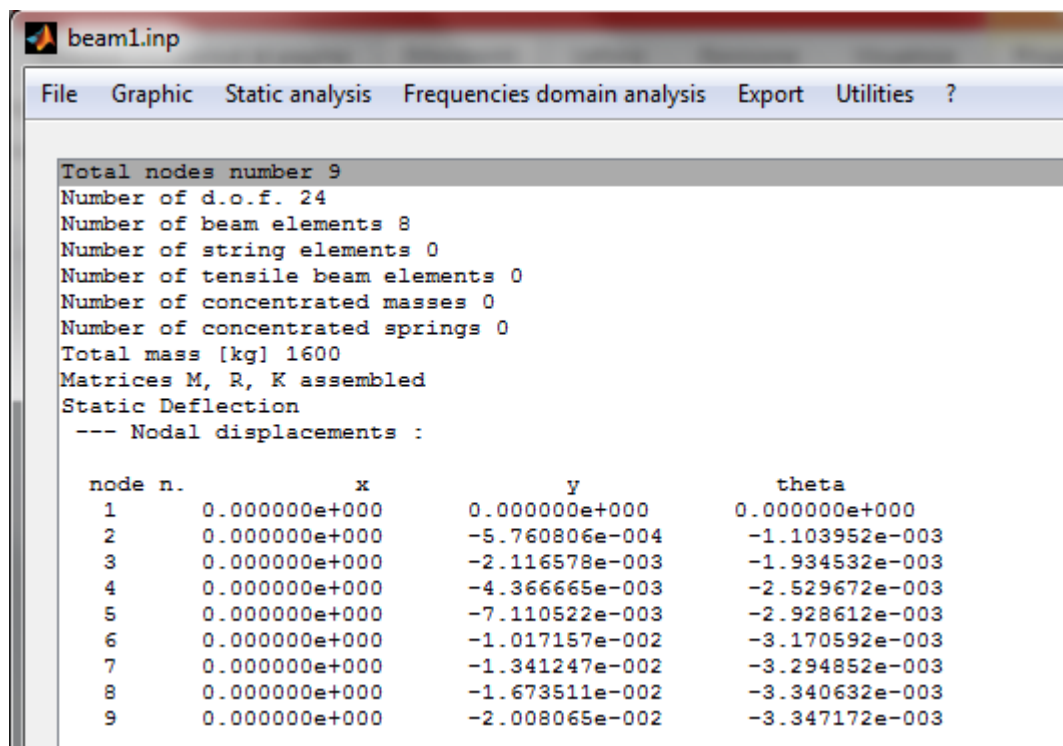


Figure 2.3

The *Concentrated loads* option allows to define a set of external forces and moments applied to the structure nodes. The software asks for the following information:

- Number of force components: this is the total number of force and moment components. For instance, if the set is constituted by a inclined force on node 5 (namely with both horizontal and vertical components), by a vertical force on node 9 and by a moment on node 9, the total number of the force and moment components is 4.
- Then, for each force/moment component, the user has to set the forced node, the direction and the amplitude. The direction is indicated by a code number: 1 for the x direction, 2 for the y and 3 for the rotation. Taking into account the software sign convention (Figure 1.5), the amplitude is positive or negative depending on the positive or negative direction of the force/moment.

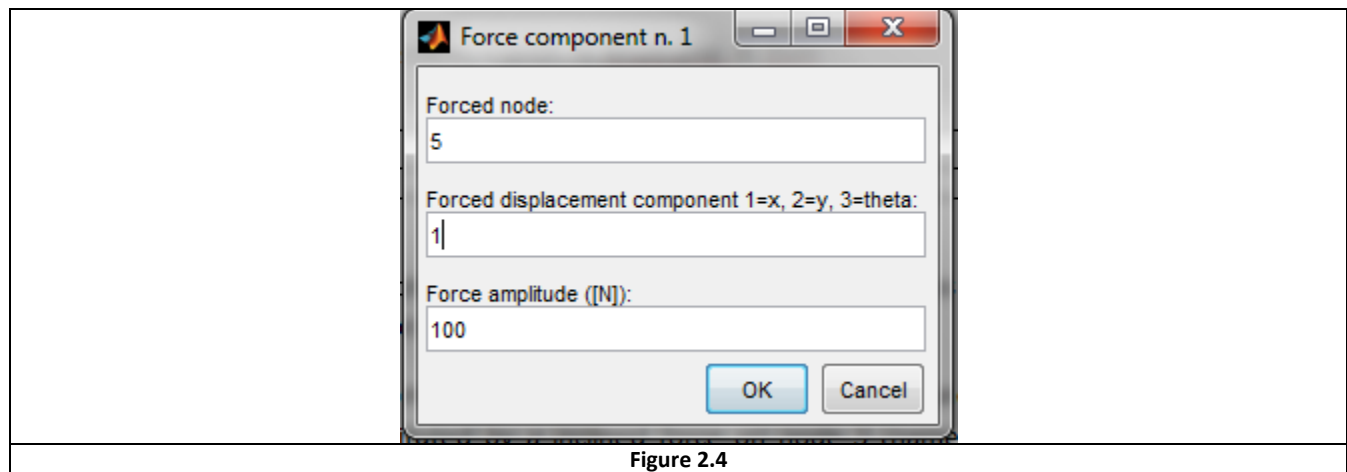


Figure 2.4

3. Natural frequencies and modes of vibration computation

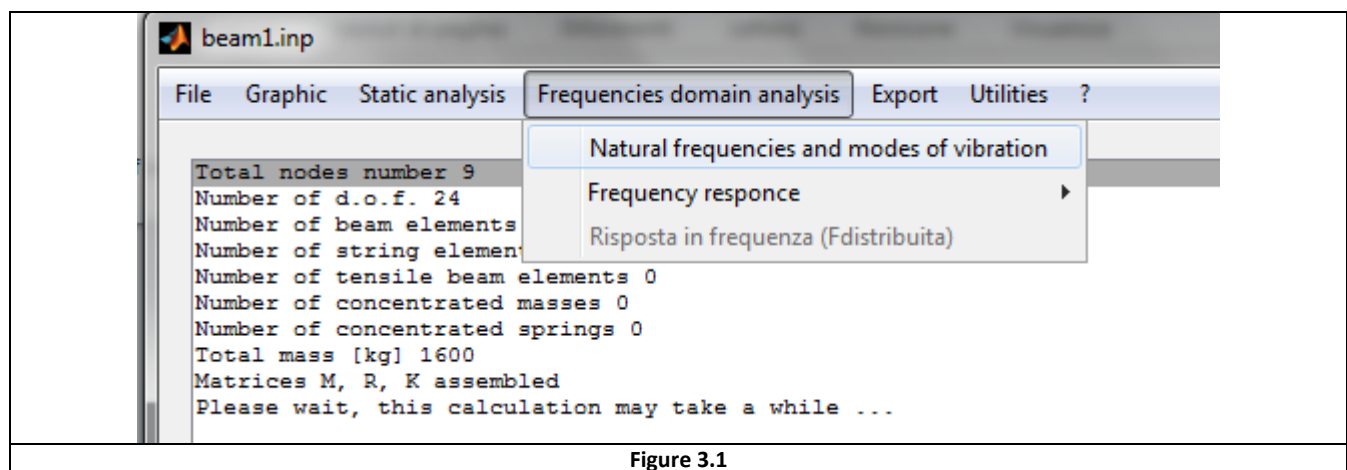


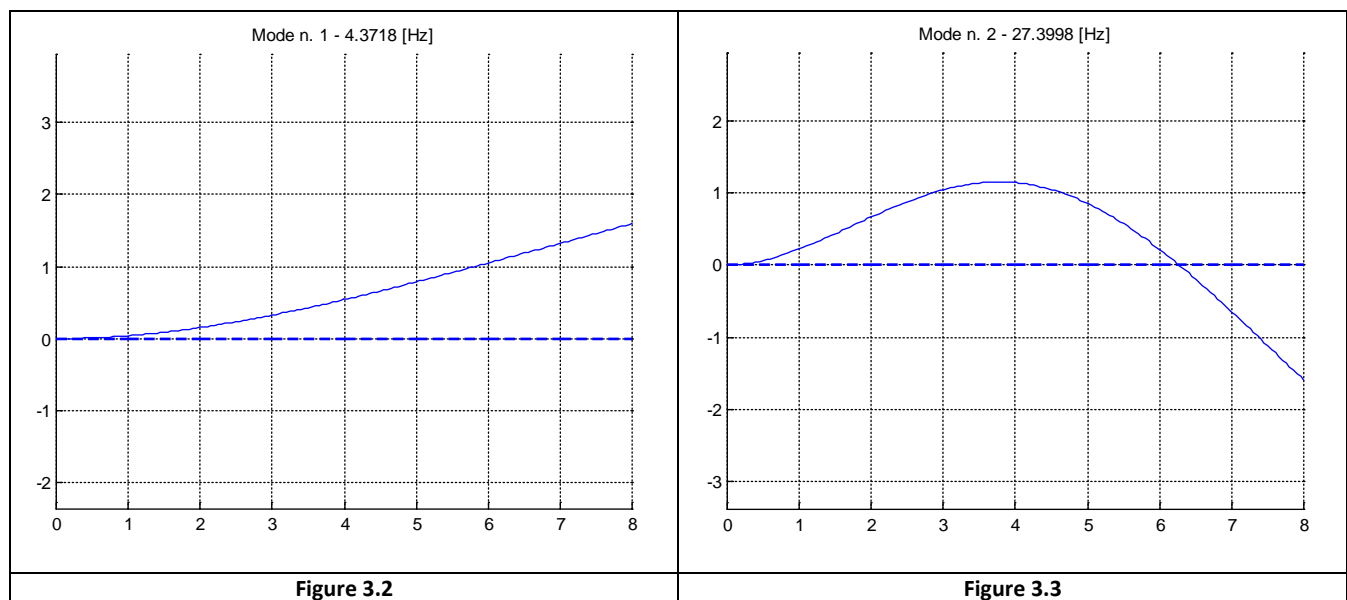
Figure 3.1

The *Natural frequencies and modes of vibration* option of the menu *Frequency domain analysis* allows for the computation of the structure's natural frequencies and modes of vibration. The modes of vibrations are

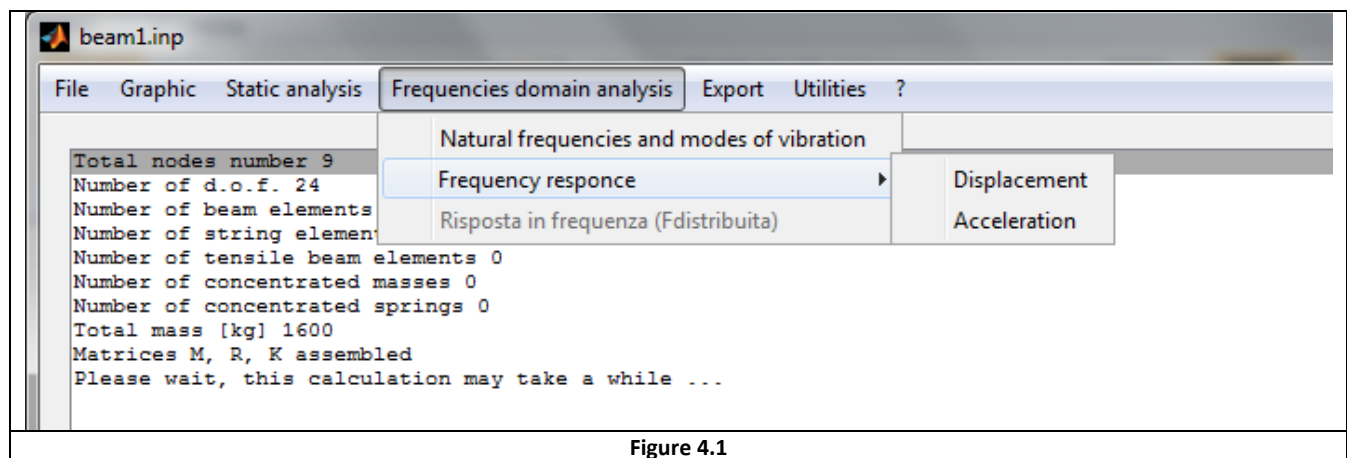
displayed using a suitable scale factor (see section 2) that can be changed with the local menu, and the correspondent nodal displacement and sections rotation values are printed in the Matlab's Command Window. The natural frequencies are printed both in the figures title and in the Matlab's Command Window.

The local menu allows the user to display the next/previous mode, up to the last one. It's worth drawing the attention to the fact that the software computes all of the modes of vibrations (N for a structure of N d.o.f.), not taking into account which is the maximum frequency that has been considered during the modelling of the structure; therefore it is up to the user to evaluate which modes are consistent with the modelling assumptions and which other are not and shall not be regarded as valid / acceptable.

Figure 3.2 and Figure 3.3 show the first two modes of vibration of the *Beam1* structure. With the data used in this example, the range of validity of the model is up to 390 Hz, so these two results can both be accepted.



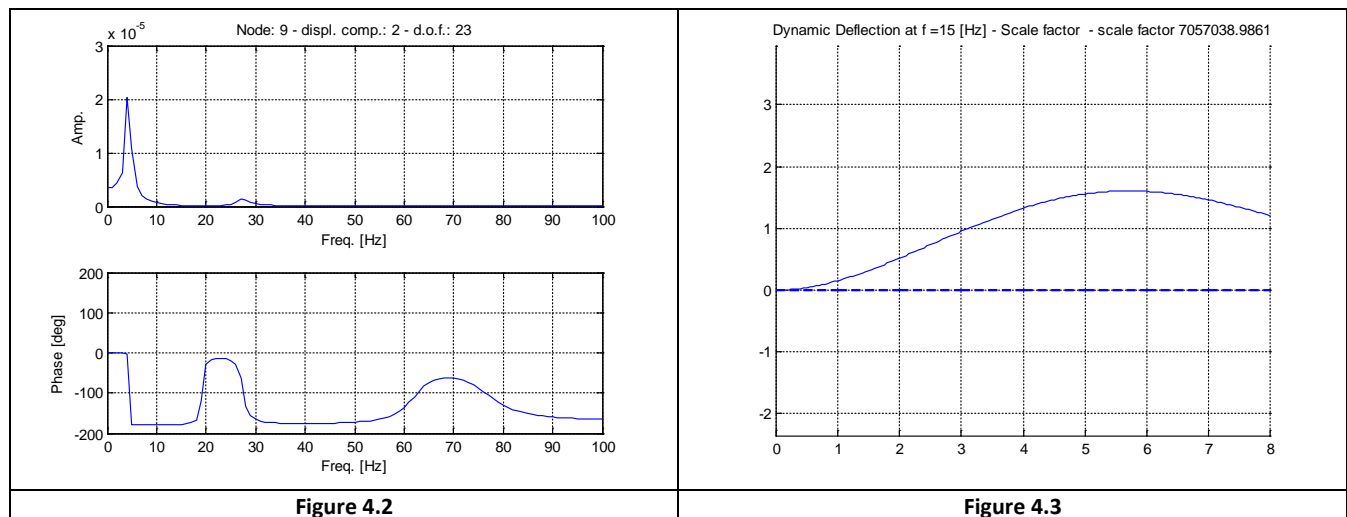
4. Frequency Response analysis



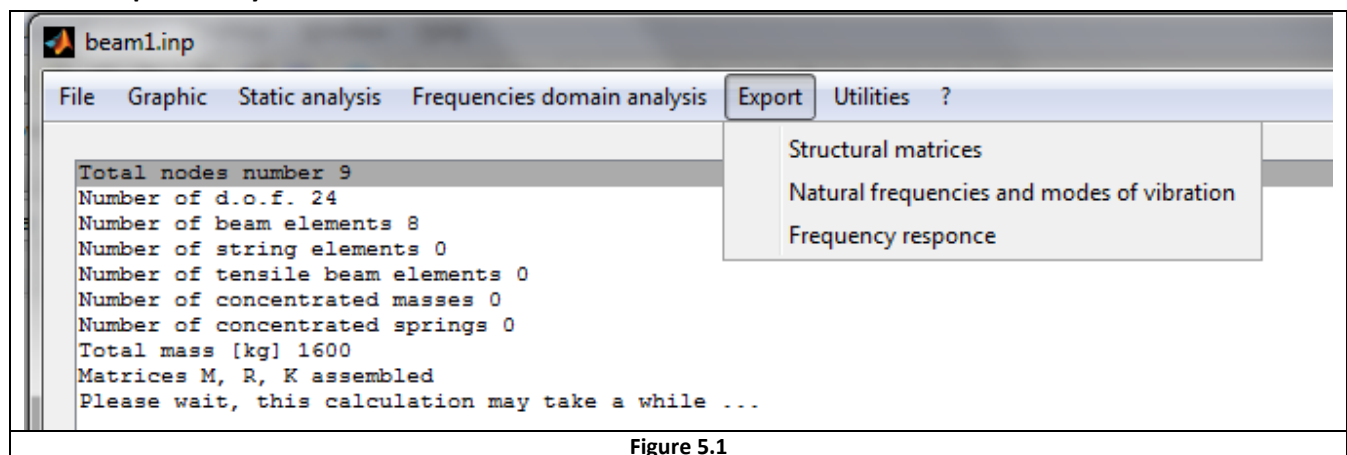
Select the option *Frequency response* of the menu *Frequencies domain analysis* to compute the frequency response of the structure. The input is a set of forces/moments applied to the nodes and the output can be either a component of generalized displacement (i.e. displacement or rotation) or generalized acceleration (i.e. linear or angular acceleration) of the nodes. A sequence of local menus allows the user to define the inputs and outputs for this analysis:

- Number of force components: is the total number of the force and moment components (see section 2 for more details)
- Then, for each force/moment component, the user has to set the forced node, the direction and the amplitude (see section 2 for more details).
- Range of frequency and step of frequency
- The output can be displayed in the form of either the Bode diagrams for a selected component of generalized displacement / acceleration, or the structure's dynamic deflection at a given frequency.

Figure 4.2 shows a linear Bode diagram of a frequency response (input: vertical force on node 9 with unit amplitude; output: vertical displacement of node 9); Figure 4.3 shows the dynamic deflection at 15 Hz.



5. Export utility



In order to perform other types of analysis not addressed by the *dmb_fem2* software, the user shall export the structural matrices to a file and work out the solution of the specific problem in MATLAB. The export menu allows to export the structural matrices and the results of the computation of the natural frequencies and of the frequency response.

The structural matrices [M], [C] and [K], together with an auxiliary matrix called the *idb* matrix (see below) are saved in a binary Matlab's file with extension **_mkr.mat*, in the same folder of the **.inp* file (for example the matrices of the *beam1.inp* structure are saved in the *beam1_mkr.mat*). The file can be loaded in the Matlab's workspace with the Matlab's *load* function. Note that the software exports the entire structural matrices, considering both the free generalized nodal displacements and those subject to constraints. It is up to the user to perform the partitioning of matrices to get the four partitions usually denoted by subscripts FF, FC, CF and CC.

The *idb* matrix is a rectangular matrix with three columns and *Nn* rows, where *Nn* is the total nodes number of the structure. It collects the numbering of the nodal coordinates for each node. The first and second column refers to displacements along the x and y direction respectively, while the third refers to the rotations. The software numbers first the free nodal coordinates and then those subject to constraint, considering the nodes order as they are written in the **.inp* file. The following table reports the *idb* matrix of the *beam1* structure (see also Figure 1.2):

	x	y	θ
1	25	26	27
2	1	2	3
3	4	5	6
4	7	8	9
5	10	11	12
6	13	14	15
7	16	17	18
8	19	20	21
9	22	23	24

Table 1: the *idb* matrix of the *beam1* structure

The *idb* matrix allows to get the number of any generalized nodal displacement of any node in the structure: for example the degree of freedom of node 5 along the vertical direction is the number 11, while the degree of constraint in the horizontal direction of the fixed clamp, namely of node 1, is n. 25. This information is needed to write, for example, a Matlab's script that applies external forces at the nodes of the structures and computes the system's response.

6. Input data file

This section describes the contents of the ascii **.inp* file defining the FEM model of the structure. The file is organized in sections called cards; each card starts and ends with keywords (e.g. **NODES* / **ENDNODES*). Lines containing comments can be inserted in the file and are identified as they bear in first column an exclamation mark character (!). Blank lines are allowed only outside the section data.

In this section the input data of the *beam1* structure will be described.

6.1 Nodes data – card *NODES

*NODES					
1	1	1	1	0.0	0.0
2	0	0	0	1.0	0.0
3	0	0	0	2.0	0.0
4	0	0	0	3.0	0.0
5	0	0	0	4.0	0.0
6	0	0	0	5.0	0.0
7	0	0	0	6.0	0.0
8	0	0	0	7.0	0.0
9	0	0	0	8.0	0.0
*ENDNODES					

This section begins with the keyword *NODES and ends with the keyword *ENDNODES. Each line is constituted by six data separated by blank spaces or tabs:

- 1st: node number
- 2nd to 4th: boundary condition codes relative to the x, y and θ directions respectively. Each code may assume only the values 0 or 1: 0 means not constrained, 1 means constrained. Hence:
 - o 0 0 0 is the code for a completely free node;
 - o 1 1 1 is the code of a node subject to a clamp;
 - o 1 1 0 is the code of a node subject to a hinge;
 - o 0 1 0 is the code of a node subject to a cart moving along the horizontal direction, and so on.
- 5th and 6th: x and y absolute coordinate of the node, measured in meters with respect to a fixed reference system. The user is free to place the reference system everywhere in the plane that contains the structure.

6.2 Beams data – card *BEAMS.

*BEAMS					
1	1	2	200	1.0e10	5E7
2	2	3	200	1.0e10	5E7
3	3	4	200	1.0e10	5E7
4	4	5	200	1.0e10	5E7
5	5	6	200	1.0e10	5E7
6	6	7	200	1.0e10	5E7
7	7	8	200	1.0e10	5E7
8	8	9	200	1.0e10	5E7
*ENDBEAMS					

This section begins with the keyword *BEAMS and ends with the keyword *ENDBEAMS. Each line is constituted by six data separated by blank spaces or tabs:

- 1st: beam number
- 2nd and 3rd: first and second node number. For example, in the *beam1* structure the second beam connects node 2 with node 3.
- 4th to 6th: linear mass [kg/m], axial stiffness EA [N] and bending stiffness EJ [N m²] of the beam element.

6.3 Damping data – card *DAMPING.

```
*DAMPING
0.1 3.0e-4
```

This section begins with the keyword *DAMPING; there isn't any closing keyword because, after the starting keyword, only a line is required with the α and β coefficients to define the structural damping matrix:

$$[C] = \alpha[M] + \beta[K]$$

Units of these coefficients are s^{-1} for α and s for β .

6.4 Rigid bodies data – card *MASSES

```
*MASSES
1 10 5.0 0.1
*ENDMASSES
```

This section allows to include in the FEM model rigid bodies attached at some nodes of the structure. The section begins with the keyword *MASSES and ends with the keyword *ENDMASSES. Each line of this section is constituted by four data separated by blank spaces or tabs:

- 1st: rigid body mass number;
- 2nd: node number where the rigid body is placed;
- 3rd and 4th: mass [kg] and moment of inertia with respect to the center of mass [kg m²].

In this example, which doesn't refer to the *beam1* structure, one rigid body has been add to node n. 10; its mass is 5.0 kg and its moment of inertia is 0.1 kg m².

6.5 Spring and damper systems data – card *SPRINGS

```
*SPRINGS
1 2 5 0 1e7 0 0 1e3 0
*ENDSPRINGS
```

This section allows to include in the FEM model spring and damper systems attached between two nodes. The section begins with the keyword *SPRINGS and ends with the keyword *ENDSPRINGS. Each line of this section is constituted by nine data separated by blank spaces or tabs:

- 1st: spring and damper system number;
- 2nd and 3rd: first and second node number where the spring and damper system is placed;
- 4th to 6th: linear stiffness along the x and y direction [N/m] and torsional stiffness [Nm/rad];
- 7th to 10th: linear viscous damping coefficient along the x and y direction [Ns/m] and torsional viscous damping coefficient [Nms/rad]

In this example, which doesn't refer to the *beam1* structure, one spring and damper system has been add between node n. 2 and nr. 5; its stiffness along the vertical direction is 1e7 N/m and its viscous damping coefficient is 1e3 Nms/rad.