LESM

Linear Elements Structure Model

Version 1.0 – May 2017

http://www.tecgraf.puc-rio.br/lesm

by

Luiz Fernando Martha (lfm@tecgraf.puc-rio.br)

Rafael Lopez Rangel (rafaelrangel@tecgraf.puc-rio.br)

Pontifical Catholic University of Rio de Janeiro – PUC-Rio
Department of Civil and Environmental Engineering and Tecgraf Institute of Technical-Scientific Software Development of PUC/Rio

Summary

1 – General Characteristics

LESM is a MATLAB program for linear-elastic, displacement-based, static analysis of two-dimensional and three-dimensional linear elements structure models, using the direct stiffness method. For each structural analysis, the program assembles a system of equilibrium equations, solves the system and computes internal forces and displacements.

The program may be used in a nongraphical version or in a GUI (Graphical User Interface) version. The nongraphical version reads a structural model from a neutral format file and prints model information and analysis results in the default output (MATLAB command window). In the GUI version, a user may create a structural model with attributes through the program graphical interface. The program can save and read a structural model data stored in a neutral format file, which can be edited using a text editor.

For educational purposes, LESM source code is based on the Object-Oriented Programming (OOP) paradigm, which can provide a clear and didactic code that may be part of a course material on matrix structural analysis.

More information about the LESM program can be found on its website: https://web.tecgraf.puc-rio.br/lesm/.

2 – Coordinate Systems

Two types of coordinate systems are used to locate points or define directions in space. Both of them are Cartesian, orthogonal and right-handed systems. Every structural model is disposed in an absolute coordinate system called global system, and each of its elements (bars) has its own coordinate system called local system.

The global system is referenced by uppercase letters while the local system is referenced by lowercase letters.

Global System

The global system is an absolute reference since its origin is fixed and it gives unique coordinates to each point in space.
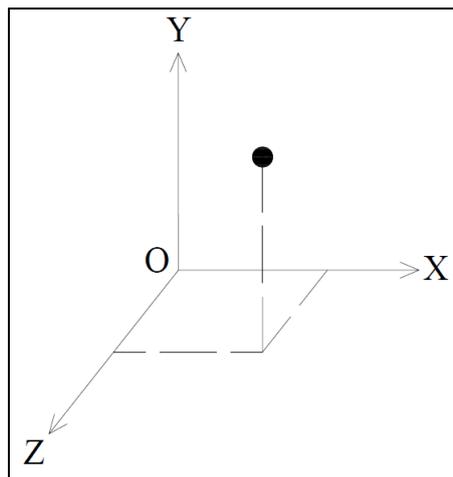

Figure 1: The global coordinate system

Local System

The local system is a relative reference since its origin is located at the beginning of the element (initial node) and the local x-axis is always in the same direction of the element longitudinal axis.
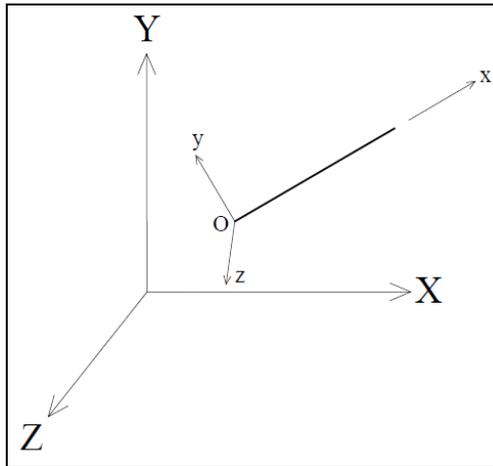
Figure 2: The local coordinate system

In LESM, the graphical representation of translations in any direction is an arrow, while the representation of rotations about an axis is a double-arrow in the axis direction, following the right-hand rule.

3 – Analysis Models

LESM considers only static linear-elastic structural analysis of 2D (plane) linear elements models and 3D (spatial) linear elements models, which can be of any of the following types of analysis models:

3.1 – Truss Analysis Model

A truss model is a common form of analysis model, with the following assumptions:

- Truss elements are bars connected at their ends only, and they are connected by friction-less pins. Therefore, a truss element does not present any secondary bending moment or torsion moment induced by rotation continuity at joints.
- A truss model is loaded only at joints, which are also called nodes. Any load action along an element, such as self-weight, is statically transferred as concentrated forces to the element end nodes.
- Local bending of elements due to element internal loads is neglected, when compared to the effect of global load acting on the truss.
- Therefore, there is only one type of internal force in a truss element: axial force, which may be tension or compression.
- A 2D truss model is considered to be laid in the global XY-plane, with only in-plane behavior, that is, there is no displacement transversal to the truss plane.
- Each node of a 2D truss model has two d.o.f.'s (degrees of freedom): a horizontal displacement in local or in global X direction and a vertical displacement in local or in global Y direction.
- Each node of a 3D truss model has three d.o.f.'s (degrees of freedom): displacements in local or in global X, Y and Z directions.
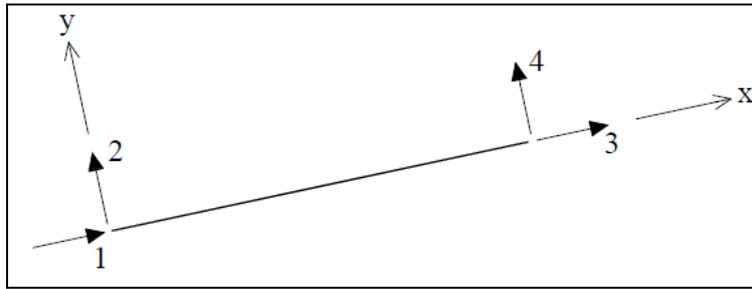
Figure 3: Degrees of freedom positive directions of a 2D truss element in local system
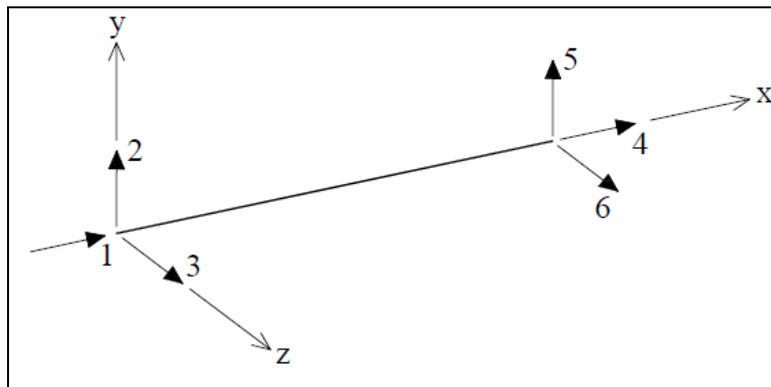

Figure 4: Degrees of freedom positive directions of a 3D truss element in local system

3.2 – Frame Analysis Model

A frame model is also made up of bars, which are called beams (horizontal bars) or columns (vertical bars). Inclined members are also called beams. In the present context, these types of bars are generically called elements. A continuous beam (an assemblage of connected beams) is considered as a frame model by LESM. These are the basic assumptions of a frame model:

- Frame elements are usually rigidly connected at the joints. However, a frame element might have a hinge (rotation liberation) at an end or hinges at both ends. A frame element with hinges at both end works like a truss element. A truss model could be seen a frame model with complete hinges at both its nodes.
- It is assumed that a hinge in a 2D frame element releases continuity of rotation about the Z-axis, while a hinge in a 3D frame element releases continuity of rotation in all directions.

- A 2D frame model is considered to be laid in the global XY-plane, with only in-plane behavior, that is, there is no displacement transversal to the frame plane.
- Internal forces at any cross-section of a 2D frame element are: axial force, shear force, and bending moment.
- Internal forces at any cross-section of a 3D frame element are: axial force, shear force, bending moment, and torsion moment.
- In 3D frame models, internal shear force and bending moment at any cross-section have components in local y-axis direction and in local z-axis direction.
- Each node of a 2D frame model has three d.o.f.'s: a horizontal displacement in local or in global X direction, a vertical displacement in local or in global Y direction, and a rotation about the Z-axis.
- Each node of a 3D frame model has six d.o.f.'s: displacements in local or in global X, Y and Z directions, and rotations about local or global X, Y and Z axes.
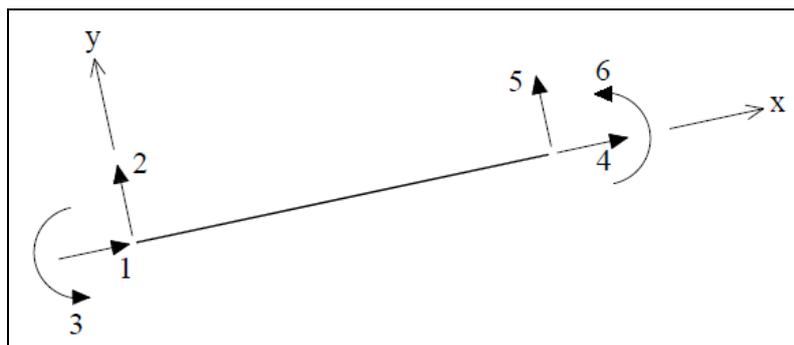

Figure 5: Degrees of freedom positive directions of a 2D frame element in local system
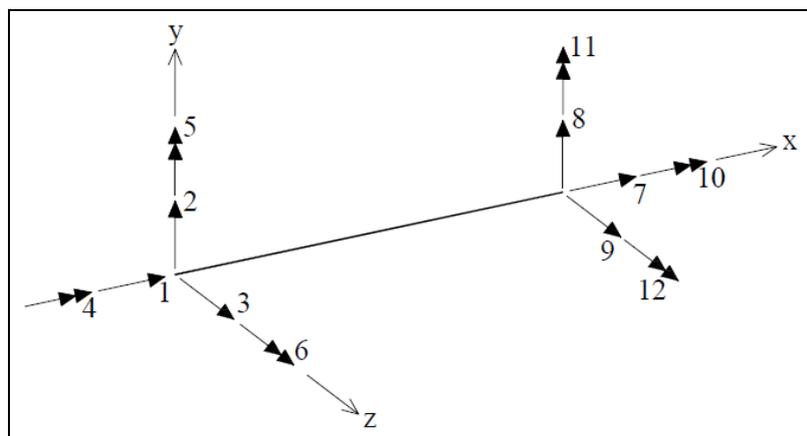

Figure 6: Degrees of freedom positive directions of a 3D frame element in local system

3.3 – Grillage Analysis Model

A grillage model is a common form of analysis model for building stories and bridge decks. Its key features are:

- It is a 2D model, which in LESM is considered in the global XY-plane.
- Beam elements are laid out in a grid pattern in a single plane, rigidly connected at nodes. However, a grillage element might have a hinge (rotation liberation) at an end or hinges at both ends.
- It is assumed that a hinge in a grillage element releases continuity of both bending and torsion rotations.
- By assumption, there is only out-of-plane behavior, which includes displacement transversal to the grillage plane, and rotations about in-plane axes.
- Internal forces at any cross-section of a grillage element are: shear force (transversal to the grillage plane), bending moment (in a plane perpendicular to the grillage plane), and torsion moment (about element longitudinal axis).
- By assumption, there is no axial force in a grillage element. The axial effect caused by thermal dilatation of elements longitudinal axes is neglected, only the bending effect caused by the temperature gradient in local z-axis is considered in this case.
- Each node of a grillage model has three d.o.f.'s: a transversal displacement in Z direction, and rotations about local or global X and Y axes.
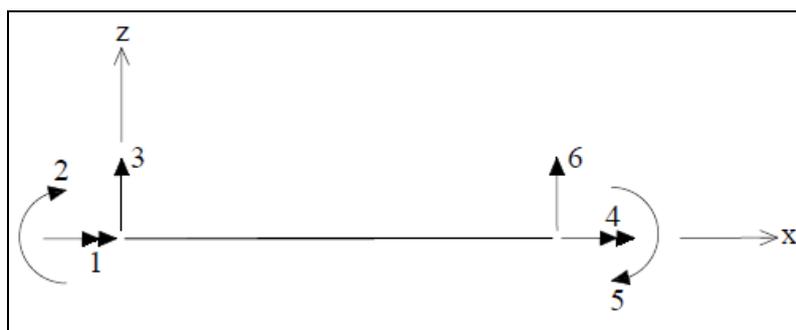

Figure 7: Degrees of freedom positive directions of a grillage element in local system

4 – Element Types

For frame and grillage models, whose members have bending effects, two types of beam elements are considered:

- **Navier (Euler-Bernoulli) Element:**

In Euler-Bernoulli flexural behavior, it is assumed that there is no shear deformation. As a consequence, bending of a linear structure element is such that its cross-section remains plane and normal to the element longitudinal axis.

- **Timoshenko Element:**

In Timoshenko flexural behavior, shear deformation is considered in an approximated manner. Bending of a linear structure element is such that its cross-section remains plane but it is not normal to the element longitudinal axis.
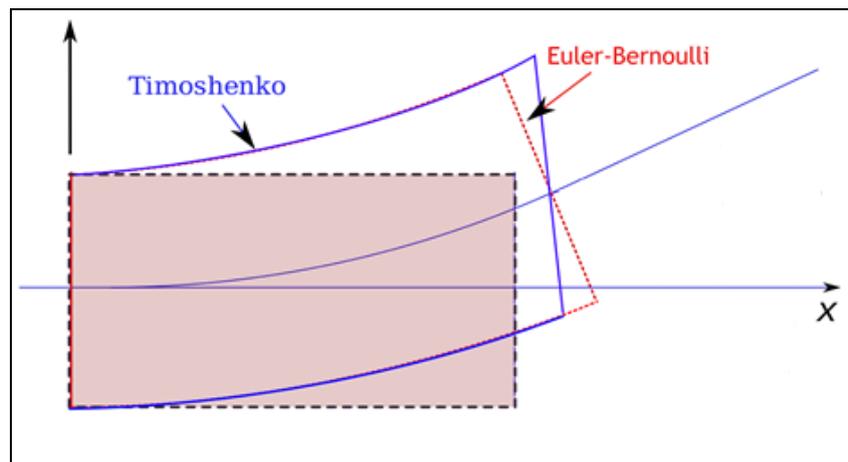

Figure 8: Comparison between Euler-Bernoulli element and Timoshenko element

In truss models, the two types of elements may be used indistinguishably, since there is no bending behavior of a truss element, and Euler-Bernoulli elements and Timoshenko elements are equivalent for the axial behavior.

5 – Local Axes of Elements

In 2D models of LESM, the local axes of an element are defined uniquely in the following manner:

- The local z-axis of an element is always in the direction of the global Z-axis, which is perpendicular to the model plane and its positive direction points out of the screen.
- The local x-axis of an element is its longitudinal axis, from its initial node to its final node.
- The local y-axis of an element lays in the global XY-plane and is perpendicular to the element x-axis in such a way that the cross-product (x-axis * y-axis) results in a vector in the global Z direction.
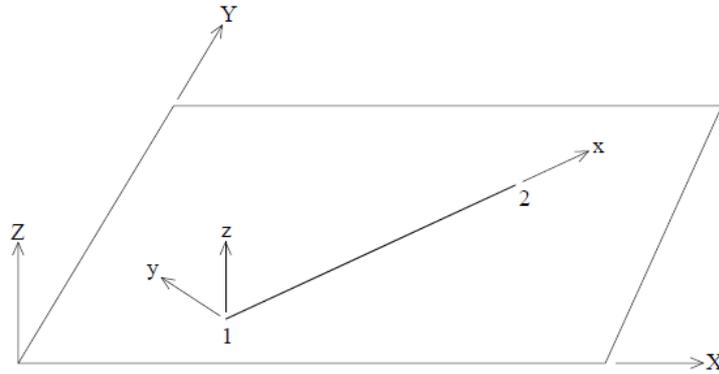

Figure 9: Local axes of elements in 2D models

In 3D models of LESM, the local y-axis and z-axis are defined by an auxiliary vector $vz = (vzx, vzy, vzz)$, which is an element property and should be specified as an input data of each element:

- The local x-axis of an element is its longitudinal axis, from its initial node to its final node.
- The auxiliary vector vz lays in the local xz-plane of the element, and the cross-product (vz * x-axis) defines the local y-axis vector.
- The direction of the local z-axis is then calculated with the cross-product (x-axis * y-axis).

In 2D models, the auxiliary vector vz is automatically set to (0,0,1). In 3D models, it is important that the auxiliary vector is not parallel to the local x-axis;

9

otherwise, the cross-product (vz * x-axis) is zero and local y-axis and local z-axis are not defined.
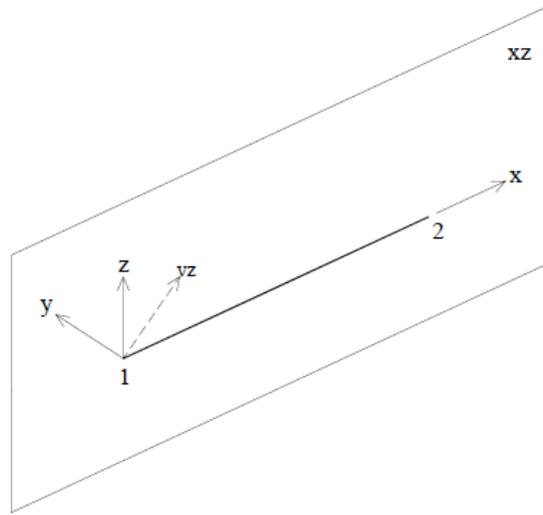

Figure 10: Local axes of elements in 3D models

Convention for bottom face and upper face of elements:

The upper face of an element, relative to the local y-axis or local z-axis, is the face turned to the positive side of the axis, while the bottom face is turned to the negative side of the axis.
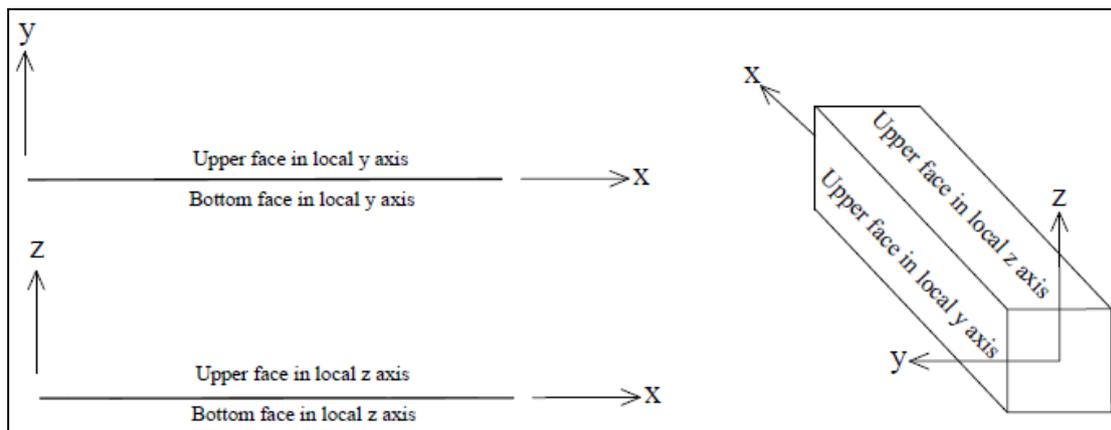
Figure 11: Bottom and upper faces of elements

6 – Materials

All materials in LESM are considered to be homogeneous and isotropic, that is, they have the same properties at every point and in all directions.

7 – Cross-sections

All cross-sections in LESM are considered to be of a generic type, which means that their shapes are not specified, only their geometric properties are provided, such as area, moment of inertia and height.

8 – Load Types

There are four types of loads in LESM:

- Concentrated nodal load in global directions.
- Uniformly distributed force on elements, spanning its entire length, in local or in global directions.
- Linearly distributed force on elements, spanning its entire length, in local or in global directions.
- Uniform temperature variation on faces of elements.

It is assumed that there is a single load case.

LESM does not work with distributed moments on elements.

In addition, nodal prescribed displacement and rotations may be specified.

8.1 – Components of Concentrated Nodal Loads

- In 2D truss models, concentrated nodal loads are force components in global X and global Y directions.
- In 3D truss models, concentrated nodal loads are force components in global X, global Y and global Z directions.
- In 2D frame models, concentrated nodal loads are force components in global X and global Y directions, and a moment component about global Z-axis.
- In 3D frame models, concentrated nodal loads are force components in global X, global Y and global Z directions, and moment components about global X, global Y and global Z axes.
- In grillage models, concentrated nodal loads are a force component in global Z direction (transversal to the model plane), and moment components about global X and global Y axes.

8.2 – Components of Distributed Loads

- In 2D truss or 2D frame models, the uniformly or linearly distributed force has two components, which are in global X and Y directions or in local x and y directions.
- In 3D truss or 3D frame models, the uniformly or linearly distributed force has three components, which are in global X, Y and Z directions or in local x, y and z directions.
- In grillage models, the uniformly or linearly distributed force has only one component, which is in global Z direction.

8.3 – Components of Thermal Loads (Temperature Variation)

- In 2D truss and 2D frame models, thermal loads are specified by a temperature gradient relative to element local y-axis and the temperature variation on its center of gravity axis.
- In 3D truss and 3D frame models, thermal loads are specified by the temperature gradients relative to element local y and z axes, and the temperature variation on its center of gravity axis.
- In grillage models, thermal loads are specified by a temperature gradient relative to element local z-axis, and the temperature variation on its center of gravity axis.

The temperature gradient relative to an element local axis is the difference between the temperature variation on its bottom face and its upper face.
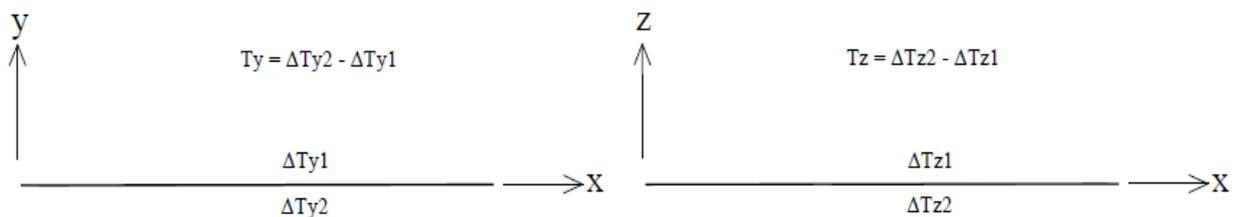

Figure 12: Temperature gradient convention

9 – Internal Forces Conventions

Axial force

Positive axial force acts in the negative direction of local x-axis on the left side of an element and in the positive direction on the right side. This means that tension force is positive while compression is negative.
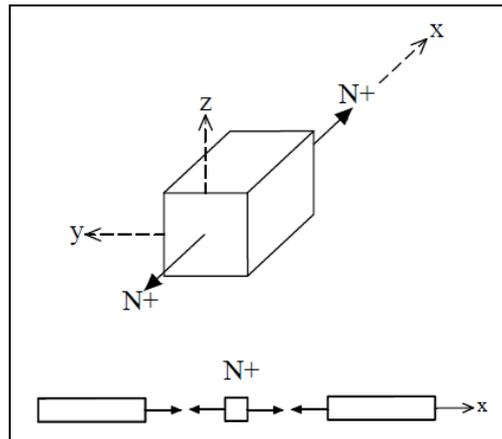
Figure 13: Positive internal axial force

Torsion moment

Positive torsion moment rotates around the negative direction of local x-axis on the left side of an element, using the right-hand rule, and in the positive direction on the right side. This means that positive torsion moment points out of the element face while negative torsion moment points inside.
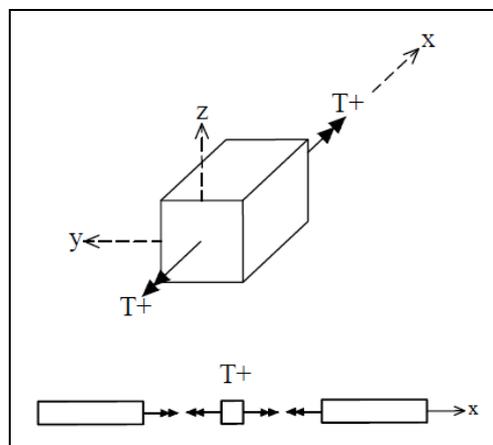

Figure 14: Positive internal torsion moment

Shear force

Positive shear force acts in the positive direction of local y-axis or z-axis on the left side of an element and in the negative direction on the right side.
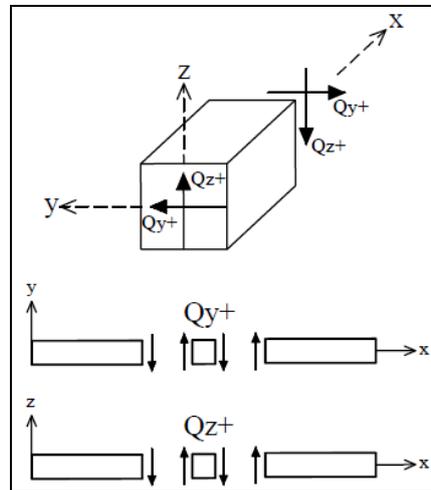

Figure 15: Positive internal shear force

Bending moment

Positive bending moment rotates around the positive direction of local y-axis and negative direction of local z-axis on the left side of an element, using the right-hand rule, and in the negative direction of local y-axis and positive direction of local z-axis on the right side.
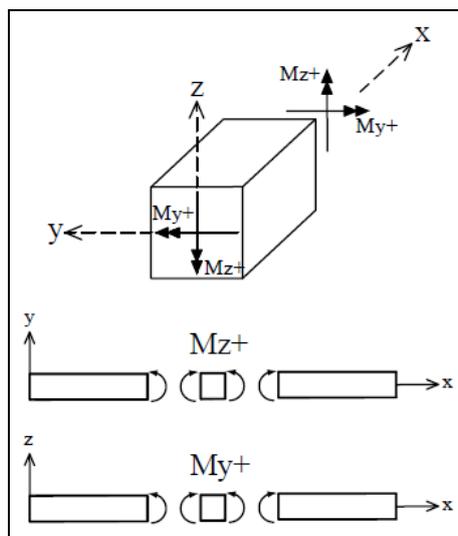

Figure 16: Positive internal bending moment

15

Diagram conventions

- Positive axial force diagram values are plotted on the upper side of elements local y-axis (in frame models). In truss models, axial force is represented only by its value next to each element, since it is always constant.
- Positive shear force diagram values are plotted on the upper side of elements.
- Positive bending moment diagram values are plotted on the tension side of elements.*
- Torsion moment is represented only by its value next to each element, since it is always constant because there is no distributed moment on elements.

* The diagram of bending moment about local y-axis (acts in local xz-plane) is plotted in local z-axis, while the diagram of bending moment about local z-axis (acts in local xy-plane) is plotted in local y-axis.

10 – Units

The units of input and output parameters of LESM follow the metric system and cannot be changed. All input data units are indicated next to each fill-in field.

- **Elasticity/Shear modulus**: Megapascal [MPa]
- **Temperature**: Celsius degree [°C]
- **Cross-section area**: Square centimeter [cm²]
- **Moment of inertia**: Centimeter to the fourth power [cm^4]
- **Cross-section height**: Centimeter [cm]
- **Model length**: Meter [m]
- **Concentrated force**: Kilonewton [kN]
- **Concentrated moment**: Kilonewton meter [kN.m]
- **Distributed force**: Kilonewton per meter [kN/m]
- **Displacement**: Millimeter [mm]
- **Rotation**: Radian [rad]

11 – Non-Graphical Version

To use the non-graphical version of LESM, it is necessary to download the program source code from the website and open the *main.m* file in any MATLAB version (preferably in a recent version).

To run the non-graphical version of the program, the path to the neutral format file with the *.lsm* extension containing target model information must be specified as a string assigned to the "fileName" variable, where indicated. If the name or the path to the input file is wrong, a message is shown in the command window.

After running this code (shortcut: F5), the model information and the analysis results will show up in the command window.

```matlab
%% Non-graphical version
% Initialize analysis driver object
drv = Drv();

% Specify file name (edit only the next line to change file path for target model)
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
fileName = 'Models/Frame2D.lsm';
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

% Open a neutral file with textual model description
lsm = fopen(fileName,'rt');

% Warn about invalid file
if lsm == -1
    fprintf(1,'Invalid input file path\n');
    return;
end

% Input model data and get print object
fprintf(1,'Pre-processing...\n');
[drv,print] = readFile(drv,lsm);

% Link print object to driver object and
% setup default output to command window
print.drv = drv;
print.txt = 1;

% Process the provided model data according to the direct stiffness method
status = drv.process();
```

Figure 17: Non-graphical version of LESM

The available non-graphical (textual) analysis results are:

- **Nodal Displacements and Rotations**: Components of nodal displacements and rotations in the degrees of freedom directions in global system.
- **Support reactions**: Forces and moments acting in constrained degrees of freedom directions of each node in global system.
- **Internal Forces at Element Nodes**: Value of the internal forces indicated in the local system directions of the degrees of freedom.*
- **Elements Internal Displacements in Local System**: Axial and transversal displacements in local directions at 10 cross-sections along each element.

* Positive convention of internal forces at element nodes:
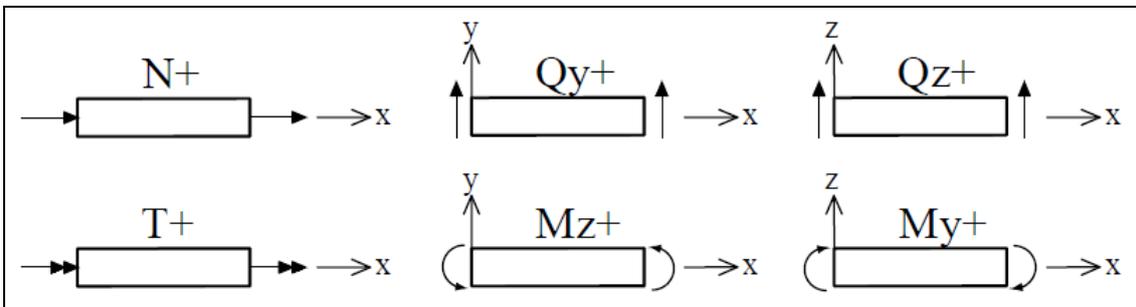
Figure 18: Positive internal forces convention in textual results

12 – User Interface

To run the graphical version of LESM, the standalone application generated by the MATLAB compiler must be downloaded from the website.

The graphical user interface of LESM is composed by a main window, where it is possible to load, create, view and analyze a structural model, and a set of auxiliary windows for getting the input data necessary to add or remove components from the model.

The area where the graphic system can draw geometric primitives to represent the structural model and display the analysis results is called canvas. In LESM, just like in any other MATLAB graphic application, this component is a tri-dimensional axes system.
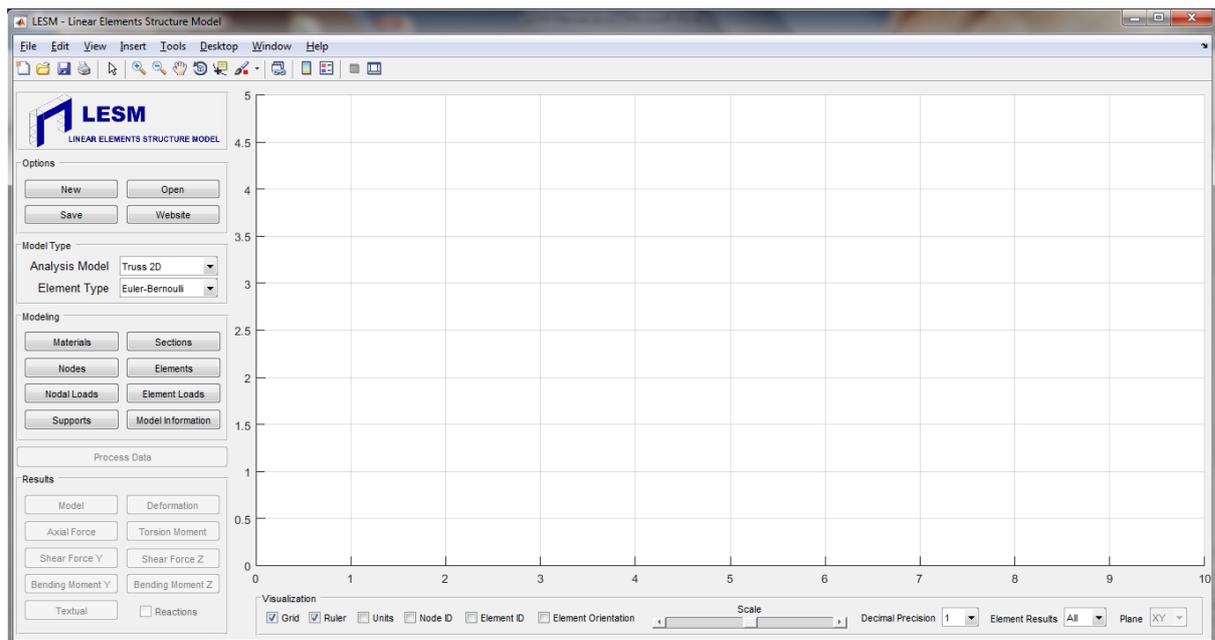
Figure 19: Main window of LESM

12.1 – The Toolbar

In the figure toolbar there are four tools that help in visualizing the models. Right-clicking on canvas when one of this tools is selected, displays a few more visualizing options.

Figure 20: The figure toolbar

Zoom in: In 2D models, it is possible to select the window of visualization. A double-click on canvas returns to the default window of visualization. In 3D models, this tool acts the same way as the Zoom out option, by approaching or moving away the camera view point. The default option in to clip the model when zooming, but it can be changed by changing the "Limits Pan and Zoom" option to the "Camera Pan and Zoom" option on the right-click menu.

Zoom Out: In 2D models, this tool returns the camera to the default window of visualization by left-clicking on canvas. In 3D models, this option acts the same way as the Zoom in option.

Pan: Moves the camera view point, keeping its distance from the model.

Rotate 3D: Rotates the camera view point. This option is only allowed in 3D models.

12.2 – Options

Figure 21: The Options panel

- **About LESM**: Opens a window with the program information.
- **New**: Displays a message to confirm that users really want to reset the current model. This implies deleting all information from the model and cleaning the canvas.
- **Save**: Creates a neutral format file with the *.lsm* extension and saves it in the selected folder.
- **Open**: Opens a dialogue window to allow users to load a file with information from a previously saved model. The program only reads a neutral format file with the *.lsm* extension. If the selected file contains valid information, the model appears on canvas, otherwise an error message shows up. In case the neutral format file has manually been edited, a file with valid information does not mean that the model parameters have consistent values what would lead to analysis errors.

12.3 – Model Type
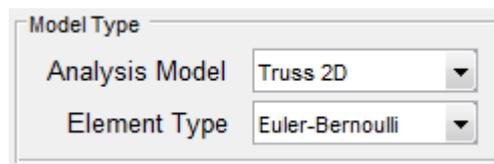
Figure 22: The Model Type panel

- **Analysis Model**: Selects which analysis model will be considered by the program. This option cannot be changed if a model is being built or analyzed (it is considered that a model is being built if there is at least one created node). To change this option, users must delete all model components (nodes and elements) or start a new model. When the analysis model option is changed, the camera view is adjusted to a viewpoint appropriate to the selected option.
- **Element Type**: Selects which beam theory the elements will follow. In LESM, a model can only have elements of the same type. Just like the "Analysis Model" option, this option cannot be changed if a model is being built or analyzed.

12.4 – Modeling

This panel is used to create or delete model components, such as materials, cross-sections, nodes, elements, loads and support conditions. When clicking a button, a new window opens to manage the creation or the editing of the selected component. These auxiliary windows permit users to fill only the input data used by the selected analysis model. When a component is created, deleted or modified, the model is automatically updated on canvas. The modelling options are always enabled so users can start building a model from the beginning or modify already existing models.

Each component (material, section, node or element) is identified by a number that represents the order in which these components were created. When a component is deleted, this order is changed, so that there are no gaps in the numbering sequence (all components with identification number higher than the number of the deleted component have their identification numbers lessened by one).
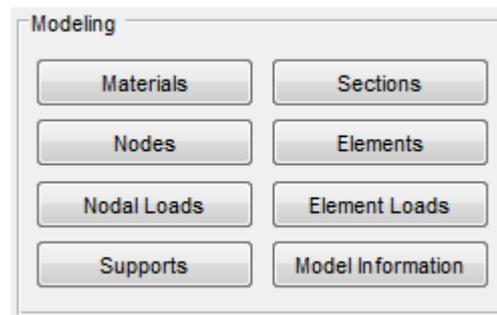
Figure 23: The Modeling panel

12.4.1 – Materials

In the window for managing materials users can create and delete these components as well as see a list with the properties of all created materials. It is only possible to delete materials that are not being used by any element. The properties for creating a material component are independent of the selected analysis model:

- **E** → Elasticity modulus;
- **ν** (nu) → Poison ration;
- **α** (alpha) → Thermal expansion coefficient;

When setting the elasticity modulus and Poison ratio, the program automatically calculates the material shear modulus.
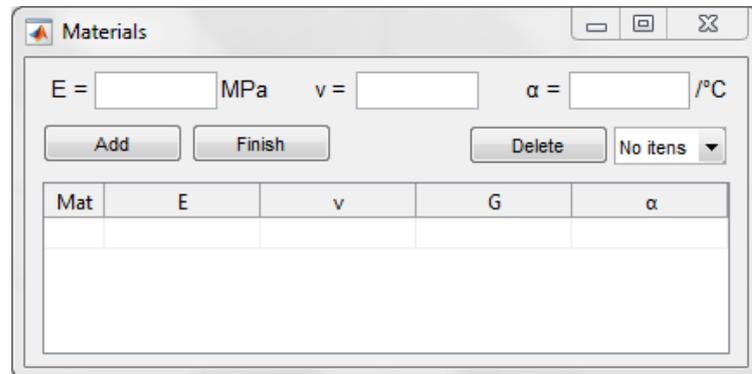
Figure 24: Window for managing materials

12.4.2 – Sections

In the window for managing cross-section users can create and delete these components as well as see a list with the properties of all created cross-sections. It is only possible to delete cross-sections that are not being used by any element. The properties required to create a cross-section, for each analysis model are:

- **Ax** → Full area (all analysis models);
- **Ay** → Effective shear area relative to local y-axis (2D frame and 3D frame);
- **Az** → Effective shear area relative to local z-axis (grillage and 3D frame);
- **Ix** → Moment of inertia relative to local x-axis – torsion inertia (grillage and 3D frame);
- **Iy** → Moment of inertia relative to local y-axis – bending inertia (grillage and 3D frame);
- **Iz** → Moment of inertia relative to local z-axis – bending inertia (2D frame and 3D frame);
- **Hy** → Height relative to local y-axis (2D truss, 2D frame, 3D truss and 3D frame);
- **Hz** → Height relative to local z-axis (grillage, 3D truss and 3D frame);
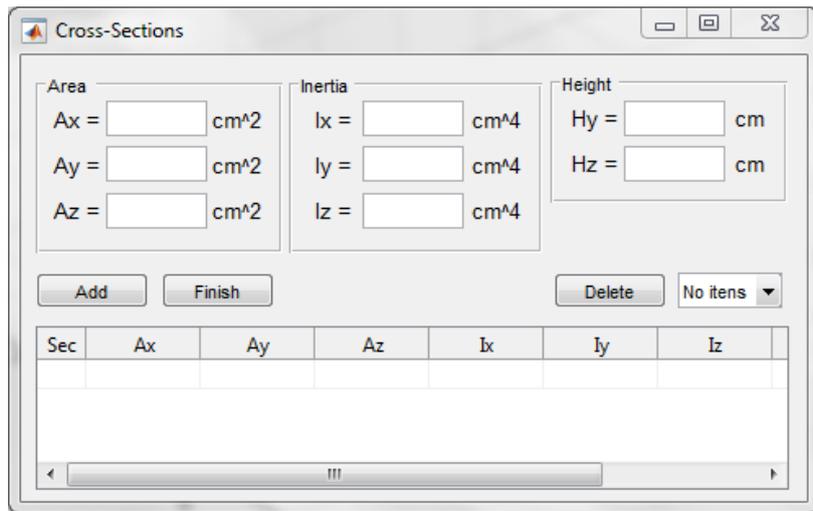
Figure 25: Window for managing cross-sections

12.4.3 – Nodes

The window for managing nodes allows users to create nodal points by setting their coordinates. It is also possible to see a list with the coordinates of all created nodes. It is not possible to create more than one nodal point with the same coordinates, if it happens an error message shows up. If a node is deleted, all the connected elements are also deleted, so a warning message is displayed to confirm this action.
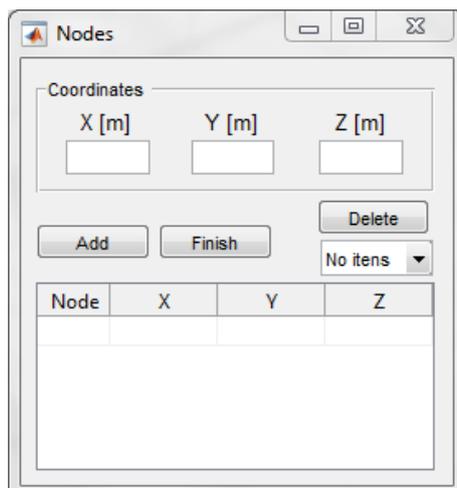
Figure 26: Window for managing nodal points

12.4.4 – Elements

To access the window that deals with elements, it is necessary to create at least one material, one cross-section, and two nodes. In this window users can create and delete elements as well as see a list with the properties of all created elements. To create one, the initial and final nodes cannot be the same; otherwise an error message shows up. The properties for creating an element are independent of the selected analysis model, except for the "Hinge 1" and "Hinge 2" options that are automatically set to "Yes" when the selected analysis model is a 2D truss or a 3D truss, and the auxiliary vector vz that is automatically set to (0,0,1) in 2D models. The required properties to create an element are:

- **Material** → Identification number of element material;
- **Section** → Identification number of element cross-section;
- **Node 1** → Identification number of initial node;
- **Node 2** → Identification number of final node;
- **Hinge 1** → Specification for hinge in the initial node;
- **Hinge 2** → Specification for hinge in the final node;
- **vz** → Auxiliary vector for defining local axes;

In 3D models, vertical elements must have its auxiliary vector changed, since the default direction (0,0,1) coincides with the local x-axis.
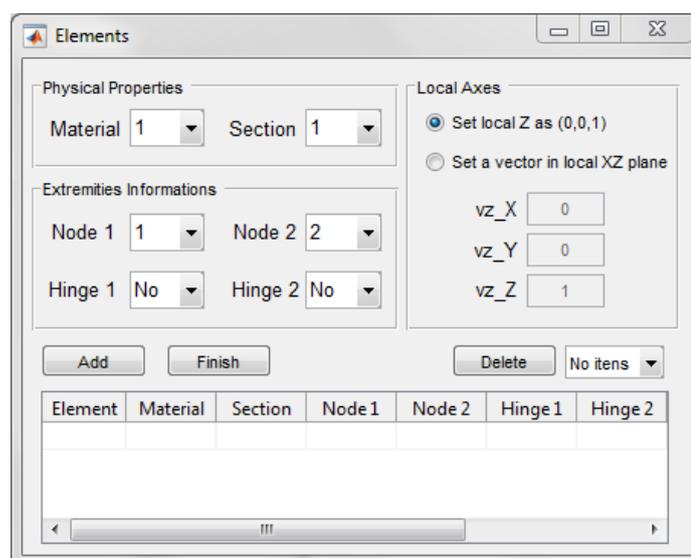
Figure 27: Window for managing elements

12.4.5 – Nodal Loads

To open the window that deals with nodal loads there must be at least one created node. Applied nodal loads are a property inherent to nodes, so they are not created components but a settable property of the node selected in the "Node" pop-up menu. The editable fields for each analysis model are:

- **Fx** → Concentrated load component in global X direction (2D truss, 2D frame, 3D truss and 3D frame);
- **Fy** → Concentrated load component in global Y direction (2D truss, 2D frame, 3D truss and 3D frame);
- **Fz** → Concentrated load component in global Z direction (grillage, 3D truss and 3D frame);
- **Mx** → Concentrated moment component about global X-axis (grillage and 3D frame);
- **My** → Concentrated moment component about global Y-axis (grillage and 3D frame);
- **Mz** → Concentrated moment component about global Z-axis (2D frame and 3D frame);
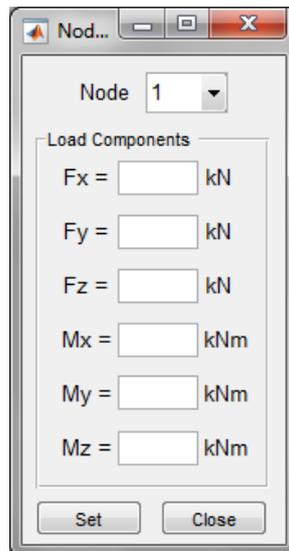
Figure 28: Window for managing nodal loads

12.4.6 – Element Loads

To open the window that deals with element distributed loads and thermal loads there must be at least one created element. Element loads are a property inherent to elements, so they are not created components but a settable property of the element selected in the "Element" pop-up menu. The "Direction" option indicates which axes system (Global or Local) the load values are specified. This option is automatically set to "Global" for distributed loads when the selected analysis model is a grillage, and always automatically set to "Local" for thermal loads. The editable fields for each analysis model are:

- **Qx** → Uniformly distributed load component in global or local X direction (2D truss, 2D frame, 3D truss and 3D frame);
- **Qy** → Uniformly distributed load component in global or local Y direction (2D truss, 2D frame, 3D truss and 3D frame);
- **Qz** → Uniformly distributed load component in global or local Z direction (grillage, 3D truss and 3D frame);
- **Qx1** → Linearly distributed load component value in the initial node in global or local X direction (2D truss, 2D frame, 3D truss and 3D frame);
- **Qx2** → Linearly distributed load component value in the final node in global or local X direction (2D truss, 2D frame, 3D truss and 3D frame);
- **Qy1** → Linearly distributed load component value in the initial node in global or local Y direction (2D truss, 2D frame, 3D truss and 3D frame);
- **Qy2** → Linearly distributed load component value in the final node in global or local Y direction (2D truss, 2D frame, 3D truss and 3D frame);
- **Qz1** → Linearly distributed load component value in the initial node in global or local Z direction (grillage, 3D truss and 3D frame);
- **Qz2** → Linearly distributed load component value in the final node in global or local Z direction (grillage, 3D truss and 3D frame);
- **ΔTx** → Temperature variation on the element center of gravity axis (2D truss, 2D frame, 3D truss and 3D frame);
- **ΔTy** → Temperature gradient relative to local y-axis (2D truss, 2D frame, 3D truss and 3D frame);

- **ΔTz** → Temperature gradient relative to local z-axis (grillage, 3D truss and 3D frame);

The element distributed loads are always drawn in the local axes system, so if a load is given in the global axes system the program automatically converts it to the local system before drawing the model.

It is possible to assign more than one type of load to the same element, which means that an element can have uniform and linear components of distributed load and a thermal load applied to it.
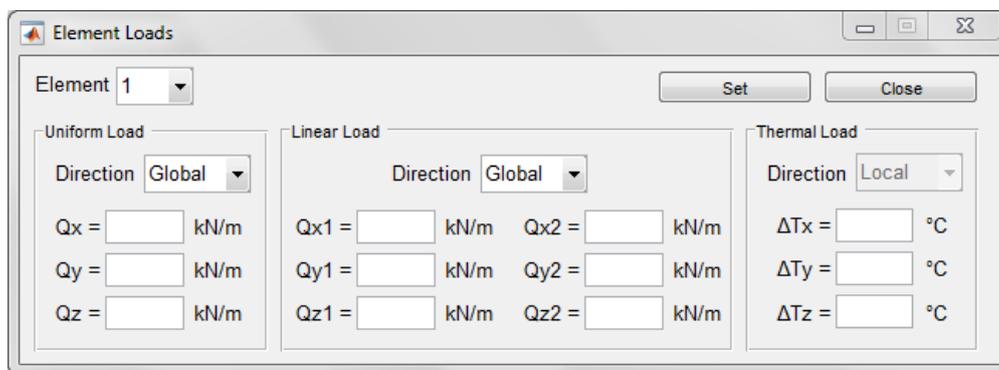
Figure 29: Window for managing element loads

12.4.7 – Supports

To open the window that deals with support conditions and prescribed displacements there must be at least one created node. These properties are inherent to nodes, so they are not created components but settable properties of the node selected in the "Node" pop-up menu. In this window, users can select a node to change its support conditions and prescribed displacements values. The degrees of freedom directions in global system are:

- **Dx** → Translation in global X direction (2D truss, 2D frame, 3D truss and 3D frame);
- **Dy** → Translation in global Y direction (2D truss, 2D frame, 3D truss and 3D frame);
- **Dz** → Translation in global Z direction (grillage, 3D truss and 3D frame);
- **Rx** → Rotation about global X-axis (grillage and 3D frame);
- **Ry** → Rotation about global Y-axis (grillage and 3D frame);
- **Rz** → Rotation about global Z-axis (2D frame and 3D frame);

Each degree of freedom of the selected analysis model can be constrained by checking the box of its direction. A constrained degree of freedom means that the node cannot have any displacement in its direction. Because all the rotations are coupled, by restricting or releasing the rotation in one direction, all rotations are automatically restricted or released.

It is only possible to set prescribed displacements and rotations in the directions of constrained degrees of freedom.
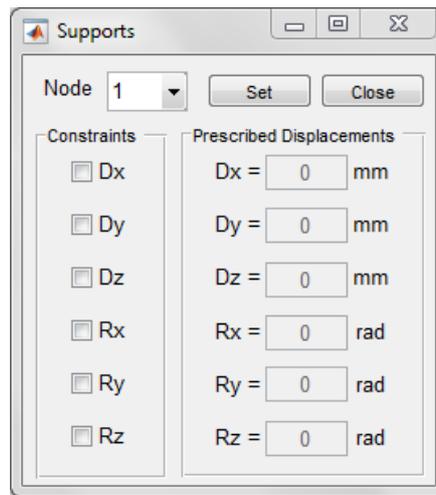

Figure 30: Window for managing support conditions

12.4.8 – Model Information

This option creates and opens a text file with the following information about the current model:

- Analysis model;
- Model description (number of nodes, elements, materials, cross-sections, loads, etc);
- Material properties;
- Cross-section properties;
- Nodal coordinates;
- Support conditions;
- Nodal prescribed displacements;
- Nodal loads;
- Elements information;

- Uniform element loads;

- Linear element loads;

- Temperature variation;

12.5 – Results
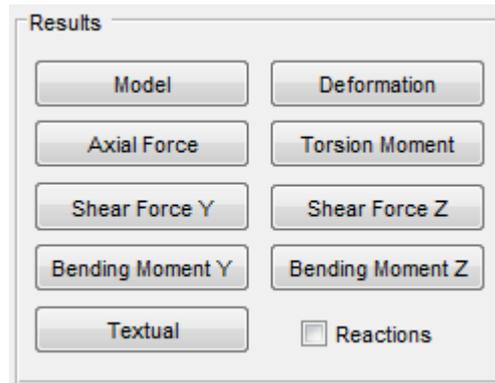
Figure 31: The results panel

The data processing button is responsible for running the code that computes the analysis results using the direct stiffness method. This button is only enabled when it is identified that a model is being built (existence of at least one nodal point). When this button is clicked and the data is successfully processed, the results options are enabled and the data processing button is disabled until a change is made to the model. Every time a model is loaded or modified, its data must be processed before checking the results. If the data is not successfully processed, which is the case of an unstable structure, a warning is issued and no changes are made.

Only the result options that provide the internal forces supported by the selected analysis model are enabled. When a button is clicked, it changes it color to indicate which result is being shown on canvas.

The interpretation of the internal forces diagrams is based on the convention adopted to determine the bottom and upper faces of elements (section 4) and the convention for positive directions of internal forces (section 8).

Model

The "Model" button is not considered as a result option, but rather an option to show only the structural model with nodes, supports, elements, loads and prescribed displacements.

Axial Force

The axial force diagram is available for 2D truss, 2D frame, 3D truss and 3D frame models. In frame models, its positive values are plotted on the upper side of local y-axis of elements. In truss models, axial forces are represented only by their values next to each element since it is always constant.

Torsion Moment

Torsion moment is available only in grillage and 3D frame models, and it is represented only by its value next to each element, since it is always constant because there is no distributed moment on elements.

Shear Force Y

Shear force that acts in the direction of local y-axis. It is an internal force characteristic of 2D frame and 3D frame models. Positive shear force diagram values are plotted on the upper side of elements relative to local y-axis.

Shear Force Z

Shear force that acts in the direction of local z axis. It is an internal force characteristic of grillage and 3D frame models. Positive shear force diagram values are plotted on the upper side of elements relative to local z axis.

Bending Moment Y

Bending moment about local y-axis (acts in the local xz-plane). The bending moment diagram about local y-axis is available for grillage and 3D frame models. Positive bending moment diagram values are plotted on the tension side of elements relative to local z-axis.

Bending Moment Z

Bending moment about local z-axis (acts in the local xy-plane). The bending moment diagram about local z-axis is available for 2D frame and 3D frame models. Positive bending moment diagram values are plotted on the tension side of elements relative to local y-axis.

Textual

By clicking this option, a text file is created and opened with the same model information given in section 11.4.8, and the same textual results given by the non-graphical version (section 10).

Reactions

This option displays the indication and values of support reactions – forces and moments acting on constrained degrees of freedom of each node in global directions.

12.6 – Visualization

Figure 32: The Visualization panel

This panel groups a couple of visualization options that helps users visualize the model and interpret the analysis results:

- **Grid**: Displays a grid on canvas.
- **Ruler**: Displays the coordinates of the global axes on the side of the canvas.
- **Units**: Displays the units of values.
- **Node ID**: Displays the identification number of each node.
- **Element ID**: Displays the identification number of each element.
- **Element Orientation**: Displays the orientation of the local axes of each element.

- **Scale**: Adjust the scale factor of the deformed configuration and the internal forces diagrams.
- **Decimal Precision**: Sets the number of decimal places exhibited by the graphical results.
- **Element Results**: Selects which element will have its internal forces diagrams displayed.
- **Plane**: Selects the plane of visualization. This option is only available in 3D models.