LESM

Linear Elements Structure Model

Version 2.0 – October 2018

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

by

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

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

Pedro Cortez Lopes (cortezpedro@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

PART 1 – PROGRAM CONSIDERATIONS

1 – General Characteristics

LESM is a MATLAB program for linear-elastic, displacement-based, static analysis of bi-dimensional and tri-dimensional linear elements structure models, using the direct stiffness method. For each structural analysis, the program assembles a system of equations, solves the system and displays the analysis results.

The program may be used in a non-graphical version or in a GUI (Graphical User Interface) version. The non-graphical 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.

The first version of the program is associated with the book *Análise Matricial de Estruturas com Orientação a Objetos* (Martha, 2018). Version 2.0 is more advanced in terms of modeling capabilities, as mouse modeling has been enabled and interactions between user and model have been improved. One of the main goals of this version update was, in fact, the achievement of user-friendliness. Also, spring supports, load cases and load case combinations were added to the program. All new features are detailed in this user guide.

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 by LESM to locate points or define directions in space. Both of these systems 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.

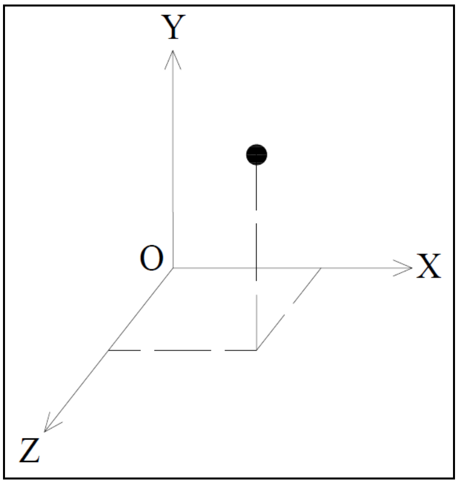


Figure 1: The global coordinate system

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.

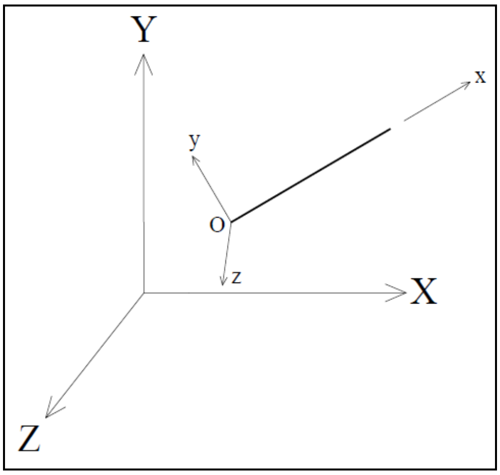


Figure 2: The local coordinate system

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.

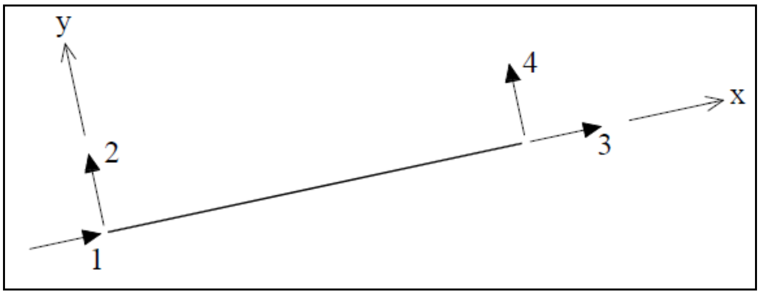


Figure 3: Degrees of freedom positive directions of a 2D truss element in local system

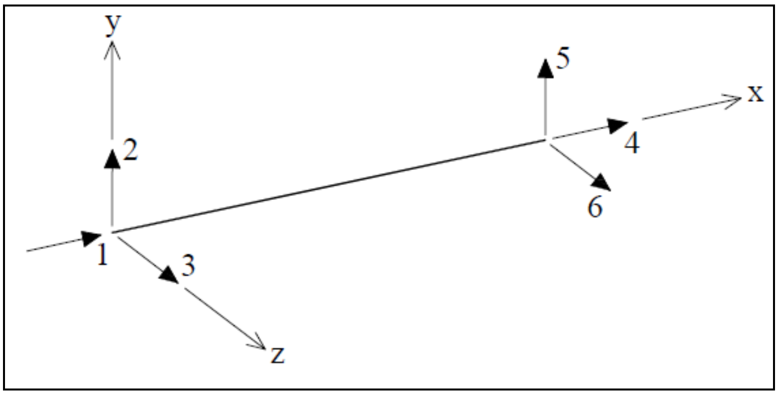


Figure 4: Degrees of freedom positive directions of a 3D truss element in local system

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.

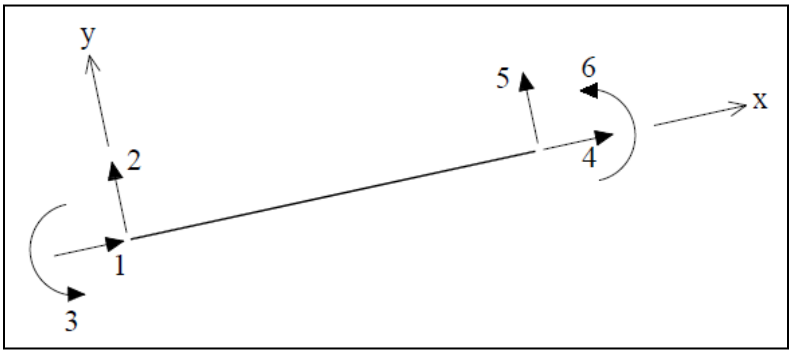


Figure 5: Degrees of freedom positive directions of a 2D frame element in local system

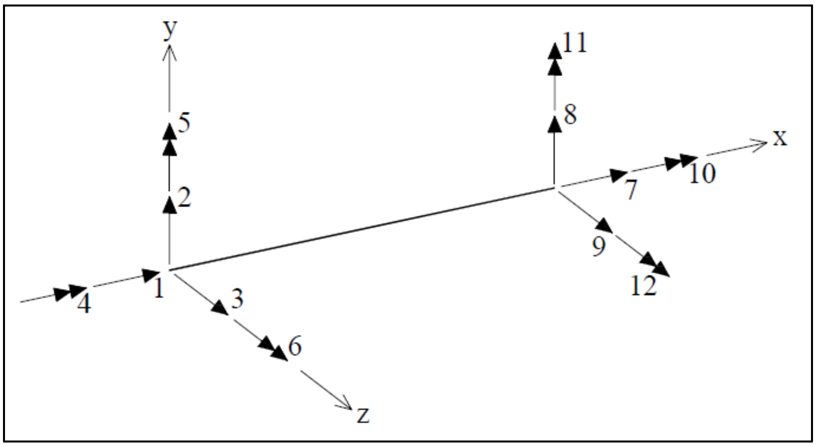


Figure 6: Degrees of freedom positive directions of a 3D frame element in local system

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.

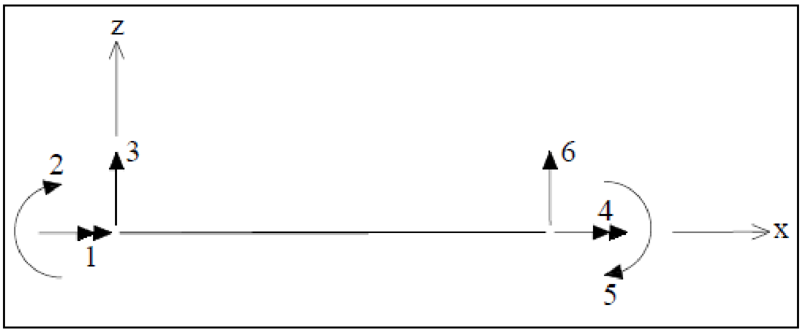


Figure 7: Degrees of freedom positive directions of a grillage element in local system

4 – Element Types

For frame and grillage models, whose members have bending effects, the two following 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.

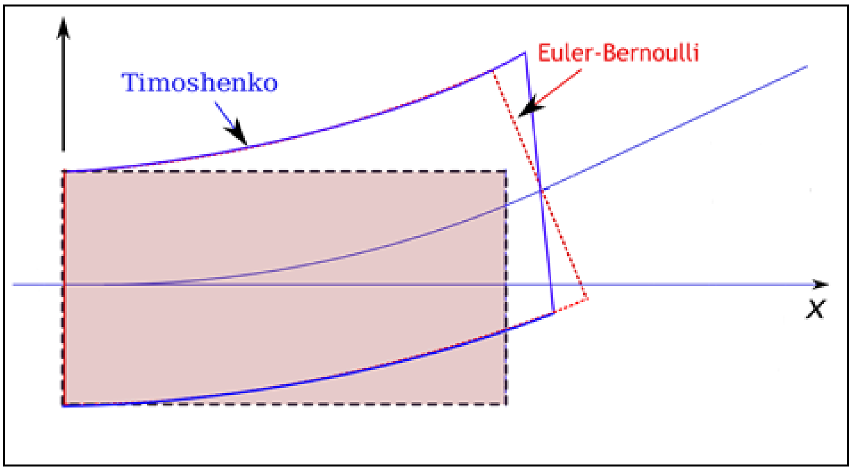


Figure 8: Comparison between Euler-Bernoulli element and Timoshenko element

For truss models, both types of elements are indistinguishable, as considerations for axial behavior are equivalent.

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.

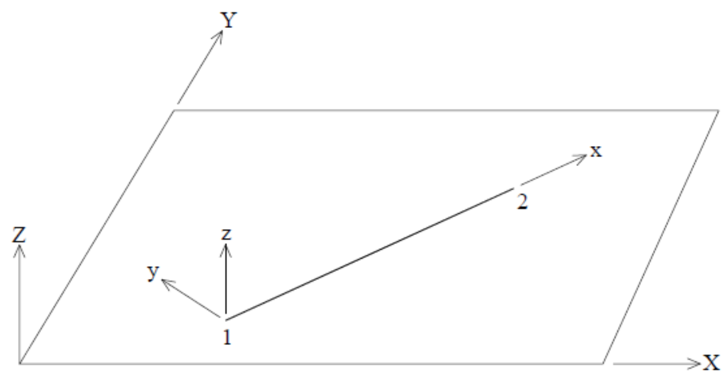


Figure 9: Local axes of elements in 2D models

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;

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

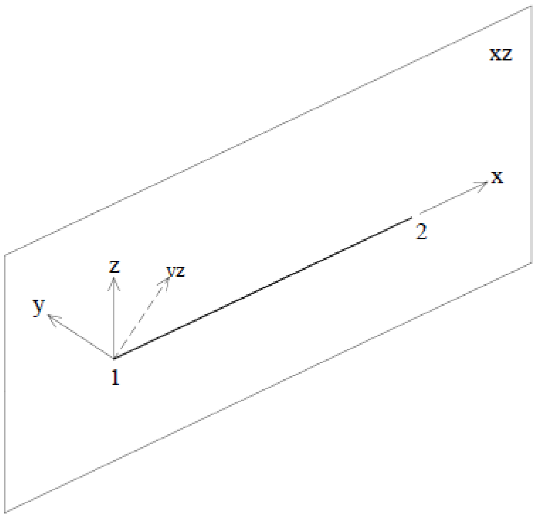


Figure 10: Local axes of elements in 3D models

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.

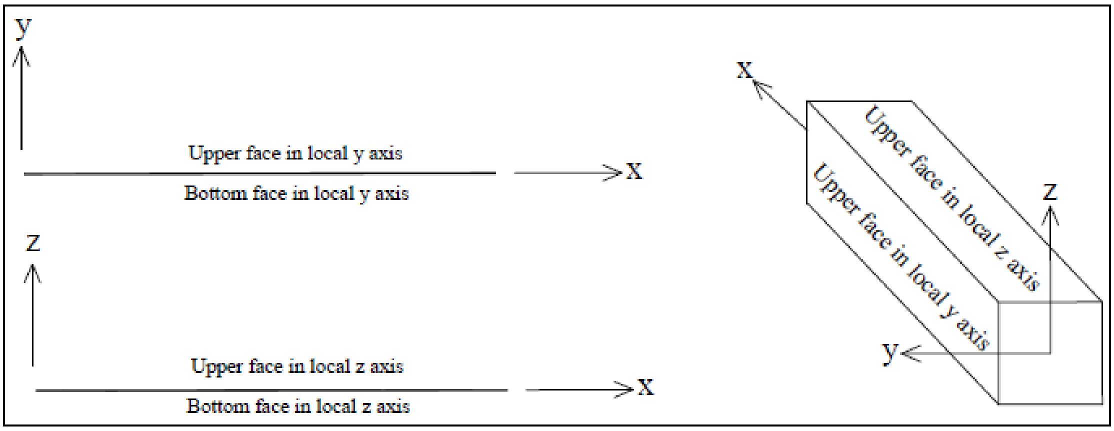


Figure 11: Bottom and upper faces of elements

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.

Different load cases may be defined and combined with specified multiplication factors. However, LESM processes only one load case/combination at a time.

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.

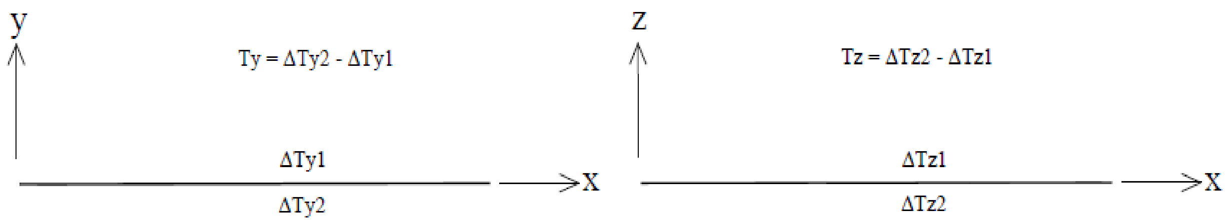


Figure 12: Temperature gradient convention

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.

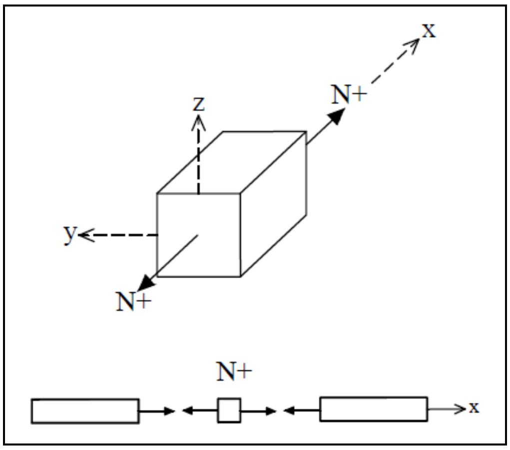


Figure 13: Positive internal axial force

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.

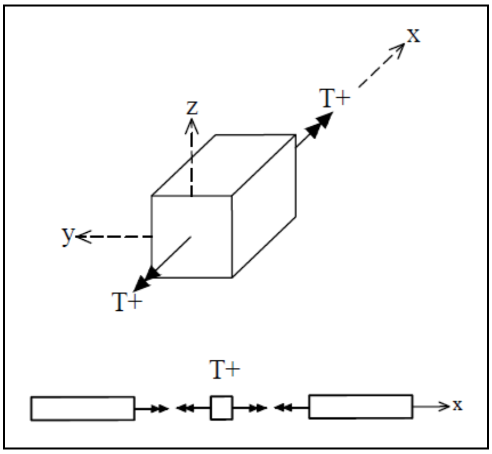


Figure 14: Positive internal torsion moment

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.

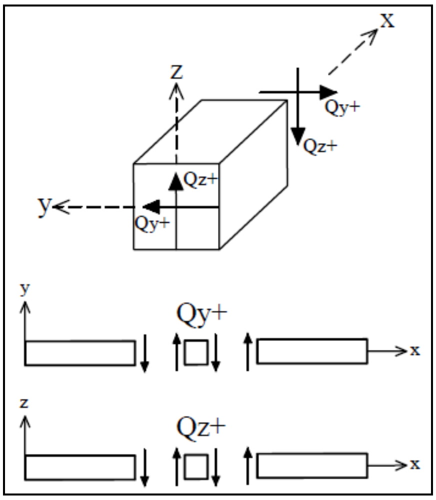


Figure 15: Positive internal shear force

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.

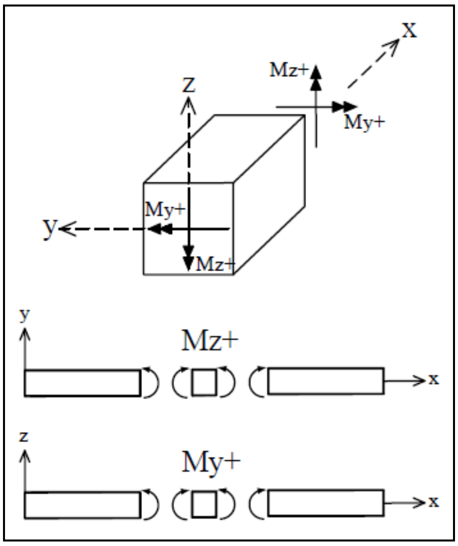


Figure 16: Positive internal bending moment

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 – Supports and Springs

In LESM, it is possible to set complete restriction of movement and/or rotation to nodes, as well as displacement and rotational springs with specified stiffness. It is considered that nodal prescribed displacements may only be assigned to ideal supports.

On models with 3D behavior, it is assumed that restrictions of rotation act on all directions.

11 – 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 [cm^2]
- **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]

PART 2 – USING THE PROGRAM

12 – 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 *lesm.m* file in any MATLAB version (preferably in a recent version).

To run the non-graphical version of the program, users must specify the mode of usage, by setting the “mode” variable as 0, on line 347 of the script, and inform a path to the neutral format file with the *.lsm* extension containing target model information, as a string assigned to the “fileName” variable, on line 350, where indicated. If the name or the path to the input file is not valid, a message is shown in the command window.

```
%% User Input
% Edit the following lines and run this script to initialize LESM

% Select LESM mode (0 = Nongraphical; 1 = Graphical)
mode = 0;

% Enter filename and path (necessary for nongraphical mode only)
fileName = 'Models/Grillage_1.lsm';

%% Program Initialization
% ATTENTION:DO NOT EDIT BEYOND THIS POINT!

clc
clearvars -except mode fileName
close all

% Switch between nongraphical and graphical mode
if mode == 0 % NONGRAPHICAL MODE
    addpath('analysis','file','print');

    % Initialize analysis driver object
    drv = Drv();

    % Open input file with model information
    fid = fopen(fileName,'rt');

    % Check for valid input file
    if fid > 0
        % Read model information
        fprintf(1,'Pre-processing...\n');
        [vs,print,~,lc,~,~] = readFile(fid,drv,mode);

        % Check input file version compatibility
        if vs == 1
            % Process provided data according to the direct stiffness method
            drv.fictRotConstraint(1); % Create ficticious rotation constraints
            status = drv.process();   % Process data
            drv.fictRotConstraint(0); % Remove ficticious rotation constraints
```

Figure 17: Non-graphical version of LESM

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

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:

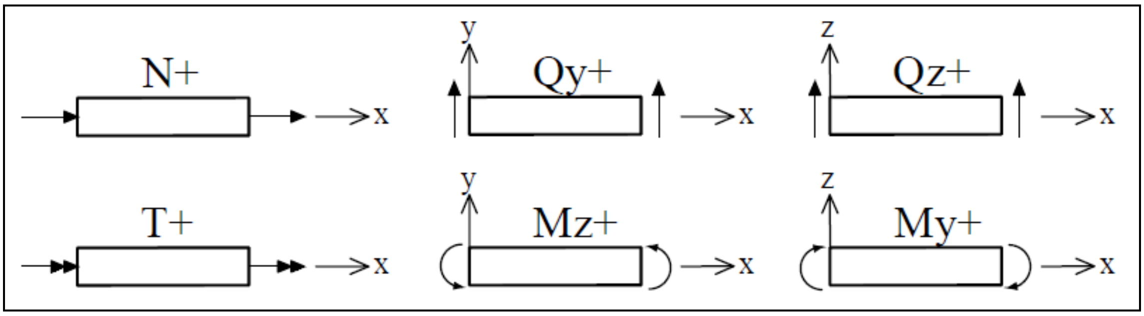


Figure 18: Positive internal forces convention in textual results

13 – User Interface

To run the graphical version of LESM, users may download the standalone executable, the packaged application or the source code, running it on the graphical mode, via MATLAB prompt.

The graphical user interface of LESM is composed by a main window, where it is possible to load, create, model through mouse interactions, view and analyze a structural model, and a set of auxiliary windows for getting the input data necessary to properly characterize 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.

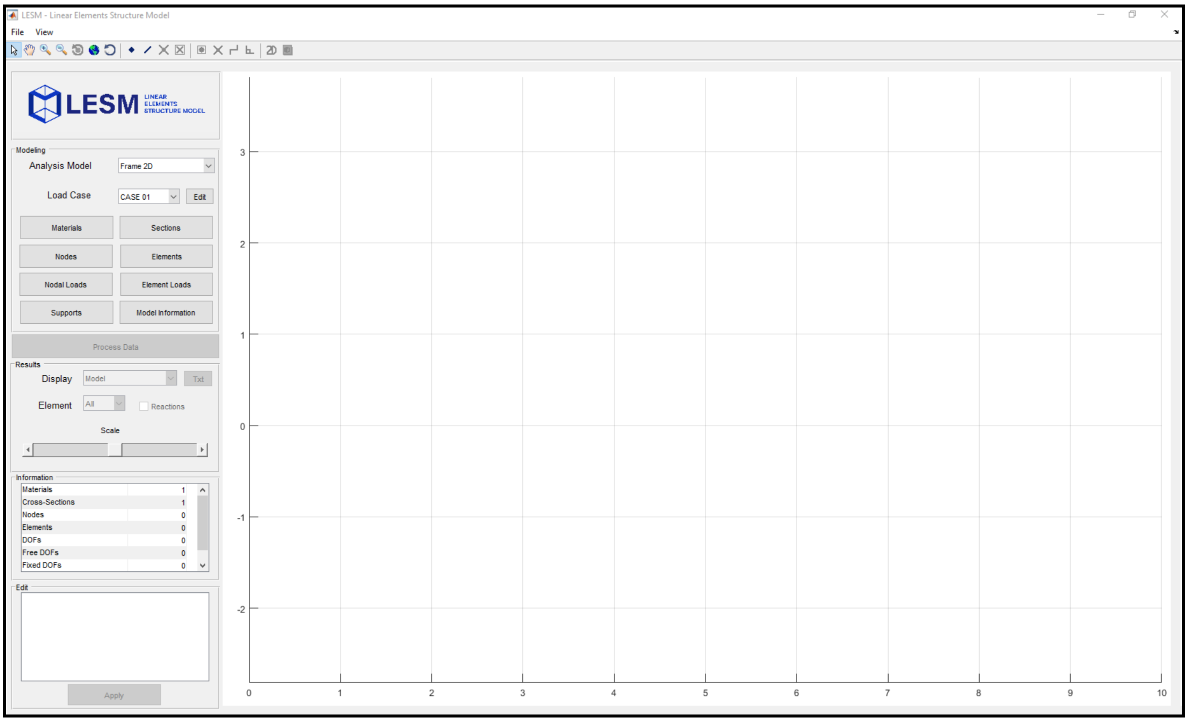


Figure 19: Main window of LESM

13.1 – Dropdown Menus

Two dropdown menus are displayed on the upper left corner of the main window, *File* and *View*, both employed to do auxiliary tasks, regarding file management and visualization options.

File

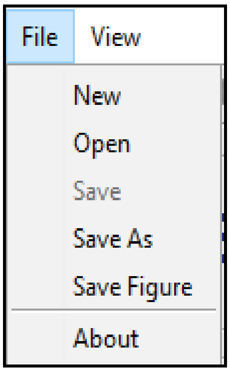


Figure 20: The File dropdown menu

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.

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 user manually edits the neutral format file prior to opening it, caution is advised, as valid information for opening the file does not guarantee that the model parameters have consistent values, what could lead to analysis errors.

Save: If model is already associated with a *.lsm* file, rewrites the file, on the same location, with the same name, saving current model information. If there is no *.lsm* file yet, does the same as *Save As*.

Save As: Creates a neutral format file with the *.lsm* extension and saves it in the selected folder with provided name.

Save Figure: Creates an image file in one of five possible formats (*.jpeg*, *.png*, *.eps*, *.emf* and *.svg*) containing a screenshot of the canvas.

About: Opens a window with the program information.

View

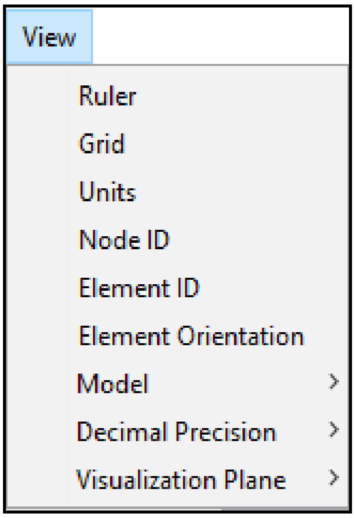


Figure 21: The View dropdown menu

Ruler: Turns axes rulers on/off.

Grid: Turns axes grid on/off.

Units: Displays/hides the units of values.

Node ID: Displays/hides the identification number of each node.

Element ID: Displays/hides the identification number of each element.

Element Orientation: Displays/hides the orientation of the local axes of each element.

Model: Opens a secondary dropdown menu, with components of the model that may or may not be displayed, by choice of the user. The checkmark on the side indicates that that component will be drawn.

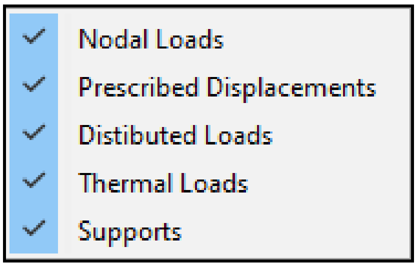


Figure 22: The Model secondary dropdown menu

Decimal Precision: Opens a secondary dropdown menu, with mutually exclusive options of how many decimal places should be used for the values displayed on canvas. Options range from zero to nine decimal places.

Visualization Plane: Opens a secondary dropdown menu, with four mutually exclusive options for points of view of the model. Users may choose between the XY, XZ and YZ planes, as well as an isometric 3D view. Any rotations of view applied to the model automatically set “3D” to be the checkmarked option. This button is only enabled for spatial models.

13.2 – The Toolbar

In the figure toolbar there are tools that help in modeling and visualizing the models.

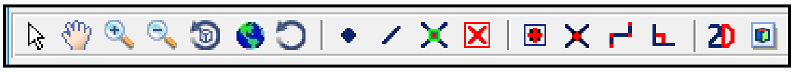


Figure 23: The figure toolbar

Visualization tools

 **Cursor**: Indicates that mouse interactions are on selection mode. Nodes and elements may be picked on the canvas, by mouse clicks, to have their information displayed on the *info panel* or to have new properties assigned to them, on the *edit panel*.

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

 **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.

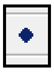
 **Zoom Out**: In 2D models, this tool returns the camera to the default window of visualization by left-clicking on canvas. In 3D models, this option acts the same way as the Zoom in option.

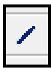
 **Rotate 3D**: Rotates the camera view point. This option is only allowed in 3D models.

 **Fit to View**: Resets canvas visualization limits as to show properly the entire model on the screen.

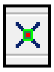
 **Refresh Model**: Redraws entire model. This tool was conceived in order to avoid potentially costly graphic procedures during mouse modeling. To adjust canvas limits after maximizing the LESM main window, users should press this button.

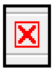
Modeling tools

 **Node**: Indicates that mouse interactions are on node insertion mode. New nodes can be inserted on the model by mouse clicks on the canvas.

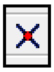
 **Element**: Indicates that mouse interactions are on element insertion mode. New elements can be inserted on the model by mouse clicks on the canvas.

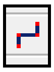
Obs.: The *Node* and *Element* toggle buttons are mutually exclusive.

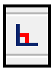
 **Solve Intersections**: Creates new nodes on all element intersections. By default, this button is disabled, becoming enabled only when there are unsolved element intersections, that is, points where elements cross other elements but are not connected by a node.

 **Delete**: Deletes selected node or element. By default, this button is disabled, becoming enabled only when a node or element is selected.

 **Snap to Grid**: Sets coordinate precision for node or element insertion and turns on cursor attraction to grid. By default, this button is disabled, becoming enabled only when *Node* or *Element* toggle buttons are on.

 **Cross Elements**: Indicates that new nodes must be automatically created on any eventual intersections when creating elements. If this toggle-button is off while inserting new elements via mouse interactions on the canvas, crossing points will be understood as unsolved intersections – elements overlap each other on the intersection, but do not cross – that may or not be solved later by the user, by pressing the *Solve Intersections* button, or inserting nodes on its coordinates. By default, this button is disabled, becoming enabled only when *Element* toggle button is on.

 **Polyline**: Enables continuous element modeling, considering the final node of the last inserted element as the first node of the next element. By default, this button is disabled, becoming enabled only when *Element* toggle button is on.

 **Ortho**: Sets coordinate precision for element insertion in order to only create new elements with the inclination of 0º, 45º, 90º, 135º, 180º, 225º, 270º and 315º. By default, this button is disabled, becoming enabled only when *Element* toggle button is on and *Snap to Grid* is off.

Special grillage modeling tools

 **Plane View**: Sets the visualization of grillage models as 2D, on the XY plane. Does not affect any aspect of the model itself, just its visualization. Users are free to choose between creating and handling grillage models in spatial perspective or in 2D.

 **Set Axes Limits**: Sets 3D canvas limits, as a box that defines the grillage modeling workspace on the spatial canvas.

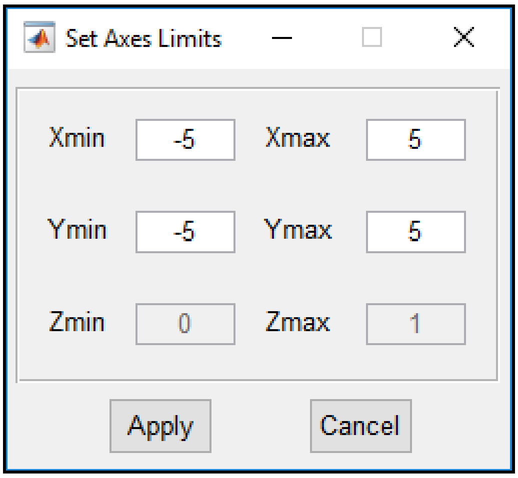


Figure 24: Set axes limits window

13.3 – Modeling Panel

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 only restrained when the current load case is a combination, this is to avoid ambiguity as to where information is being set. Aside from that, the modeling options are always enabled so users can start building a model from the beginning or modify already existing models.

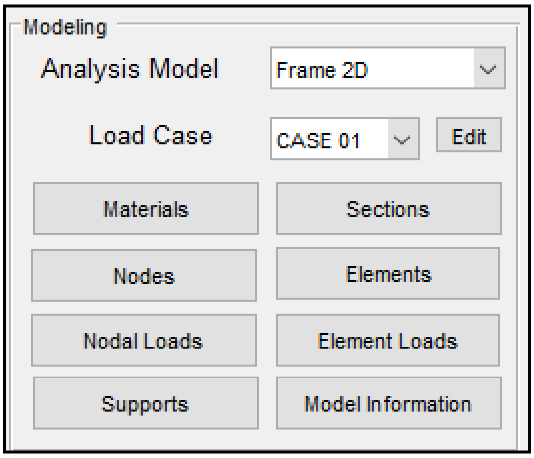


Figure 25: The Modeling panel

- **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.
- **Load Case**: Selects the current load case, or combination, to be analyzed and have loading conditions assigned to it. Load cases may be created and combined on the auxiliary window opened by clicking on the *edit* button.

Each model component (this is valid for 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).

13.3.1 – Edit Load Cases and Combinations

In the window for managing load cases users can create, combine and delete load cases as well as manage combinations between them and see a list with all the existing cases and combinations. A default load case "CASE 01" is created upon initialization of a model. Users may work with this load case or delete it and create other ones, with preferred given names. Note that there must always be, at least, one load case, for that reason, a warning is issued if the user attempts to delete all cases at once, and the *Delete* button on the *Load Cases* panel remains disabled while there is only one load case.

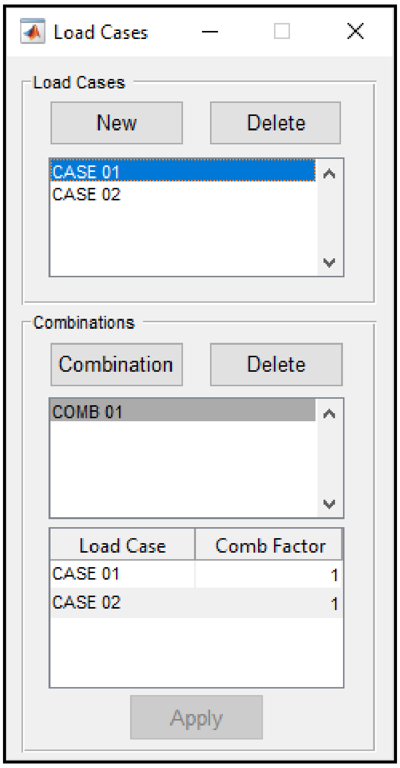


Figure 26: Window for managing load cases and combinations

To create load case combinations, users must select the targeted load cases on the list box located on the *Load Cases* panel, by clicking on them while holding the *Ctrl* or *Shift* key on the keyboard (the first one for specific items, the second for item intervals), then click on the *Combination* button, on the *Combinations* panel. The new combination will have its multiplication factors displayed on the table located on the *Combinations* panel. Default multiplication factors are set as 1.0, users are free to specify different values by editing them on the table and pressing the *Apply* button. The factors may be accessed at any time by selecting the respective combination on the list box. If the data has already been processed, any changes made to the current load case or combination will disable result visualization options and enable the *Process Data* button, on the main window of the program, so that users can rerun the analysis with the new information.

13.3.2 – Materials

In the window for managing materials users can create, modify and delete these components and view a list with the properties of created materials. It is only possible to delete materials that are not being used by any element. All changes made to previously defined materials must be confirmed by pressing the *Apply* button. 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 computes the material shear modulus.

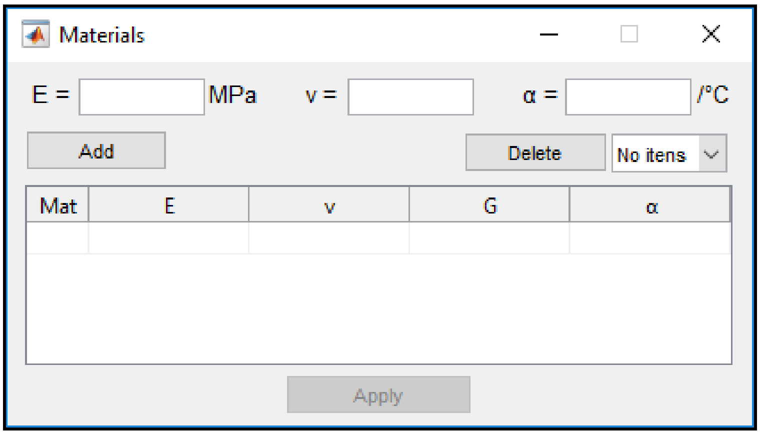


Figure 27: Window for managing materials

13.3.3 – Sections

In the window for managing cross-section users can create, modify 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. All changes made to previously defined cross-sections must be confirmed by pressing the *Apply* button. The properties required to create a cross-section, for each analysis model are the following.

- **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);

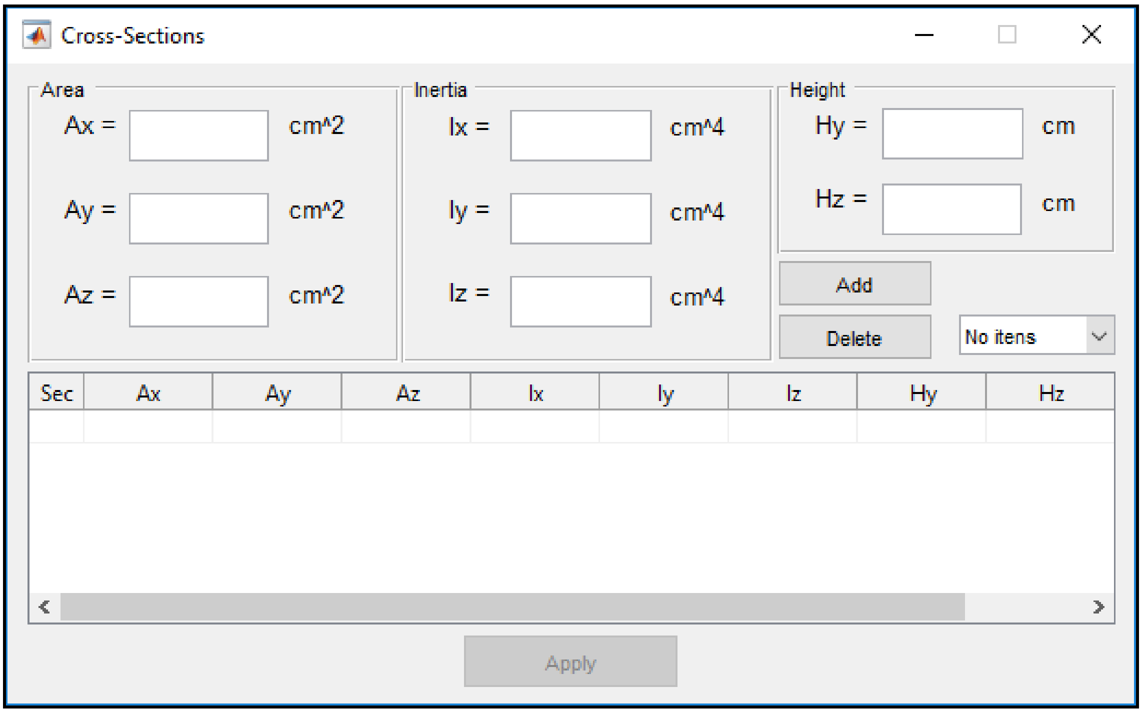


Figure 28: Window for managing cross-sections

13.3.4 – 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 is tried, 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.

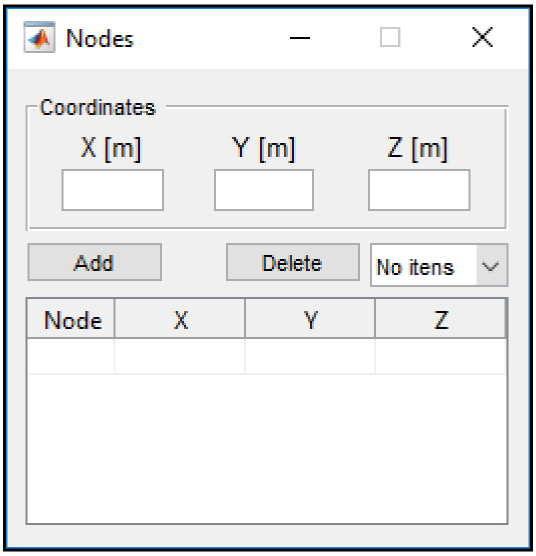


Figure 29: Window for managing nodal points

13.3.5 – 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, modify 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 deformability considerations, "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;
- **Shear Deformation →** Specification for beam flexural behavior theory to be considered (No : Bernoulli-Euler , Yes : Timoshenko);
- **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.

Any changes made to previously defined elements must be confirmed by pressing the *Apply changes* button. Changes made to a specific element may be applied to multiple others by pressing the *Apply to multiple elements* button and informing the ones that must be changed, while the target element is selected on the table.

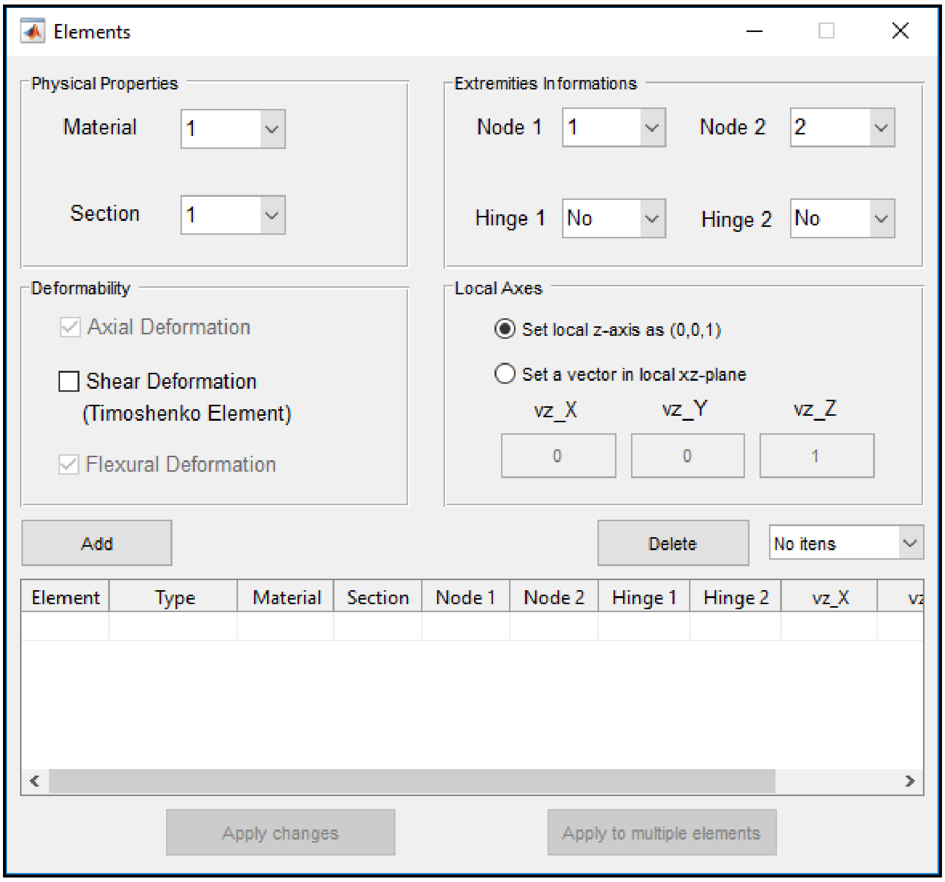


Figure 30: Window for managing elements

13.3.6 – 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 of the model, but a settable property of the node selected in the "Node" pop-up menu. All loads are set to the current load case. 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);

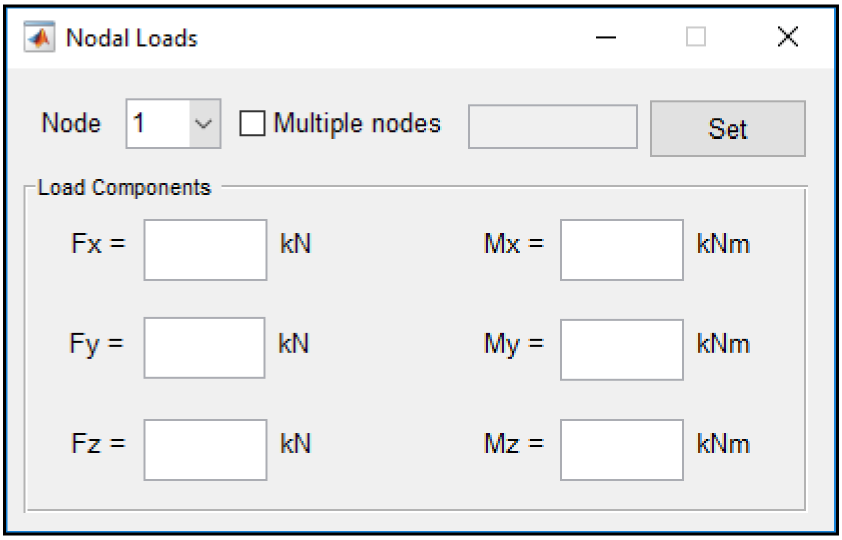


Figure 31: Window for managing nodal loads

Nodal loads may be applied to multiple nodes at once, by checkmarking the *Multiple nodes* checkbox and informing to which nodes must the load be set, on the adjacent editable text box. Specific nodes and node intervals may be given as an input in the text box. For example, by entering “1; 5-7;10”, the load will be set to nodes 1, 5, 6, 7 and 10. Users may also type “All” to apply the load to all existing nodes on the model. Any inconsistent text input will trigger an error message.

13.3.7 – 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 of the model, 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. All loads are set to the current load case. 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.

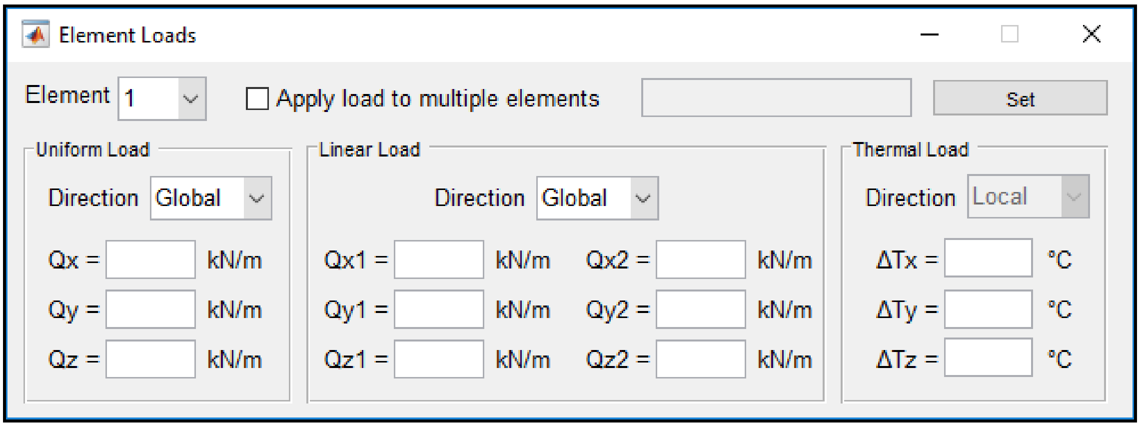


Figure 32: Window for managing element loads

Element loads may be applied to multiple elements at once, by checkmarking the *Apply load to multiple elements* checkbox and informing to which elements must the load be set, on the adjacent editable text box. Specific elements and element intervals may be given as an input in the text box. For example, by entering “4-6; 8”, the load will be set to elements 4, 5, 6 and 8. Users may also type “All” to apply the load to all existing elements. Any inconsistent text input will trigger an error message.

13.3.8 – 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 of the model, but settable properties of the node selected in the "Node" pop-up menu. In this window, users can select a node, or multiple nodes, to change its support conditions, prescribed displacements values and spring stiffness. 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 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. If the spring stiffness checkbox is marked, the constraint on that direction is unmarked, that is, those are mutually exclusive options. Because all the rotations are coupled, by restricting, releasing or assigning a spring stiffness to the rotation in one direction, rotations in all directions are automatically changed as well. It is only possible to set prescribed displacements in the directions of constrained degrees of freedom. All prescribed displacements are associated with the current load case.

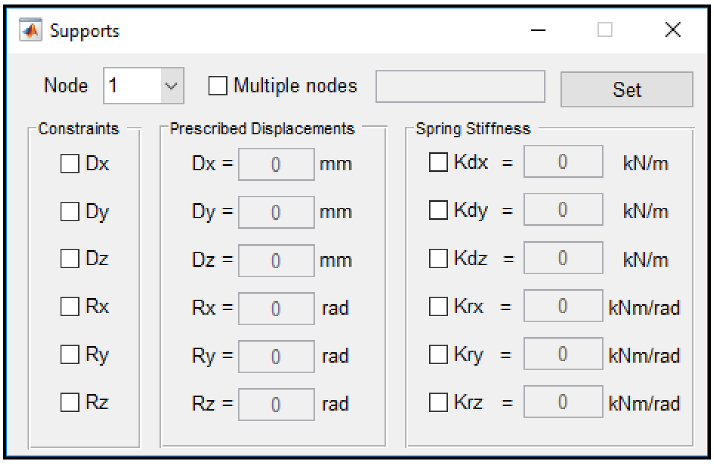


Figure 33: Window for managing support conditions

Support conditions may be applied to multiple nodes at once, by checkmarking the *Multiple nodes* checkbox and informing to which nodes must the support condition be set, on the adjacent editable text box. Specific nodes and node intervals may be given as an input in the text box. For example, by entering “1; 5-7;10”, the support condition will be set to nodes 1, 5, 6, 7 and 10. Users may also type “All” to apply the support condition to all existing nodes on the model. Any inconsistent text input will trigger an error message.

13.3.9 – 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;

13.4 – Data Processing and Results

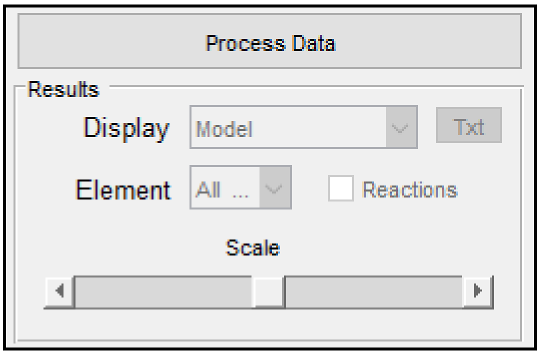


Figure 34: Process Data button and Results panel

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. The loading conditions for data processing are assumed to be according to the current load case or load case combination. 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.

The *Display* pop-up menu contains the visualization options of available results that provide the internal forces supported by the selected analysis model and the deformed configuration. When an option is selected, the drawing on canvas will change to show the requested result diagram.

The *Txt* button generates a *.txt* file with the same model information given in section 13.3.9 and the analysis results, including displacements, internal forces and support reactions.

The *Element* pop-up menu lists all elements for users to choose specific ones for results visualization.

The *Reactions* checkbox signals if reactions - forces and moments acting on constrained degrees of freedom of each node in global directions - must be drawn or not, being marked or not, respectively.

The *Scale* slider controls the plotting scale of result diagrams. When the selected option on the *Display* pop-up menu is the deformed configuration, an additional editable text box is made visible next to the slider, for users to enter specific scale values.

The options of result visualization are detailed on the following topics. The interpretation of the internal forces diagrams is based on the convention adopted to determine the bottom and upper faces of elements (section 5) and the convention for positive directions of internal forces (section 9).

Model

The “Model” option is not considered as a result, 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.

13.5 – Information Panels

Two information panels are located on the lower left corner of the main window of the program. Their purpose is to provide users accessible feedback of updated information about the model and to assist on quick modeling tasks performed on selected nodes or elements.

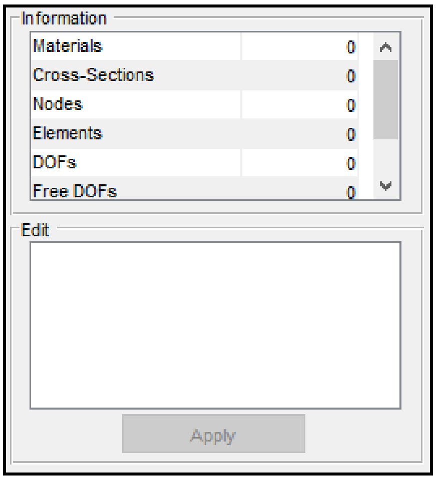


Figure 35: Information panels

The one on top contains static (non-editable) information regarding the model or the selected node or element, including result values of the selection, after data processing, according to the current visualization option on the *Display* pop-up menu.

The lower panel, labeled as *Edit*, is, by default, blank and disabled, being enabled when a node or element is selected. It is filled with the editable information of the selection, according to the selected analysis model, that is, concentrated loads and restrictions of movement and rotation – for nodes – , and the consideration or not of shear deformability, end rotation releases (hinges), material, cross-section, distributed and thermal loads – for elements. Any changes made are required to be confirmed by pressing the *Apply* button, which becomes enabled once a modification is identified.

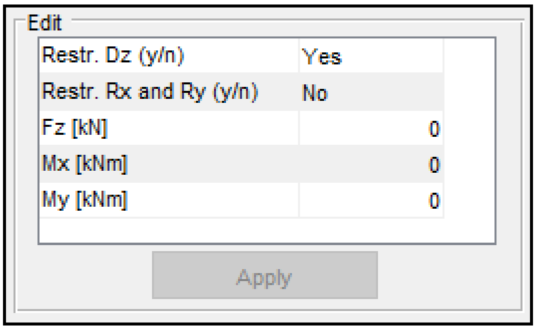


Figure 36: Editable informations of a node on a grillage model

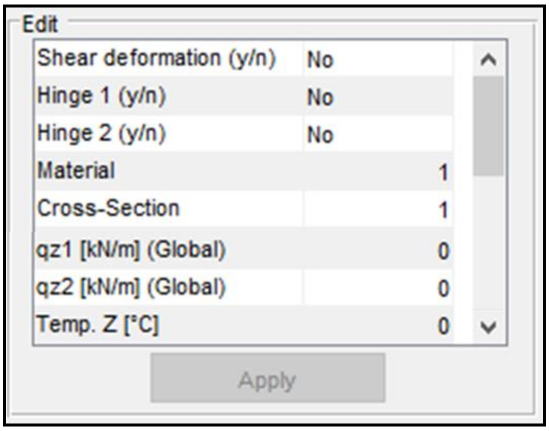


Figure 37: Editable informations of an element on a grillage model

14 – Interactive Mouse and Keyboard Functionalities

Version 2.0 of the LESM program brings significant improvements on interactive modeling capabilities, now being possible to create and handle models via mouse interactions and user-friendly tools such as the aforementioned toolbar and editable information panel. In addition, fifteen keyboard shortcuts were added to assist on managing models faster.

14.1 – Mouse Modeling

There are three mouse modeling modes on the program. One is the selection mode, which takes place when the *Cursor* toggle-button is pressed on the toolbar, and the other two are the node and element insertion modes, associated to the *Node* and *Element* toggle-buttons, respectively.

Selection

The selection mode lets users pick nodes or elements on the canvas through mouse clicks to access its respective data, being possible to modify some of said data on the editable information panel, as seen in section 13.5. It is also possible to delete the selected object, by pressing the *Delete* button on the toolbar or the *Delete* key on the keyboard. Selection functionalities are fully enabled for 2D and 3D models.

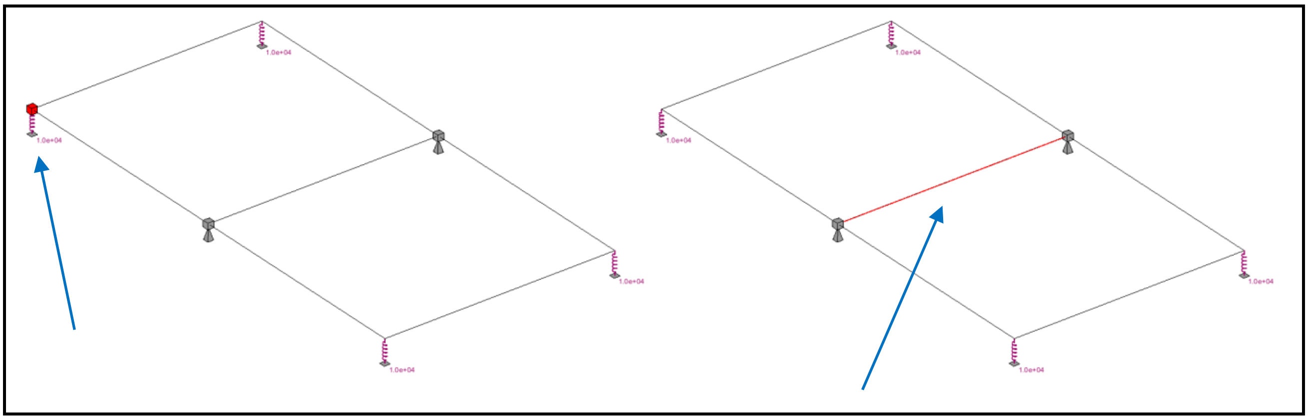


Figure 38: Node and element selection visual feedback

Node insertion

The node insertion mode allows users to insert new nodes to the model by using mouse clicks on the canvas. On 2D and grillage models, nodes may be created anywhere within the canvas limits – it is important to properly set limits of the spatial canvas for grillages, using the *Set Axes Limits* tool, as seen on section 13.2. The process may be aided by turning on the *Snap to Grid* tool, in order to obtain precise nodal coordinates. On 3D frame and 3D truss models, nodes may only be inserted via mouse clicks on points inside existing elements, splitting them in two. That is to avoid uncertainty when dealing with point location on a spatial canvas.

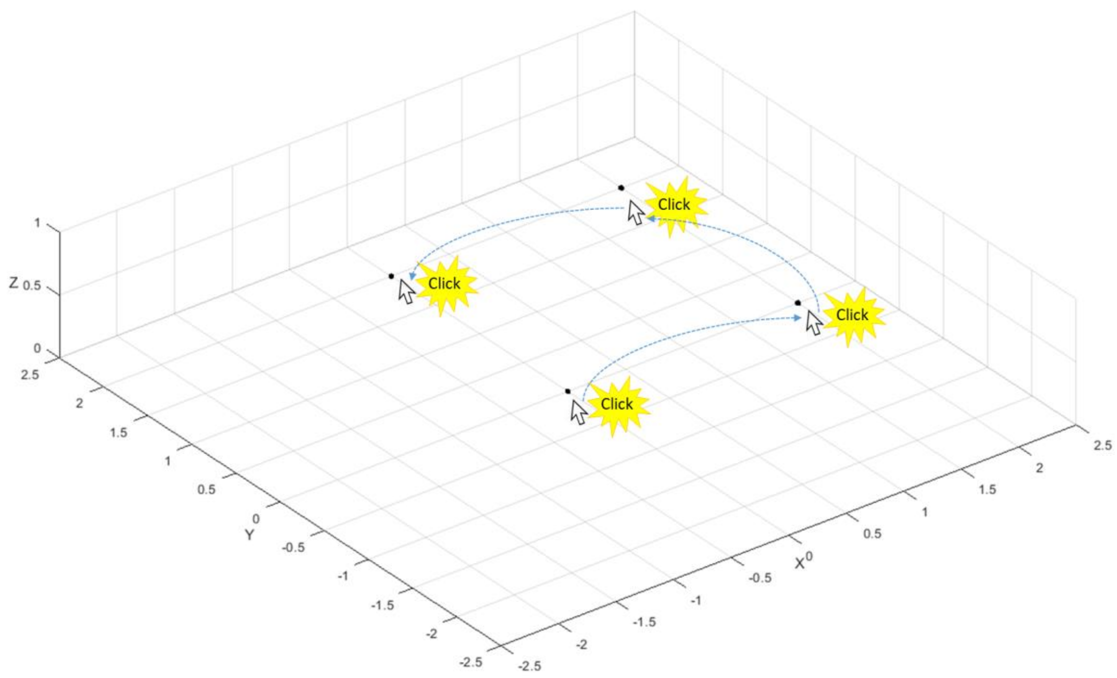


Figure 39: Node insertion via mouse clicks on a grillage model

Element insertion

The element insertion mode lets users enter new elements in the model by using mouse clicks and movements on the canvas. On 2D and grillage models, elements may be created anywhere within the canvas limits. The element ends may be previously defined nodes picked through mouse clicks or new nodes (clicks on empty space) created with the element. The process may be aided by the modeling tools available on the toolbar, presented on section 13.2. On 3D frame and 3D truss models, elements may only be inserted via mouse clicks on points inside existing elements or existing nodes.

That is to avoid uncertainty when dealing with point location on a spatial canvas. In this cases, only the *Cross Elements* and *Polyline* tools are available.

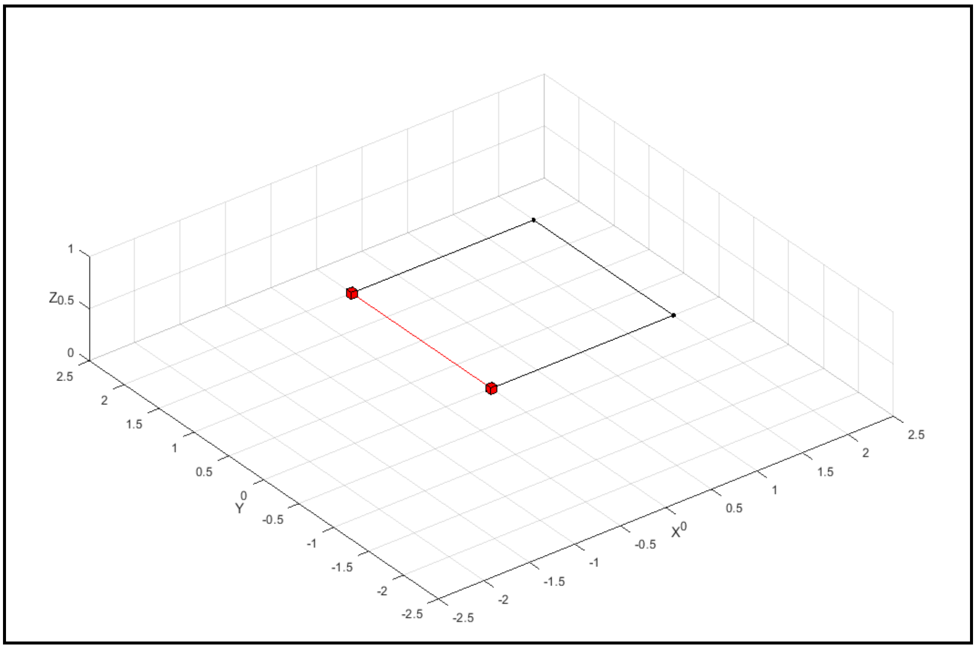


Figure 40: Element insertion visual feedback via mouse clicks and movements on a grillage model

14.2 – Keyboard Shortcuts

The following shortcuts are used by just pressing the respective key, there is no need to press the *Ctrl* key simultaneously.

- **Return (Enter)** → Runs analysis, if the *Process Data* button is enabled;
- **Escape** → Turns off node or element insertion mode – turns on selection mode – or unselects any selected node or element;
- **Delete** → Deletes any selected node or element, may issue to warning messages regarding the model consistency (if deleting a node that is connected to elements, for example), asking for confirmation to perform deletion;
- **C** → Calls auxiliary load cases and combinations window;
- **E** → Turns on element insertion via mouse;
- **G** → Turns on cursor attraction to grid if node or element insertion mode is on;
- **I** → Turns on intersection solving while inserting elements via mouse;
- **L** → Turns on polyline mode while inserting elements via mouse;
- **M** → Calls auxiliary materials window;
- **N** → Turns on node insertion via mouse;
- **O** → Turns on orthogonal mode while inserting elements via mouse;
- **P** → Calls auxiliary nodal loads window;
- **Q** → Calls auxiliary element loads window;
- **R** → Calls auxiliary supports window;
- **S** → Calls auxiliary cross-sections window;