LESM

Linear Elements Structure Model

Version 1.1 – February 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)

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 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. Furthermore, the available public version of the code has incomplete parts so students can fill them as assignments.

All the answers for the expressions and algorithms of the incomplete parts of the code can be found in the book *Análise Matricial de Estruturas com Orientação a Objetos* (Martha, 2018).

More information about the LESM program can be found on its website: http://www.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.

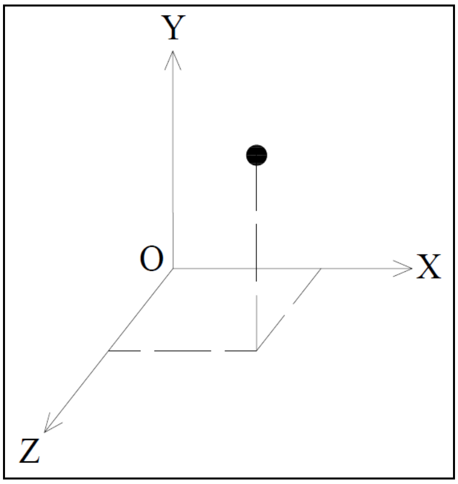

Figure 1: The global coordinate system

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.

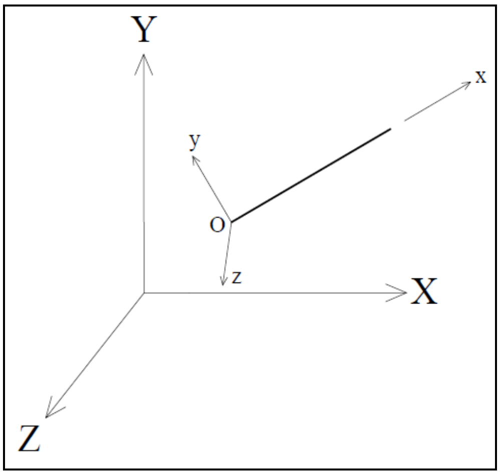

Figure 2: The local coordinate system

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.

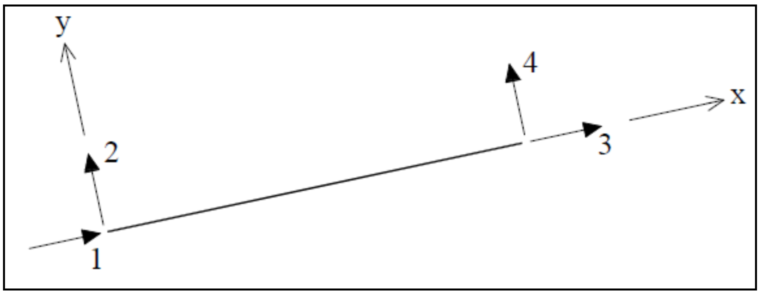

Figure 3: Degrees of freedom positive directions of a 2D truss element in local system

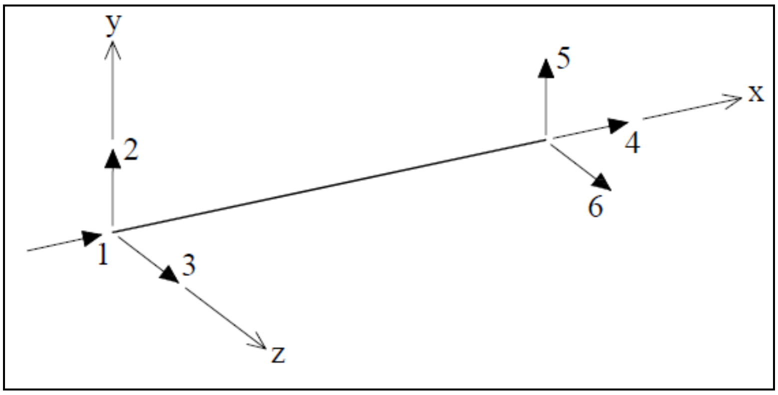

Figure 4: Degrees of freedom positive directions of a 3D truss element in local system

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.

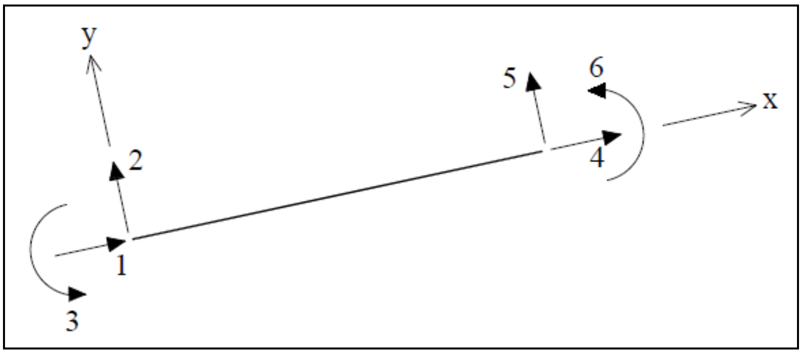

Figure 5: Degrees of freedom positive directions of a 2D frame element in local system

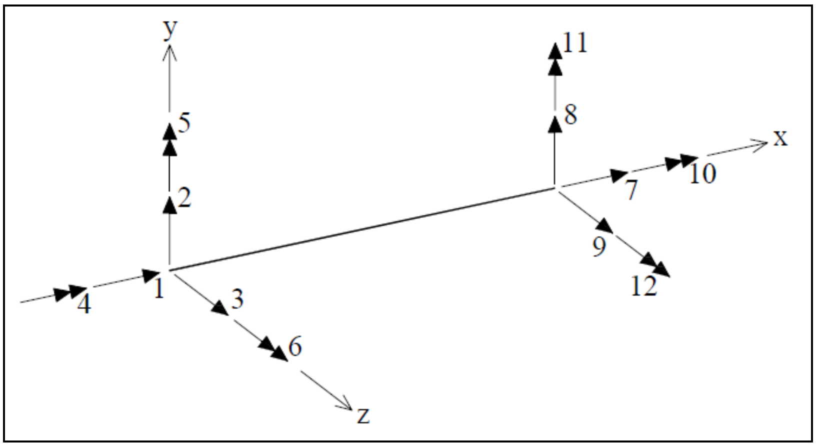

Figure 6: Degrees of freedom positive directions of a 3D frame element in local system

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.

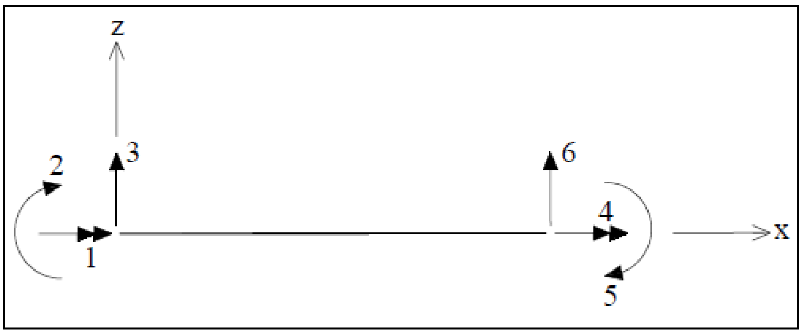

Figure 7: Degrees of freedom positive directions of a grillage element in local system

4 – Element Types

For frame and grillage models, whose members have bending effects, the two following types of beam elements are considered. Members created through mouse modeling are set as Timoshenko elements by default.

- **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.

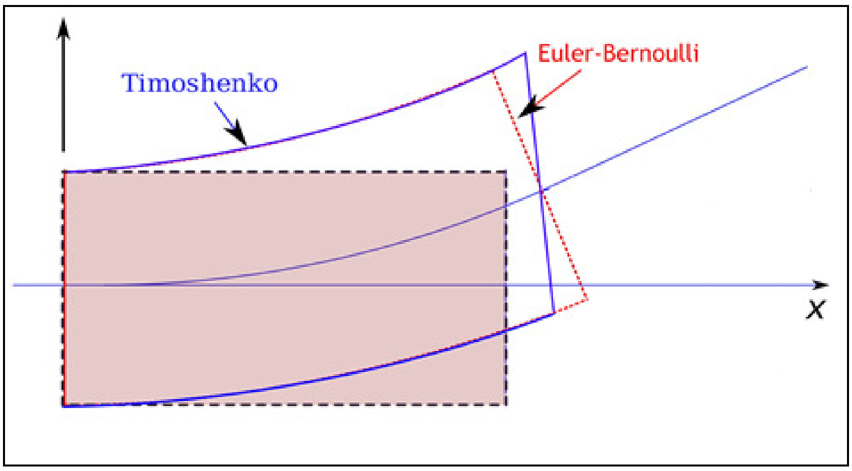

Figure 8: Comparison between Euler-Bernoulli element and Timoshenko element

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.

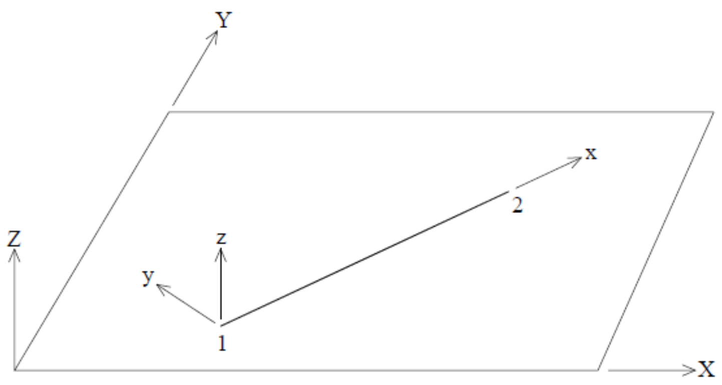

Figure 9: Local axes of elements in 2D models

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;

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

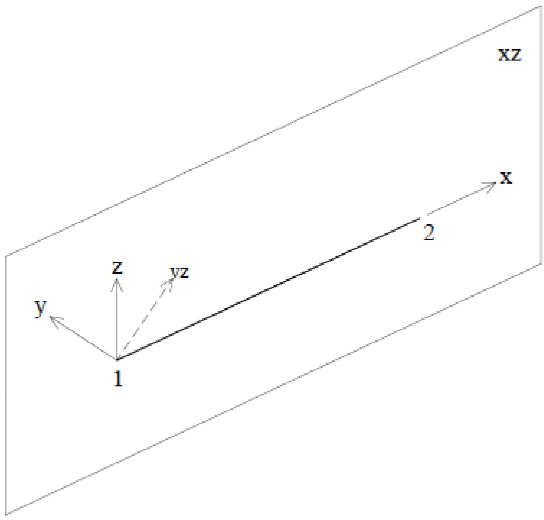

Figure 10: Local axes of elements in 3D models

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.

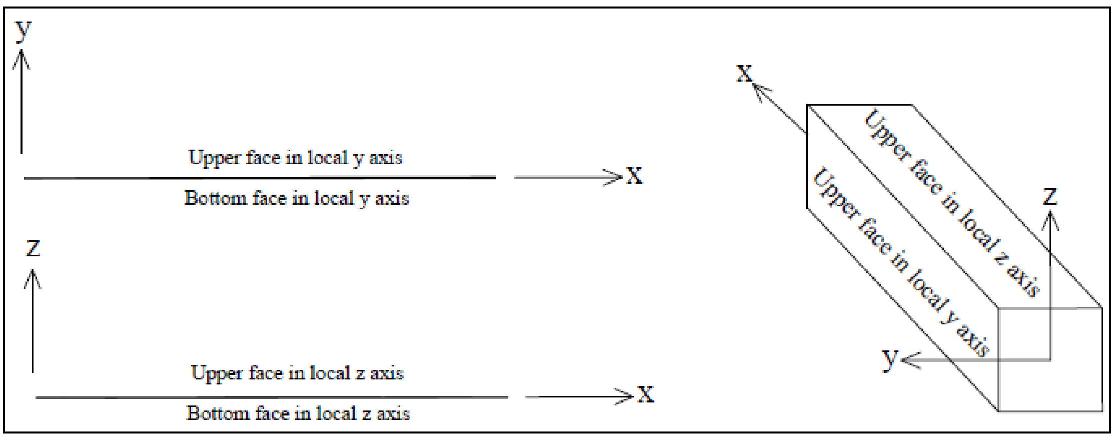

Figure 11: Bottom and upper faces of elements

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.

This version (v1.1) of 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.

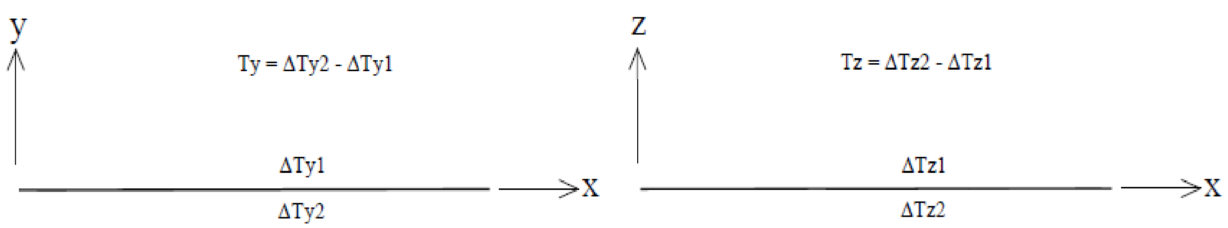

Figure 12: Temperature gradient convention

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.

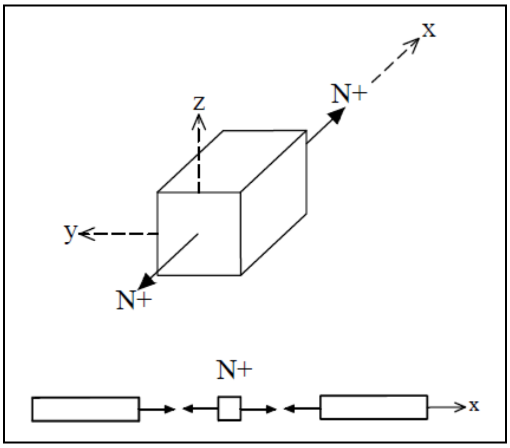

Figure 13: Positive internal axial force

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.

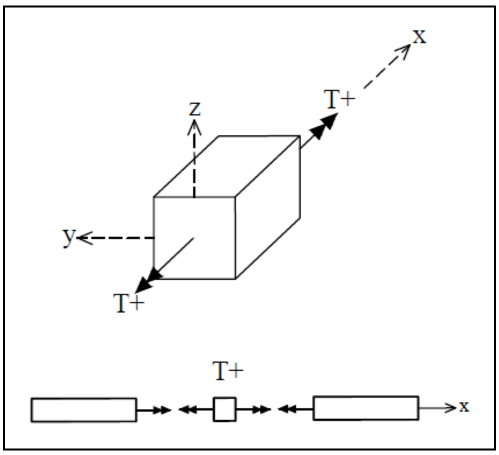

Figure 14: Positive internal torsion moment

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.

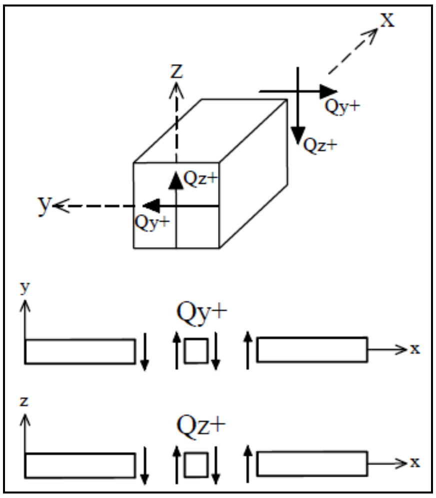

Figure 15: Positive internal shear force

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.

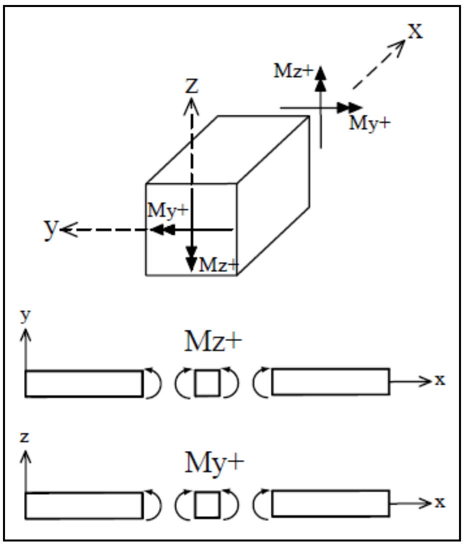

Figure 16: Positive internal bending moment

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 – 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]

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.

```
%% 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:

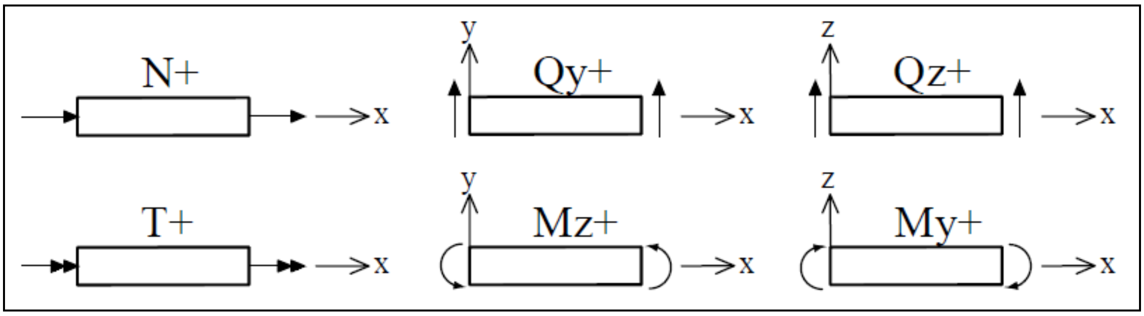

Figure 18: Positive internal forces convention in textual results