FTOOL

Interactive-Graphics Program for Structural Analysis

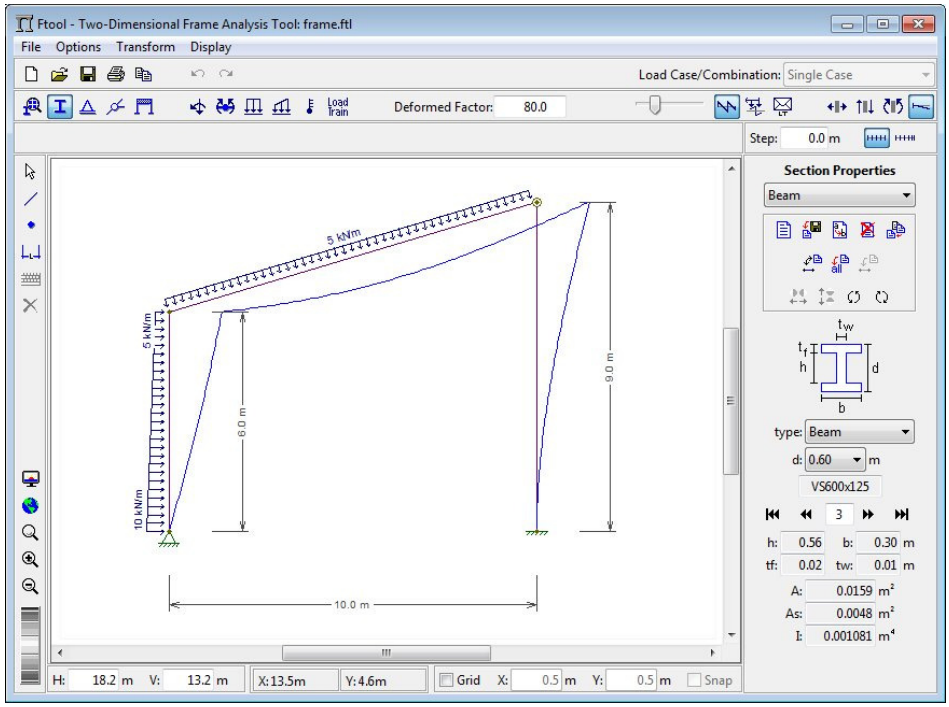
Educational Version 3.00
Lisboa Version
August 2012
http://www.tecgraf.puc-rio.br/ftool

English translation of manual of version 2.11:
Professor Robert H. Dodds, Jr.
Mr. Adam Carlyle
Department of Civil & Environmental Engineering
University of Illinois at Urbana-Champaign
November 2003

Prof. Luiz Fernando Martha - PUC-Rio Rua Marquês de São Vicente, 225 - CEP 22453-900 - Rio de Janeiro, RJ
Tel. +55 21 2512-5984 - Fax. +55 21 3527-1848 - E-mail: lfm@tecgraf.puc-rio.br - URL: http://www.tecgraf.puc-rio.br/~lfm

TABLE OF CONTENTS

FTOOL in Use



FTOOL Authorship

Luiz Fernando Martha
Associate Professor
Pontifical Catholic University of Rio de Janeiro (PUC-Rio)
Department of Civil Engineering and
Computer Graphics Technology Group (Tecgraf/PUC-Rio)
Rua Marquês de São Vicente, 225, Gávea
22453-900 - Rio de Janeiro, RJ - BRASIL
Phone: 55+21+3527-1190
Fax: 55+21+3527-1195
e-mail: lfm@tecgraf.puc-rio.br
URL: http://www.tecgraf.puc-rio.br/~lfm

General Concept

FTOOL is a program designed for instruction on the analysis of plane frame structures. It fulfills a need not often explored by educational programs that usually focus on analytical (numerical) techniques, or by the educational versions of commercial programs. The objective of FTOOL is to motivate learning of structural behavior. Educational experiences in this area have shown that knowledge of structural behavior is essential for better understanding of the analytical methods learning process.

FTOOL provides a simple analysis program that merges, in the same interface, resources for effective creation and manipulation of the model (pre-processing), linked to a fast and effective code for visualization of the results (post-processing).

Downloading FTOOL

- Ftool Version 3.00 for Windows 32 bits (also works in Windows 64 bits): http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftool300win32.zip.
- Ftool Versão 3.00 for Linux - Ubuntu 10.04 (Kernel 2.6 & libc 2.13) - GTK 2 - 32 bits: http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftoo300linux26_32.tgz.
- Ftool Versão 3.00 for Linux - Ubuntu 10.04 (Kernel 2.6 & libc 2.13) - GTK 2 - 64 bits: http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftoo300linux26_64.tgz.
- Ftool Versão 3.00 for Linux - Ubuntu 12.04 (Kernel 3.2 & libc 2.14) - GTK 2 - 32 bits: http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftoo300linux32_32.tgz.
- Ftool Versão 3.00 for Linux - Ubuntu 12.04 (Kernel 3.2 & libc 2.14) - GTK 2 - 64 bits: http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftoo300linux32_64.tgz.
- Download of this guidebook in CHM (*Compiled HTML Help*):
  http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftoolman300-pt.chm (Portuguese)
  http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftoolman300-en.chm (English)
- Download of this guidebook in PDF:
  http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftoolman300-pt.pdf (Portuguese)
  http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftoolman300-en.pdf (English)
- Download of tutorial for the creation and analysis of a simple frame with FTOOL (PDF format, in English): http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftool300tutorialframe.pdf
  FTOOL model of tutorial: http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftool300frame.ftl
- Download of tutorial for the creation and analysis of a truss with FTOOL (PDF format, in English): http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftool300tutorialtruss.pdf
  FTOOL model of tutorial: http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftool300truss.ftl
- Download of tutorial for the creation of a bridge model with two load-trains, and visualization of critical load-train positions along influence lines and of internal force envelops with FTOOL (PDF format, in English): http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftool300tutorialloadtrain.pdf
  FTOOL model of tutorial: http://www.tecgraf.puc-rio.br/ftp_pub/lfm/ftool300 bridge.ftl

Background

FTOOL (Two-dimensional Frame Analysis Tool) was initially developed through an integrated research project coordinated by Professor Marcelo Gattass from the Computer Department of PUC-Rio and leader of the Computer Graphics Technology Group (Tecgraf/PUC-Rio). Professor Gattass received support form CNPq (National Council for Scientific and Technologic Development). The responsiblility for the program is now with Professor Luiz Fernando Martha from the Department of Civil Engineering of the PUC-Rio. Former undergraduate students Eduardo Thadeu Leite Corseuil, Vinícius Samu de Figueiredo and Adriane Cavalieri Barbosa, from Department of Civil Engineering of PUC-Rio, contributed to the development of the program as research assistants from March of 1991 until December of 1992. The program was developed originally on a DOS platform and went through some initial improvements until April of 1995.

Waldemar Celes Filho, professor of the Computer Science Department of PUC-Rio, worked on development of the HED (*Half-Edge Data structure*) function library for the internal data structure. The program's numeric module of analysis received a contribution from the former PhD student of PUC-Rio Ivan Fábio Menezes, (currently professor of the Department of Mechanical Engineering of PUC-Rio).

From 1997 to 1998, FTOOL was revised by Professor Luiz Fernando Martha, who used the interface system IUP and the graphics system CD, both developed by Tecgraf/PUC-Rio. This graphics interface allows the program to be executed on both Windows and Unix. In February of 1998, version 2.00 was launched. Successive versions of FTOOL were then introduced, each one with some improvements.

In August 2000, version 2.08 was also launched on the Linux platform. In version 2.09 (January of 2001) specific procedures for supporting instruction on structural analysis were incorporated: inextensible and infinitely rigid members, and member end moments loading. In the 2.10 version (November 2001) display of influence lines were added. In version 2.11 (August, 2002), the definition of member cross-sections was improved. Other improvements include property attributes and the ability of FTOOL to import model attributes from other files, option to select the member side (tension or compression) to draw bending moment diagrams, and option to draw result values transversely to the members.

Version 2.12 (February, 2008) added a load-train menu with options to create and manipulate lists of generic load-trains, with variable number of concentrated, distributed loads (full and empty cars), and external and internal live loads. In that version, it was created an option to display critical positions of load-trains in relation to current displayed influence lines and an option to display envelops of internal forces due to load-trains. In the last version, version 3.00 (August, 2012), the IUP graphics interface adopted by FTOOL was updated to reflect recent evolution in native interfaces of Windows and Linux. The graphics interface of the Linux version had a considerable improvement because it uses the GTK package. Besides several improvements, one important implementation was the consideration of shear deformations in members (Timoshenko beams). Other important improvement was the elimination of auxiliary file (extension ".pos") that was used to link the program with its analysis module (solver). This file may still be saved, but has to be explicitly demanded by the user.

Throughout its use, FTOOL has shown to be a suitable tool for engineering instruction and has been used in Structural Analysis, Concrete Structures and Steel Structures courses of Civil Engineering in several Brazilian and overseas universities.

New features of the latest version 3.00 (August 2012)

- Graphics interface of program was updated to reflect recent evolution in native interfaces of Windows and Linux. Following this evolution, IUP (www.tecgraf.puc-rio.br/iup) graphics interface system adopted by FTOOL was updated, and in the Linux version package GTK is used. This provides a better user-interface than the previous one that was based on MOTIF.
- Created option for member shear deformation (Timoshenko beams). By default, a member is created without shear deformation.
- Added Poisson ratio to material properties.
- Added effective shear area parameter to all types of cross-section.
- Modified menu that specifies member deformation constraints. There are two exclusive main options: flexible member and rigid member. For flexible member, there are two non-exclusive options: toggle for axial deformation and toggle for shear deformation. When a new model is created, the default toggle options are flexible member with axial deformation of with no shear deformation. If a model is read from a file, and all its members have shear deformation allowed, the shear deformation toggle is set on.
- Set up display of members with shear deformation using a different color (dark magenta).
- When doing post-processing of deformed configuration, and a member is picked to consult results, made available cross-section rotation value in prompt messages. Also, added rotation values to exported deformed configuration result files.
- Eliminated material specific weight property because it has not been used.
- Created options to select display sizes of support, load, and text on the canvas (three options for each: small, medium, and big). Supports and hinges are treated equally for display size definitions.
- Changed display of distributed loads such that the arrow size proportionally reflects load intensity.
- Modified display of member orientation arrow: its size is half of the size used in the previous version.
- Created option to run analysis with or without saving neutral file to pass data to internal analysis module (file with extension ".pos"). The default value of this option is to run analysis without saving a file. The default value of this option is reset when opening an existing model.
- Modified display of influence line to draw a red mark at the reference cross-section.
- Allowed the use of blank spaces in the title of all attributes.
- Prevented modification of orientation of cross-section in post-processing mode.
- Modified display of influence line diagram such that it is only drawn along the members that belong to the chain of members of the load-train path. Before, an influence line diagram was drawn along all members. In case there is no current load-train, the influence line diagram is still drawn along all members.
- Eliminated automatic launch of credits dialog when program is activated.
- Added two key control options: CTRL+Z for undoing pre-processing operation, and CTRL+Y or CTRL+R for redoing last pre-processing undoing operation.
- Fixed mistake in Gerdau-Aço Minas cross-section table, *I Shape* family, depth 610 mm: the number of profiles with this depth (6) was not correct and the navigation was leaking to other family.
- Fixed mistake in Gerdau-Aço Minas cross-section table, *H Shape* family: the program was not allowing the selection of profiles with depth greater than 210 mm.

New features of version 2.12 (February 2008)

- Implemented structural analysis for load-trains. Reference: Envelopes of internal forces due to load-trains in bridges using an evolution strategy. by Gisele C. Holtz, Luiz F. Martha, and Luiz E. Vaz (www.tecgraf.puc-rio.br/ftool/doc/ HoltzWCSMO2005.pdf).
- Created load-train menu: options to create and manipulate lists of generic load-trains, with variable number of concentrated, distributed loads (full and empty cars), and external and internal live loads.
- Created display of critical positions of load-trains in relation to current displayed influence line. The critical positions are the ones that result in minimum and maximum values for the specified internal force result, at the selected section, due to the current selected load-train.
- Created option to visualize internal force minimum and maximum envelop diagrams for the current selected load-train.
- In "Options" drop-down menu, added toggle to add permanent single load effects to load-train envelops.
- Made a complete revision of Linux version graphics interface: redefined fonts and sizes of user-interface elements.
- Improved user-interface dialogs for data input through the keyboard. These dialogs use Ftool's number formatting and unit convention.
- Renamed cross-section table NBR Welded I-shapes to Welded I-shapes (BR).
- Renamed cross-section table Usilight I-shapes to Electro-Welded I-shapes (BR).
- Added two new cross-section tables: Gerdau AçoMinas I-shapes (BR) and ASCI Parallel Flange I-shapes.
- Created options to export a formated text file with analysis results along a chain of members. The results are exported either in step values or in display resolution.
- Modified policy for selection and unselection of entities (members, nodes, or dimension lines): if the SHIFT key is pressed during pick or fence selection, the selection status of a found entity is inverted. Before a selected entity would remain selected if the SHIFT key was pressed.
- Modified treatment of dimension line selection such that new dimension lines are set unselected by default. Before, new dimension lines were set selected.
- Modified dimension lines display such that arrows tip do not appear when size of dimension line on screen is small.
- Modified launch of user-interface dialog for keyboard input of coordinates during mouse entry. For keyboard input, press the middle mouse button with CRTL key pressed (berfore this was done with the middle button and without the CRTL key).
- Modified control of mouse tolerance box such that now this control is made only using the right mouse button with CRTL key pressed (before the middle mouse button could also be used).
- Implemented zooming in and out of visualization window using the mouse middle button wheel.
- Implemented translation of visualization window dragging the mouse with middle button pressed.
- Implemented automatic translation of visualization window during selection of the second node of a member.
- Modified exported image file in PS and EPS formats to display in landscape format, using vector fonts instead of raster fonts, and with a resolution of 600 DPI.
- Added options to export image file in PDF, DGN, and CD-Metafile formats.
- Set up resolution of 24 pixels per milimeter for exported image files in DXF, PDF, and DGN formats.
- Decreased font size in exported image files.

- Eliminated display of node marks and grid points in exported image files.
- Increased the thickness of lines in exported images files.
- Increased maximum allowed number of member to 480. Before, the maximum allowed number was 96.

New features of version 2.11 (August 2002)

- Improved definition of beam member cross-sections. Several types of cross-sections are available, each one defined by specific parameters. Two I-shape section tables are incorporated (Brazilian specifications).
- Created an option to change the member side for displaying bending moment diagrams. Bending moment diagrams may now be displayed on the tension side (Brazilian convention) or on the compression side (common U.S.A. convention) of the members. The adopted sign convention for bending moments is unique: positive moments causing compression on the top fibers and tension on the bottom fibers. When adopting the tension side option, positive bending moments are displayed on the bottom side; and, when adopting the compression side option, positive bending moments are displayed on the top side.
- Created an option to change the member side for displaying influence lines. There two plotting conventions: positive influence line values plotted on the bottom side of member, or influence line plot convention follows corresponding internal force diagram convention.
- Created options to import global parameters (units system, visualization parameters, etc.) and attributes (material and cross-section property tables, load tables, etc.) from another file created by FTOOL. Properties may be imported globally or each table may be imported individually.
- Created an option to display result values along the members.
- Created an option to display step values on diagrams.
- A scaling factor was made available in the graphics interface for displaying internal force diagrams. This scale is defined in terms of units of internal force per unit of length.
- Fixed a code problem when adjusting the model image on screen. Adjustment now takes into account displayed diagrams to compute model image size.
- Fixed a problem with long file names.

New features of version 2.10 (November 2001)

- Created influence line response mode.
- Created step size for querying member diagram response values.
- Changed member result message such that it lists result values at each step along a member (if a step size is defined).
- Created internal force sign convention message dialog.
- Changed display of diagram result values such that values are displayed with signs, with the exception of bending moment values, which by default are displayed with no sign. Bending moment signs may be displayed optionally.
- Changed display of reaction forces and moments, adding a dash line in the center of the arrows.
- Fixed a mistake in length conversion from inch to meter (used internally).

New features of version 2.09 (January 2001)

- Created member displacement constraints. The user may specify members with no axial deformation or infinitely rigid members.
- Created member end moments loading.

- Implemented computation of member end rotation at hinged ends.
- Supports with prescribed displacements are shown with each constraint component displayed independently (simple supports for horizontal and vertical prescribed displacement and a plate for prescribed rotation).
- Several modifications were made to improve the quality of the model image. The primary one being the following: in the Windows version, the model is displayed in double buffering, that is, the image is updated at once on the screen, with a better visual result. The display of internal force diagrams was also improved: in each member, the diagram is offset by eventual hinges at the ends.

New features of version 2.08 (August 2000)

- Released Linux version.
- Created member temperature variation loading. The user specifies a temperature variation at the top edge of the cross section (edge at the positive local *y*-axis side) and a temperature variation at the bottom of the cross section (edge at the negative local *y*-axis side). A new material parameter, the thermal expansion coefficient, and two additional cross section properties, the cross section depth and the center of gravity height, were added to support this capability.
- Created dimension lines for distance annotation on the structure image.
- Created an option to open a structure file using drag-and-drop operation in the program screen. This option only works for the Windows version.
- Modified the program to allow creation of attributes (material parameters and section properties) and loads prior to creating a structure. In this way, the user may keep a file with the attributes and loads that are often used, as well as units and number formatting, and use this file as a starting point for a new structure.
- Fixed a mistake in the sign of bending moments that was shown in the top message box and in the information for members that had right-to-left or top-to-bottom orientations. The sign was not consistent with the convention that a positive bending moment causes tension in the lower fiber of the cross section (or right fiber for vertical members). By mistake, the sign was defined based on the direction of the local *y* axis of the member.

New features of version 2.07 (April 2000)

- Fixed a mistake in version 2.06; it was not displaying the correct values of internal force diagrams for units different than [kN] for axial and shear forces, and different than [kNm] for bending moments. The values were unintentionally displayed using the internal units of the program.
- Modified the default number formatting for displacement values such that now the default format is an exponential number with 3 decimal places.
- Fixed number formatting for nodal coordinates, visualization window sizes, and grid steps when number formatting of length values was modified by the user.
- Fixed number formatting for internal force diagrams values when number formatting of force or moment values was modified by the user.
- Modified fence selection such that, when no object is selected, selection is automatically geared to beam members.

New features of version 2.06 (February 2000)

- Created *Units & Number Formatting* item in the *Options* menu, which launches a dialog for units and number formatting configuration. The user may specify the units of each parame-

ter in FTOOL, as well as its number formatting. There are options to set default SI units (International System), default US units (United States System), or all units in kilo-Newtons and meters. The user may overwrite the default unit of any parameter.

- Replaced *Member Properties* menu by *Material Parameters* menu and *Section Properties* menu, which handles geometric parameters of cross sections. Created default Concrete and Steel material parameters.
- Created spring supports (translation and rotation springs).
- Replaced vector font text display by raster font text display on the drawing window. Exported images still use vector fonts, with the exception of postscript formats.
- Created options to display load and reaction values in addition to the load and reaction drawings.
- Created an option to display loads in conjunction to the response diagrams and deformed configuration.
- Modified info message box to use multi-line texts instead of just passive label texts as in the previous versions. This allows the user to copy the info message to the clipboard and paste it into a text editor. The right mouse button may be used to copy the message to the clipboard.

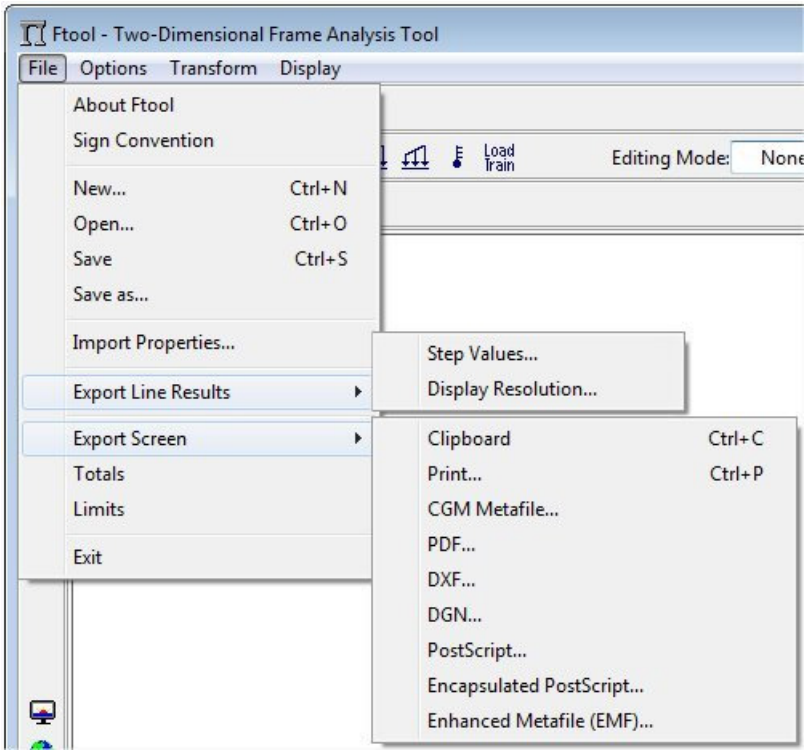
Additional Credits

- The first version of this manual (for version 2.07) was created by Luis Fernando Kaefer, former MSc and PhD student of the Polytechnic School of the University of São Paulo (EPUSP), Department of Engineering of Structures and Foundations (PEF) and Laboratory of Computational Mechanics (LMC).
- The implementation of multiple types of transverse sections in the FTOOL was contributed by Christiana Niskier, former undergraduate and MSc student in Civil Engineering in PUC-Rio.
- The update of manual for version 2.11 was contributed by Pedro Cordeiro Marques, former undergraduate student in Civil Engineering in PUC-Rio.
- The development of internal force envelops due to load-trains was performed by the researcher of Tecgraf/PUC-Rio Gisele Cristina da Cunha Holtz, based on her MSc thesis "*Traçado Automático de Envoltórias de Esforços em Estruturas Planas devido a Cargas Móveis utilizando um Algoritmo Evolucionário*" (in English, "Envelopes of Internal Forces due to Load-Trains in Bridges using an Evolution Strategy"), Department of Civil Engineering, PUC-Rio, 2005. Gisele Holtz actively worked in version 2.12 of FTOOL.
- The consideration of members with shear deformation (Timoshenko beams), introduced in version 3.00 of FTOOL, use analytical solutions that were developed by Rodrigo Bird Burgos, a civil engineer graduated from PUC-Rio, with MSc and PhD degrees also from PUC-Rio, and currently an assistant professor in UERJ, State University of Rio de Janeiro, Brazil.
- Many improvements in graphics interface of version 3.00 were made by Antonio Scuri, who is responsible for the development of IUP user interface system.
- The conclusion of version 3.00 of FTOOL was only possible because the author took a sabbatical license during the first semester of 2012 in IST – Instituto Superior Técnico – of Lisbon. This is reason for the name "Lisboa" of this version. The author is very grateful to many IST professors, in particular to José Paulo Moitinho de Almeida, Carlos Tiago Fernandes, and Orlando Pereira, for the fruitful discussions related to FTOOL and structural analysis.

File Operations

The *File* Menu

The drop down *File* menu controls file operations in FTOOL.

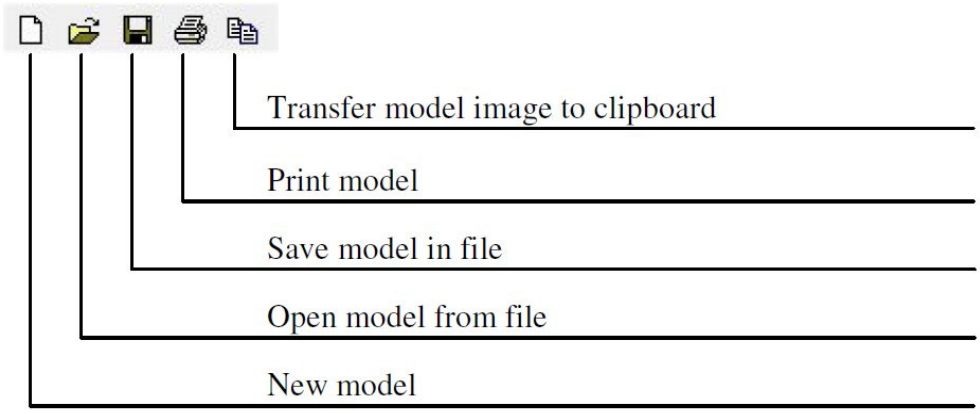


The *File* menu allows the user to:

- Get information about current version of program and authorship (*About Ftool*);
- Show the Sign Convention for the internal processing associated with FTOOL (*Sign Convention*) – see details of the sign convention in the post-processing section (*Results*);
- Create a new model (*New*);
- Open a model from an existing file on a disk (*Open*). FTOOL uses the extension ".ftl" in its model files;
- Save the current model in a file (*Save*) or under a different name (*Save as*);
- Import properties from another FTOOL file (*Import Properties*);
- Export to a file results (internal force diagrams, deformed configuration, influence lines, or internal force envelops due to load-trains) along a chain of selected members. These results may be saved in the indicated file using the current step that is specified for visualizing results (*Step Values* – see post-processing section – *Results*) or the step that is used to display diagrams on the screen (*Display Resolution*);
- Export images from the screen (*Export Screen*) to the Windows transfer area (*Clipboard*) or to files with specific formats;
- Verify the total number of members and nodes in the model (*Totals*);
- Establish the limits of the work window (*Limits*);
- Exit the program (*Exit*).

The *Import Properties* option in the *File* menu reads all the global parameters (units system, visualization parameters, etc.) into the current model, as well as all the attributes (property of materials and section tables, etc.) that exist in another file created by FTOOL. When this option is selected the program asks the user to indicate the file to import. Attribute tables can also be imported separately (see sections *Node and Member Attributes* and *Defining the Applied Loads*)

The most commonly used commands in the *File* menu are arranged in a toolbar at the top of the screen:



Exporting screen display images through the *Clipboard*

For best results, follow these steps:

- Choose the option *Export Screen/Clipboard* in the *File* menu, press the  key, or press *Ctrl+C*. This will copy all images on the main screen (not just what is currently selected).

- Open the program into which you want to import the image, MS-Word for example. In MS-Word for versions, select option *Paste* in menu *Edit* or press CTRL+V. In Windows, it is possible to choose the option *Paste Special*, then, the option *Picture (Windows Metafile)* or the option *Picture (Enhanced Metafile)*.

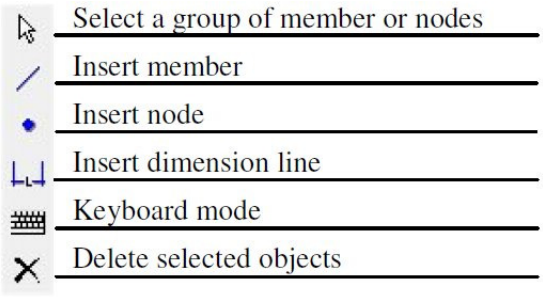
- Edit the pasted figure. You can make the lines thicker (1/2 pt or 3/4 pt are good options), change the colors of lines, etc.

- If you are printing to a black and white printer, best results are obtained if the option *Black and White Image* is selected before exporting the image. Use the option *Black Foreground* from menu *Display*.

Model Creation and Manipulation

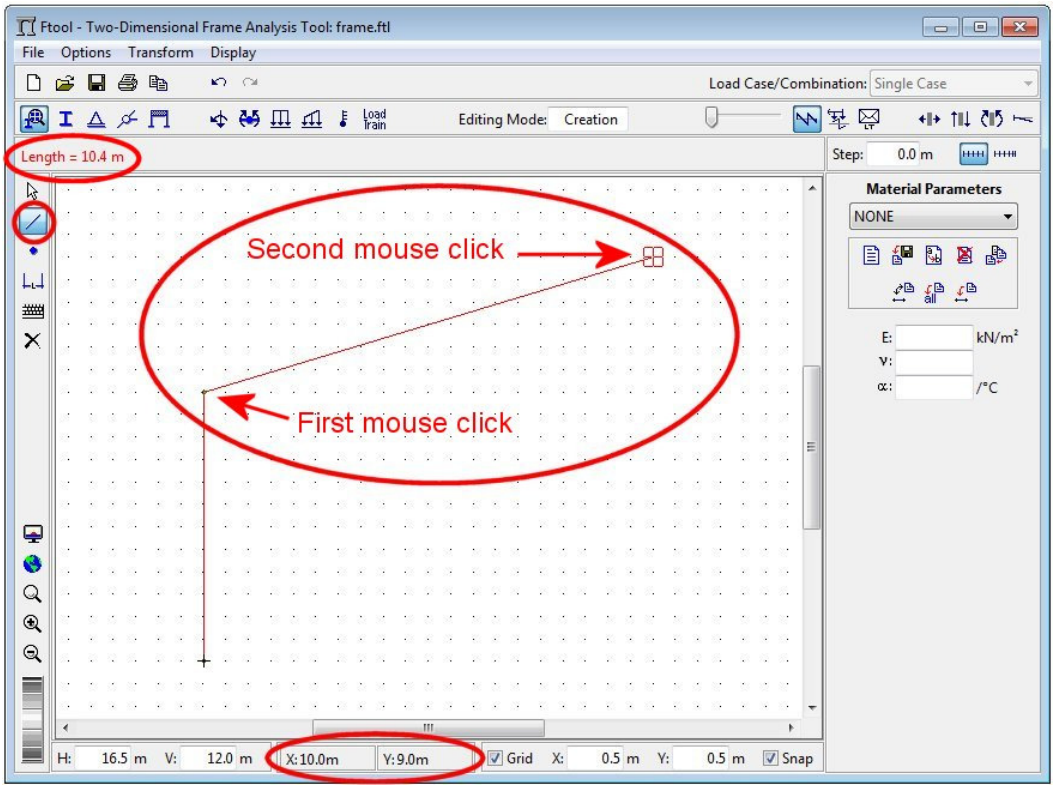
The *Edit* Toolbar

The *Edit* toolbar groups the buttons for the creation and modification of the model.



For the insertion of members, nodes, or dimension lines in FTOOL, the program automatically snaps the mouse cursor to an existing unit (a node or a member). The creation process can be facilitated by enabling a *Grid* of points and using the *Snap* feature. (See section on *Visualization Controls*).

Members and Nodes



Member and node creation is straightforward. To insert a member, select the button and click on two points within the model display area. Nodes are created instantly at the ends of the member. If

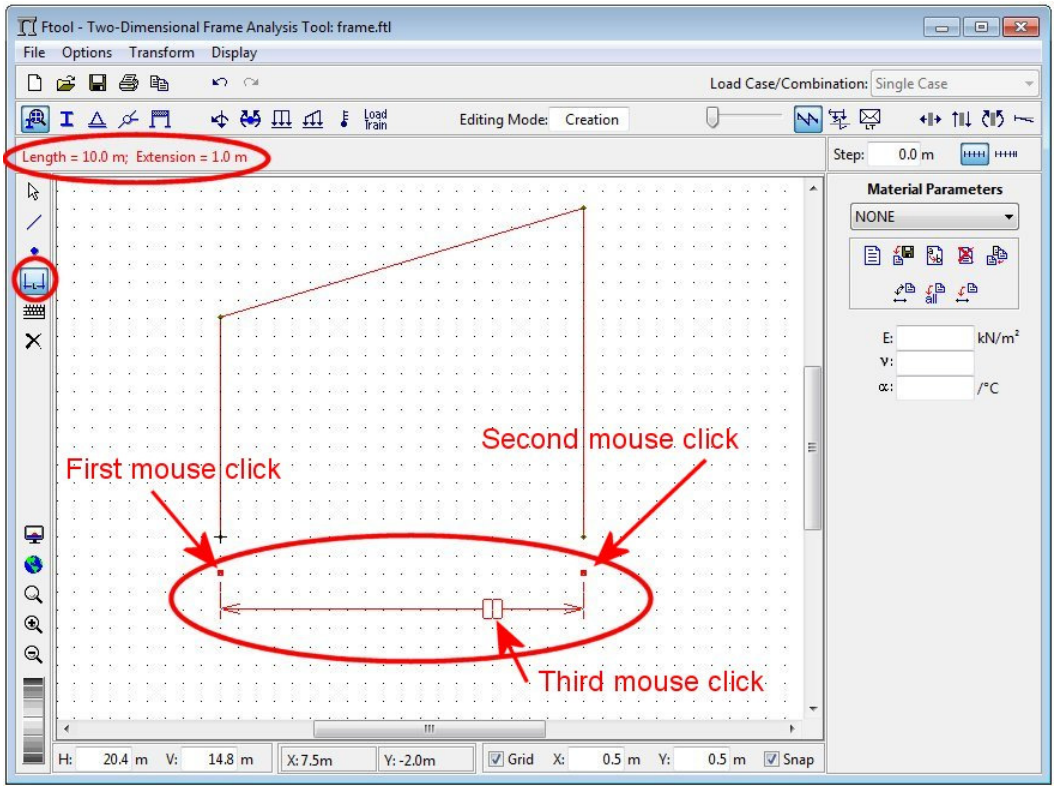
the inserted member intersects an existing one, a new node at the intersection of the two members is created automatically and the two members are automatically subdivided.

In the same way, a node is created by selecting the button and clicking with the mouse at a point within the model display area. If the clicked point is on an existing member, the member is divided in two members with insertion of the new node.

The insertion of "lines" with the mouse for creation of members is made with two clicks: one for the first node of the member and the next click for the second node. Usually the creation of lines using a mouse follows the rule "press button - drag mouse – release button". The "two clicks" method used in FTOOL allows the user to stop the insertion of members after the first point by clicking with the right mouse button or by pressing *Esc* on the keyboard. This type of method also allows the user to zoom in or move the drawing window after entering the first node and before entering with the second, etc.

The coordinates of the mouse cursor position are indicated in a text box located in the bottom of the screen. (see *Visualization Controls* section). While a member is inserted, prior to the second mouse click, a message bar on the top of the screen displays the member length.

Creation of Dimension Lines

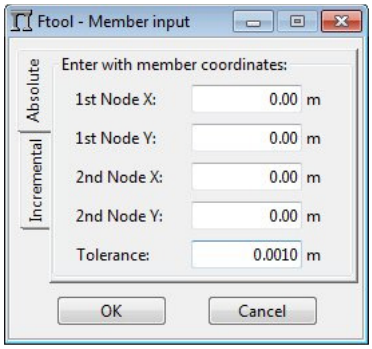


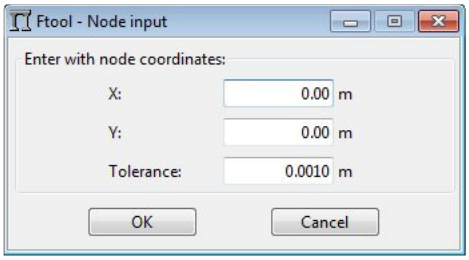
Dimension Lines are auxiliary lines used to indicate distances on the structural model. To insert a dimension line, select the button in the Edit toolbar and click three points on the screen. The two first points are the control points to specify the distance to be listed by the dimension. The third point defines where the dimension line will be located. During the construction of the line, the program updates the dimension line on the screen until the user enters the third point.

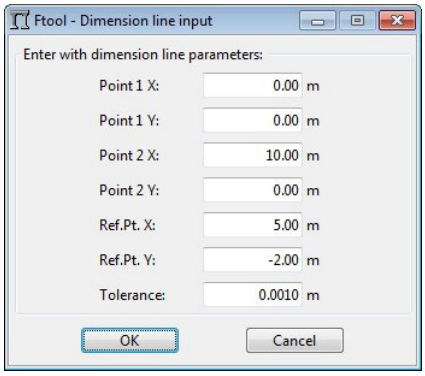
The creation of a dimension line through the "three clicks" method allows the user to suspend the insertion of the line before entering the final points, by clicking with the right mouse button or by pressing *Esc* on the keyboard. This type of interaction also allows the user to zoom in or to move the viewing window after entering the first point or the second point and before entering the third.

Keyboard Mode

After selecting the button , member, nodes and dimension lines can be created by entering coordinates into the pop-up box fields, as shown in the pictures below. The *Tolerance* value is used for the "attraction" of existing nodes/members (*never* use null value for tolerance).







Selection Mode

The button turns on "selection" mode. In this mode, right-clicking the mouse on a member or node displays its attributes in a menu on the side of the screen. Use the left button to select an item (the program never allows members and nodes to be selected simultaneously).

The selection of a set of members or a set of nodes can be made clicking the left button of mouse concurrently with the SHIFT key. A set of members/nodes can also be selected by defining an enclosing rectangle. To do this, press and hold the left button of the mouse and drag it to define the rectangle. Release the mouse button to complete the rectangle.

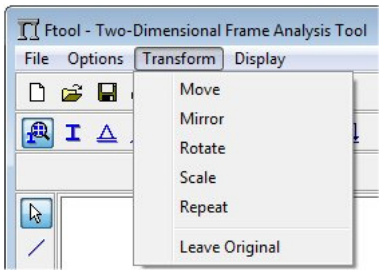
Once nodes and members are selected, actions can be executed upon them. To delete entities push the button. To transform the selected entities, use the *Transform* menu. The user can also apply attributes or loads to the selected members or nodes – this is discussed later.

Undo and *Redo*

The *Undo* option cancels previous actions. The *Redo* option re-executes the last cancelled action.

The *Transform* Menu

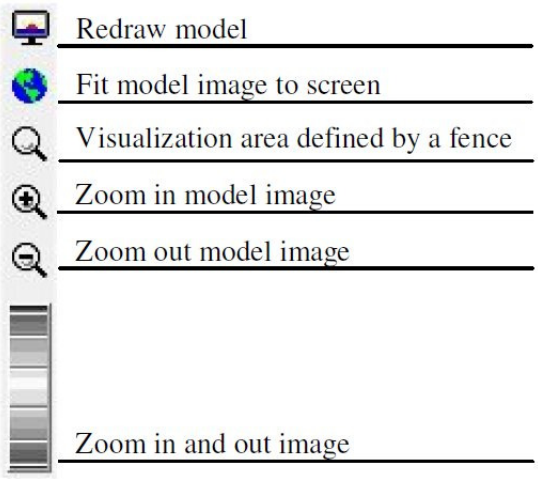
The *Transform* menu lists options to manipulate the nodes or members that have already been created. There are options to move (*Move)*, to reflect (*Mirror)*, to rotate (*Rotate*), to apply a scale factor (*Scale*) and to repeat the last transformation (*Repeat*). If the option *Leave Original* is selected, the transformation is applied on a copy of the selected entities.



Visualization Controls

The *Visualization Control* Toolbar

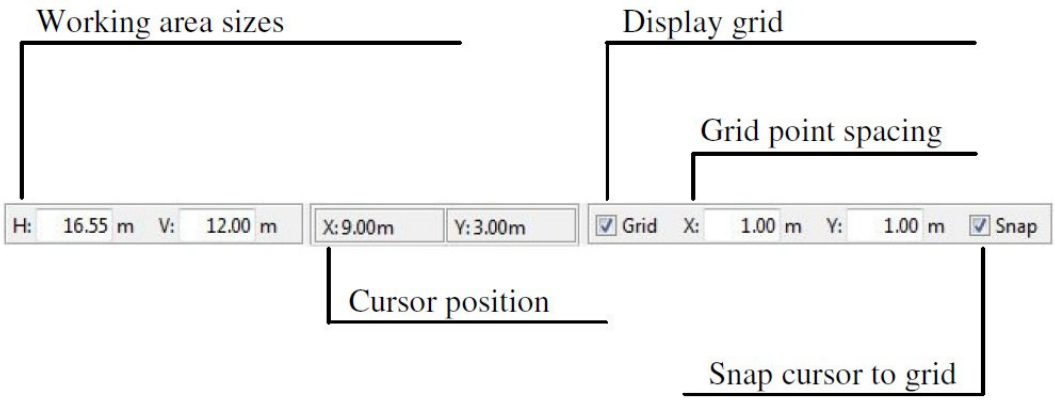
This toolbar groups all the controls for the model visualization window.



The option to *Fit and center on the screen* scales the structure image to fill the screen. The scale of the image on the screen can be modified in several ways, the first of which zooms based on a defined zoom rectangle (*Define a rectangle to focus the display*). The entry of two opposing corners of the zoom rectangle is completed with "two clicks" of the mouse. The "two clicks" approach permits the user to cancel the visualization area after the first point of the rectangle by clicking with the right button of the mouse or by pressing *Esc*. If the two mouse clicks are at the same point on the screen there will be a "central" zoom at this point. The *Zoom In* icon increases the size of the model on the screen, while *Zoom Out* button decreases the size. The dial at the base of the toolbar allows for real-time zoom control of the model.

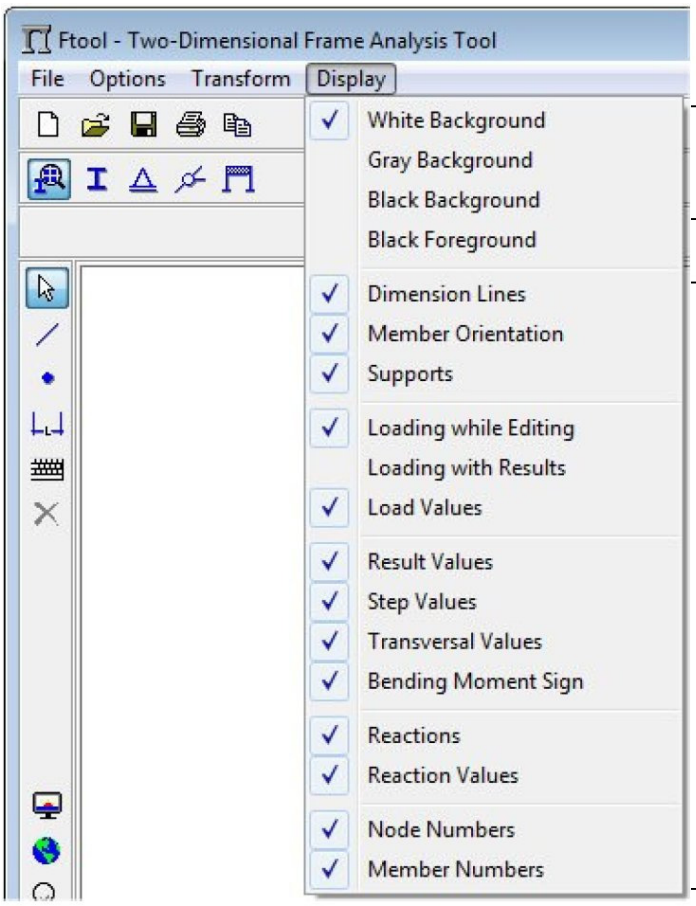
Coordinate Control

In this toolbar at the bottom of the display, the *H* and *V* text fields show the size of the visualization window and enable the user to alter these values. The *X* and *Y* fields show the position of the cursor in the screen. The *Grid* check box enables the option to display a grid of points on the screen, and the *Snap* check box activates the "attraction" of the cursor to the points of the grid.



The *Display* Menu

This menu lists all the items that may be displayed. The user can choose the background color, for example. A check mark indicates the display quantity is active. Just click on list items with the mouse. For each background color selected there are different colors related to the members and nodes of the model. Another option here is to work with all the members of the model in black and white. Thus the image of the model can be printed effectively on a black and white printer. Furthermore, it is possible to specify which attributes should be shown on the screen during the use of the program. It is important to emphasize that certain options are only applied to pre-processing and others only to post-processing.



Screen background	White
	Gray
	Black
Display all entities in black	

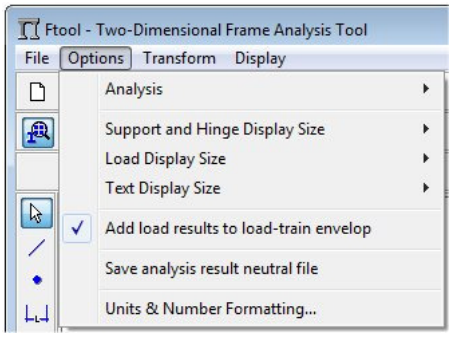
Display	
	Dimension lines
	Member orientation
	Supports
	Loads in edition mode
	Loads in result visualization mode
	Load values
	Result values (member extremities)
	Result step values
	Result values transversal to members
	Bending moment signs
	Support reactions
	Support reaction values
	Node numbers (only with results)
	Member numbers (only with results)

Configurations

The *Options* Menu

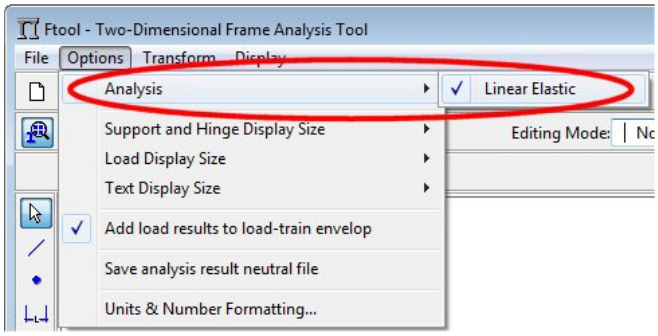
In FTOOL it is possible to configure:

- Sizes of supports, hinges, loads, and text in the display of a model on the screen;
- Option to consider or not permanent load results in the calculation of envelops of internal forces due to load-trains;
- Option to save auxiliary file (extension ".pos") that is used internally to link the program with is analysis module (solver). By default, this option is not activated, and it is reset when a new model is initialized or open from file.
- Unit systems and the format for numerical values.



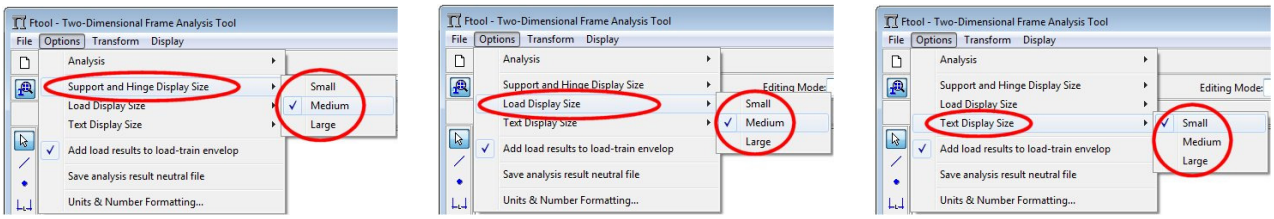
Analysis type

In the current version of FTOOL, the only type of analysis is linear-elastic:



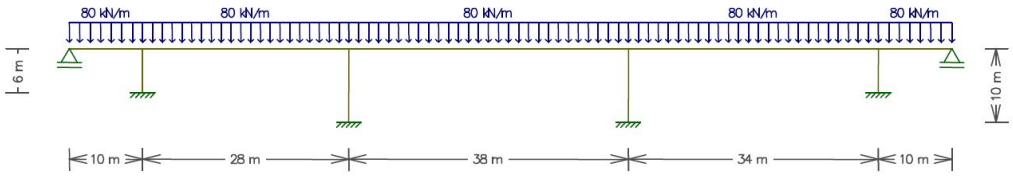
Sizes of supports, hinges, loads, and text in the display

There are itens in *Options* menu that allow the configuration of sizes of supports, hinges, loads, and text in the display of a model on the screen:

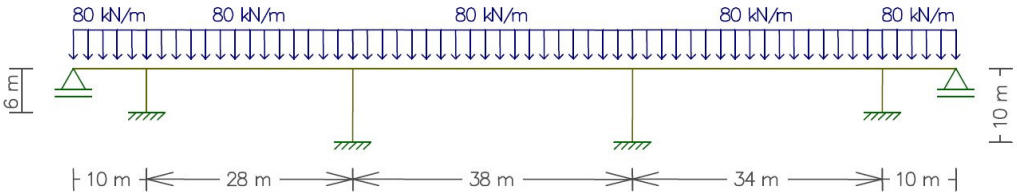


For each type of entity, display size may be *Small*, *Medium*, or *Large*. An example of the results of this display size configuration may be seen in the model of a bridge:

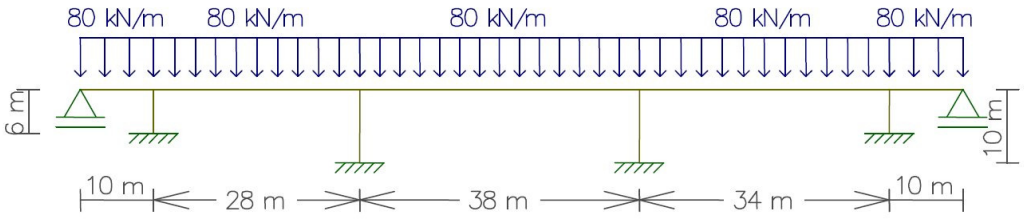
Small sizes



Medium sizes

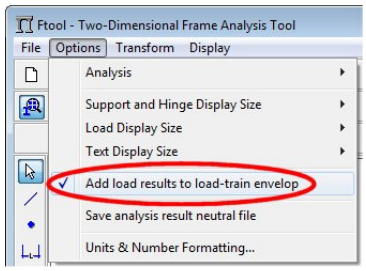


Large sizes



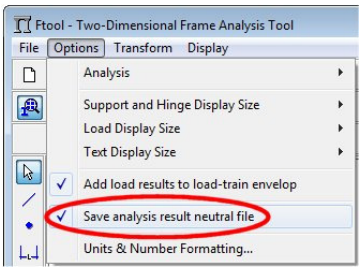
Superposition of permanent load and load-train results

FTOOL, by default, adds permanent load effects to the calculation of internal force envelops due to load-trains. This option may be deactivated through one item of *Options* menu. The default configuration is reset when a new model is created or opened from a file.



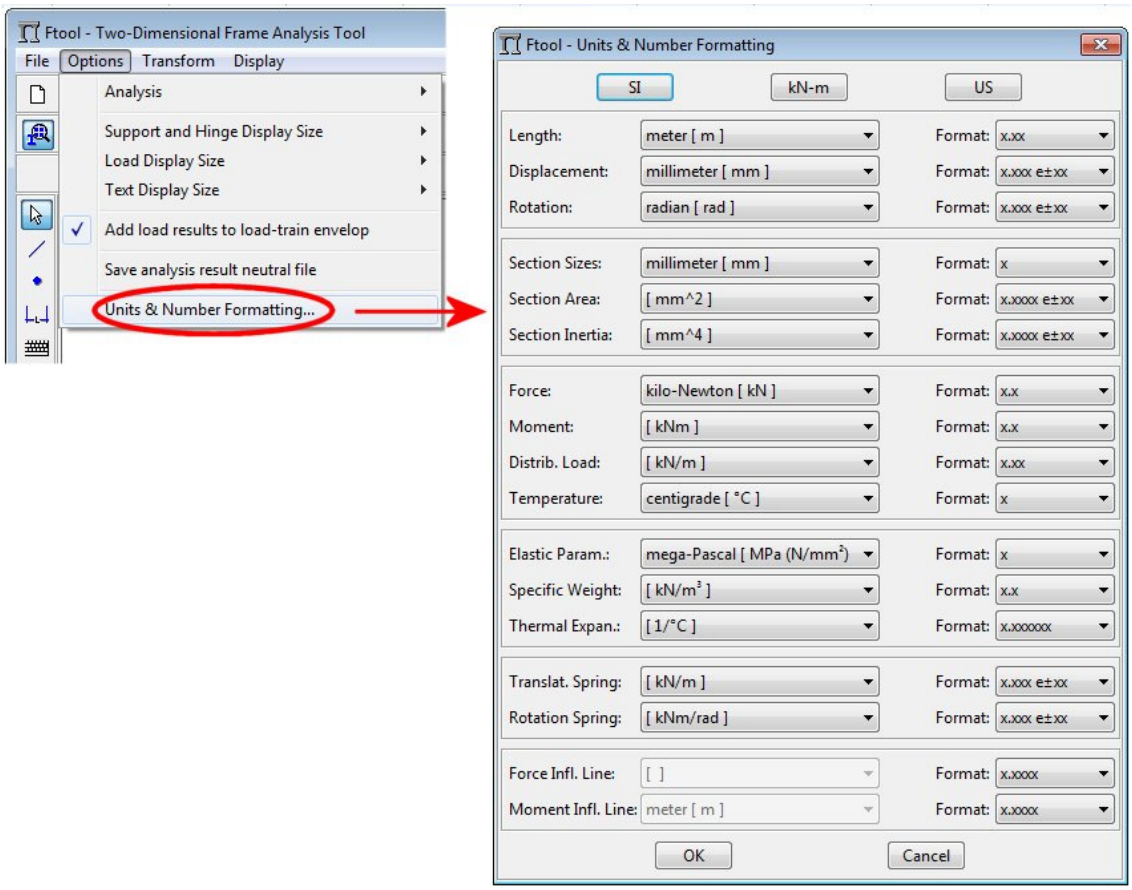
Solver file

FTOOL has an internal module to perform the analysis (solver) of current model. It is possible to save an auxiliary file with the data that is passed to the analysis module. In version previous to version 3.00, this file was the only option to feed the analysis. After version 3.00, this communication is done through RAM memory. However, there is an item in the *Options* menu that allows this file to be saved. The file has the same name as the current model file name, but with the extension ".pos". This option is deactivated when a new model is created or open from a file.



Formatting Units and Numbers

Through the sub-menu *Units & Number Formatting*, the user selects physical units for parameters used in the structural analysis and the format for the display of numerical values for these parameters. Options exist to specify units in SI (International System), US (American System), or units in kilo-Newtons and meters. To specify a system of units and the display format, select the corresponding button at the top of the box. The user can always switch the system of units or formatting as needed during an analysis.



Unit Systems

The following tables show the unit systems implemented in FTOOL for each parameter and the corresponding factors of conversion for the units used internally by the program (indicated in **boldface**).

The first unit of each parameter appears automatically when the user selects SI units (International System) or US units (American System).

The conversions for units that have ton-force (tf) are: 1 tf = 10^3 kg·g.

The gravity acceleration adopted is g = 9.81 m/sec^2.

- Pascal Units:
  1 Pa = 1 N/m^2
  1 kPa = 10^3 N/m^2 = 1 kN/m^2
  1 MPa = 10^6 N/m^2 = 10^3 kN/m^2 = 1 N/mm^2
  1 GPa = 10^9 N/m^2 = 10^6 kN/m^2 = 1 kN/mm^2

Parameter	SI units			US units		
	Symbol	Name	Factor of ref. SI unit	Symbol	Name	Factor of ref. SI unit
Length	**m**	meter	1.0	ft	foot	0.3048
	cm	centimeter	0.01	in	inch	0.0254
	mm	millimeter	0.001			
Displacement	mm	millimeter	0.001	in	inch	0.0254
	cm	centimeter	0.01	ft	foot	0.3048
	m	meter	1.0			
Rotation	**rad**	radian	1.0	rad	radian	1.0
	deg	degree	π/180.0	deg	degree	π/180.0
Section Size	mm	millimeter	0.001	in	inch	0.0254
	cm	centimeter	0.01	ft	foot	0.3048
	m	meter	1.0			
Section Area	mm^2		0.000001	in^2	$inch^2$	0.0254^2
	cm^2		0.0001	ft^2	$foot^2$	0.3048^2
	m^2		1.0			
Section inertia	mm^4		1.0e-12	in^4	$inch^4$	0.02544
	cm^4		1.0e-08	ft^4	$foot^4$	0.30484
	m^4		1.0			
Force	**kN**	kilo-Newton	1.0	kip	kilo-pound	4.448
	N	Newton	0.001	lb	pound	0.004448
	tf	ton (weight)	9.81			
Moment	**kNm**		1.0	ft-k	kip·foot	1.356
	Nm		0.001	ft-lb	foot·pound	0.001356
	tfm		9.81	in-k	inch·kip	0.11298
	kNcm		0.01	in-lb	inch·pound	0.00011298
	Ncm		0.00001			
	tfcm		0.0981			
	kNmm		0.001			
	Nmm		0.000001			
	tfmm		0.00981			

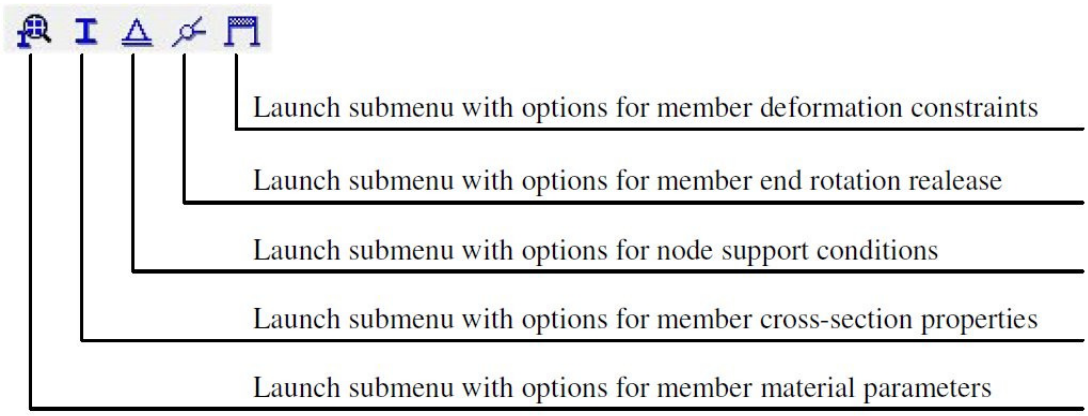
Parameter	SI units			US units		
	Symbol	Name	Factor of ref. SI unit	Symbol	Name	Factor of ref. SI unit
Distributed	**kN/m**		1.0	k/ft	kip/foot	14.593
Load	N/m		0.001	lb/ft	pound/foot	0.014593
	tf/m		9.81	k/in	kip/inch	175.1
	kN/cm		100.0	lb/in	pound/foot	0.1751
	N/cm		0.1			
	tf/cm		981.0			
	kN/mm		1000.0	kN/mm		1000.0
	N/mm		1.0	N/mm		1.0
	tf/mm		9810.0	tf/mm		9810.0
Temperature	**℃**	centigrade	1.0	℉	Fahrenheit	(T-32)×5/9
Elasticity	MPa	mega-Pascal	1000.0	ksi	kip/inch^2	6895.0
Modulus	GPa	giga-Pascal	1000000.0	psi	pound/inch^2	6.895
	tf/mm^2		9810000.0	k/ft^2	kip/foot^2	47.878
	N/cm^2		10.0	lb/ft^2	pound/foot^2	0.047878
	kN/cm^2		10000.0			
	tf/cm^2		98100.0			
	Pa	Pascal (N/m^2)	0.001			
	kN/m^2	kilo-Pascal	1.0			
	tf/m^2		9.81			
Specific Weight	**kN/m^3**		1.0	pcf	pound/foot^3	0.1571
	N/m^3		0.001	k/ft^3	kip/foot^3	157.1
	tf/m^3		9.81	lb/in^3	pound/inch^3	271.434
	kN/cm^3		1000000.0	k/in^3	kip/inch^3	271434.0
	N/cm^3		1000.0			
	tf/cm^3		9810000.0			
	kN/mm^3		1.0e+09			
	N/mm^3		1000000.0			
	tf/mm^3		9.81e+09			
Thermal Expan.	**1/℃**		1.0	1/℉		1.8

Parameter	SI units			US units		
	Symbol	Name	Factor of ref. SI unit	Symbol	Name	Factor of ref. SI unit
Translat. Spring	**kN/m**		1.0	k/ft	kip/foot	14.593
Stiffness	N/m		0.001	lb/ft	pound/foot	0.014593
	tf/m		9.81	k/in	kip/inch	175.1
	kN/cm		100.0	lb/in	pound/foot	0.1751
	N/cm		0.1			
	tf/cm		981.0			
	kN/mm		1000.0			
	N/mm		1.0			
	tf/mm		9810.0			
Rotat. Spring	**kNm/rad**		1.0	ft-k/rad	foot·kip/rad	1.356
Stiffness	Nm/rad		0.001	ft-lb/rad	foot·pound/rad	0.001356
	tfm/rad		9.81	in-k/rad	inch·kip/rad	0.11298
	kNcm/rad		0.01	in-lb/rad	inch·pound/rad	0.00011298
	Ncm/rad		0.00001	ft-k/deg	foot·kip/deg	244.08/π
	tfcm/rad		0.0981	ft-lb/deg	foot·pound/deg	0.24408/π
	kNmm/rad		0.001	in-k/deg	inch·kip/deg	20.3364/π
	Nmm/rad		0.000001	in-lb/deg	inch·pound/deg	0.0203364/π
	tfmm/rad		0.00981			
	kNm/deg		180.0/π			
	Nm/deg		0.18/π			
	tfm/deg		1765.8/π			
	kNcm/deg		1.8/π			
	Ncm/deg		0.0018/π			
	tfcm/deg		17.658/π			
	kNmm/deg		0.18/π			
	Nmm/deg		0.00018/π			
	tfmm/deg		1.7658/π			

Node and Member Attributes

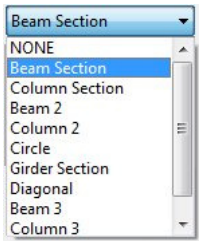
Control of Node and Member Attributes

The buttons of this toolbar display a set of sub-menus to create and assign properties for nodes and members of the model. These sub-menus appear along the side of the screen.

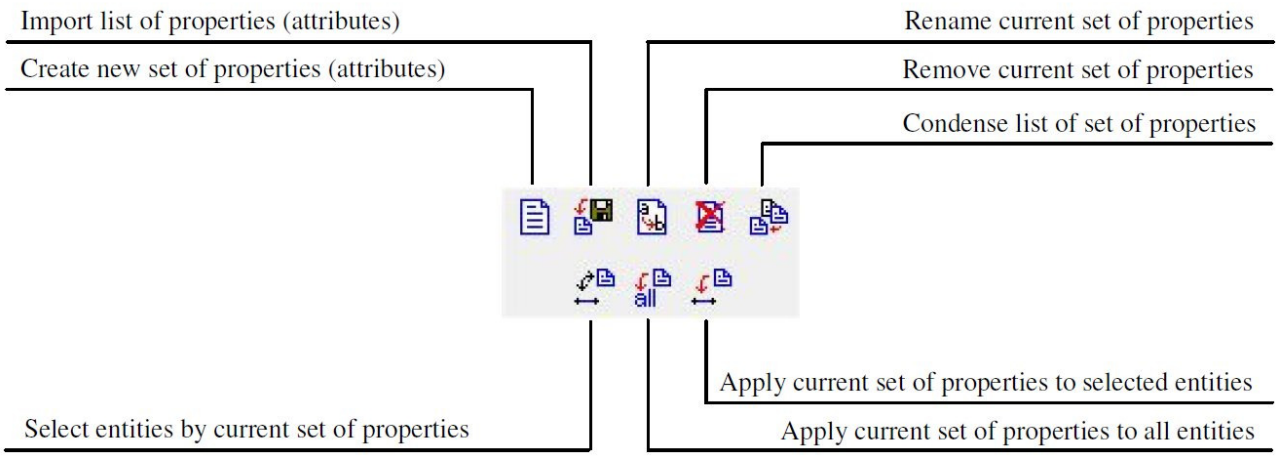


Features common to sub-menus

Sub-menus for manipulation of the material parameters, member cross-section properties and values of loads all have common features. The *drop-down* list (next figure) selects a set of properties that is defined by a unique name. These property values are automatically shown in the sub-menus for editing.



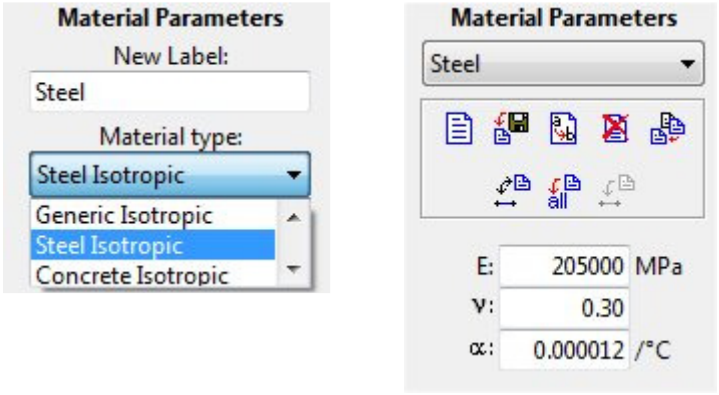
The icons shown in the next figure allow the user to manage these property sets.



To create a new set of properties, select the button and assign the new set a unique name. The button imports a list of property sets from another file generated by FTOOL. Duplicate property sets in the imported file are ignored. The internal function that compresses the property sets just eliminates those which are not in use in the current model.

Material Parameters

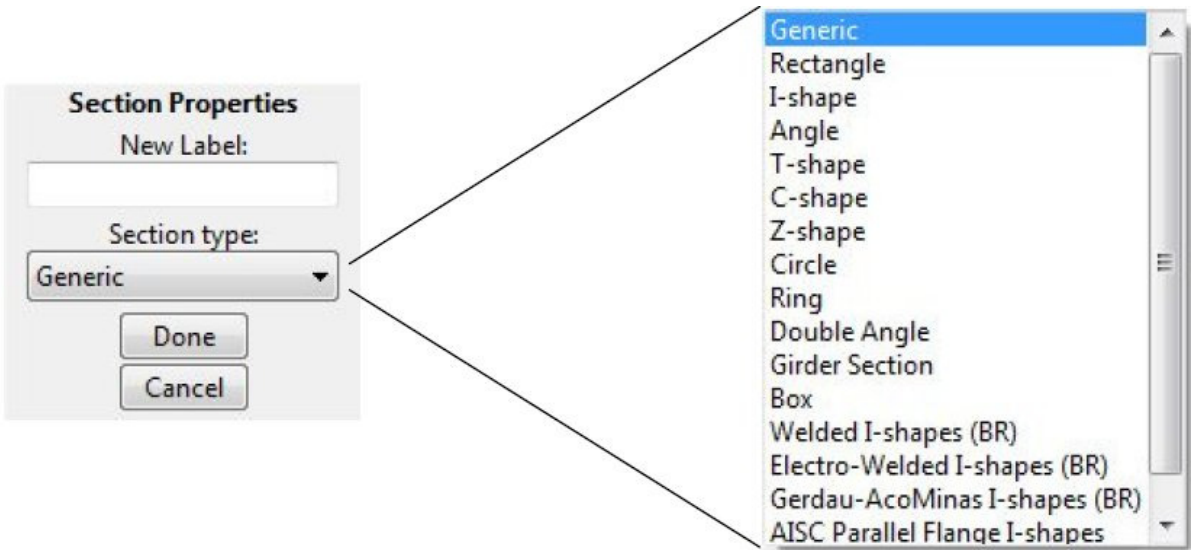
To create a new set of material parameters, select button and specify a unique name (figure below to the left). Default values of material parameters are available for a generic isotropic material, steel, or concrete.



Material parameters include Young's modulus (elasticity modulus E), Poisson ratio (ν) – used for members with shear deformation, and coefficient of thermal expansion (α) – used in case temperature loadings are applied.

Member Section Properties

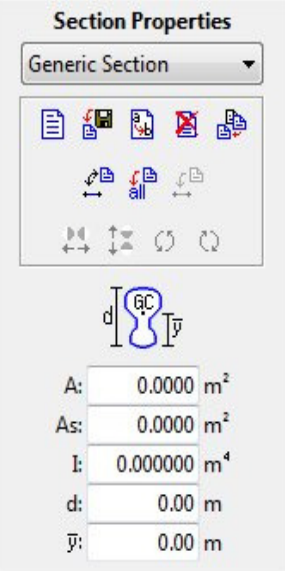
The button creates a new set of cross-section properties. Available *section types* are shown in the drop-down list. One of the following must be chosen: *Generic*, *Rectangle*, *I-shape*, *Angle*, *T-shape*, *C-shape, Z-shape*, *Circle*, *Ring, Double Angle*, *Girder Section*, *Box*, *Welded I-shapes (BR)* , *Electro-Welded I-shapes (BR)*, *Gerdau-Aço Minas I-shapes (BR)*, or *AISC Parallel Flange I-shapes*.



Based on the type of section selected, the parameters that define the features of that section are then defined along the side of the screen under the schematic drawing.

In the case of the *Generic* section (shown here) the parameters listed are the following:

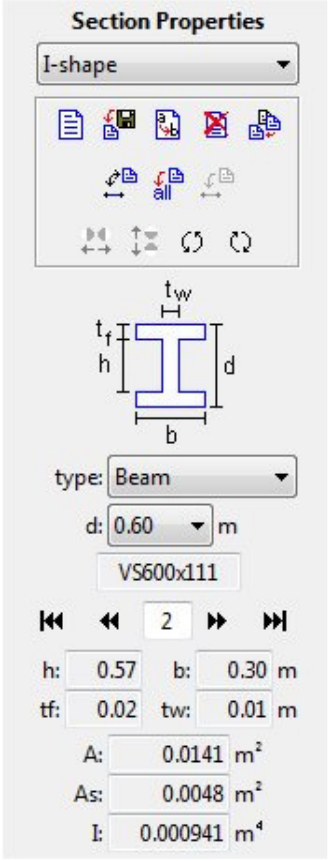
- *A*: area
- *As*: shear effective area
- *I*: moment of inertia (second moment of the area)
- *d*: section depth
- $\bar{y}$: centroid position.



Defaults: only *A* and *I* are required. *As* is required for members with shear deformation. *d* and $\bar{y}$ are necessary for members with thermal loads.

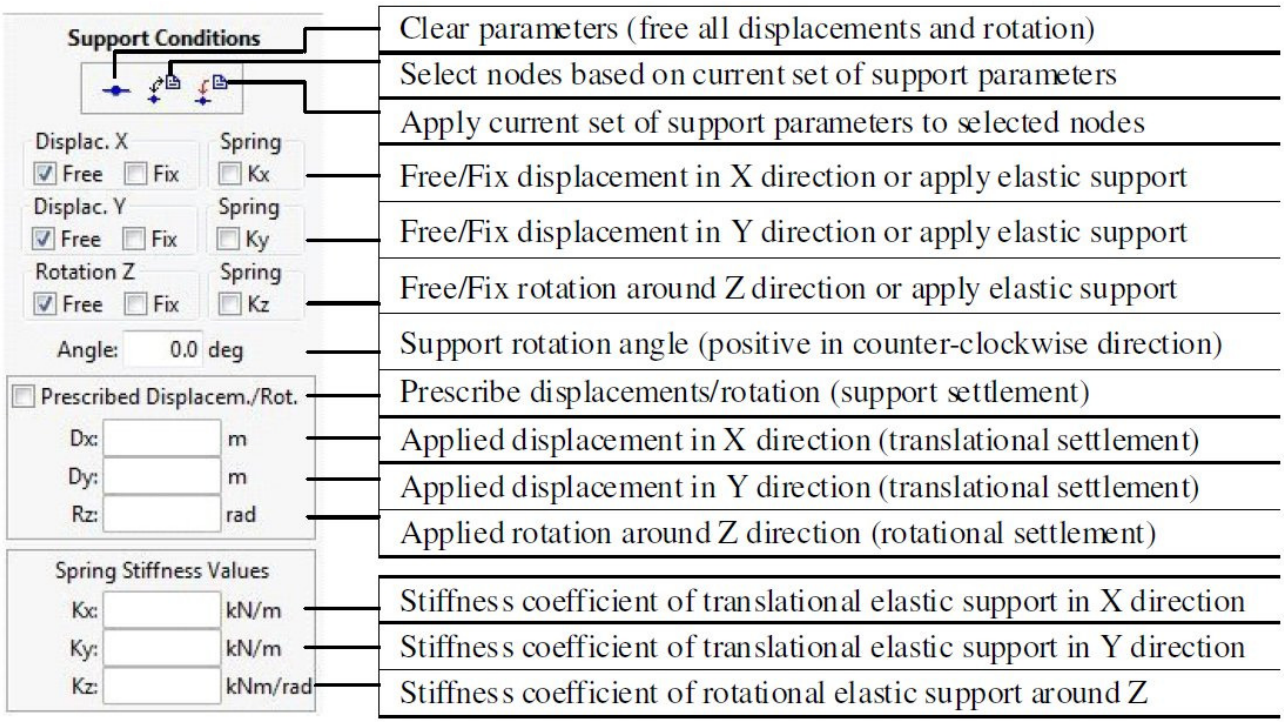
For the section type *Profile - Welded-I (BR)*, (shown here), choose the type of profile (*Beam, Column* or *Beam-Column*) and the section depth, *d*.

FTOOL contains data from NBR standard profiles, and will provide the remaining data. The user only needs to select specific requirements via the drop-down lists.



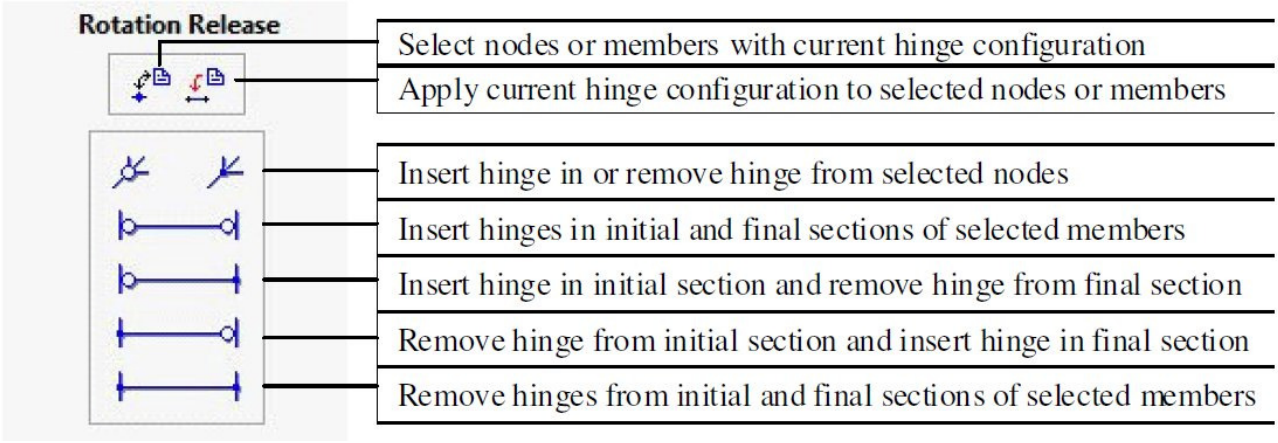
Support Conditions

Through this submenu, the user defines constraints on components of displacements in the *x* and *y* directions and the rotation around the *z*-axis. The support orientation angle (for skew supports) is also defined, as well as any prescribed values of displacement or rotation. Non-rigid supports may be modeled by specifying values for corresponding stiffness of linear-elastic springs.



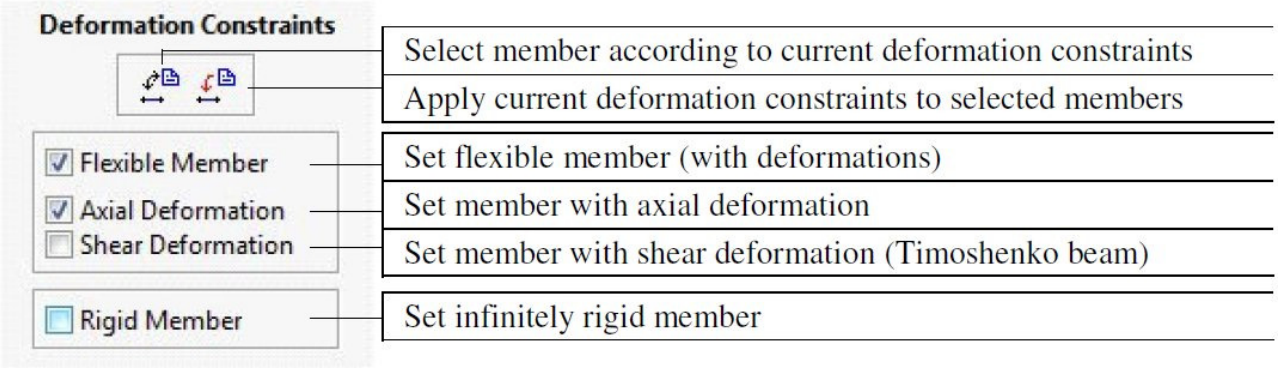
Connection of Member Ends to Nodes

This sub-menu allows the user to introduce hinges at member ends or at the model nodes. By default, members are rigidly connected to the structure nodes.



Constraints on Member Deformations

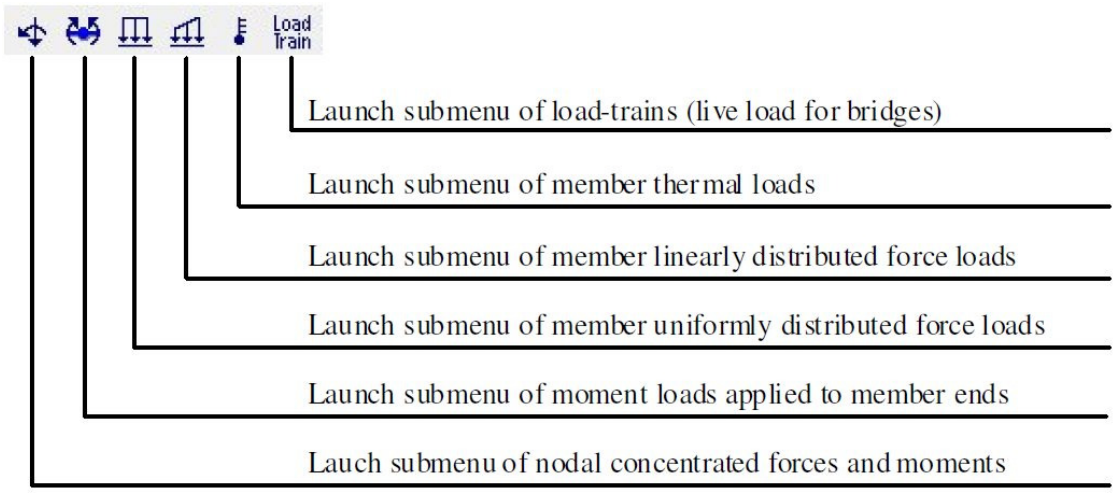
This sub-menu restricts deformations within members. There are two exclusive main options: flexible member and infinitely rigid member. For flexible member, there are two non-exclusive options: toggle for axial deformation and toggle for shear deformation. When a new model is created, the default toggle options are flexible member with axial deformation of with no shear deformation. If a model is read from a file, and all its members have shear deformation allowed, the shear deformation toggle is set on.



Defining the Applied Loads

The *Load Control* Toolbar

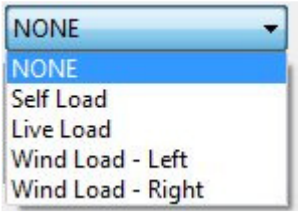
Sub-menus are also available to define or modify various nodal and member loadings. These sub-menus appear along the side of the screen. The types of loads available are the following: concentrated loads at nodes, moments applied at nodes, moments applied at the ends of members, uniformly or linearly distributed loads along members, variations of temperature applied to members, and global load-trains (live load for bridges).



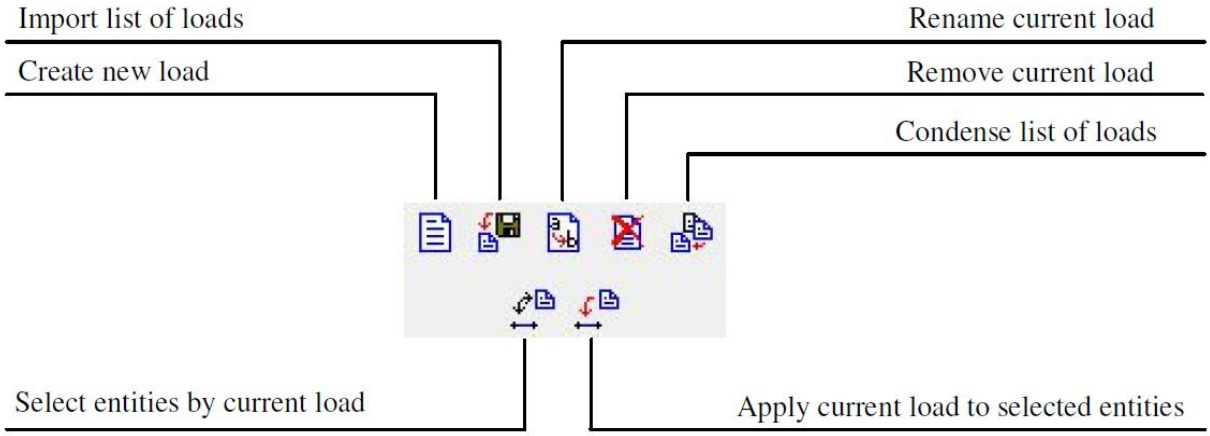
General Information

- **Multiple Load Definitions**
  All the loads applied to a structure act simultaneously.

- **Load Definition**
  The loads definition system follows the same procedures as the one that defines member properties. A type of load associated with a user-supplied name is created and added to the corresponding load list. The figure below shows a *drop-down* list of distributed loads defined by the user in a specific analysis. The values of load associated with the selected name are automatically displayed in the fields of submenu and can be edited.



The icons shown in the figure below are used to manipulate the load in a list:



The current load will be applied to the selected elements (members or nodes). The members of interest must be selected and the load is applied through the button for members, or the button for the nodes.

- **Coordinate System in FTOOL**
  There is a system of structural global axes in FTOOL and a system of local axes for each one of the members. In the global system, the global *X*-axis is horizontal and positive from left to right; the global *Y*-axis is vertical and positive from the bottom to the top; and the global *Z*-axis is always positive outward from the display. In a member's local coordinate system, the local *x*-axis coincides with the longitudinal axis of the member, with the positive direction following creation of the member; that is, from the initial node to the end node. The local *x*-axis direction can be displayed by selecting the *Member Orientation* option in the *Display* menu. The local *y*-axis is perpendicular to the *x*-axis. The *z*-axis for a member is always positive outward from the display. The positive direction of local *y* then follows the right-hand rule of vector cross products: $y = z \times x$.

- **Application of concentrated loads**
  Concentrated loads (forces and moments) can be applied only on nodes of the structure. Of course concentrated loads could be applied along the span of a member. However, for user-interface simplicity, it was adopted a police of only applying concentrated loads at nodes. If it is necessary to apply a concentrated load on a member, insert a new node at the desired position, thereby dividing the member into two members. Concentrated loads are always applied in directions of the global axes of the structure, positive when the forces have directions of the global axis, and negative when they have the opposite direction. Positive concentrated moments are applied in a counter-clockwise orientation.

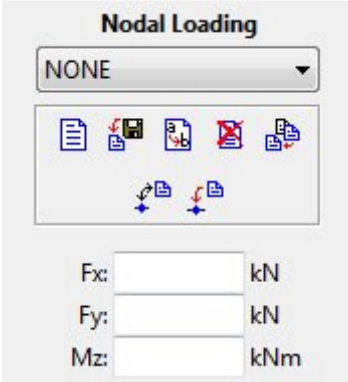
- **Orientation for distributed member loads**
  Distributed loads along a member may be specified in the framework of global coordinates or in the framework of member local coordinates. The loads are positive if they coincide with the direction of the global or local axes.

- **Partially distributed member loads**
  New nodes can be introduced along a member to apply distributed loads that act on a portion of the member length. As for concentrated loads (see above), this is done for user-interface simplicity.

- **Load removal from nodes and members**
  To remove a load from from selected nodes or members, select the first element (NONE) of the corresponding load *drop-down* list and apply this to the selected entities. In other words, “to remove a load from selected entities in FTOOL is to apply nothing”.

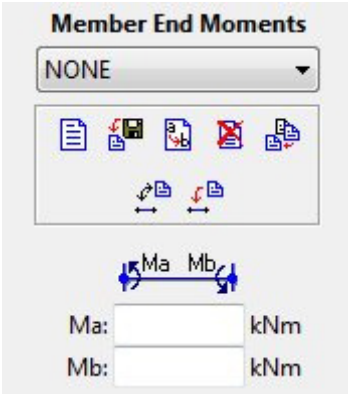
Nodal Loads

This sub-menu allows the user to define the concentrated loads on structure nodes. It uses the global coordinate system.



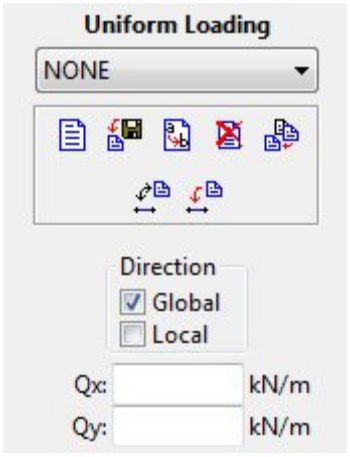
Moments Acting on Member Ends

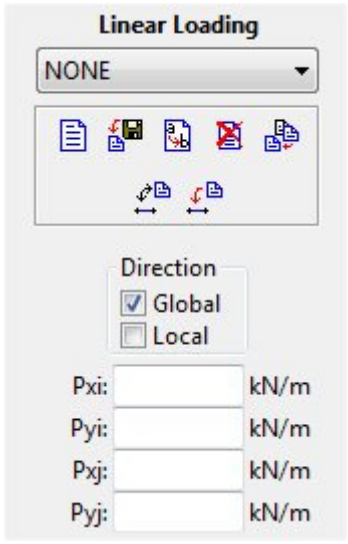
This sub-menu allows the user to define concentrated moments at the ends of members. Moments applied in a counterclockwise orientation are positive. "Ma" denotes the moment applied at the "initial" node of the member, while "Mb" is the moment applied at the "ending" node of the member.



Linear and Uniformly Distributed Loads

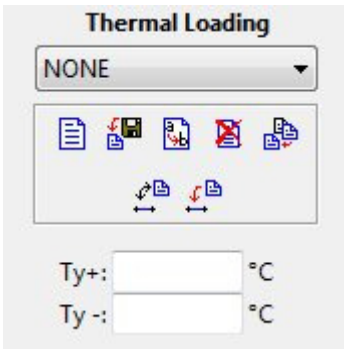
This sub-menu allows the user to define linearly varying or uniformly distributed loads over a member. The user can specify the global or local member coordinate system for the loading direction.





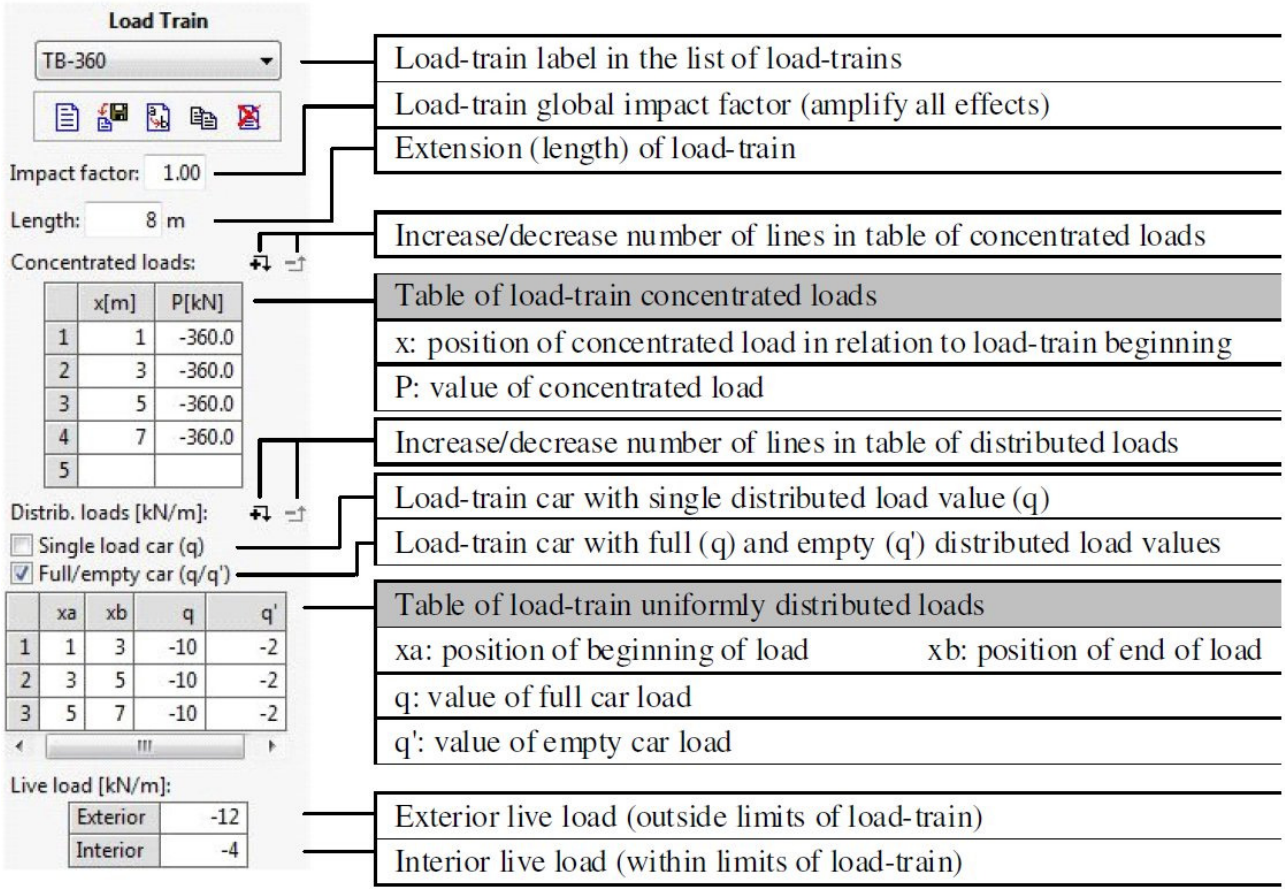
Thermal Loads (Temperature Changes)

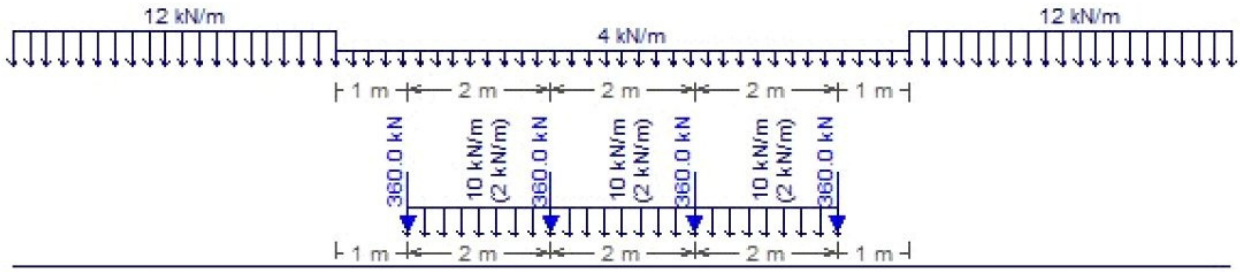
This sub-menu allows the user to define a linear temperature gradient over the member depth. The user specifies the temperature on the section's top edge (i.e., on the positive side of local *y*-axis) and on the bottom edge (i.e., on negative side of local *y*-axis). FTOOL must have access to the section depth to impose this loading – even for "generic" sections.



Load-Trains (Vehicle Live Loads for Bridges)

This sub-menu allows the creation of live loads for bridges that are used to calculate envelops of internal forces (see *Results* section). A load-train is composed of concentrated forces, uniformly distributed forces and live loads (representing the population of small vehicles on a bridge). Concentrated and distributed loads are assumed in the vertical top-bottom direction. Therefore, according to the sign convection of FTOOL, all load values are negative. In case the user does not enter a negative sign for a load value, the program automatically changes the sign of this value.





- **Impact factor**
  This is an amplification factor that multiplies globally all the effects of a load-train and allows the consideration of a dynamic effect on the structure. The value of this factor should be always greater than one.

- **Load-train length**
  This extension length limits the application of concentrated, distributed and internal live loads.

- **Concentrated forces**
  Load-train concentrated forces are specified using a matrix with two columns, with the following parameters:
  - x – position of concentrated force in relation to load-train beginning;
  - P – value of a concentrated force.

  It is not allowed to create more than one concentrated force at a single position, or outside the limits of the load train (specified by its length). To add a concentrated force to the current load-train, first enter the force position then its value. As other concentrated forces are inserted, they are ordered according to their position. At the top of the matrix, there are two buttons that allow increase or decrease the number of lines of the matrix.

- **Uniformly distributed forces**
  The matrix of distributed forces varies according to the type of load-train. In case of a load-train that has single values of distributed forces, the matrix has three columns, with following parameters:
  - xa – initial distributed force position in relation to load-train beginning;
  - xb – end distributed force position in relation to load-train beginning;
  - q – value of a distributed force.

  In case of a load-train that has double value (full and empty car) of distributed loads, the matrix has four columns, with the following parameters:
  - xa – initial distributed force position in relation to load-train beginning;
  - xb – end distributed force position in relation to load-train beginning;
  - q – value of a distributed force for a full car in a rail-road load-train;
  - q' – value of a distributed force for na empty car in a rail-road load-train.

  Overlap of distributed forces is not allowed. Initial and end positions of distributed forces must be within the load-train extension (length). To add a distributed force to the load-train, first enter its initial and end positions then the load value (or values). In the case of a rail-road load-train, the first load value to be inserted is q and then q'. As other distributed forces are inserted, they are ordered according to their initial and end positions. When xa is greater than xb, or q is less than q', these values are automatically inverted. It is possible to modify the type of load-train, even after distributed forces have been inserted. When transforming a load-train with single value distributed force to a load-train with double distributed load value, the value of q is replicated to q'.

- **Live loads**
  There are two types of live loads that represent the population of small vehicles on a bridge:
  - Exterior: is applied outside the limits (length) of current load-train;
  - Interior: is applied within the limits (length) of current load-train.

  Live loads may be applied partially along the load-train path on the structure. The portions of the path on which live loads are applied depend on influence lines. These portions are defined to maximize or to minimize a target effect. The maximum value of an effect is obtained applying the live load only on the positive portions of the influence line of that effect; and the minimum value is obtained applying the live load only on the negative portions of the influence line.

Results

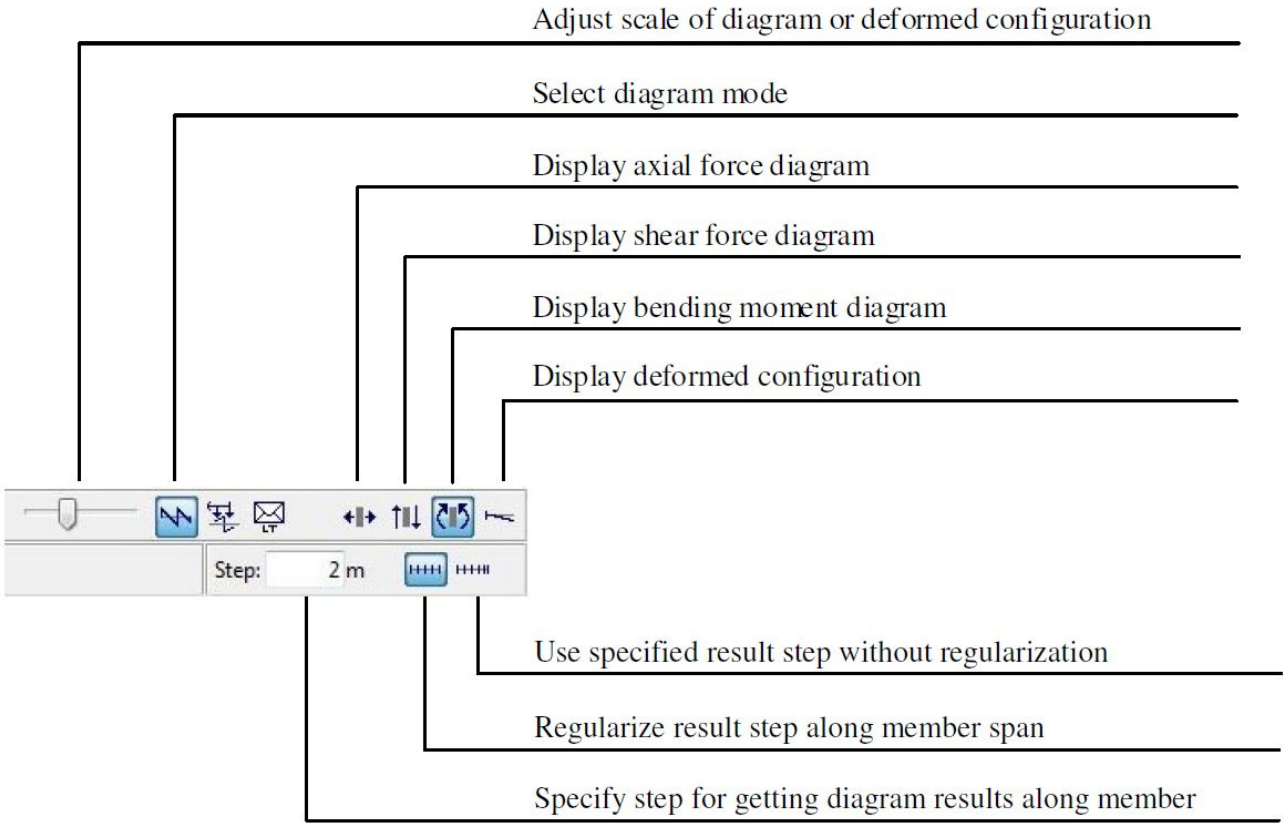
Result Modes

There are three modes for obtaining and visualizing analysis results for structures in FTOOL. These are the *Diagram*, *Influence Line*, and *Load-Train Envelop* modes of operation. Select the desired mode of operation using buttons of results toolbars, as described in the sequel.

In *Diagram* and *Load-Train Envelop* modes, FTOOL automatically analyzes the structure using the stiffness method to construct the requested diagrams. Just clicking on the appropriate button switches the display to show the desired result. In *Influence Line* mode, the program prompts the user to indicate a cross-section on a member to display the desired influence line.

Diagram Toolbar

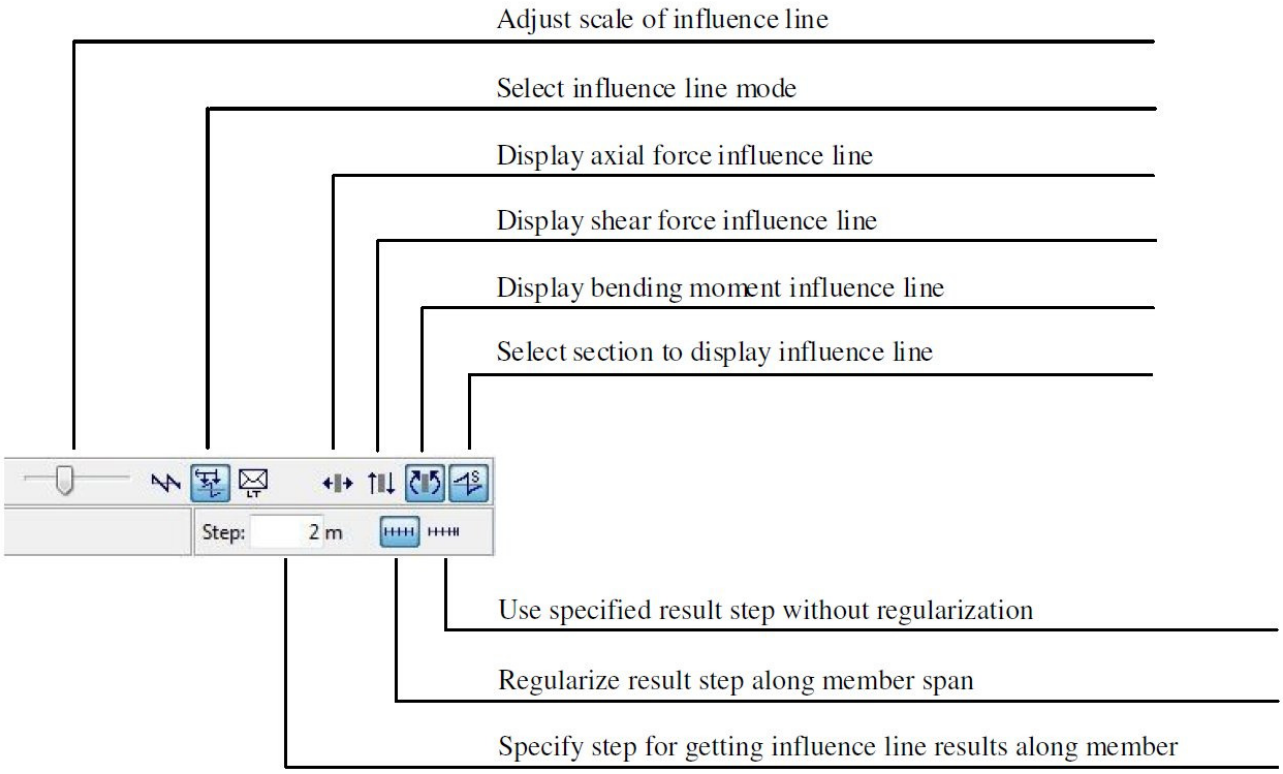
Diagram result mode considers the visualization of internal force diagrams (axial forces, shear forces, bending moments) and deformed configuration.



To get numerical results of a diagram, move the mouse cursor to a position on a member and click the left mouse button. The desired values will be displayed on the top line message. Additional results may be consulted clicking the right mouse button. These results will be displayed on the lateral area of the screen (see section *Numerical Display of Results*).

Influence Line Toolbar

In *Influence Line* mode, first click on a button for the desired type of influence line (*e.g.*, shear force or bending moment). Then place the mouse cursor at some point along a member and click the left mouse button. FTOOL imposes the required unit displacement or rotation discontinuity to create the desired influence line, which is displayed on the structural model. To display an influence line, the program ignores the applied loads on the structure. An influence line represents the effect of a unitary vertical force that traverses the members of the structure. The unitary force has always a vertical downward orientation, no matter the orientation of a member on which the force passes.



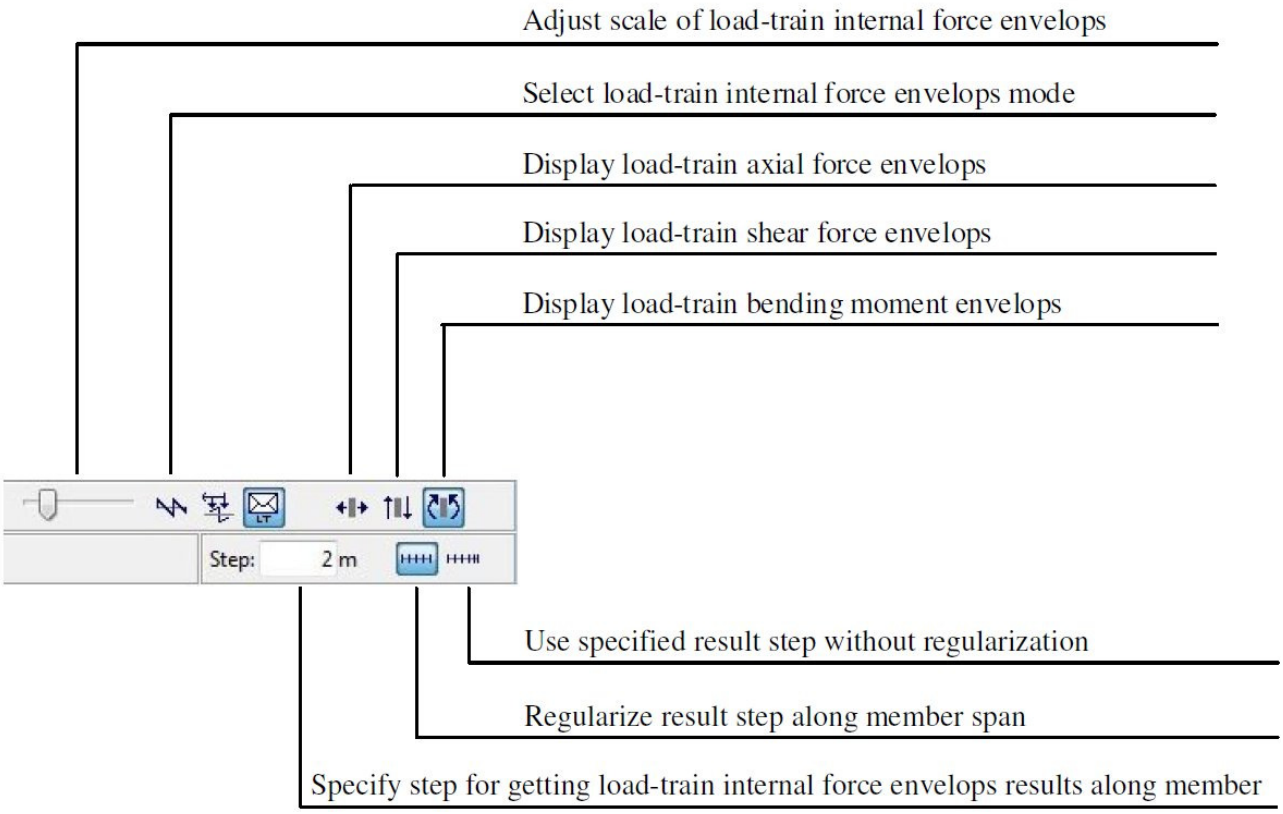
In case there is a currently selected load-train (vehicle live load for bridges), the critical positions of this load-train are displayed above and below the displayed influence line. These critical positions are the ones that cause the minimum (negative) and the maximum (positive) target result values on the desired cross-section. In this case, the influence line is displayed only along the members of the structure that belong to the load-train path.

When it is not possible to automatically define a load-train path of members along the structure, the program prompts the user to go selection mode and to select a group of members that form a continuous and smooth path for the load-train. When the user provides a valid load-train path, the program stores the corresponding group of members and the user will be prompt again to specify a load-train path only in case the model is modified.

As long as the button to select a cross-section (rightmost button of the toolbar) is active, FTOOL displays a new influence line for each selected section. To avoid the selection of another section, deactivate this button. When this button is not active, point results along the influence line may be consulted by just selecting a target position using the left mouse button. The desired value will be displayed on the top message bar. Additional member results may be consulted using the right mouse button (see section *Numerical Display of Results*).

Load-train Envelops Toolbar

The *Load-train Envelop* mode is used to display envelops of limiting internal force values along a structure. Envelops are diagrams of minimum and maximum values of an internal force (axial, shear or bending moment) for the current load-train (vehicle live load for bridges). If there is no current load-train, or if no load-train path is defined, the program does not activate the *Load-train Envelop* mode. The load-train path, if not automatically defined, has to be defined by the user selecting a continuous and smooth chain of members prior to getting an envelop result.



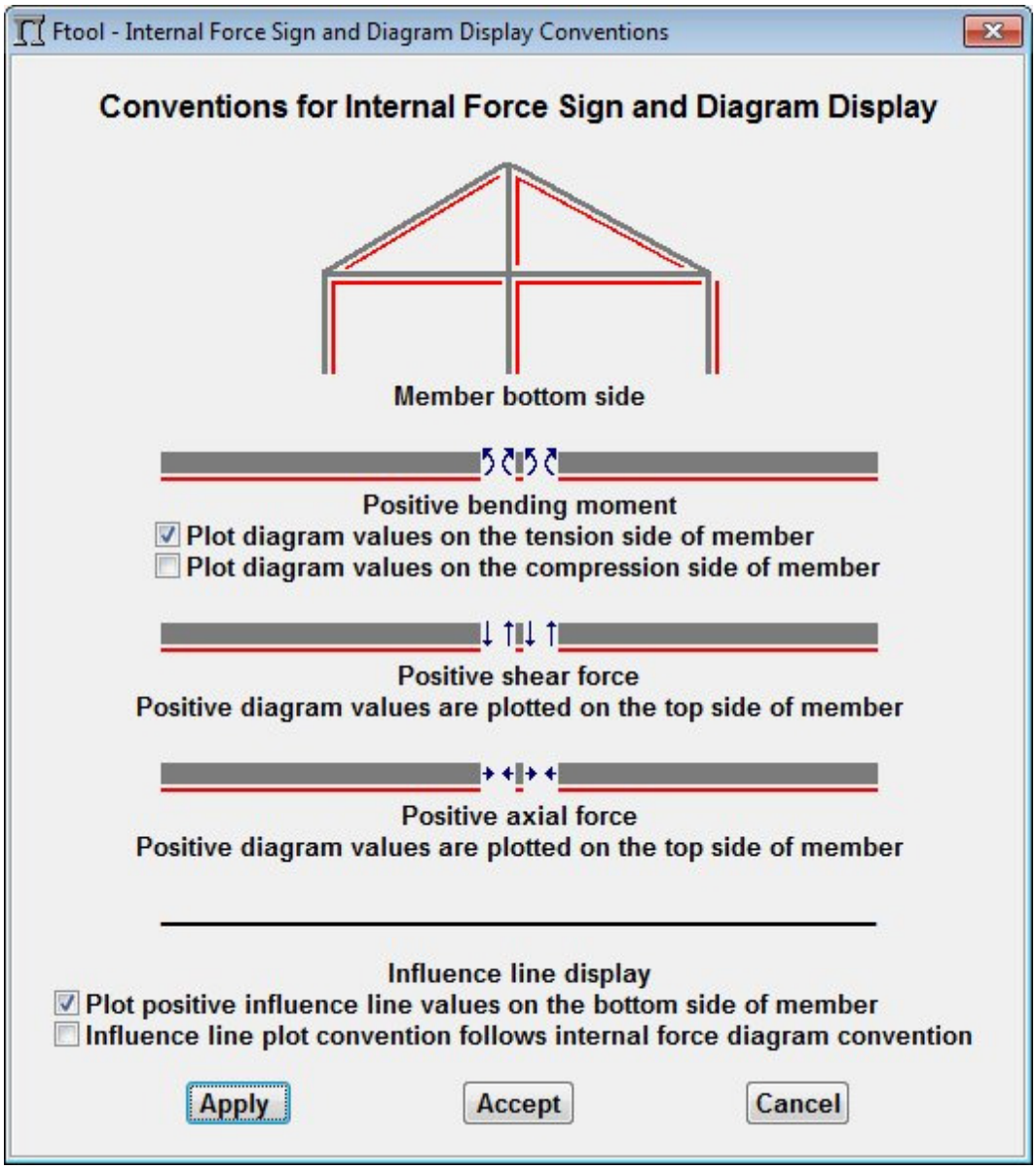
Envelops of internal forces are calculated adding the effects of permanent (fixed) loads to the current load-train (vehicle live load) effects. The diagram of internal forces due to permanent loads are displayed as dashed lines on the envelop image. In the *Options* menu (see *Configurations* section), there is an item to activate and deactivate the superposition of permanent and vehicle live load effects in the envelop calculation. By default, this superposition is activated. This option is reset when a new model is created or when a model is opened from a file.

The construction of the load-train envelops interpolates minimum and maximum results calculated at cross-sections automatically selected along the members of the structure. For each selected section, FTOOL builds the corresponding influence line and finds the critical positions of the current load-train that cause the minimum and maximum vales at that section. Envelops are diagrams that linearly interpolate the minimum and maximum values calculated at the selected cross-sections, superimposed to the permanent load diagram. The same step used to build internal force diagrams, deformed configuration, or influence lines is adopted to select the cross-sections for the construction of the load-train envelops.

Point results of load-train envelops may be obtained by just selecting a target position using the left mouse button. The minimum and maximum values at that point are shown in the top message bar.

Sign Convention for Internal Forces and Moments

The sign convention for internal forces and moments can be displayed in the program by selecting the option *Sign Convention* on the *File* menu. A window appears on the screen which shows the sign convention. The user may alter the sign convention for resulting diagrams and influence lines.



- **Top and Bottom Fibers**
  The sign convention for internal forces and moments depends on the definition of the "lower" and "upper" fibers of the members. In horizontal and inclined members, the lower fibers are in the bottom when looking at the screen with the vertical axis in its natural position (up). In vertical members, the lower fibers are on the right. The figure at the top of the sign convention display indicates lower fibers of a structure that contains members with all possible directions.

- **Sign Convention for Diagrams**
  FTOOL adopts the following convention for the signs of the internal forces and moments and for the drawing of diagrams.

  - *Normal forces (axial):*
    Normal positive forces are tension forces.

  - *Shear forces:*
    Shear forces on the diagram below are positive.

  - *Bending Moments:*
    The sign convention shown below is adopted: positive moments causing compression on the top fibers of members and tension on the bottom fibers. Bending moment diagrams may be displayed on the tension side (Brazilian convention) or on the compression side (common U.S.A. convention) of the members. Use the *File → Sign Convention* menu to select the "tension" or "compression" convention for the plotting of bending moment diagrams. When adopting the tension side option, positive bending moments are displayed on the bottom side; and, when adopting the compression side option, positive bending moments are displayed on the top side

- **Plotting Influence Lines**
  Influence Lines of internal forces in the selected sections follow the sign convention explained above. The standard for plotting influence lines can be defined in the sign convention sub-menu. There are two options: *Plot positive influence line values on the bottom side of member*, or *influence line plot convention follows corresponding internal force diagram convention.*

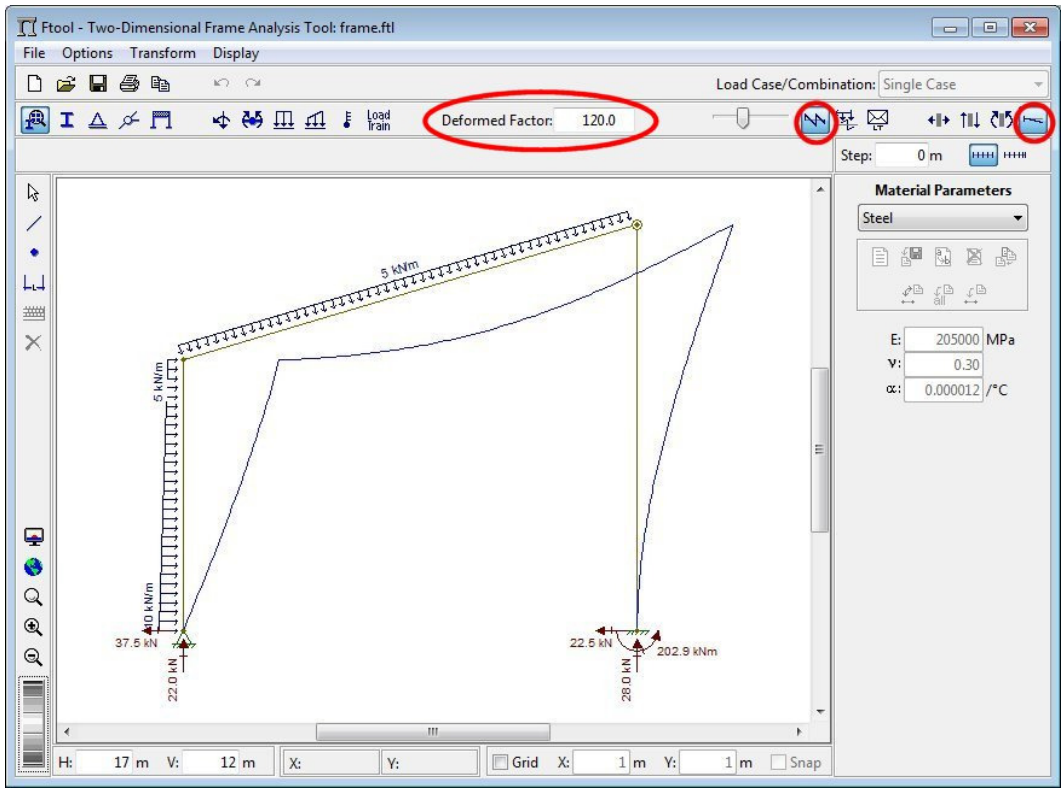
Scaling of Diagrams and Influence Lines

The internal force diagrams, the deformed configuration, the influence lines, and the envelops have a default initial scale on the screen to create a reasonable image. This scale can be modified using the slider bar at the left of buttons of the *Results* toolbars. Use the slider bar to modify the scaling of displayed diagrams as desired.

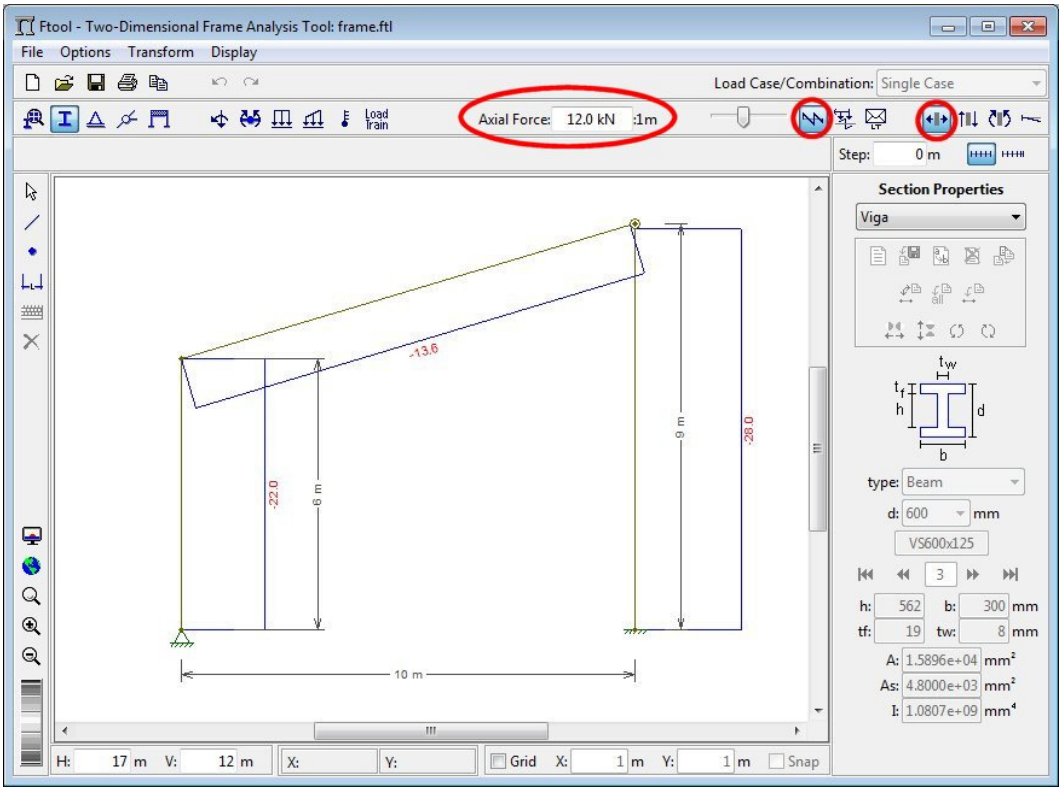
The scaling of the results may also be adjusted by the user editing its value on a text box in the top toolbar, between the load attribute buttons and the result scale slide. In *Diagram* and *Load-train Envelop* modes, the result scale factor is defined in terms of units of the corresponding internal force per unit of length. In deformed configuration, the scale factor is a displacement amplification factor (*Deformed Factor*). In influence lines, the scale factor is a non-dimensional parameter that controls the influence line display scale in relation to the model.

The images on the following sections highlight the text box on the graphics interface that is used to edit the result scale factor and the buttons that are used to obtain the desired result visualization.

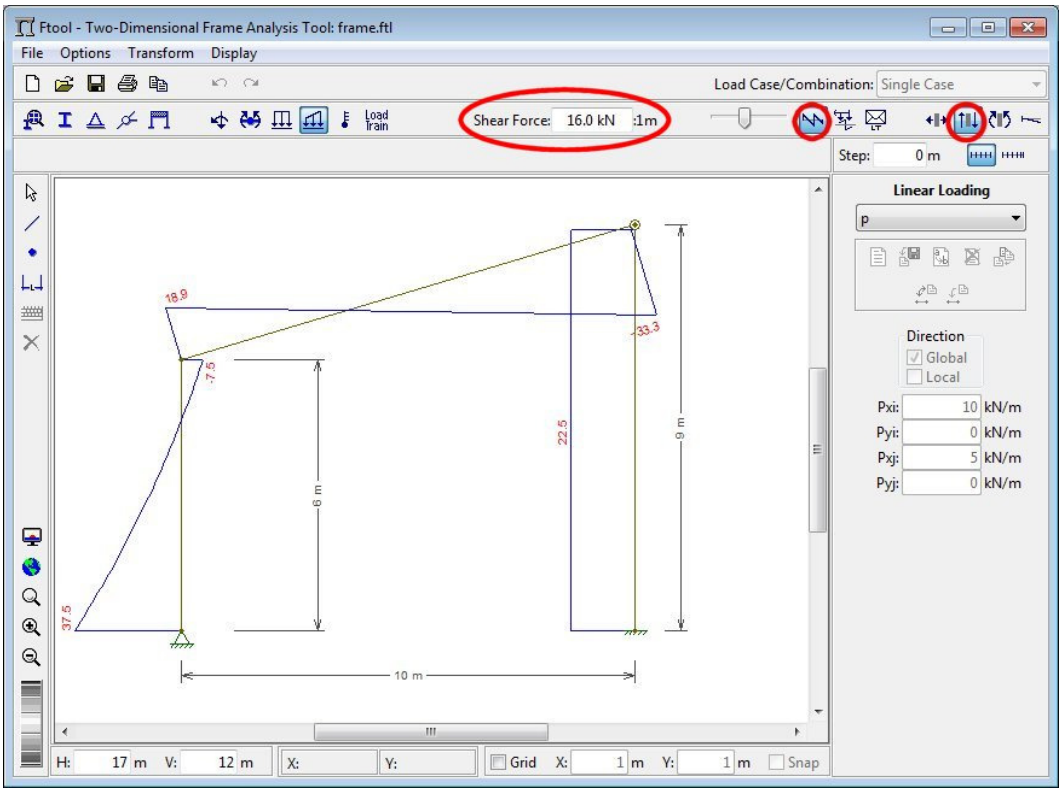
Deformed configuration



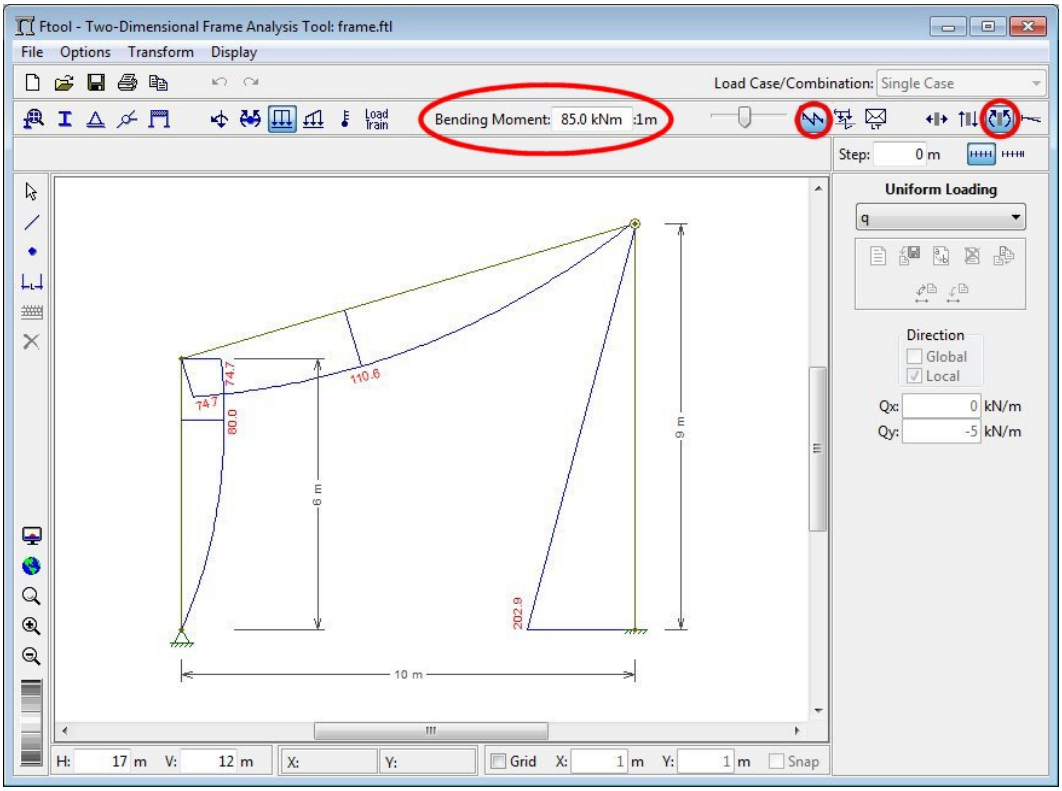
Axial force diagram



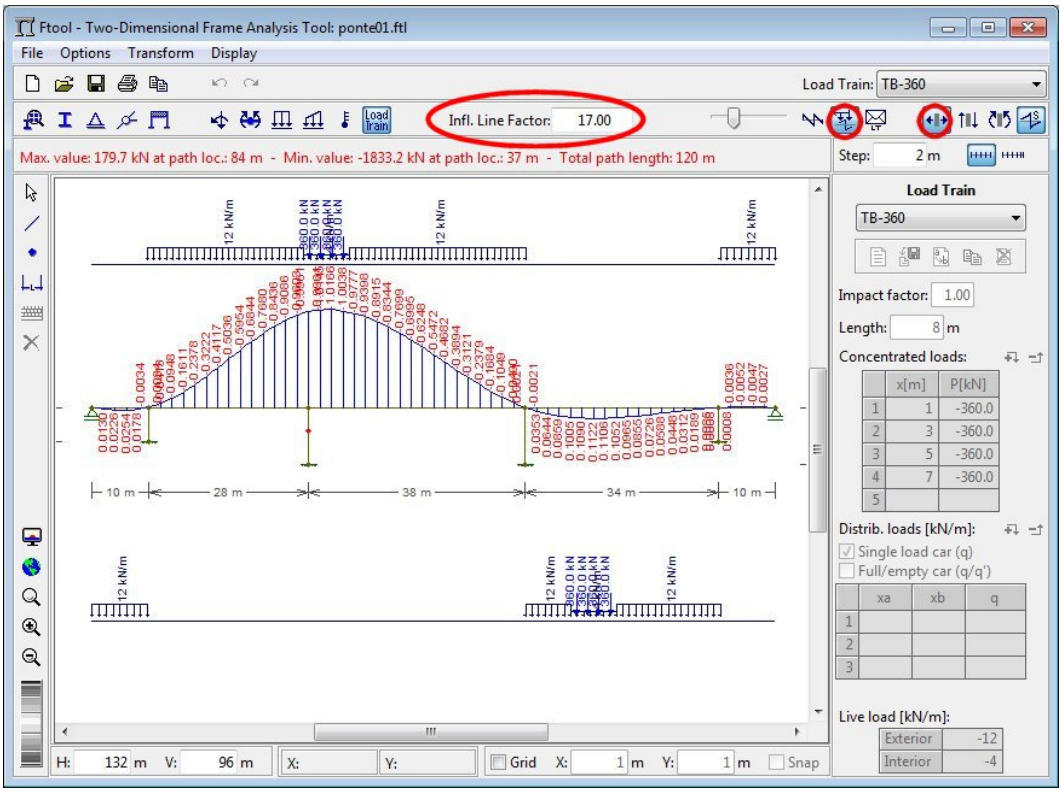
Shear force diagram



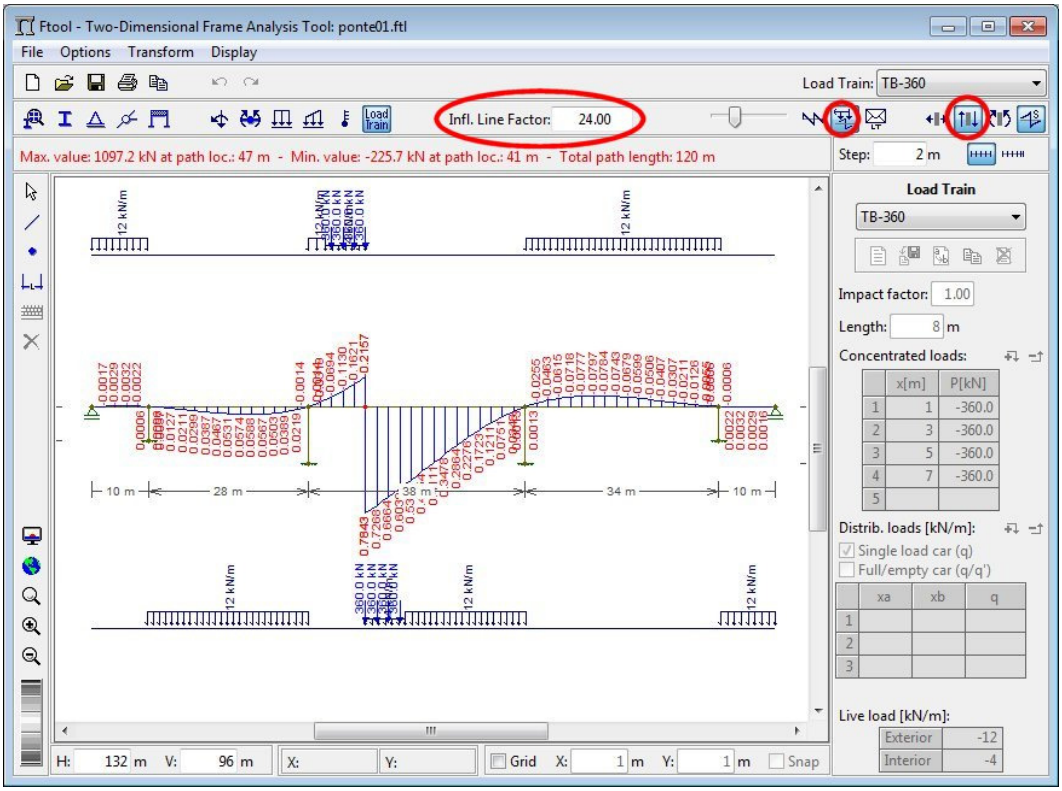
Bending moment diagram



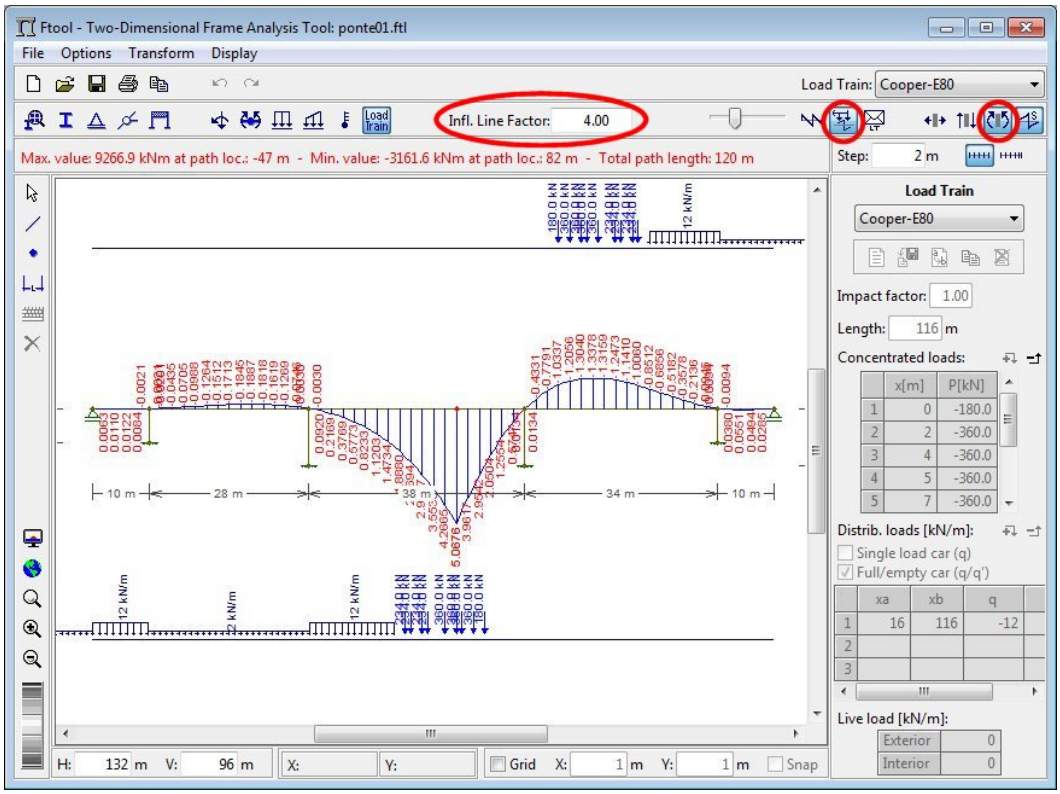
Axial force influence line



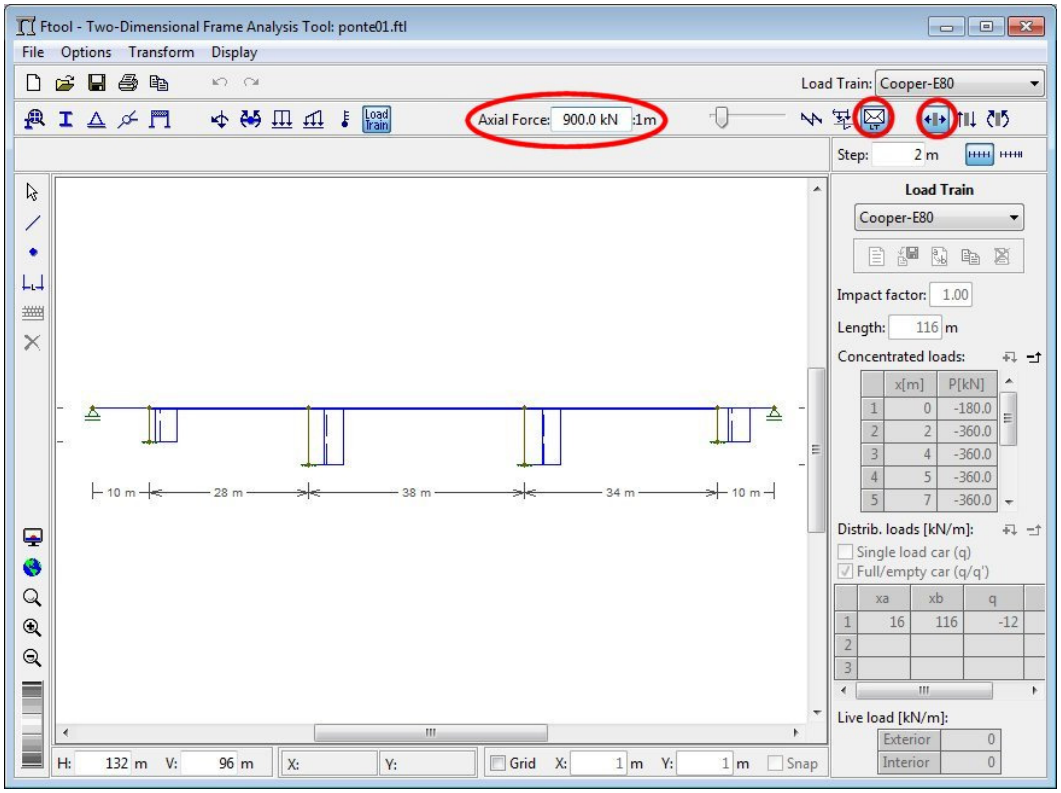
Shear force influence line



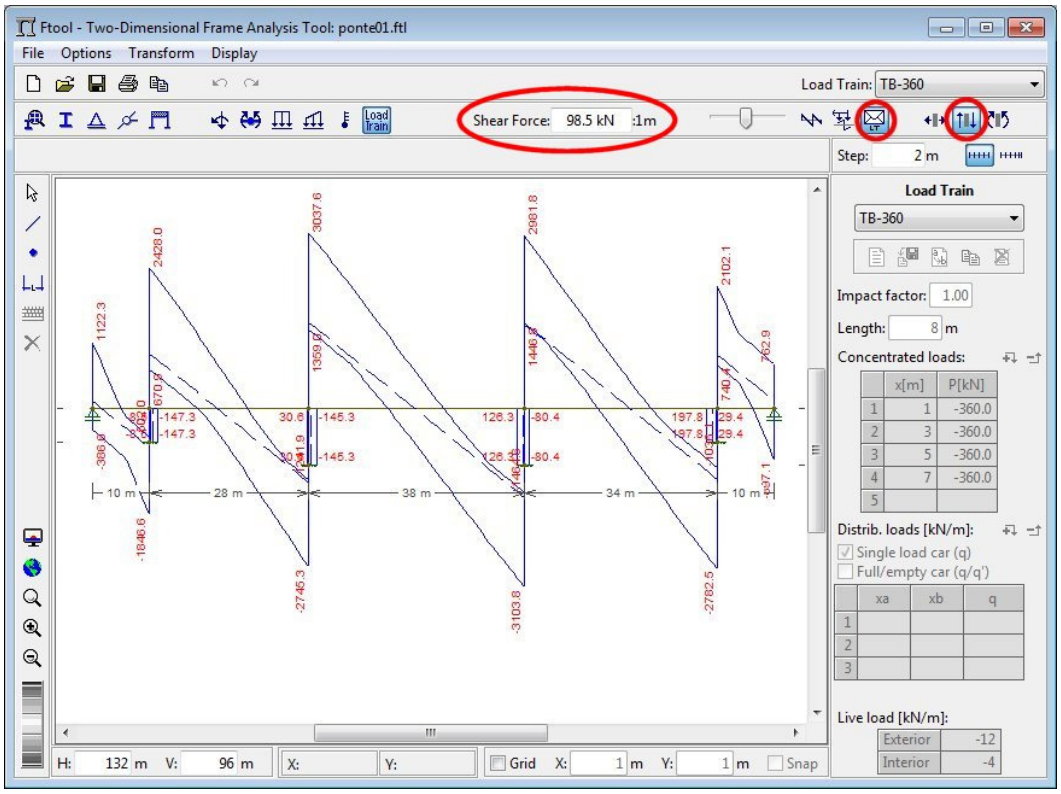
Bending moment influence line



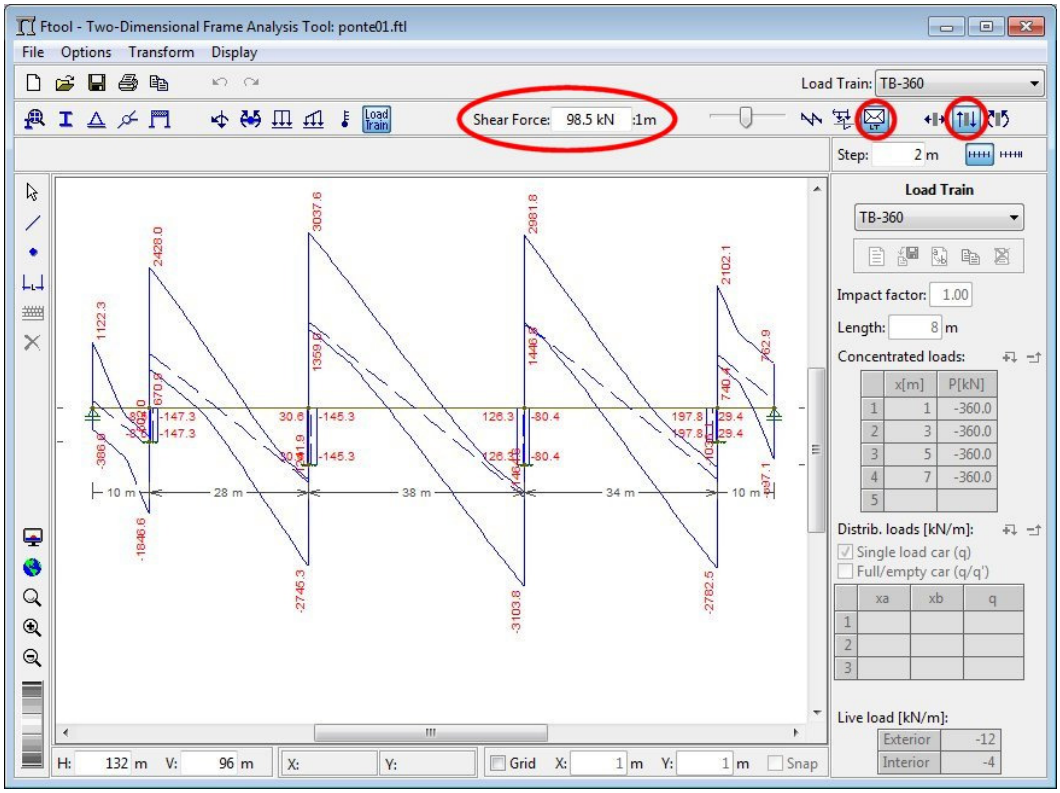
Axial force load-train envelops



Shear force load-train envelops



Bending moment load-train envelops



Numerical Display of Results

In *Diagram* mode, clicking a point on a member with the mouse left button displays the value of the corresponding diagram for that point in the top message bar above the drawing area. In this mode, clicking a node with the mouse right button displays result information, such as displacements, rotation, or reaction forces, related to the selected node in the right-most area of the screen.

In *Influence Line* mode, if the option to select a new cross-section is NOT active, clicking a point on a member with the mouse left button displays the value of the corresponding influence line in the top message bar.

In *Load-train Envelop* mode, the minimum and maximum envelop values at the selected point are display in the top message bar.

In all the modes, if the right button of mouse is used, additional information related to the selected member results appears in the right-most area of the screen. If a step for the displayed result is defined (through the *Step Values* option in the *Display* menu – see section *Visualization Controls*), the step results of the diagram for the selected member will be shown in this area.

The image below shows an example of the visualization of a bending moment diagram with step values indicated in the display and in the lateral area for the selected member.