FTOOL

Interactive, Graphical Program For Structural Analysis

Educational Version 2.11
August 2002
http://www.tecgraf.puc-rio.br/ftool

English Translation by:
Professor Robert H. Dodds, Jr.
Mr. Adam Carlyle
Department of Civil & Environmental Engineering
University of Illinois at Urbana-Champaign

November 2003

TABLE OF CONTENTS

FTOOL IN USE 3

FTOOL AUTHORSHIP 3
GENERAL CONCEPT 3
DOWNLOADING FTOOL 4
BACKGROUND 4
NEW FEATURES OF THE LATEST VERSION 2.11 (AUGUST 2002) 5
NEW FEATURES OF VERSION 2.10 (NOVEMBER 2001) 6
NEW FEATURES OF VERSION 2.09 (JANUARY 2001) 6
NEW FEATURES OF VERSION 2.08 (AUGUST 2000) 7
NEW FEATURES OF VERSION 2.07 (APRIL 2000) 7
NEW FEATURES OF VERSION 2.06 (FEBRUARY 2000) 8
ADDITIONAL CREDITS 8

FILE OPERATIONS 9

THE *FILE* MENU 9
EXPORTING SCREEN DISPLAY IMAGES THROUGH THE *CLIPBOARD* 10

MODEL CREATION AND MANIPULATION 11

THE *EDIT* TOOLBAR 11
MEMBERS AND NODES 11
CREATION OF DIMENSION LINES 11
KEYBOARD MODE 12
SELECTION MODE 12
UNDO AND *REDO* 12
THE *TRANSFORM* MENU 13

VISUALIZATION CONTROLS 14

THE *VISUALIZATION CONTROL* TOOLBAR 14
COORDINATE CONTROL 14
THE *DISPLAY* MENU 15

CONFIGURATIONS 16

THE *OPTIONS* MENU 16
FORMATTING UNITS AND NUMBERS 16
UNIT SYSTEMS 17

NODE AND MEMBER ATTRIBUTES 21

CONTROL OF NODE AND MEMBER ATTRIBUTES 21
FEATURES COMMON TO SUB-MENUS 21
MATERIAL PARAMETERS 22
MEMBER SECTION PROPERTIES 22
SUPPORT CONDITIONS 24
CONNECTION OF MEMBER ENDS TO NODES 25
CONSTRAINTS ON MEMBER DEFORMATIONS 25

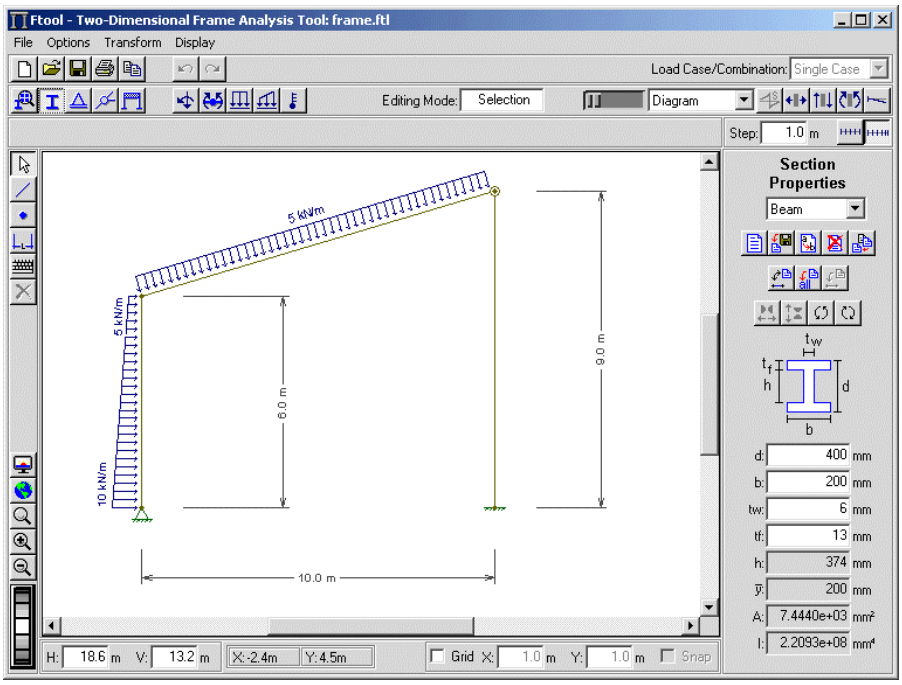
DEFINING THE APPLIED LOADS 26

THE *LOAD CONTROL* TOOLBAR 26
GENERAL INFORMATION 26
NODAL LOADS 28
MOMENTS ACTING ON MEMBER ENDS 28
LINEAR AND UNIFORMLY DISTRIBUTED LOADS 29
TEMPERATURE CHANGES 29

RESULTS 30

THE *RESULTS* TOOLBAR 30
RESULT MODES 30
SIGN CONVENTION FOR INTERNAL FORCES AND MOMENTS 31
SCALING OF DIAGRAMS AND INFLUENCE LINES 33
NUMERICAL DISPLAY OF RESULTS 34

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+3114-1190
Fax: 55+21+3114-1195
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). Users of this **educational version** are free to apply and re-distribute the program as they wish. However, neither the author nor the PUC-Rio University or any other related institution is responsible for the incorrect use of the program and its results. The author and PUC-Rio have no legal responsibility for any damage caused directly or indirectly to a person or a company, resulting from the application of any information or the utilization of the program. Users are responsible for all conclusions made through the program. There is no commitment of satisfactory performance or any kind of warranty.

Downloading FTOOL

- Ftool Version 2.11 for Windows:
  ftp://ftp.tecgraf.puc-rio.br/pub/users/lfm/ftool211win.zip
- Ftool Version 2.11 for Linux:
  ftp://ftp.tecgraf.puc-rio.br/pub/users/lfm/ftool211linux.tgz
  the available version is compatible to library glibc 2.0 for Linux and operates libraries of OSF Motif (http://www.openmotif.org).
- Download of this guidebook in CHM (*Compiled HTML Help*):
  ftp://ftp.tecgraf.puc-rio.br/pub/users/lfm/ftool211pt.chm (Portuguese)
  ftp://ftp.tecgraf.puc-rio.br/pub/users/lfm/ftool211en.chm (English)
- Download of this guidebook in Portuguese (PDF format):
  ftp://ftp.tecgraf.puc-rio.br/pub/users/lfm/ftoolman211pt.pdf (Portuguese)
  ftp://ftp.tecgraf.puc-rio.br/pub/users/lfm/ftoolman211en.pdf (English)

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. Graduate 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 PhD student of PUC-Rio Ivan Fábio Menezes, (currently professor of the Department of Computer Science 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 small 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 the last version, 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. 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 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 bet-

ter 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 parameter 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.

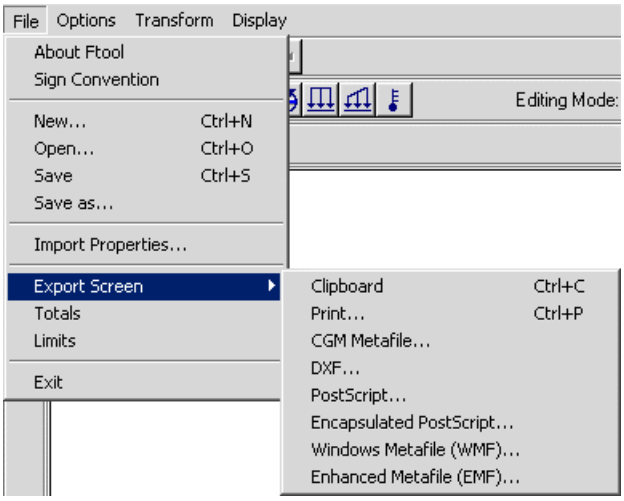
Additional Credits

- The first version of this manual (for version 2.07) was created by Luis Fernando Kaefer, 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, graduate student in Civil Engineering in PUC-Rio.
- The update of manual for version 2.11 was contributed by Pedro Cordeiro Marques, undergraduate student in Civil Engineering in PUC-Rio.

File Operations

The *File* Menu

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

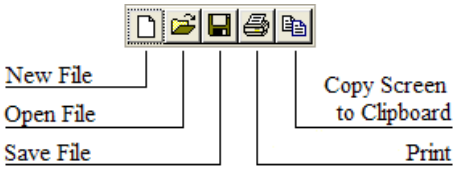


The *File* menu allows the user to:

- Show the Sign Convention for the internal processing associated with FTOOL (*Sign Convention*) – see details of the sign convention in the Post-Processing section;
- Create a new model (*New);*
- Open a model from an existing file on a disk (*Open*);
- Save the current model in a file (*Save*) or under a different name (*Save as*);
- Import properties from another FTOOL file (*Import Properties*);
- 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 limit 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 NT, 2000 or XP, select option *Paste* in menu *Edit* or press *Ctrl+V*. It is possible to choose the option *Paste Special*, then, the option *Picture*. In Windows, DO NOT select the option *Picture (Enhanced Metafile)* – this creates an extremely large image. For Windows, the option *Picture* generates a small image that can be scaled by pressing the mouse button over one of the corners of the selected image.

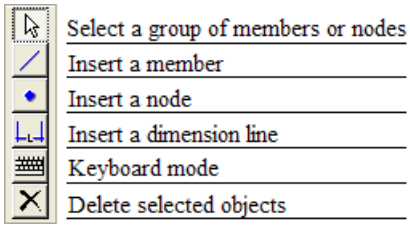
- Edit the pasted figure. In Windows 95 or 98, circles resulting from rotation or moments might appear black when editing the figure. It has not been discovered how to prevent this. To correct it, select the object and change the background color to white (or to the color of the background). This problem does not occur for Windows NT, 2000 or XP. 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.

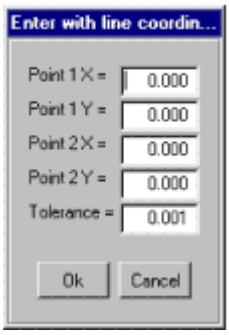
Creation of Dimension Lines

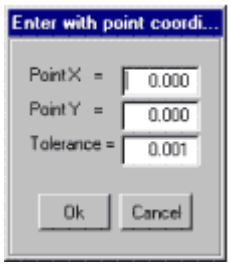
Dimension Lines are auxiliary lines used to indicate distances on the structural model. To insert a dimension line, select the corresponding button in the Edit toolbar and click three points on the screen. The two first points are the control points to specify the distance to be listed by the dimension. 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  , nodes and members can be created by entering coordinates into the pop-up box fields. 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 (See section below). 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.

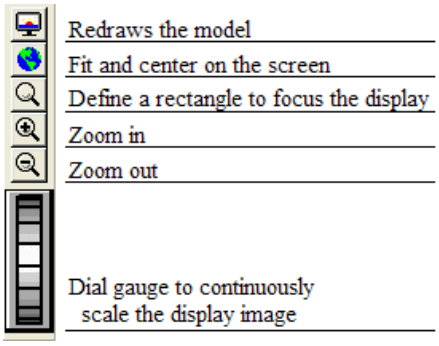
The *Transform* Menu

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

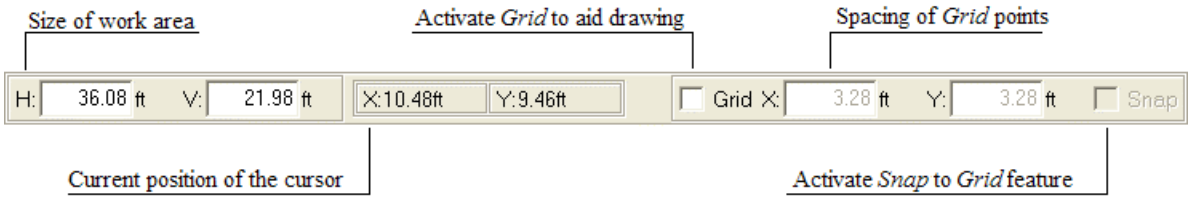
This toolbar groups all the controls for the model visualization window.



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.

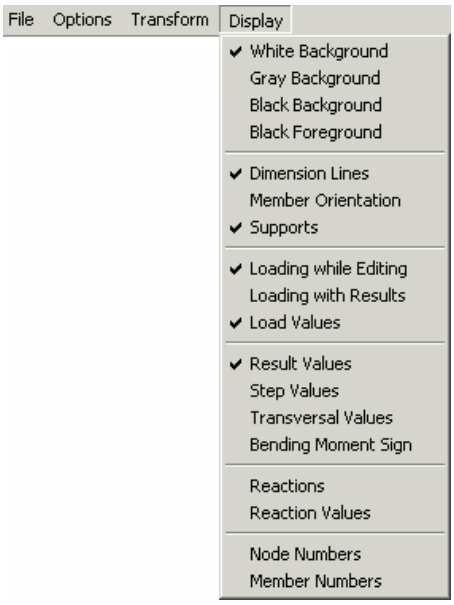
Coordinate Control

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

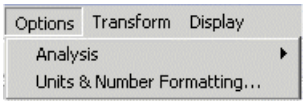
This menu lists all the items that may be displayed. The user can choose the background color, for example. A check mark indicates the display quantity is active. Just click on list items with the mouse. For each background color selected there are different colors related to the members and nodes of 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.



Configurations

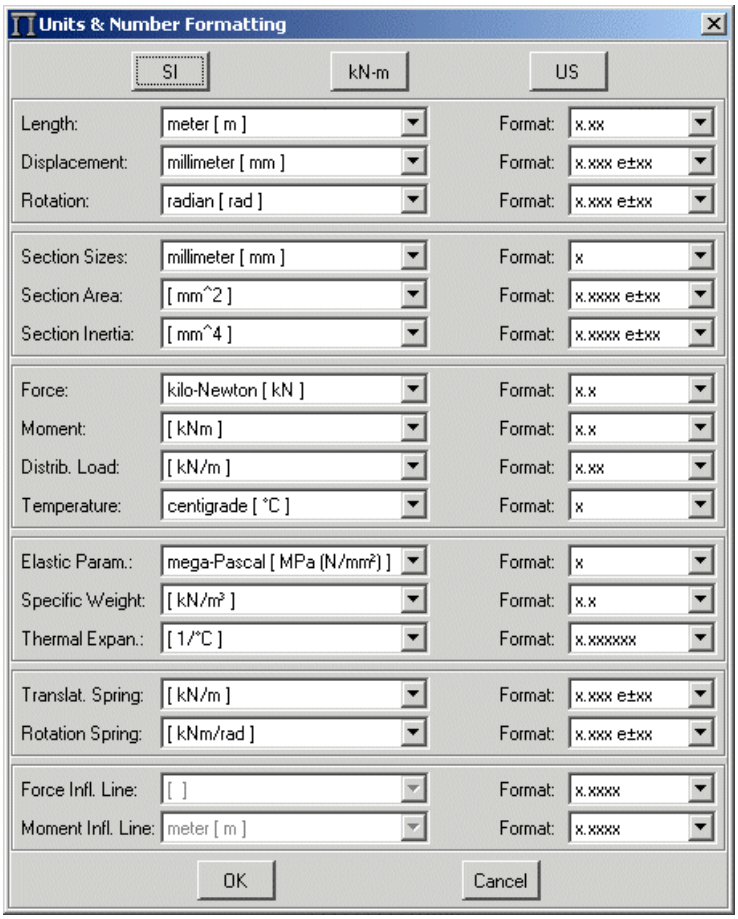
The *Options* Menu

Currently, the options in FTOOL configurable by the user include the physical unit systems and the format for numerical values. The option *Units & Number Formatting* within the *Options* menu accesses parameters for these configurations.



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			

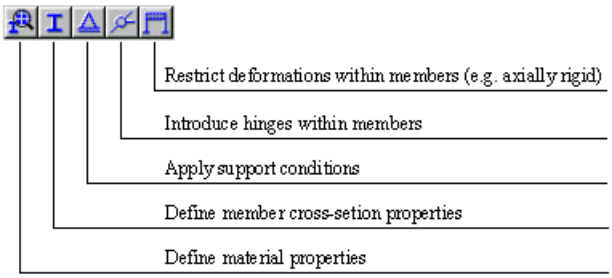
Parameter	**SI units**			**US units**		
	Symbol	**Name**	**Factor of ref. SI unit**	**Symbol**	**Name**	**Factor of ref. SI unit**
Distributed	**kN/m**		1.0	k/ft	kip/foot	14.593
Load	N/m		0.001	lb/ft	pound/foot	0.014593
	tf/m		9.81	k/in	kip/inch	175.1
	kN/cm		100.0	lb/in	pound/foot	0.1751
	N/cm		0.1			
	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	**°C**	centigrade	1.0	°F	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/°C**		1.0	1/°F		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

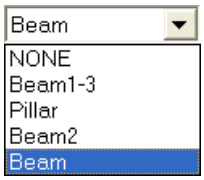
Control of Node and Member Attributes

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.

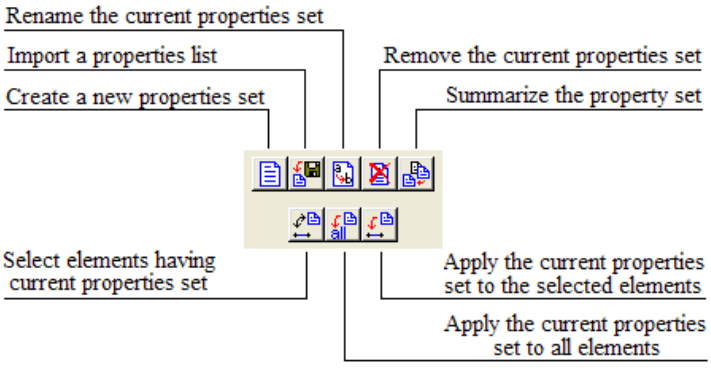


Features Common to Sub-menus

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.

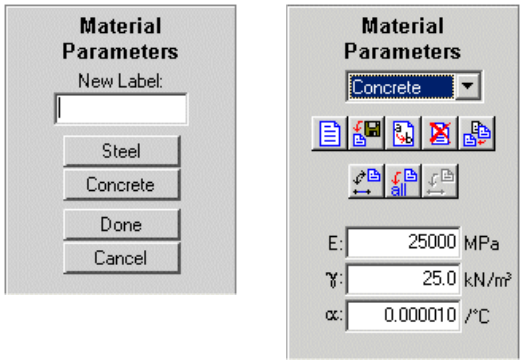


To create a new set of properties, select the button  and assign the new set a unique name. The button  imports a list of property sets from another file generated by FTOOL. Duplicate property sets in the imported file are ignored. The internal function

that compresses the property sets just eliminates those which are not in use in the current model.

Material Parameters

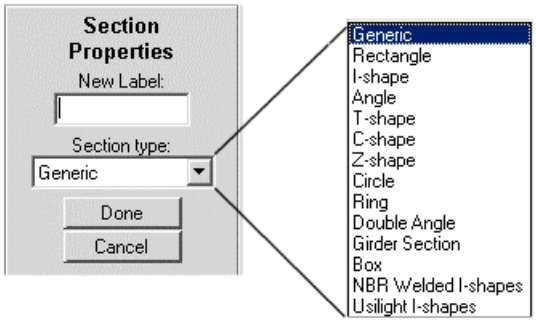
To create a new set of material parameters, select button  and specify a unique name (figure below to the left). Default values of material parameters are available for steel or concrete.



Material parameters include Young's modulus, specific weight (not currently used by FTOOL) and coefficient of thermal expansion (if 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*, *Box*, *NBR welded I-shapes* or *Usilight I-shapes*.

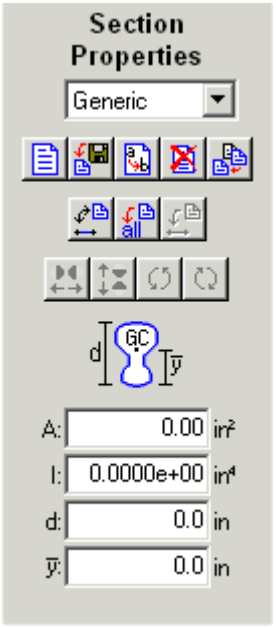


Based on the type of section selected, the parameters that define the features of that section are then defined along the side of the screen under the schematic drawing.

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

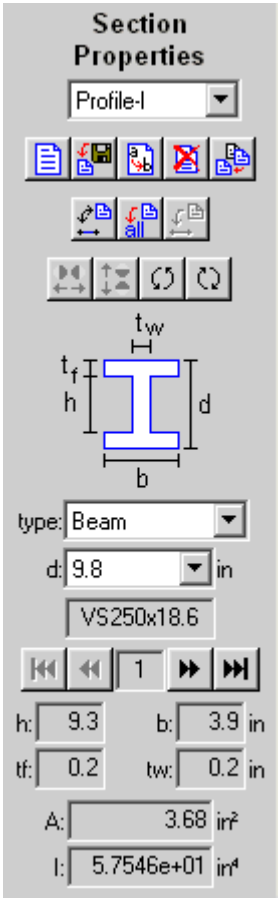
- A: Area
- I: Moment of Inertia (Second Moment of the Area)
- d: Section depth
- $\bar{y}$: Centroid position.

Defaults: only A and I are required.



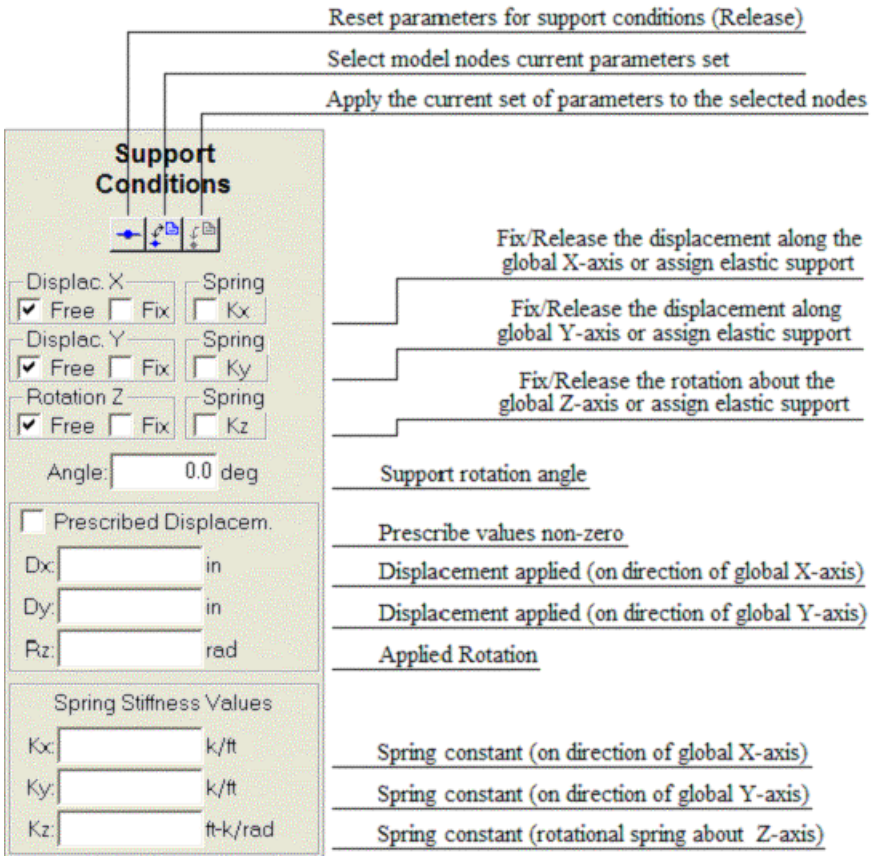
For the section type *Profile - NBR Welded-I*, (shown here), choose the type of profile (*Beam, Column* or *Beam-Column*) and the depth of the section, *d*.

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.



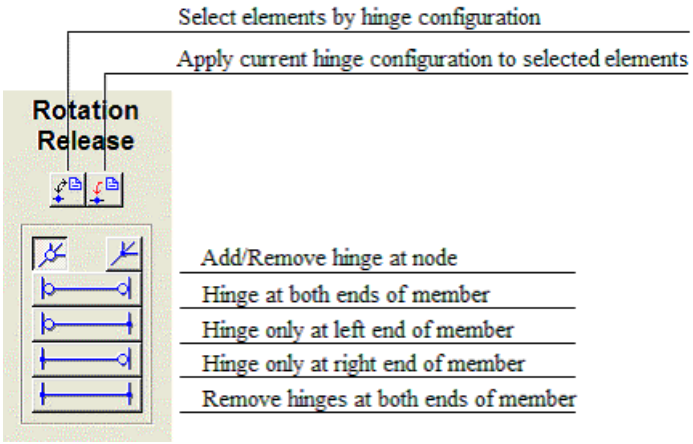
Support Conditions

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.



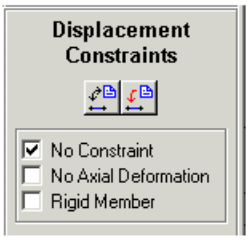
Connection of Member Ends to Nodes

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

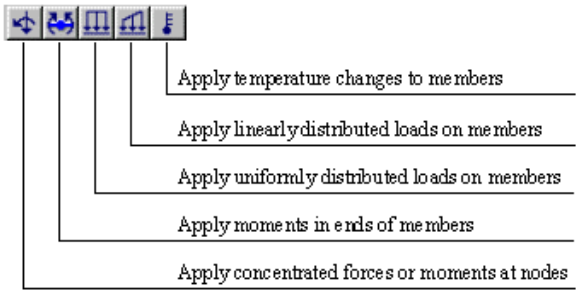
This sub-menu restricts deformations within members. It is possible either to restrict axial deformations of a member or to consider a member as infinitely rigid.



Defining the Applied Loads

The *Load Control* Toolbar

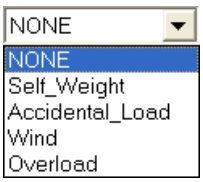
Sub-menus are also available to define or modify various nodal and member loadings. These sub-menus appear along the side of the screen. The types of loads available are the following: concentrated loads at nodes, moments applied at nodes, moments applied at the ends of members, uniformly or linearly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



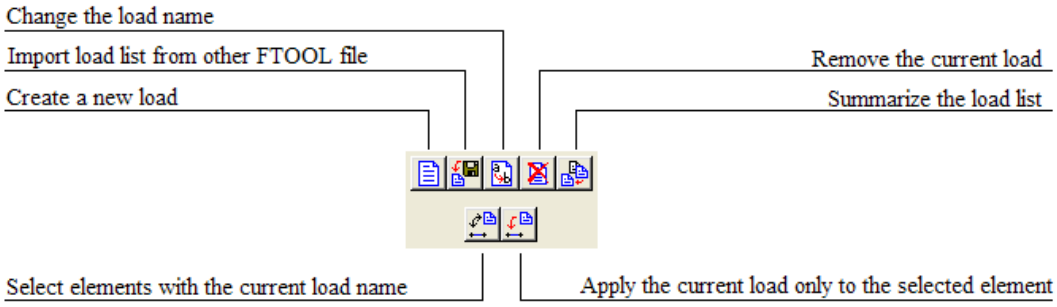
General Information

- **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 *X*-axis is horizontal and positive from left to right; and the *Y*-axis is vertical and positive from the bottom to the top. 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: $Y = Z \times X$.

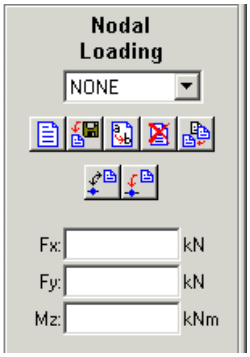
- **Application of concentrated loads**
  Concentrated loads (forces and moments) can be applied only on nodes of the structure. This is a current limitation of the program. If it is necessary to apply a concentrated load on a member, insert a new node at the desired position, thereby dividing the member 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).

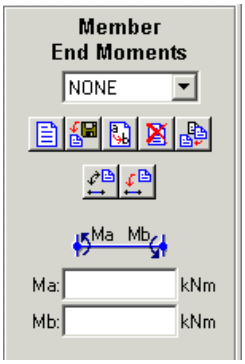
Nodal Loads

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



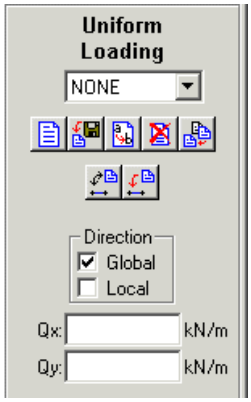
Moments Acting on Member Ends

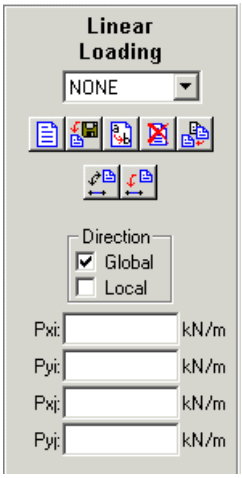
This sub-menu allows the user to define concentrated moments at the ends of members. Moments applied in a counterclockwise orientation are positive. "Ma" denotes the moment applied at the "initial" node of the member, while "Mb" is the moment applied at the "ending" node of the member.



Linear and Uniformly Distributed Loads

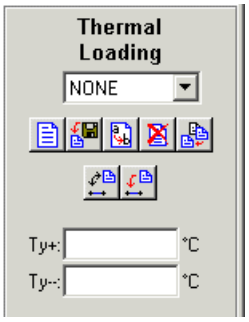
This sub-menu allows the user to define linearly varying or uniformly distributed loads over a member. The user can specify the global or local member coordinate system for the loading direction.





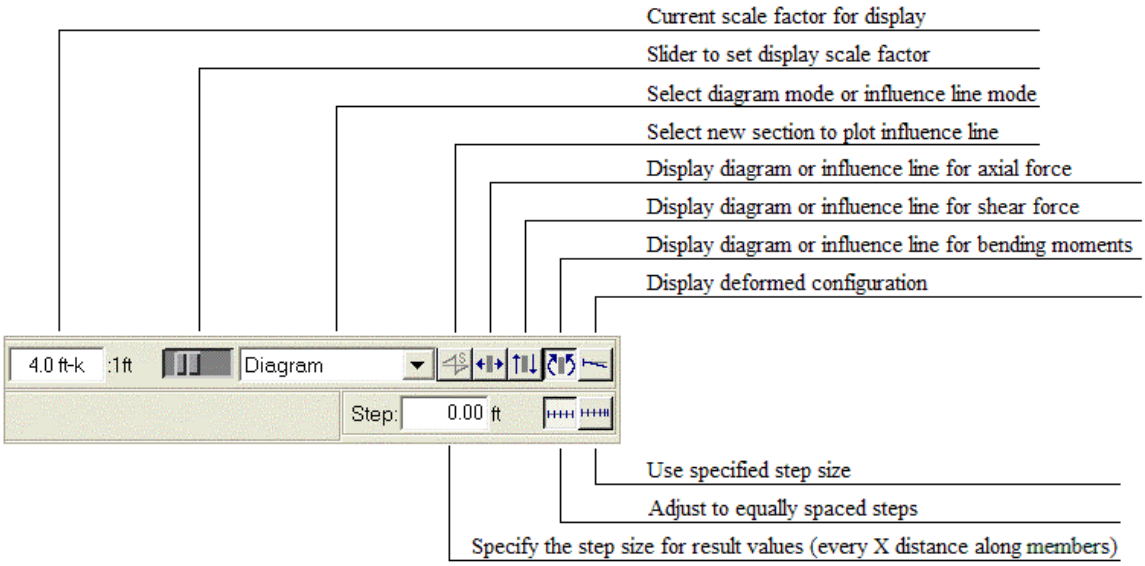
Temperature Changes

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.

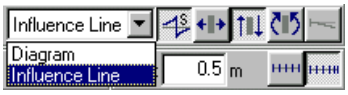


Results

The *Results* Toolbar



Result Modes



There are two methods to obtain and display analysis results for structures in FTOOL. These are the *Diagram* and *Influence Line* modes of operation. Select the desired mode of operation using the drop-down menu shown above.

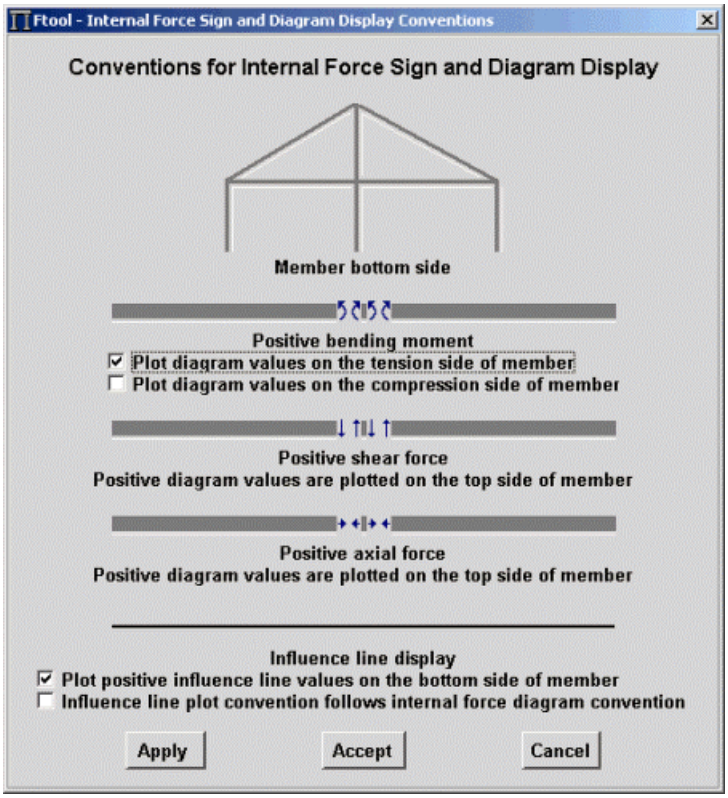
In *Diagram* mode, FTOOL automatically analyzes the structure as required using the stiffness method to construct the requested diagrams: axial forces, shear forces, bending moments, or deformed shape. Use the slider bar to modify the scaling of displayed diagrams as desired. Just clicking on the appropriate button switches the display to show the quantity desired. Values at locations along members can be displayed. Place the mouse cursor over a point on the member and click the right mouse button. Values are shown in a window created at the right side of the display at specified increments along the member.

In *Influence Line* mode, the applied loads on the structure are ignored. First click on the button for the desired type of influence line (*e.g.*, shear or moment). Now place the mouse cursor at some point along a member and click the left mouse button. FTOOL imposes the required unit loading and displacement discontinuity to create the influence

line, which is displayed on the structural model. Move the mouse cursor to a position on another member and click the right mouse button. This displays numerical values of the influence line along that member.

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.

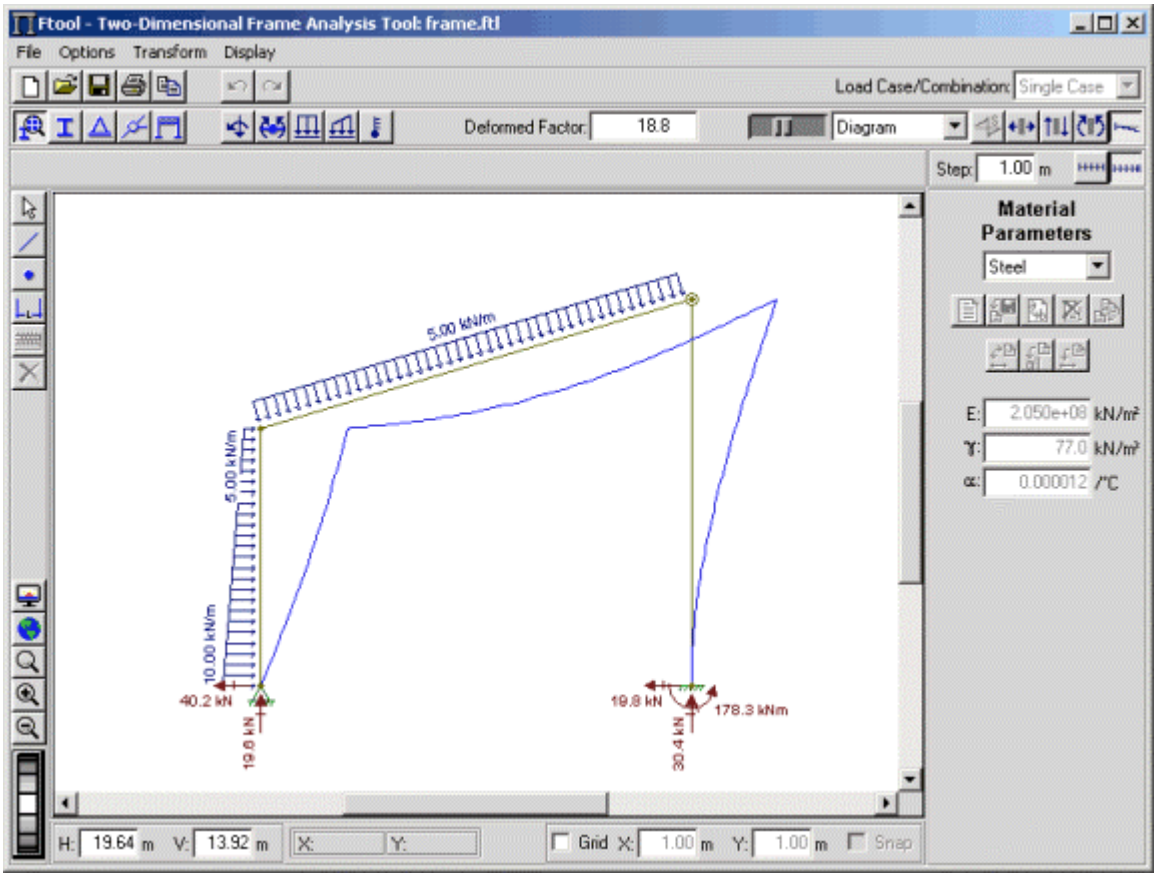
  - *Bending Moments:*
    The sign convention shown below is adopted: positive moments causing compression on the top fibers of members and tension on the bottom fibers. Bending moment diagrams may be displayed on the tension side (Brazilian convention) or on the compression side (common U.S.A. convention) 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

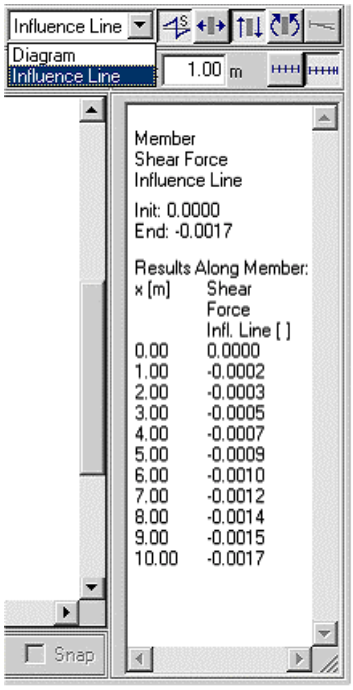
The force and moment diagrams and the deformed configuration have a default initial scale on the screen to create a reasonable image. This scale can be modified using the slider bar at the left of the *Results* toolbar.

Similarly, the influence lines also have a single initial scale. This scale can be modified using the slider bar at the left of the *Results* toolbar.



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 message bar above the drawing area. If the right button of mouse is used, additional information related to the diagram appears in the rightmost area of the screen. If a step for the displayed result is defined, the results of the diagram for the selected member will be shown in this area.



In *Diagram* mode, right-click the mouse on a node to show the pertinent information for the node, such as displacements and support reactions.

Influence line values along a member are displayed using the same procedure.