



EngiLab Beam.2D ML v1.20
Linear Static Analysis of Plane Frames Program

USER MANUAL

v1.10 Nov. 29, 2004



www.engilab.com

© 2004 EngiLab Co.

*Concern for man himself and his fate
must always form the chief interest of all technical endeavors...
Never forget this in the midst of your diagrams and equations.*

Albert Einstein

License & Copyright

EngiLab Beam.2D ML Program

© 1999-2004 EngiLab Co.

EngiLab Beam.2D ML is copyrighted and all rights are reserved. The license for use is granted to the purchaser by EngiLab Co. as a single user license and does not permit the program to be used on more than one computer at one time. Copying of the program to other media is permitted for back-up purposes only as long as all copies remain in the possession of the purchaser.

EngiLab Beam.2D ML User Manual and Tutorial

© 2004 EngiLab Co.

All rights reserved. No part of this publication may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any language in any form or by any means, without the written permission of EngiLab Co. EngiLab Co. reserves the right to revise this publication from time to time and to make changes to the contents without obligation to notify any person or organization of such changes.

Disclaimer of warranty – Terms of use

This software and any documentation is provided by the copyright holders and contributors "as is", without any guarantee made as to its suitability or fitness for any particular use. The program may contain bugs, so its use is at the user's risk. To the best of our knowledge the software is accurate and complies with the standards of good engineering practice. We have made every effort to make the use of this program as reliable and safe as possible. However, no responsibility whatsoever is accepted to any person or company whatsoever, nor is any duty or obligation owed to them as regards the accurate and safe use of this software or part thereof. No liability of any nature is accepted for any error, bug or omission contained in the software package whether caused by any negligence on the part of the author or otherwise howsoever.

If, for any reason, you do not agree to these terms of use, do not use the software and destroy all copies of it. No EngiLab distributor, or agent, or employee is authorized to make any modification, extension, or addition to this warranty.

Contents

Contents	4
Figures index	6
Tables index	6
1. Introduction	7
1.1 Overview.....	7
1.2 Analysis capabilities	7
1.2.1 Continuous (multi-span) beam model example	7
1.2.2 Plane frame model example	8
1.2.3 Plane truss model example.....	8
1.3 Multilingual operation.....	9
1.4 System requirements.....	9
1.5 Installation	9
1.6 Removing the program	10
1.7 Technical support	10
2. Using the program.....	11
2.1 System of units	11
2.2 The program's environment.....	12
2.3 Files commands	12
2.4 View commands	13
2.5 Model commands	13
2.6 Post-processing view commands	13
2.7 View options command	14
2.7.1 View options: General pane.....	14
2.7.2 View options: Model pane.....	14
2.7.3 View options: M, Q, N pane	15
2.8 Analyze command	15
2.9 Data – Results command	15
3. Model	16
3.1 Model components	16
3.2 Setting up a model and viewing the analysis results	16
3.2.1 Setting up a model.....	16
3.2.2 Once the model is set up	17
3.2.3 Setting up a model graphically.....	17
3.3 E,A,I Element Groups	17
3.4 Nodes	18
3.5 Elements.....	19
3.6 Nodal loads.....	19
3.7 Elemental loads	20
4. Program tools	21
4.1 Move nodes tool	21
4.2 Merge nodes tool.....	21
4.3 Divide elements tool	22
4.4 Create arc tool	23
4.5 Check tool.....	23
4.5.1 Check for Coincident nodes.....	23
4.5.2 Check for Coincident elements.....	24
4.6 Calculator tool.....	24

5.	Analysis and post-processing	25
5.1	Analysis	25
5.1.1	<i>Errors during the analysis process.....</i>	<i>25</i>
5.2	M, Q, N Diagrams	26
5.3	Deformation	26
5.4	Model data and analytical results	27
5.4.1	<i>Node displacements</i>	<i>28</i>
5.4.2	<i>Element forces</i>	<i>29</i>
5.4.3	<i>Constraint – spring reactions</i>	<i>30</i>
5.4.4	<i>Exporting data and the analysis results</i>	<i>30</i>
6.	Useful information.....	31
6.1	Ready-to-analyze example files	31
6.2	Tips on hinges and truss modeling.....	31
6.2.1	<i>External hinges</i>	<i>31</i>
6.2.2	<i>Internal hinges</i>	<i>32</i>
6.3	Tips on modeling structures with symmetries.....	34
6.3.1	<i>Symmetrical structure (axis) + Symmetrical loads.....</i>	<i>34</i>
6.3.2	<i>Symmetrical structure (axis) + Anti-Symmetrical loads.....</i>	<i>34</i>
6.3.3	<i>Symmetrical structure (point) + Symmetrical loads.....</i>	<i>35</i>
6.3.4	<i>Symmetrical structure (point) + Anti-Symmetrical loads</i>	<i>35</i>

Figures index

Figure 1. Continuous beam model example.....	7
Figure 2. Plane frame model example.....	8
Figure 3. Plane truss model example.....	8
Figure 4. The program's environment	12
Figure 5. View options window, General pane	14
Figure 6. View options window, Model pane	14
Figure 7. View options window, M,Q,N pane.....	15
Figure 8. Model components.....	16
Figure 9. E,A,I Element Groups window.....	17
Figure 10. Nodes window	18
Figure 11. Types of nodes.....	18
Figure 12. Elements window	19
Figure 13. Types of elements.....	19
Figure 14. Nodal loads window	19
Figure 15. Elemental loads window.....	20
Figure 16. Move nodes tool window.....	21
Figure 17. Merge nodes tool window.....	21
Figure 18. Merge nodes example.....	22
Figure 19. Divide elements tool window.....	22
Figure 20. Create arc tool window.....	23
Figure 21. Check for coincident nodes tolerance control window	23
Figure 22. Windows calculator window	24
Figure 23. Data and results window	27
Figure 24. Data and results window, Node displacements pane.....	28
Figure 25. Data and results window, Element forces pane	29
Figure 26. Data and results window, Constraint – spring reactions pane	30
Figure 27. Two ways of modeling external hinges with one connecting element.....	32
Figure 28. Modeling external hinges with more than one connecting element.....	32
Figure 29. Modeling 'partial' internal hinges	33
Figure 30. Modeling 'partial' internal hinges	33
Figure 31. Symmetric structure (axis) + symmetrical loads.....	34
Figure 32. Symmetric structure (axis) + anti-symmetrical loads.....	34
Figure 33. Symmetric structure (point) + symmetrical loads	35
Figure 34. Symmetric structure (point) + anti-symmetrical loads	35

Tables index

Table 1. Units example (Input data).....	11
Table 2. Units example (Output results)	11

Chapter 1

1. Introduction

1.1 Overview

EngiLab Beam.2D ML is an easy-to-use yet powerful engineering tool for the **linear static analysis of plane frames for Windows**. It features a Full GUI (Graphical User Interface) for pre-processing or post-processing and uses the **Finite Element Method (FEM)** for plane frames for its analysis purposes.

1.2 Analysis capabilities

EngiLab Beam.2D ML can analyze continuous (multi-span) beams, plane frames and plane trusses. The program can handle models of up to 200 nodes, 300 elements and 300 E,A,I Element Groups.

1.2.1 Continuous (multi-span) beam model example

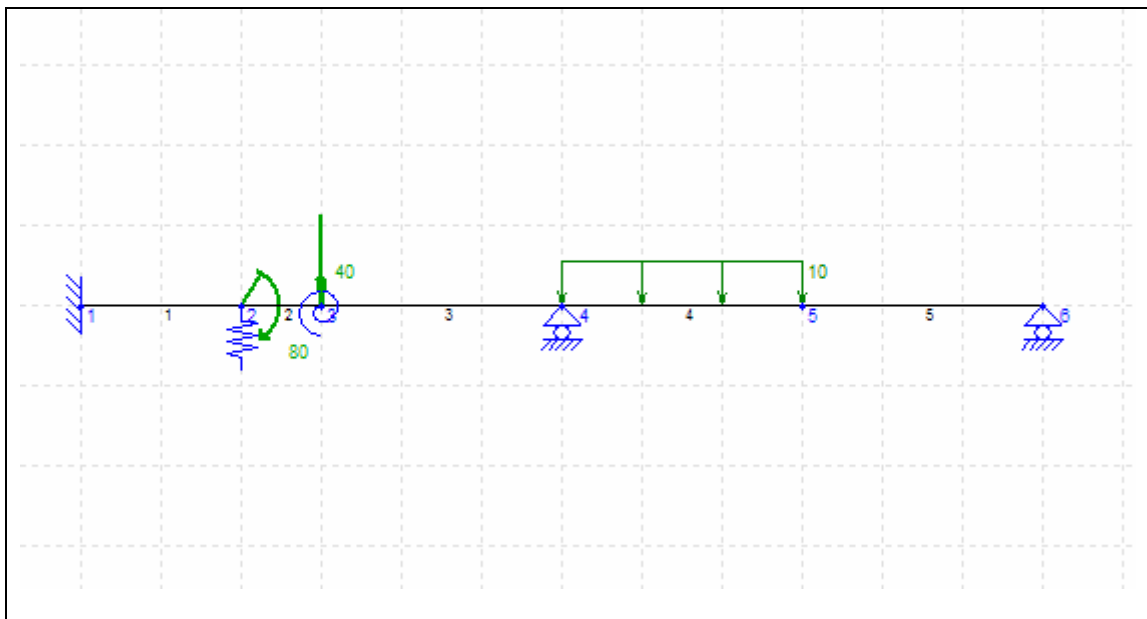


Figure 1. Continuous beam model example

1.3 Multilingual operation

The Multilingual (ML) Edition features full multilingual operation. Languages supported in this version include:

- English
- French
- Italian
- Spanish
- Bahasa Malaysia
- Bulgarian
- Czech
- Danish
- Dutch
- Finnish
- Greek
- Macedonian
- Norwegian
- Polish
- Portuguese (Brazilian)
- Swedish
- Simplified Chinese
- Traditional Chinese
- Turkish

1.4 System requirements

O/S:	Windows 95/98/ME/NT/2000/XP or later compatible version
Display:	Color monitor with a minimum resolution of 640 x 480 pixels Works best with resolutions of 800 x 600 pixels and higher
Disk Space:	Approx. 25 MB of free disk space for installation
Memory:	Minimum free system memory of 16MB when program is run

1.5 Installation

Run the Setup program provided (Setup.exe) and follow the on-screen instructions. After the installation a shortcut will be placed in the Start Menu (Start > Programs > Beam.2D ML).

Note that in order to minimize the package size, the Visual Basic 6.0 SP5 Runtime files have been excluded from the installation package. These files are required for the program to work properly. If you don't have them or you are unsure, you can download them from the Microsoft Corp. Web Site at:

<http://download.microsoft.com/download/vb60pro/Redist/sp5/WIN98Me/EN-US/vbrun60sp5.exe>

1.6 Removing the program

If you wish to uninstall the program, the recommended method of removal is:

- Go to the Windows 'Control Panel' and click the 'Add or Remove Programs' icon (Start > Settings > Control Panel > Add/Remove Programs).
- Select EngiLab Beam.2D ML from the list of programs and click 'Change/Remove'.
- When prompted whether you are sure you wish to remove EngiLab Beam.2D ML, select YES.

This will remove the program and all of its components from your system.

1.7 Technical support

For technical support on EngiLab Beam.2D ML you may contact the developers team via e-mail at support@engilab.com. You may also find additional information, new releases and others on the web at www.engilab.com.

Chapter 2

2. Using the program

2.1 System of units

Specific units are not currently supported in EngiLab Beam.2D ML, as the program uses a consistent approach to measuring units. **Consistent** means that the user must choose specific measuring units only for the two basic quantities which are **DISTANCE and FORCE**. All quantities that are derived from them must then follow the measuring units that these two use.

If, for example, the user chooses to use the S.I. system (Distance in meters and force in Newton) then all data must be given as:

Quantity	Unit used
x, y node coordinates	m
Youngs modulus E	Pa = N/m ²
Cross section area A	m ²
Cross section moment of inertia I	m ⁴
Nodal force F	N
Nodal moment M	N*m
Elemental load f	N/m
Elastic constants Kx, Ky	N/m
Elastic constant Kz	N*m (/RAD)

Table 1. Units example (Input data)

The results will also comply with that system, thus they will be reported as shown in the table below.

Quantity	Unit used
Node displacement	m
Node z-rotation	RAD (See note)
Axial, shear force at element end i, j	N
Moment at element end i, j	Nm
Constraint reaction x, y (force)	N
Constraint reaction (moment)	N*m
Spring reaction x, y (force)	N
Spring reaction (moment)	N*m

Table 2. Units example (Output results)

Practically, **ANY other consistent system of units can be used** (English, Imperial, etc). In the above example, **N and m can be replaced with any other appropriate unit** (for example, pounds and inches for the English system).

Note that rotations are always given in RADIANS.

2.2 The program's environment

The program features a user friendly intuitive graphical environment. The main commands are shown in the figure below.

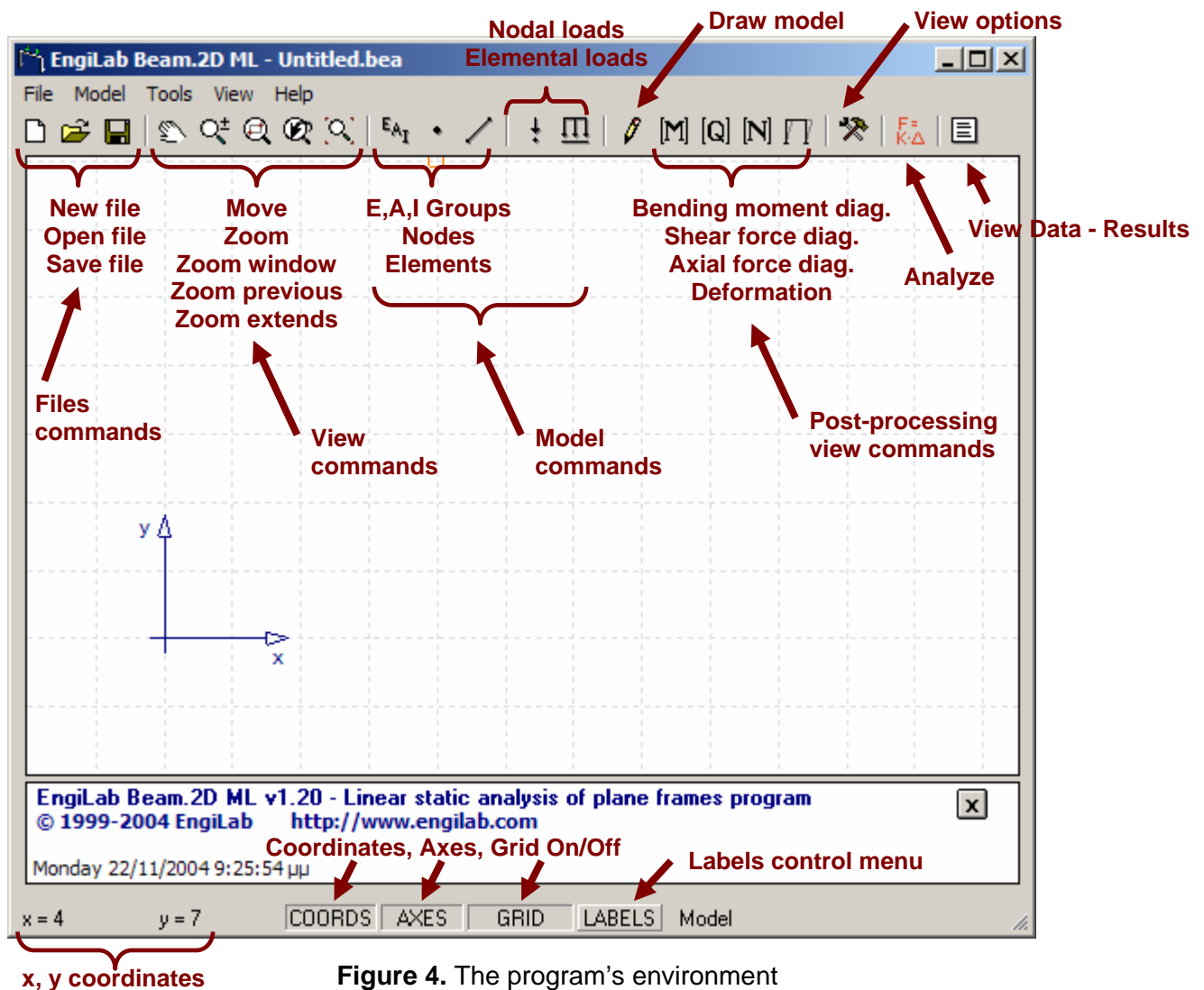





Figure 4. The program's environment

For detailed information on setting up a model, see section 3.2 (Setting up a model and viewing the analysis results).

2.3 Files commands

Buttons:   






Use these buttons to:

-  Start a new model
-  Open an existing .bea EngiLab Beam.2D input file
-  Save the current model as a .bea EngiLab Beam.2D input file

2.4 View commands






Buttons:     

Use these buttons to:






-  Move the drawing
-  Zoom in and out
-  Zoom into a specified window
-  Return to the previous view
-  Zoom extends

When you press the Move, Zoom or Zoom window buttons, you enter the Zoom mode. You can exit the zoom mode by right-clicking on the picture and then clicking 'Exit'.

2.5 Model commands

Buttons:     

Use these buttons to:





-  Bring up the 'E,A,I Element Groups' window to specify the material and cross section data
-  Bring up the 'Nodes' window to specify the nodes coordinates, constraints and spring properties
-  Bring up the 'Elements' window to specify the group each element belongs to, element connectivity and rotation releases
-  Bring up the 'Nodal loads' window to specify the nodal forces or moments
-  Bring up the 'Elemental loads' window to specify the elemental uniform loads

For detailed information about each model command, see Chapter 3 (Model).

2.6 Post-processing view commands

Buttons:    

Use these buttons **after the analysis has been performed** to:

-  Draw the bending moment diagram
-  Draw the shear force diagram
-  Draw the axial force diagram
-  Draw the deformed model

In order to return to the normal model view, press the 'Draw model' button .

2.7 View options command

Button: 

Use this button to:

-  Bring up the 'View options' window to control the picture view settings.

2.7.1 View options: General pane

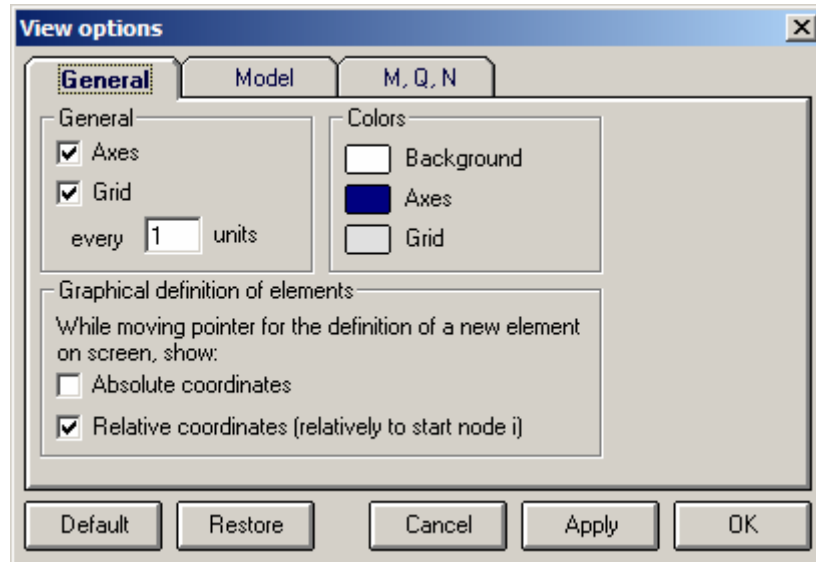


Figure 5. View options window, General pane

2.7.2 View options: Model pane

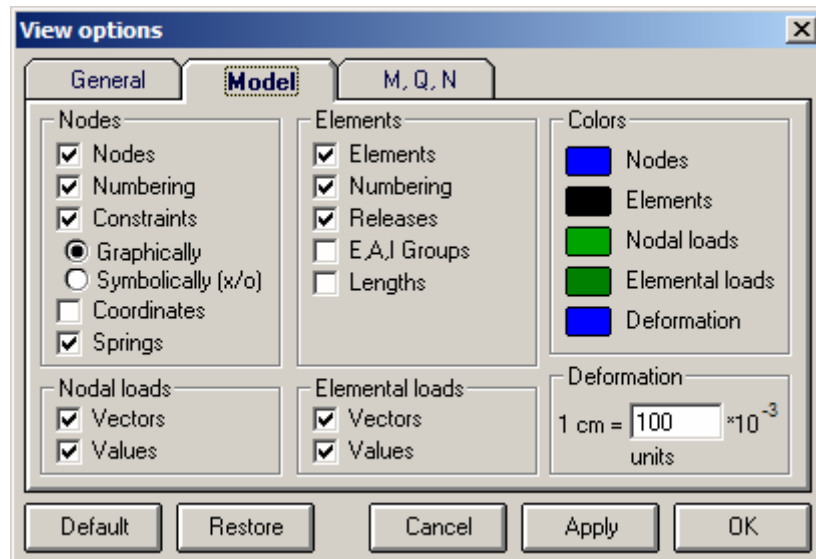


Figure 6. View options window, Model pane

2.7.3 View options: M, Q, N pane

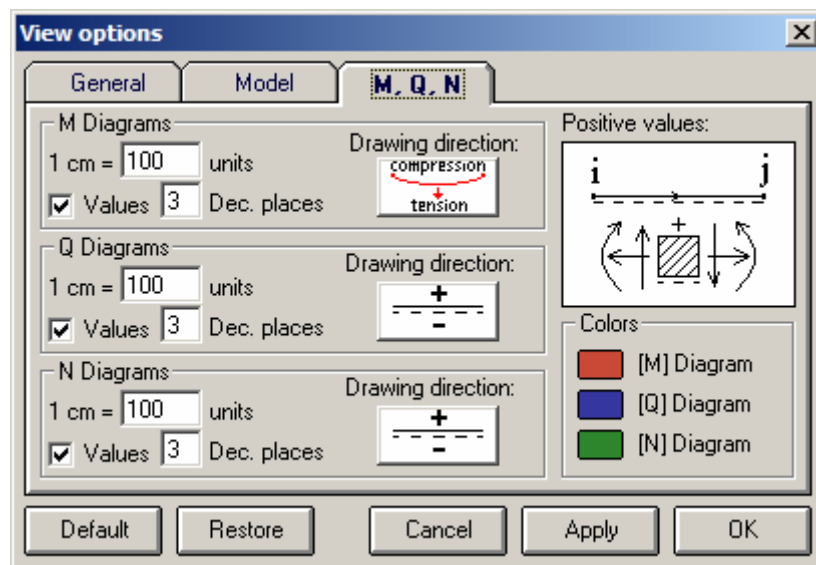


Figure 7. View options window, M,Q,N pane

2.8 Analyze command

Button:

Use this button to:

- Perform the Finite Elements Analysis of the model

For detailed information on the analysis, see Chapter 5 (Analysis and post-processing).

2.9 Data – Results command

Button:

Use this button to:

- Bring up the 'Data – Results' window with analytical information on the model data and the analysis results (Node displacements, Element forces, Constraint reactions).

For detailed information on the data and results, see Section 5.4 (Model data and analytical results).

Chapter 3

3. Model

3.1 Model components

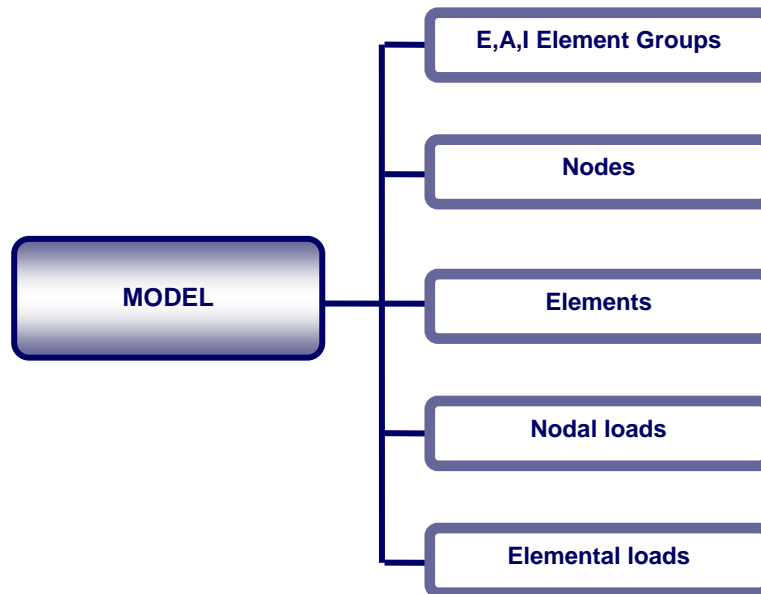







Figure 8. Model components

A model consists of 5 components, as seen in the figure above.







3.2 Setting up a model and viewing the analysis results

3.2.1 Setting up a model

The user can set-up a model using the following simple steps:

1. Click  to: Create one or more **E,A,I Element Groups**. Each element must belong to an E,A,I element group which describes the material (E) and cross section (A, I) properties of the element.
2. Click  and  to: Create **nodes and elements**. Also, define the constrained Degrees Of Freedom (DOFs), fixed [xxx], pinned [xxo], roller [xoo] or [oxo], etc, springs and any rotation releases of elements.
3. Click  and  to: Define the **nodal and elemental loads**.

3.2.2 Once the model is set up

1. Click  to: **Analyze** the model.
2. Click , ,  to: **Draw the M, Q, N diagrams** (bending moment, shear force and axial force diagrams).
3. Click  to: **Draw the deformed state of the model**
4. Click  to: **View the data and analytical results** (node displacements, element forces and constraint reactions).

See also Chapter 5 (Analysis and post-processing) for additional information.

3.2.3 Setting up a model graphically

A model can be set-up analytically using the procedure described above, or graphically using the mouse pointer and the mouse buttons on screen:

- Left-clicking on screen you can define a free node at the mouse pointer location.
- If you left-click on screen, hold down the button and then release it at another location, you can define an element and two nodes at ends i, j.
- Double-clicking on a node or near it, you can change the constraints of the node (node must be located on the grid).
- Right-clicking near a node or an element, brings up a pop-up window for quick access to the nodes, elements, nodal loads and elemental loads control windows. You can turn off the Grid (by clicking on 'GRID' on the program's status bar) in order to 'catch' nodes that are not located on the grid.

Note that the E,A,I Element Groups as well as springs, loads (nodal or elemental) and the element releases can be defined only analytically.

3.3 E,A,I Element Groups

Button: 

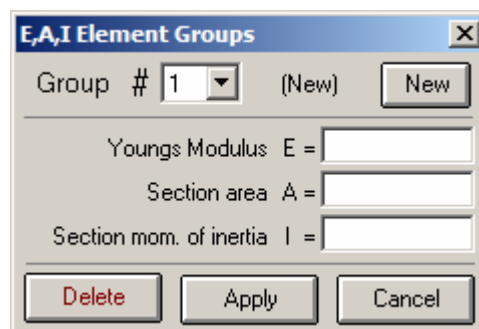


Figure 9. E,A,I Element Groups window

Each element of a model belongs to an E,A,I Element Group which describes the material and cross section properties of the element; the material's Young's Modulus E, the cross section area A and the cross section moment of inertia I.

Note: You cannot delete an E,A,I Element group if there are elements that belong to it.

3.4 Nodes

Button: 

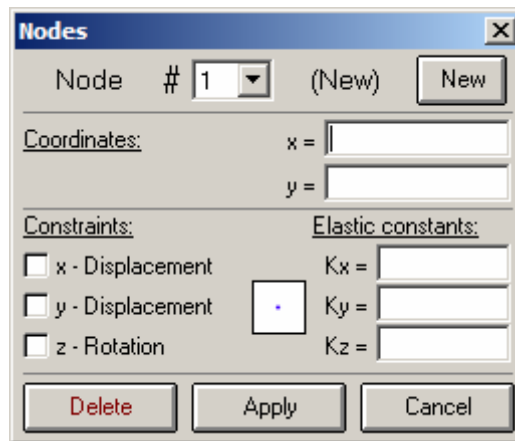





Figure 10. Nodes window

Each node is located at a (x, y) point (Global coordinate system) in the xy plane, and has 3 Degrees of Freedom (DOFs):

-  x – displacement (1)
-  y – displacement (2)
-  z – rotation (3)

Each DOF can be either (a) free to move, or (b) fully fixed (displacement = 0), or (c) elastic (spring), given the stiffness K and following the rule $F(\text{reaction}) = K \cdot u(\text{displacement})$.

Based on the constraints, there are totally 8 types of nodes, as shown in the figure below.

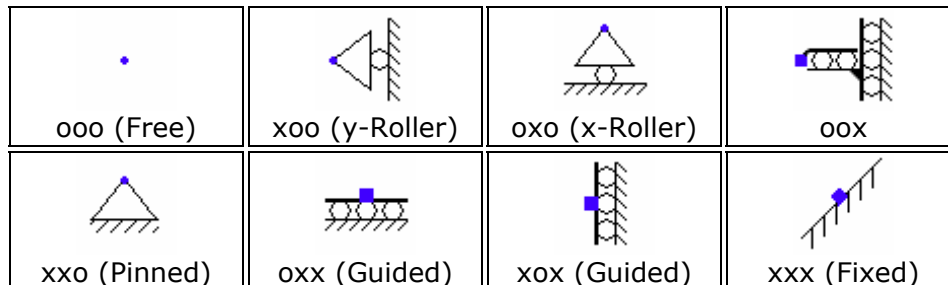


Figure 11. Types of nodes

Where **x** denotes a **Constrained DOF** (translation equals to zero) and **o** denotes a **free DOF**.

Each free DOF can have an **elastic constant** (K_x , K_y or K_z), which makes it a **spring** (elastic) DOF, with force (spring reaction) proportional to deflection at that location. If K is omitted the DOF is assumed to be totally free to move. A DOF that has an elastic constant cannot be totally constrained (x).

Note: If you DELETE a node, all elements that are connected to that node will be deleted also. The program will ask for confirmation if there are elements to be deleted also.

3.5 Elements

Button: 

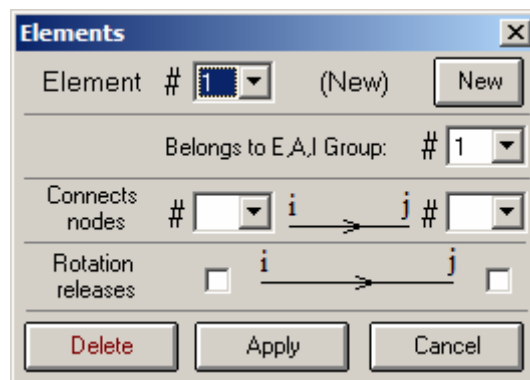


Figure 12. Elements window

Each element belongs to an E,A,I element group and connects two nodes with each other. It can have rotation releases at each one of its ends (start i or/and end j), thus there are totally 4 types of elements, as shown in the figure below.

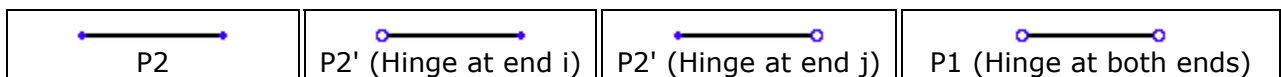


Figure 13. Types of elements

3.6 Nodal loads

Button: 

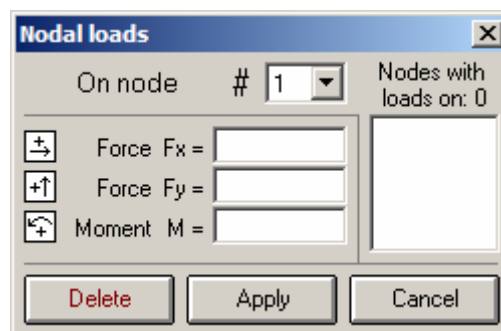





Figure 14. Nodal loads window

At each node there can be 3 types of loads (Global axes):

-  Force Fx
-  Force Fy
-  Moment M

Note: Point loads can only be Nodal loads (acting on nodes). To define a load (Fx, Fy and/or M) at a specific point, first a node must have been defined at the specified location.

3.7 Elemental loads

Button: 

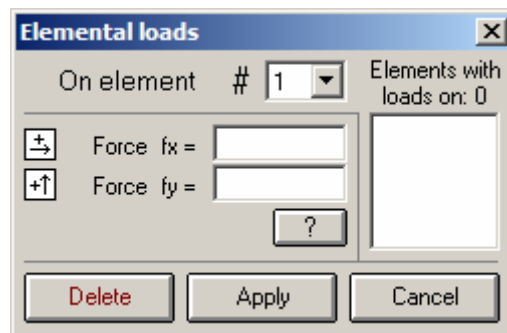




Figure 15. Elemental loads window

At each element there can be 2 types of loads (Global axes):

-  Uniform load fx
-  Uniform load fy

Note: The loads are applied along the element and must be given as Force per Length of the element.

Chapter 4

4. Program tools

4.1 Move nodes tool

Program Menu > Tools > Move nodes

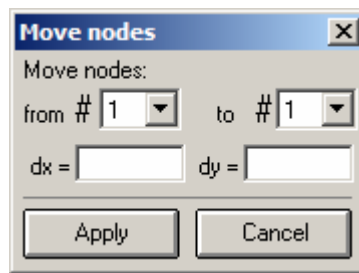


Figure 16. Move nodes tool window

Using this tool you can move nodes, one or more at a time, giving the Vector $\{V\}=\{dx, dy\}$. You can specify the start (*from*) and finish (*to*) node to which this action will apply. For moving a single node, these values have to be the same (for example, *from #3 to #3* for node #3)

4.2 Merge nodes tool

Program Menu > Tools > Merge nodes

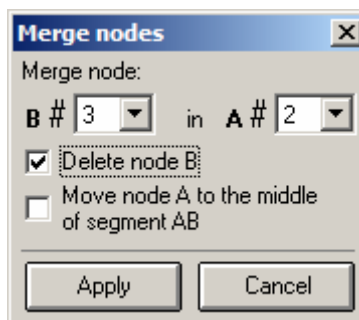
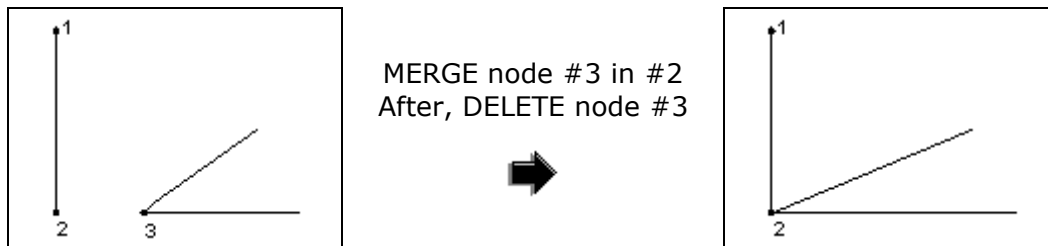


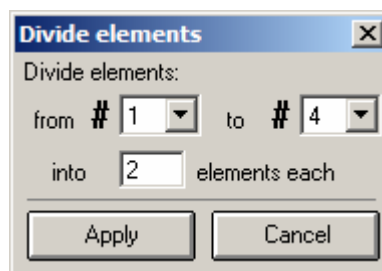
Figure 17. Merge nodes tool window

Using this tool you can merge a node (B) into another (A).

- You can choose whether node B will be deleted after the action is performed or not.
- You can choose whether node A will be moved to the middle of segment AB after the action is performed or not.

Example:**Figure 18.** Merge nodes example**4.3 Divide elements tool**

Program Menu > Tools > Divide elements

**Figure 19.** Divide elements tool window

Using this tool you can divide elements, one or more at a time, specifying the number of elements (segments) into which each element will be divided. You can specify the start (*from*) and finish (*to*) element to which this action will apply. For dividing a single element, these values have to be the same (for example, *from #3 to #3* for element #3)

4.4 Create arc tool

Program Menu > Tools > Create arc

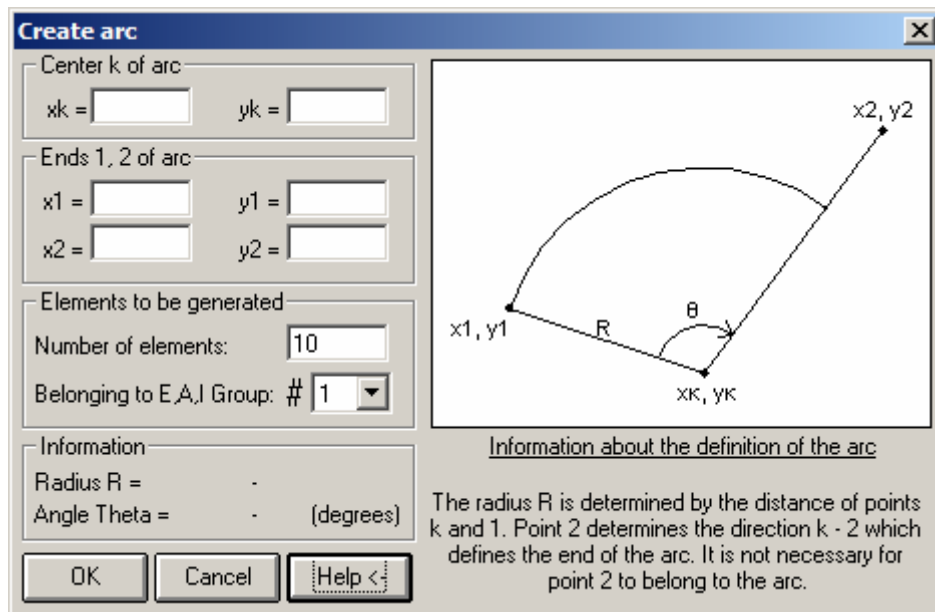


Figure 20. Create arc tool window

Using this tool you can create arcs using a number of linear elements. The number of elements to be generated determines the detail of the arc.

4.5 Check tool

4.5.1 Check for Coincident nodes

Program Menu > Tools > Check > Coincident nodes

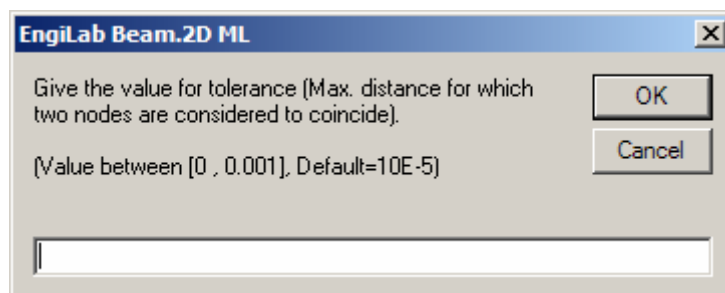


Figure 21. Check for coincident nodes tolerance control window

Due to a mistake during the model set-up process, there may be nodes that coincide (located at exactly the same point x,y) or almost coincide (very small distance between them). These nodes are likely to cause problems to the analysis process and are most times difficult to notice on screen.

This tool is for detecting such nodes. The user must define the **Tolerance**, the maximum distance for which two nodes are considered to coincide. Then the program will calculate the distances between all nodes of the model and warn the user for nodes too close to each other according to the Tolerance criterion. If two nodes are found to coincide, you can use the 'Merge nodes' tool to merge one node into another.

4.5.2 Check for Coincident elements

Program Menu > Tools > Check > Coincident elements

Due to a mistake during the model set-up process, there may be elements that coincide (connecting both the same nodes i, j). These elements are most times difficult to notice on screen.

This tool is for detecting such elements. The program checks the connectivity between all elements of the model and warns the user for elements that coincide.

4.6 Calculator tool

Program Menu > Tools > Calculator

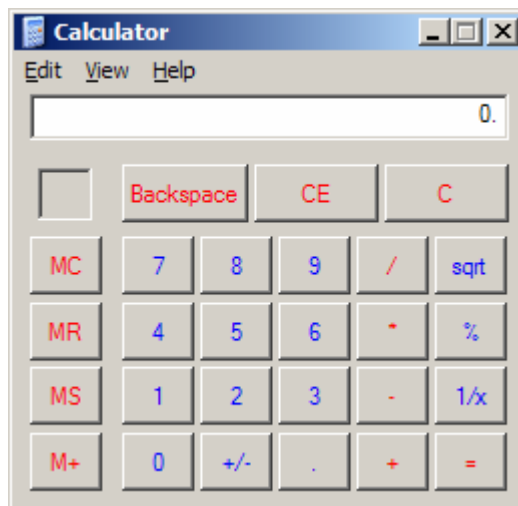


Figure 22. Windows calculator window

This brings up the Windows Calculator for numeric calculations. The calculator window can be opened (Normal or minimized) and used whenever necessary while working with EngiLab Beam.2D ML.

Chapter 5

5. Analysis and post-processing

5.1 Analysis

Button: 

The analysis is performed using the **Finite Element Method (FEM)** for plane frames. During the analysis process, the program forms the model Global Stiffness Matrix $[K]$ as well as the vector of the External Forces $\{F\}$. Then, the linear equations system $\{F\}=[K]\{D\}$ is solved in order for the Displacement Vector $\{D\}$ to be calculated: $\{D\}=[K]^{-1}\{F\}$.

If the model is statically well-defined then the Stiffness Matrix $[K]$ is reversible (Matrix Determinant greater than zero) and no problems will occur during the analysis process.

5.1.1 Errors during the analysis process

Computational errors may occur during the analysis process. In that case, the program warns the user with the following messages:

'An error occurred during the analysis process. Make sure that the structure is well defined (not a mechanism) and can be analyzed.'

This error occurs due to the model stiffness matrix being non-reversible. There is a zero-element appearing at the matrix diagonal, the determinant of the model stiffness matrix equals to zero, the matrix is non-reversible thus the linear equations system cannot be solved for displacements. Most likely the model is a mechanism.

'The analysis has been carried out successfully, yet some of the resultant displacements have excessive values. The structure is probably a mechanism.'




This case does not have to do with a computational error occurring during the analysis process. The analysis has been carried out successfully yet some of the resultant displacements have excessive values (greater than 10^7) which means that probably the structure is a mechanism. In this case the model stiffness matrix is in fact non-reversible but during the analysis process instead of the right zero value, a very small non-zero value appears at the matrix diagonal due to computational approximation errors. As a result, the solver finds a solution (with no practical interest) to the linear equations system yet the resultant displacements have excessive values.

Possible reasons for such warning messages:

- There are nodes that are not connected to each other with elements
- The constraint set is inadequate
- Some parts of the structure are not connected to each other and as a result the stress cannot be transferred from one part of the structure to another
- The stress cannot be transferred from one part of a structure to another due to the type of the connecting elements (P2', P1 cases)
- Due to some other reason the structure is a mechanism

5.2 M, Q, N Diagrams

After setting up the model and analyzing it (Clicking the 'Analyze' button ) the user can draw the M, Q, N diagrams.

- Click  to: **Draw the M diagram** (Bending moment diagram)
- Click  to: **Draw the Q diagram** (Shear force diagram)
- Click  to: **Draw the N diagram** (Axial force diagram)

Note: Values for M, Q, N are given only at nodal points. This means that if you want to have values for M, Q, N within an element you have to use the 'Divide elements' tool.

5.3 Deformation

After setting up the model and analyzing it the user can draw the deformed state of the model.

- Click  to: **Draw the deformed model**

EngiLab Beam.2D ML uses a finite element approach for analyzing the structure. According to the Finite Element Method, results are computed for nodes. Loads within elements (elemental loads) are distributed to the connecting nodes and the analysis results give the **Node displacements** (Vector {D}).

In fact, beam elements for a frame structure are normally bent, which means that they are curved. At the deformed state of the model, elements are not linear, as they were before. In order to find the intermediate displacements within elements and draw the deformed model with accuracy, EngiLab Beam.2D ML uses beam element shape functions. The model's deformed shape is drawn based on node displacements and beam elements shape functions.

Notes:

- EngiLab Beam.2D ML v1.20 calculates and draws the intermediate displacements of any element more accurately than previous versions of the program that used approximations in the case of elements with uniform loads on.
- P1 elements (with hinges at both ends) with no elemental loads on are not bent (not curved) as they are stressed only axially.

- Deformation values are not given on screen, for the deformed shape to be clearer. To see the analytical results, click the 'Data - Results' button.
- The MAX and MIN values for deformation shown on screen MAY NOT BE ACCURATE, as they refer to NODAL DISPLACEMENTS only. No intermediate displacements within elements are taken into consideration when calculating the MAX and MIN values for displacements. Use the 'Divide elements' tool to add Nodes in order to have a more accurate result.

5.4 Model data and analytical results

After setting up the model and analyzing it the user can see the analytical results.

- Click  to: **Bring up the window with the data and analytical results**

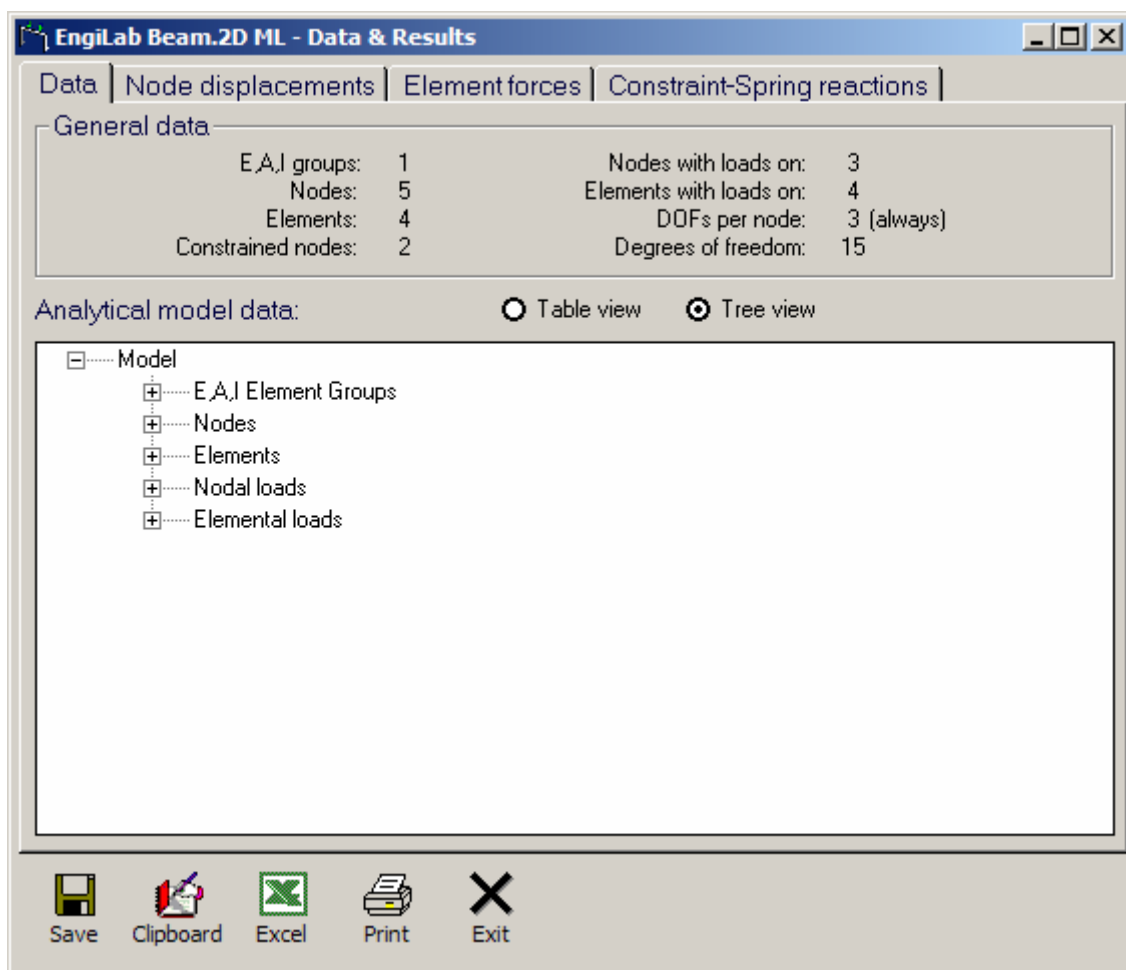


Figure 23. Data and results window

This window shows the model data (first pane) in general and analytical form and the analysis results in numerical form. There are two views available for the analytical model data: Table view and Tree view. The analysis results are given in three panes: Node displacements, Element forces and Constraint - spring reactions.

5.4.1 Node displacements

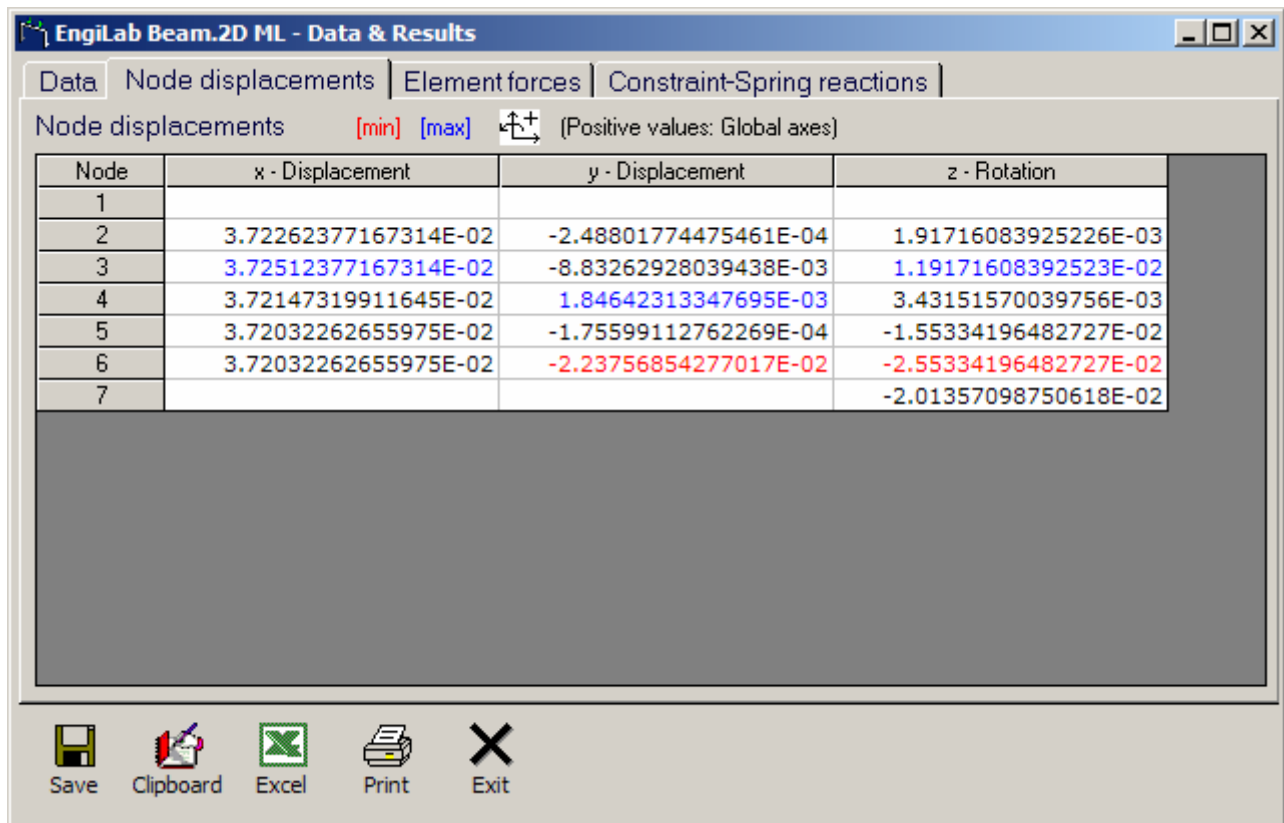



Figure 24. Data and results window, Node displacements pane

Node displacements are given in the Global system of coordinates. The green color refers to elastic (spring) DOFs' displacements, while the red color refers to the minimum and the blue color refers to the maximum displacement.

5.4.2 Element forces

EngiLab Beam.2D ML - Data & Results				
Data Node displacements Element forces Constraint-Spring reactions				
Element forces		Positive values: 		
Element	i, j	Axial force	Shear force	Moment
1	1	24.8801774475461	7.69885488660674	14.9184195634004
	2	-24.8801774475461	-7.69885488660674	15.8769999830265
2	2	10.00000000000018	-20	-20
	3	-10.00000000000018	20	0
3	2	2.30114511339525	4.88017744754611	4.12300001697345
	4	-2.30114511339525	-4.88017744754611	5.63735487811876
4	4	2.30114511339252	-15.1198225524539	-5.63735487811876
	5	-2.30114511339252	15.1198225524539	-24.602290226789
5	5	0	20	20
	6	0	-20	-1.4210854715202E-14
6	5	35.1198225524539	2.30114511339453	4.60229022678906
	7	-35.1198225524539	-2.30114511339453	7.105427357601E-15






 Save
  Clipboard
  Excel
  Print
  Exit

Figure 25. Data and results window, Element forces pane

Element forces are given in the local system of axes for every element at element ends i (start) and j (end).

5.4.3 Constraint – spring reactions

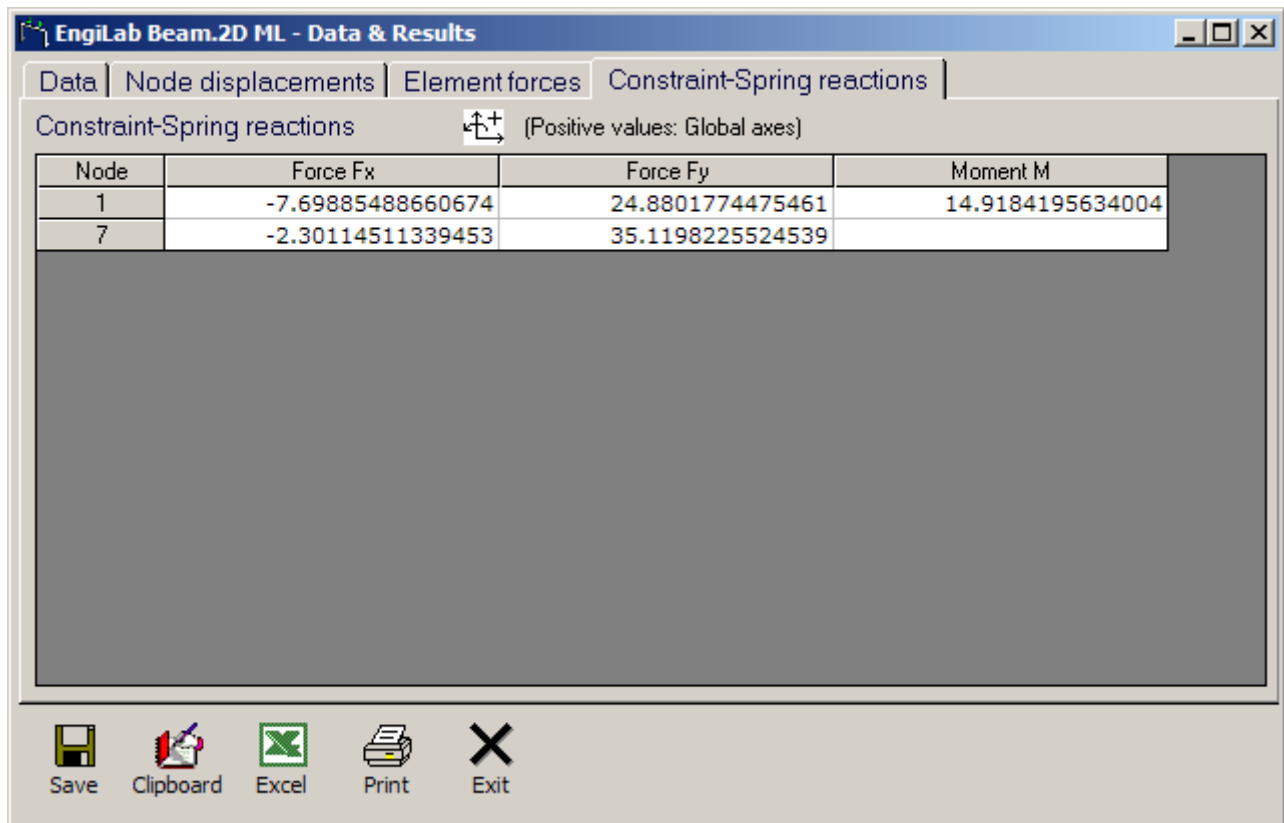






Figure 26. Data and results window, Constraint – spring reactions pane

Constraint and spring reactions are given in the Global system of coordinates. The **green** color refers to **spring** reactions.

5.4.4 Exporting data and the analysis results

The user can export the data and/or the results:







-  To a text file (clicking Save).
-  To the Windows Clipboard (clicking Clipboard), in order to paste them into another application (Such as Microsoft Word or any other editor).
-  To Microsoft Excel (clicking Excel), **Microsoft Excel must have been installed on the user's system.**
-  To a Printer (clicking Print).

Chapter 6

6. Useful information

6.1 Ready-to-analyze example files

In the folder \Examples (which is located where the program was installed), you can find ready-to-analyze EngiLab Beam.2D input files (*.bea) that you can open and analyze within EngiLab Beam.2D ML.

- From the File menu, select **Open**.
- Open the **Examples** folder.
- Select a .bea file to open.
- After the file is opened, click the 'Analyze'  button to analyze the model.
- After the analysis has been carried out:
 - Click , ,  to: **Draw the M, Q, N diagrams** (bending moment, shear force and axial force diagrams).
 - Click  to: **Draw the deformed model**
 - Click  to: **View the analytical results** (node displacements, element forces and constraint reactions).

6.2 Tips on hinges and truss modeling

The hinges of a model is an issue which requires our attention. There are two types of hinges, the internal and the external ones.

6.2.1 External hinges

A. One connecting element

An **external hinge** to which only **one** element is connected can be given in a model using two possible ways (the result of the analysis will be the same):

- (1) As a partially constrained node (1 - xxo) connecting a P2 element with no rotation release at the specific end.
- (2) As a fully constrained node (1 - xxx) connecting a P2' element with a rotation release at specific end.

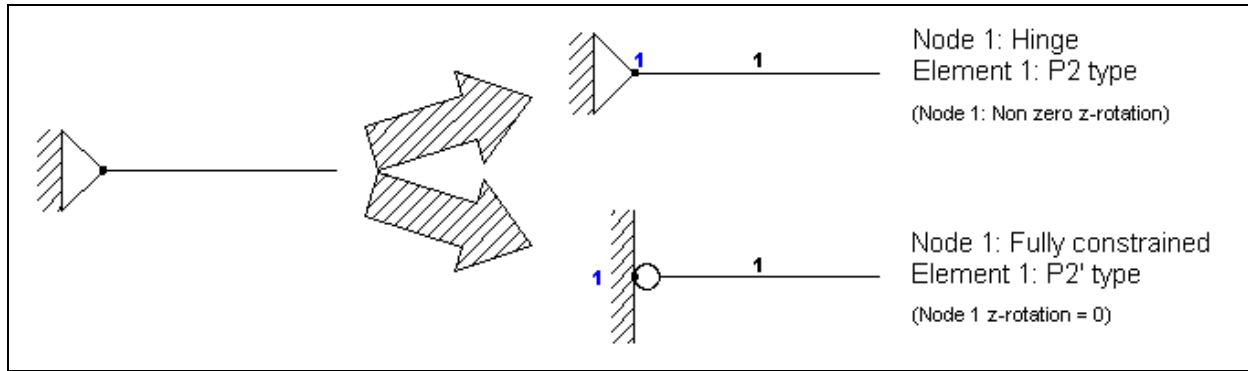


Figure 27. Two ways of modeling external hinges with one connecting element

Note that only by using the first option (1) you can have the rotation of the element at end 1 in the analysis results.

B. More than one connecting elements

An **external hinge** to which more than one element are connected must be given in a model as shown in the figure below.

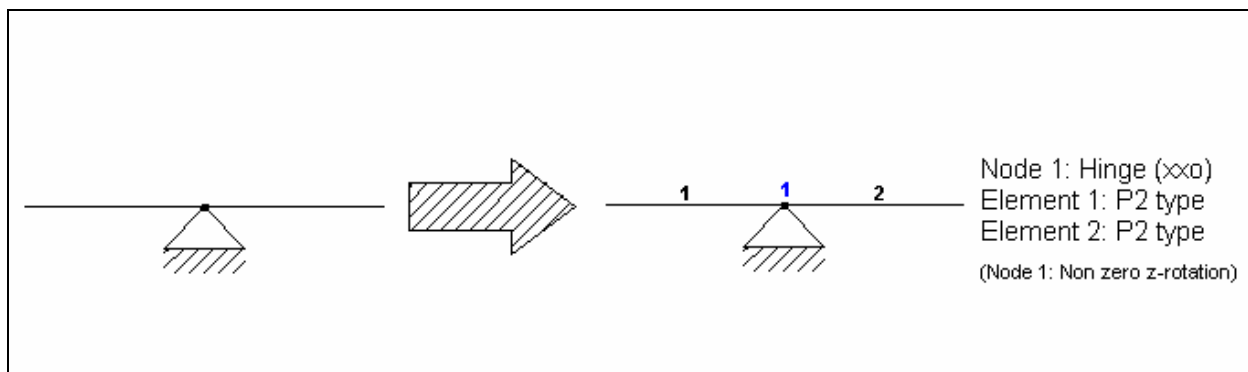
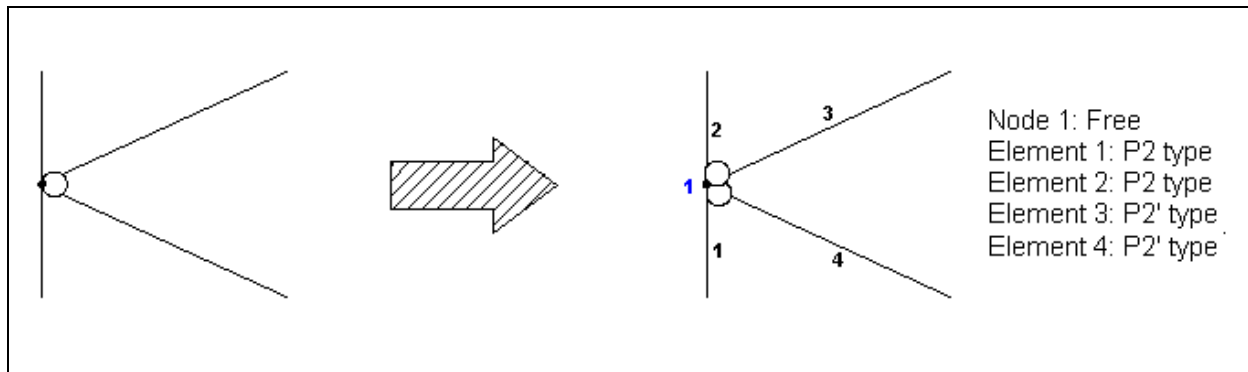


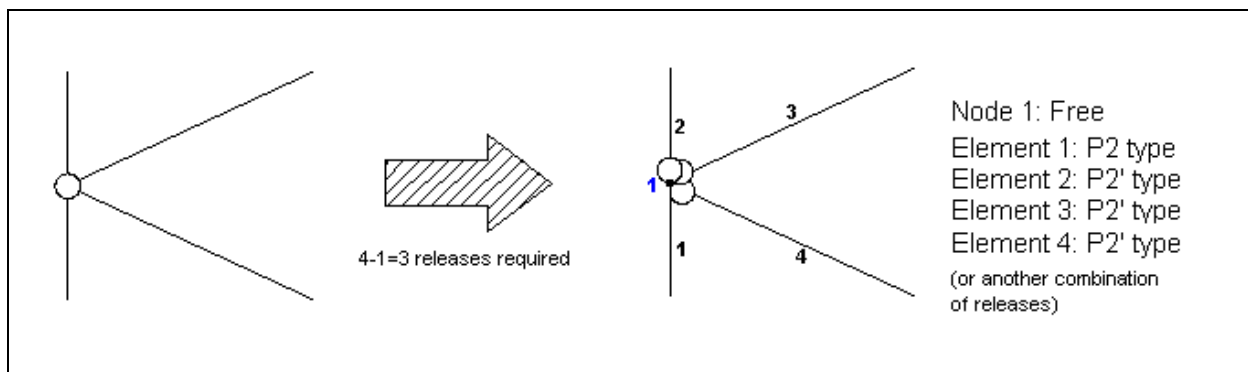
Figure 28. Modeling external hinges with more than one connecting element

6.2.2 Internal hinges

An **internal hinge** must be always given as a rotation release of one or more elements. There can be a 'partial' internal hinge that applies to only some of the connecting elements, or a 'full' internal hinge that applies to all connecting elements.

A. 'Partial' internal hinge**Figure 29.** Modeling 'partial' internal hinges

In the case of a partial internal hinge that applies only to some of the connecting elements, these elements have to be given as P2' elements (with a hinge at the specific end), while the others are given as P2 elements as shown in the figure above.

B. 'Full' internal hinge**Figure 30.** Modeling 'full' internal hinges

A 'full' internal hinge that applies to all connecting elements means that for each of the connecting elements the bending moment value at the specific element end equals to zero. **Such a hinge is used for modelling truss structures.** To model that case, you need as many as ('connecting elements' - 1) releases for the connecting elements. Only one of the connecting elements will then have no rotation release at the specific end (P2 type), no matter which of them, as shown in the figure above.

Important: In the case of a 'Full' internal hinge, each connecting element has its own rotation at the hinge end. Note that the program only calculates the rotation of the P2 connecting element. In the above example, only the rotation of element 1 at end 1 will be calculated, yet the rotation of the elements 2, 3 and 4 at end 1 will not be given in the analysis results. By using different combination of releases, one can get the rotations of any connecting element separately.

6.3 Tips on modeling structures with symmetries

Symmetrical structures with symmetrical / anti-symmetrical loads on can be analyzed modelling half the structure. See the examples that follow.

6.3.1 Symmetrical structure (axis) + Symmetrical loads

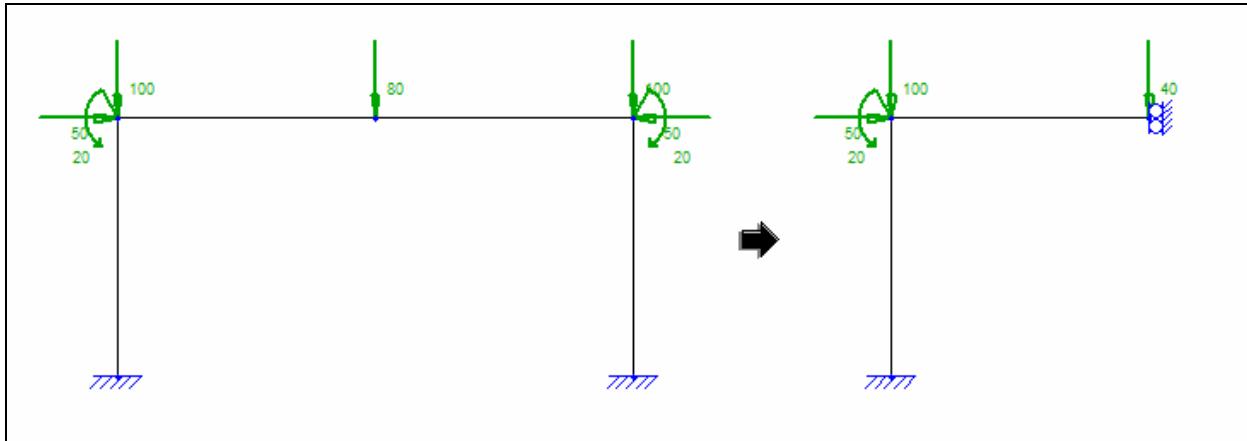


Figure 31. Symmetric structure (axis) + symmetrical loads

For the above example, half the structure can be analyzed using a (xox) constraint for the node on the symmetry axis. Note that if there is a load applied on that node, **half the load has to be applied on the constrained node** of the second model.

6.3.2 Symmetrical structure (axis) + Anti-Symmetrical loads

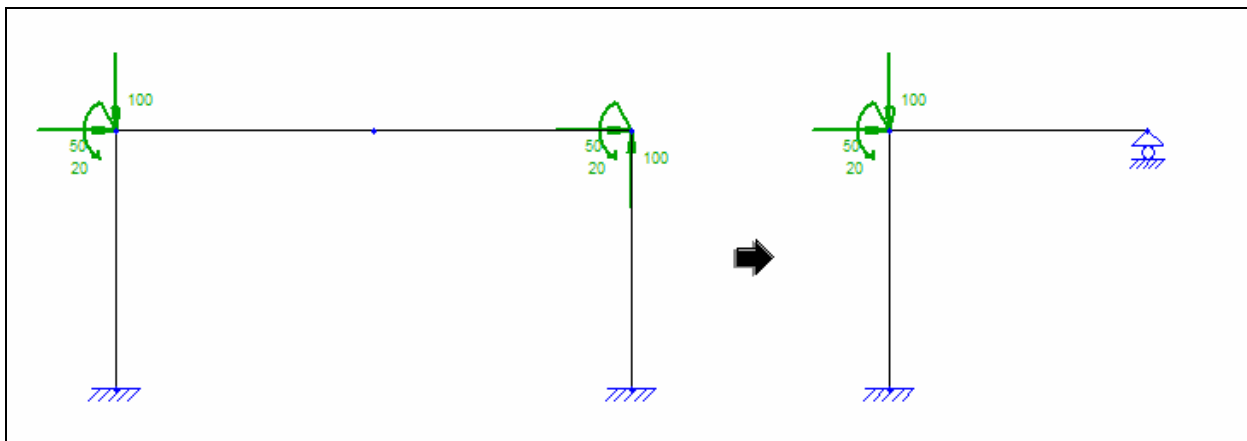


Figure 32. Symmetric structure (axis) + anti-symmetrical loads

For the above example, half the structure can be analyzed using an (oxo - Roller) constraint for the node on the symmetry axis.

6.3.3 Symmetrical structure (point) + Symmetrical loads

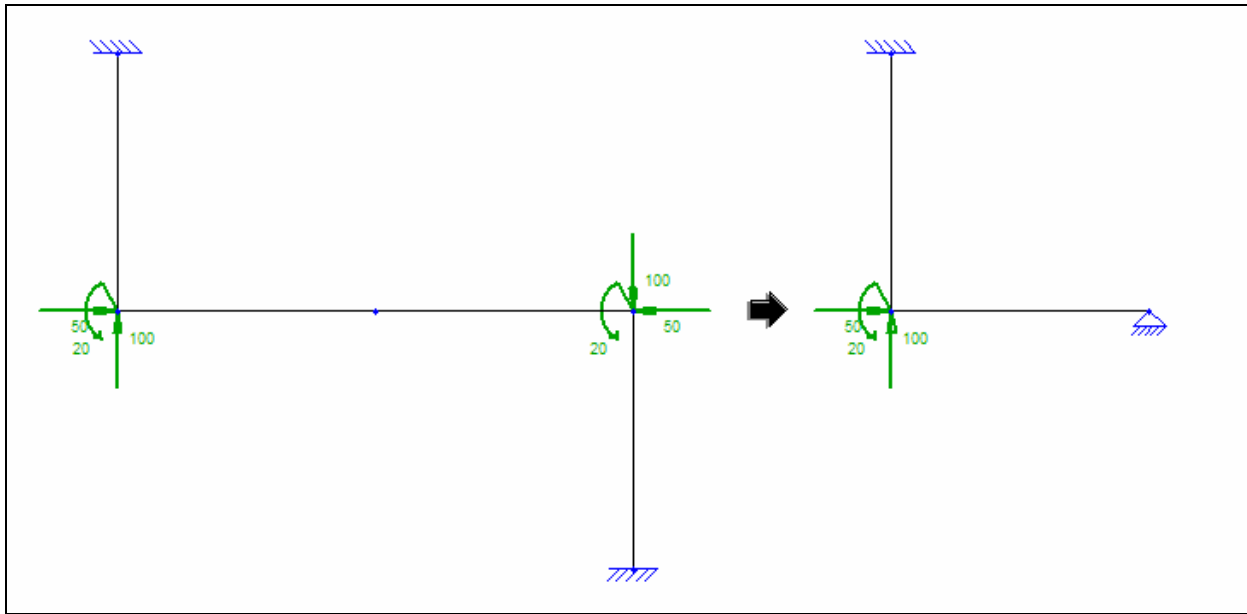


Figure 33. Symmetric structure (point) + symmetrical loads

For the above example, half the structure can be analyzed using a (xxo - Pinned) constraint for the node on the symmetry point.

6.3.4 Symmetrical structure (point) + Anti-Symmetrical loads

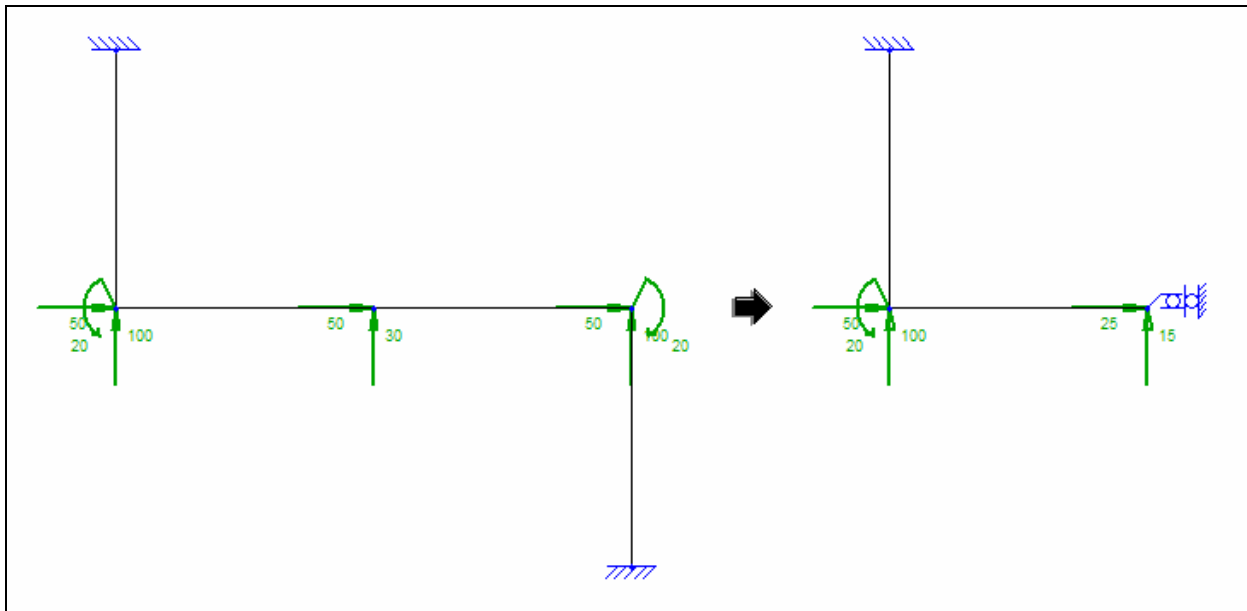


Figure 34. Symmetric structure (point) + anti-symmetrical loads

For the above example, half the structure can be analyzed using an (oox) constraint for the node on the symmetry point. Note that if there is a load applied on that node, **half the load has to be applied on the constrained node** of the second model.