



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

TUTORIAL

v1.00 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. It may contain bugs, so the use of this program is at the user's risk. To the best of our knowledge this 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	5
Tables index	6
1. Introduction	7
2. Starting the program	7
2.1 Choosing the system of units	9
3. Specifying the E,A,I Element Groups	10
4. Drawing nodes and elements on screen	11
5. Modifying Element #2 properties	15
6. Adding the constraints	18
6.1 Add a pinned constraint with a rotational spring for node #1	18
6.2 Add a fixed constraint for node #4	20
6.3 Add a roller constraint with an axial spring for node #6	22
7. Adding loads	23
7.1 Adding the nodal loads	23
7.2 Adding the elemental load	26
8. Creating the arc	28
9. Checking for coincident nodes	31
10. Merging the coincident nodes	34
11. Analysis	37
12. Post-processing	37
12.1 Draw the deformed model	37
12.2 Drawing the bending moment diagram	40
12.3 Drawing the shear force diagram	41
12.4 Drawing the axial force diagram	42
12.5 Back to normal model view	43
13. Displaying the analytical results	44
13.1 Node displacements	45
13.2 Element forces	46
13.3 Constraint - Spring reactions	47
13.4 Exporting data and the analysis results	47

Figures index

Figure 1. The model to be analyzed.....	7
Figure 2. Select the program from the Windows programs list.....	8
Figure 3. EngiLab Beam.2D ML start-up screen	8
Figure 4. EngiLab Beam.2D ML program window	9
Figure 5. E,A,I Element Groups window.....	10
Figure 6. Defining the first element on screen.....	12
Figure 7. Nodes #1, #2 and Element #1 have been defined	13
Figure 8. Defining the second element.....	14
Figure 9. Elements #1 to #5 have been defined.....	15
Figure 10. Selecting Element #2 from the pop-up menu	16
Figure 11. Element #2 properties	16
Figure 12. Modifying element #2 properties	17
Figure 13. Element #2 properties have been modified.....	17
Figure 14. Selecting Node #1 from the pop-up menu.....	18
Figure 15. Node #1 properties.....	19
Figure 16. Modifying Node #1 properties	19
Figure 17. Node #1 properties have been modified	20
Figure 18. Modifying Node #4 properties	21
Figure 19. Node #4 properties have been modified	21
Figure 20. Modifying Node #6 properties	22
Figure 21. Node #6 properties have been modified	23
Figure 22. Selecting Loads on Node #2 from the pop-up menu.....	24
Figure 23. Loads on Node #2.....	24
Figure 24. Loads on Node #2 have been specified.....	25
Figure 25. A horizontal load on Node #2 has been added to the model	25
Figure 26. Loads on Node #5 have been specified.....	26
Figure 27. Selecting Loads on element #4 from the pop-up menu.....	26
Figure 28. Loads on Element #4	27
Figure 29. Loads on Element #4 have been specified	27
Figure 30. A uniform load has been added on Element #4	28
Figure 31. Selecting 'Create arc' from the Tools menu of the program.....	29
Figure 32. Create arc window.....	29
Figure 33. Create arc expanded window.....	30
Figure 34. Specifying the arc properties.....	30
Figure 35. The arc has been created	31
Figure 36. Selecting 'Check for coincident nodes' from the Tools menu of the program	32
Figure 37. Tolerance control input box.....	32
Figure 38. Tolerance value message box	33
Figure 39. Coincident nodes #2 and #7.....	33
Figure 40. Coincident nodes #3 and #17.....	33
Figure 41. Check for coincident nodes result	33
Figure 42. Selecting 'Merge nodes' from the Tools menu	34
Figure 43. Merge nodes window	34
Figure 44. Specifying the node to be merged.....	35
Figure 45. Node #7 has been merged in Node #2	35
Figure 46. Specifying the second node to be merged.....	36
Figure 47. The second node has been merged.....	36
Figure 48. The model is ready.....	37
Figure 49. Deformed state of the model.....	38
Figure 50. Deformed state of the model (adjusted scale).....	39

Figure 51. Bending moment diagram	40
Figure 52. Shear force diagram	41
Figure 53. Axial force diagram.....	42
Figure 54. Return to the basic model view	43
Figure 55. The Data & Results window	44
Figure 56. Node displacements	45
Figure 57. Element forces	46
Figure 58. Constraint – spring reactions.....	47

Tables index

Table 1. Input data units	10
Table 2. Results units	10

1. Introduction

To introduce you to the concepts and techniques of EngiLab Beam.2D ML, this tutorial will describe a simple frame modeling, analysis and post-processing, step by step. The frame to be analyzed is shown in the figure below.

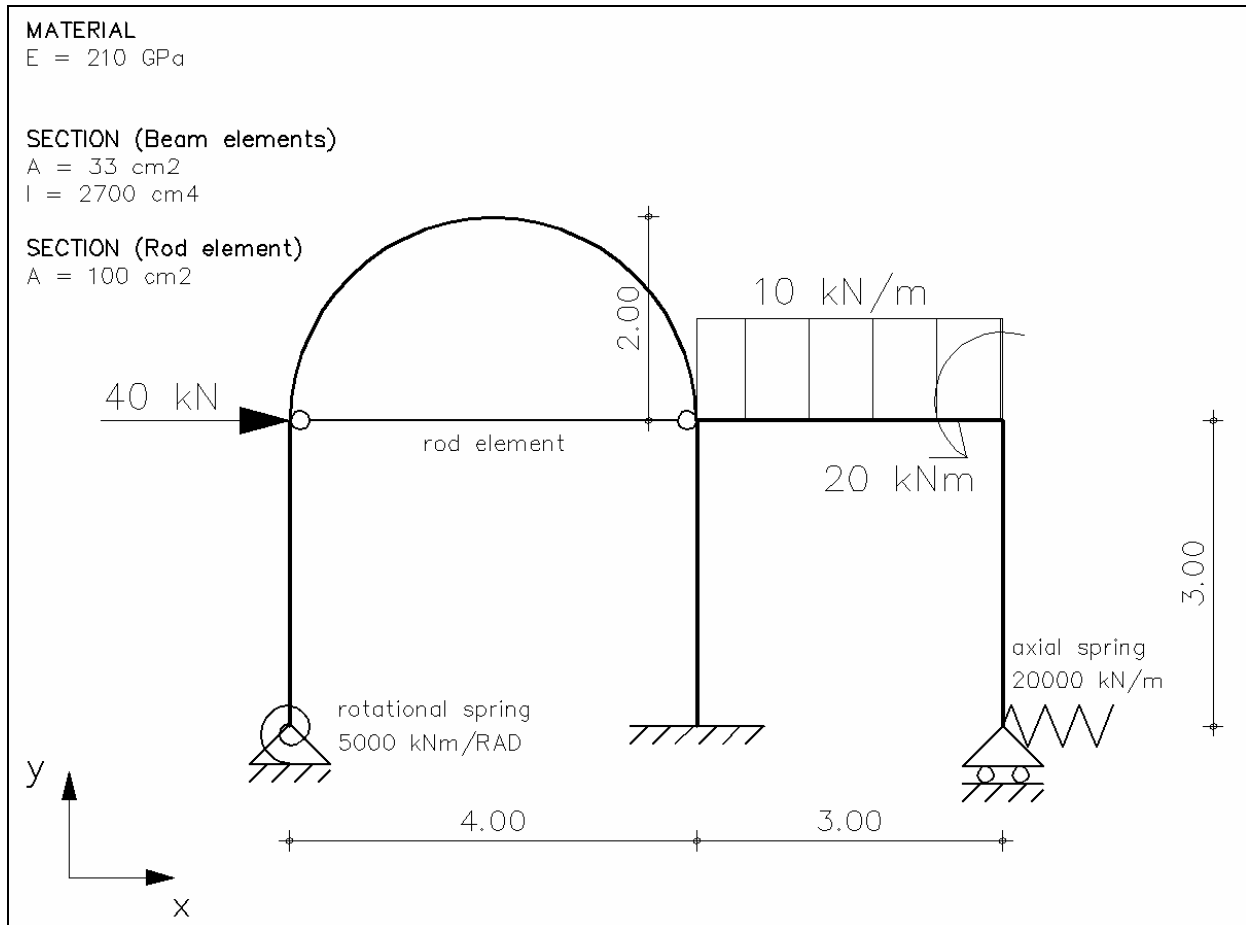


Figure 1. The model to be analyzed

2. Starting the program

Open the Windows Start menu and select 'Programs' > 'EngiLab Beam.2D ML' > 'EngiLab Beam.2D ML', as shown in the figure below.

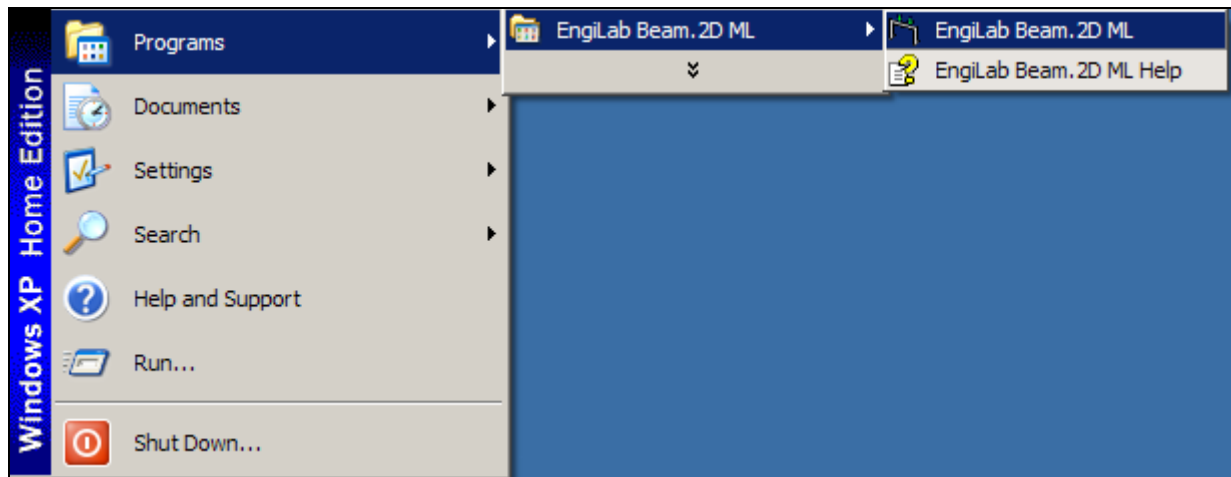


Figure 2. Select the program from the Windows programs list

This will launch the program and the start-up screen will appear.

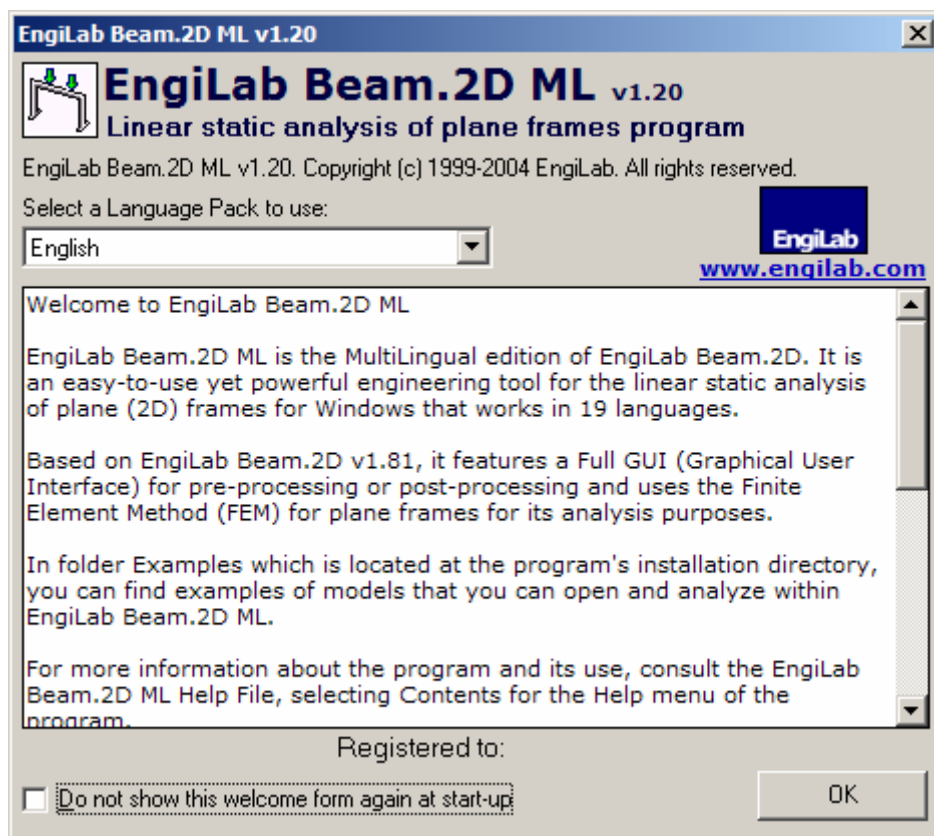


Figure 3. EngiLab Beam.2D ML start-up screen

Press **OK** to start using the program. The basic program window will appear, as shown below.

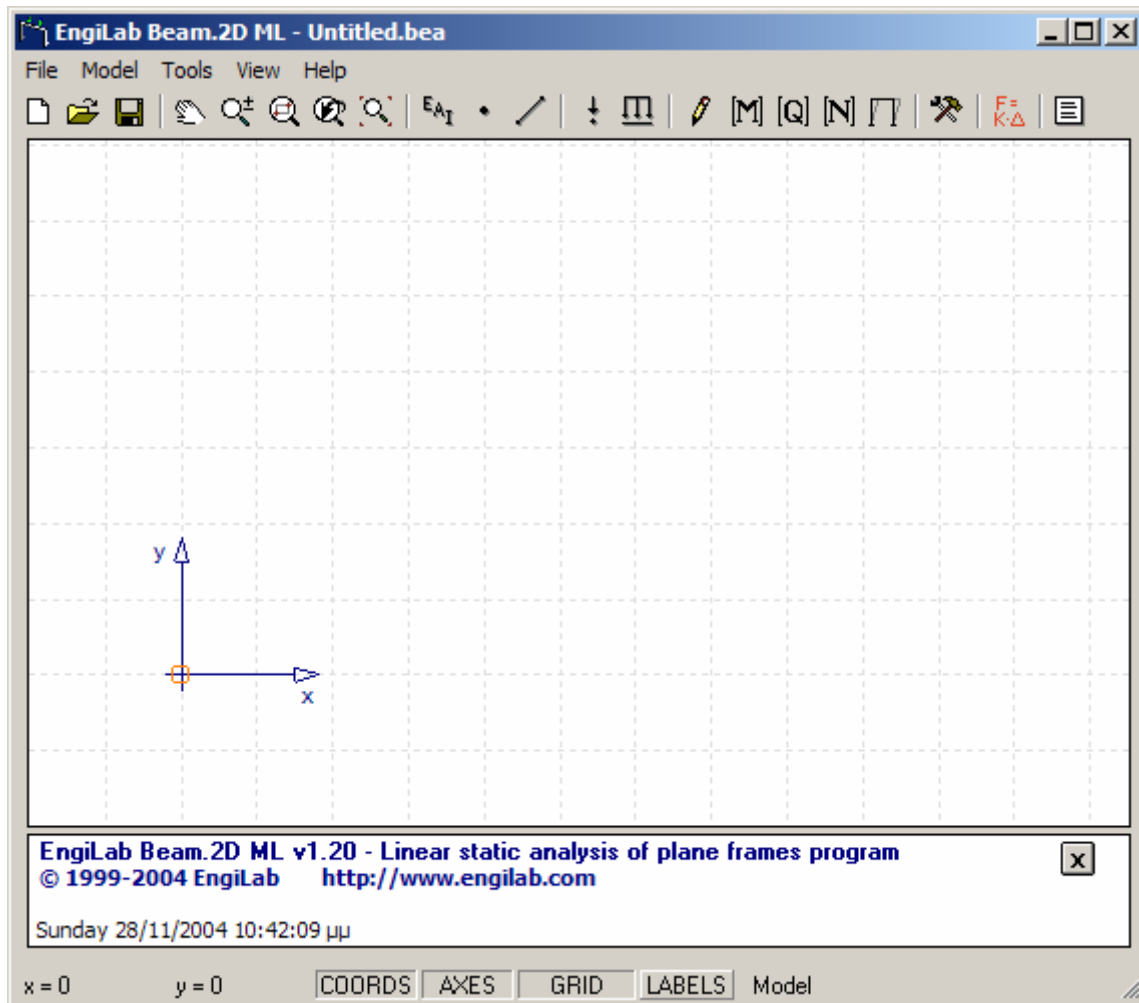


Figure 4. EngiLab Beam.2D ML program window

2.1 Choosing the system of units

The program uses a consistent system of units, which means you have to choose specific measuring units only for the two basic quantities, **DISTANCE** and **FORCE**. All quantities that are derived from them must then follow the measuring units that these two use.

In this example, we will use:

- **m** (meters) for Distance
- **kN** (kilo Newton) for force.

You do not have to tell the program that you are going to use **m** for distances and **kN** for forces. Yet, any quantity that is given to the program from now on has to comply with **m** and **kN**, as shown in the table below.

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

Table 1. Input data units


As a result, the analysis results will also comply with that system, as shown in the table below.

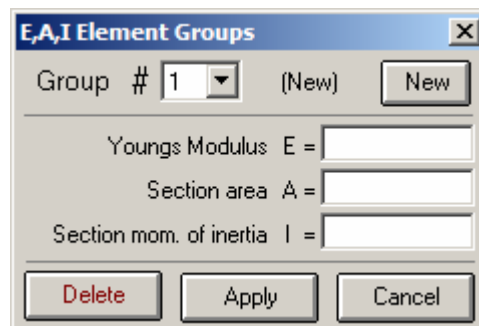
Quantity	Unit used
Node displacement	m
Node z-rotation	RAD
Axial, shear force at element end i, j	kN
Moment at element end i, j	kNm
Constraint reaction x, y (force)	kN
Constraint reaction (moment)	kN*m
Spring reaction x, y (force)	kN
Spring reaction (moment)	kN*m

Table 2. Results units

Note that rotations are always given in **Radians**.

3. Specifying the E,A,I Element Groups

Two E,A,I Element Groups are needed, one for the frame elements and one for the rod element (the one with hinges at both ends). Press the E,A,I Element Groups button  from the program's toolbar at the top. This will bring up the E,A,I Element Groups window, as shown below.

**Figure 5.** E,A,I Element Groups window

Type in the E,A,I properties for the beam elements that will belong to **Group #1**. Be sure you use the right units:

$$E = 210 \text{ GPa} = 210 \cdot 10^9 \text{ Pa} = 210 \cdot 10^9 \text{ N/m}^2 = \mathbf{210,000,000 \text{ kN/m}^2}$$

$$A = 33 \text{ cm}^2 = 33 (10^{-2} \text{ m})^2 = 33 \cdot 10^{-4} \text{ m}^2 = \mathbf{0.0033 \text{ m}^2}$$

$$I = 2700 \text{ cm}^4 = 2700 (10^{-2} \text{ m})^4 = 2700 \cdot 10^{-8} \text{ m}^4 = \mathbf{0.000027 \text{ m}^4}$$

Press **Apply**. Now type in the E,A,I properties for the rod element that will belong to **Group #2**:

$$E = \mathbf{210,000,000 \text{ kN/m}^2}$$

$$A = 100 \text{ cm}^2 = 100 (10^{-2} \text{ m})^2 = 100 \cdot 10^{-4} \text{ m}^2 = \mathbf{0.01 \text{ m}^2}$$

$$I = \mathbf{1}$$

* The moment of inertia (I) property of the rod element isn't taken into account, as the rod element is not bent, as it is stressed only axially, so any numeric value can be used for the moment of inertia. We use the value of **1**.

Press **Apply** and then **Cancel**, as there are no other E,A,I Element Groups to define. Two E,A,I Element Groups have now been defined.

4. Drawing nodes and elements on screen

If you move the mouse pointer on screen you will notice that the **Grid** is activated (as shown in the program's status bar at the bottom of the main window), which means that only nodes located directly on grid points can be defined on screen.

Any point can be used for the definition of the first node. We will use the point located at **{x=3, y=0}** as a starting point. Move the mouse pointer to that location. Press the left mouse button (that creates **Node #1** at **{x=3, y=0}**) and **hold it down**.

While holding the left mouse button, move the mouse pointer **upwards** until you read **@0, 3** (relative coordinates, relatively to start node #1 at **{x=3, y=0}**). That means you have moved 3 distance units along the y axis. The absolute coordinates (in the program's status bar) should now read **{x=3, y=3}**. The picture should be as shown below.

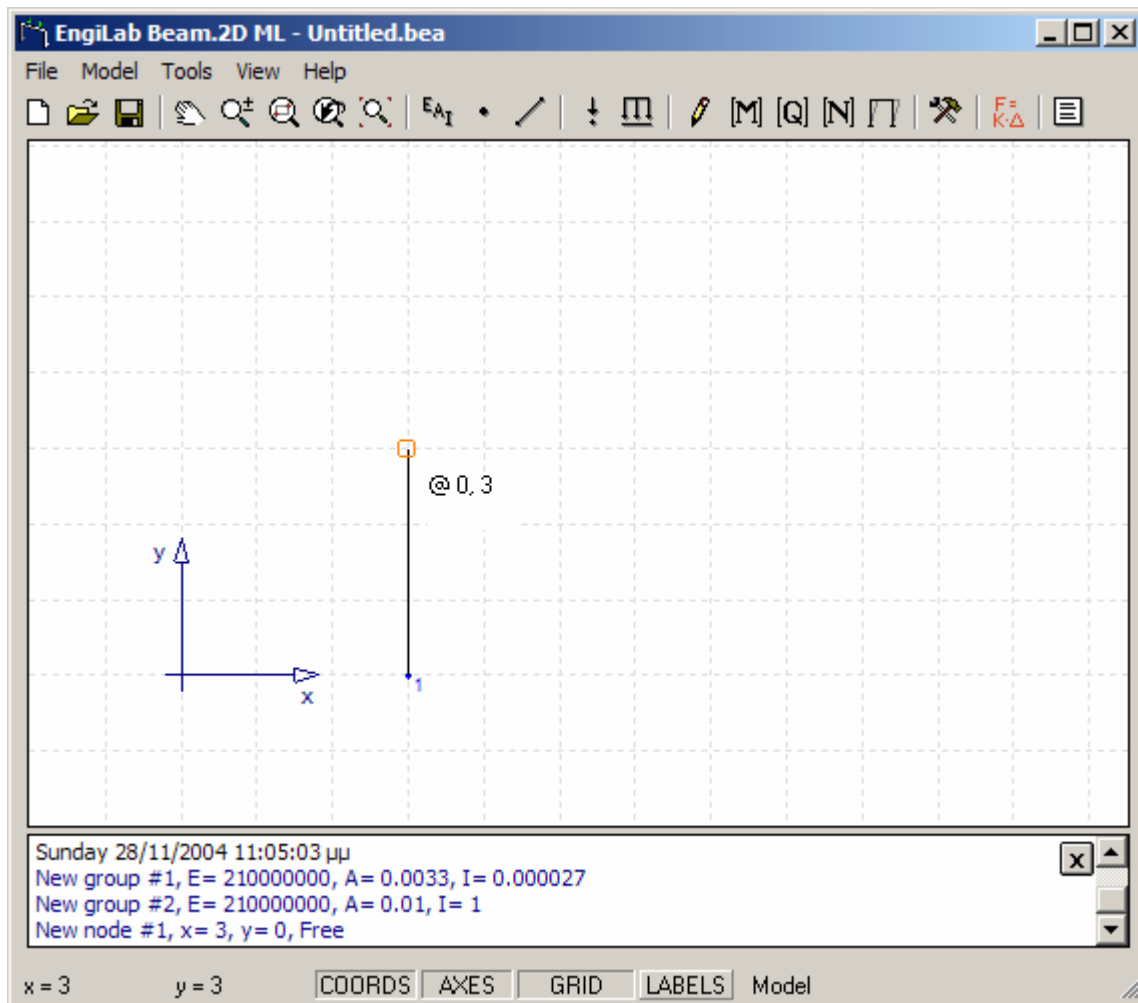


Figure 6. Defining the first element on screen

Now, **release the left mouse button**. This creates **Node #2** at $\{x=3, y=3\}$ and **Element #1** connecting nodes **#1** and **#2** as shown below.

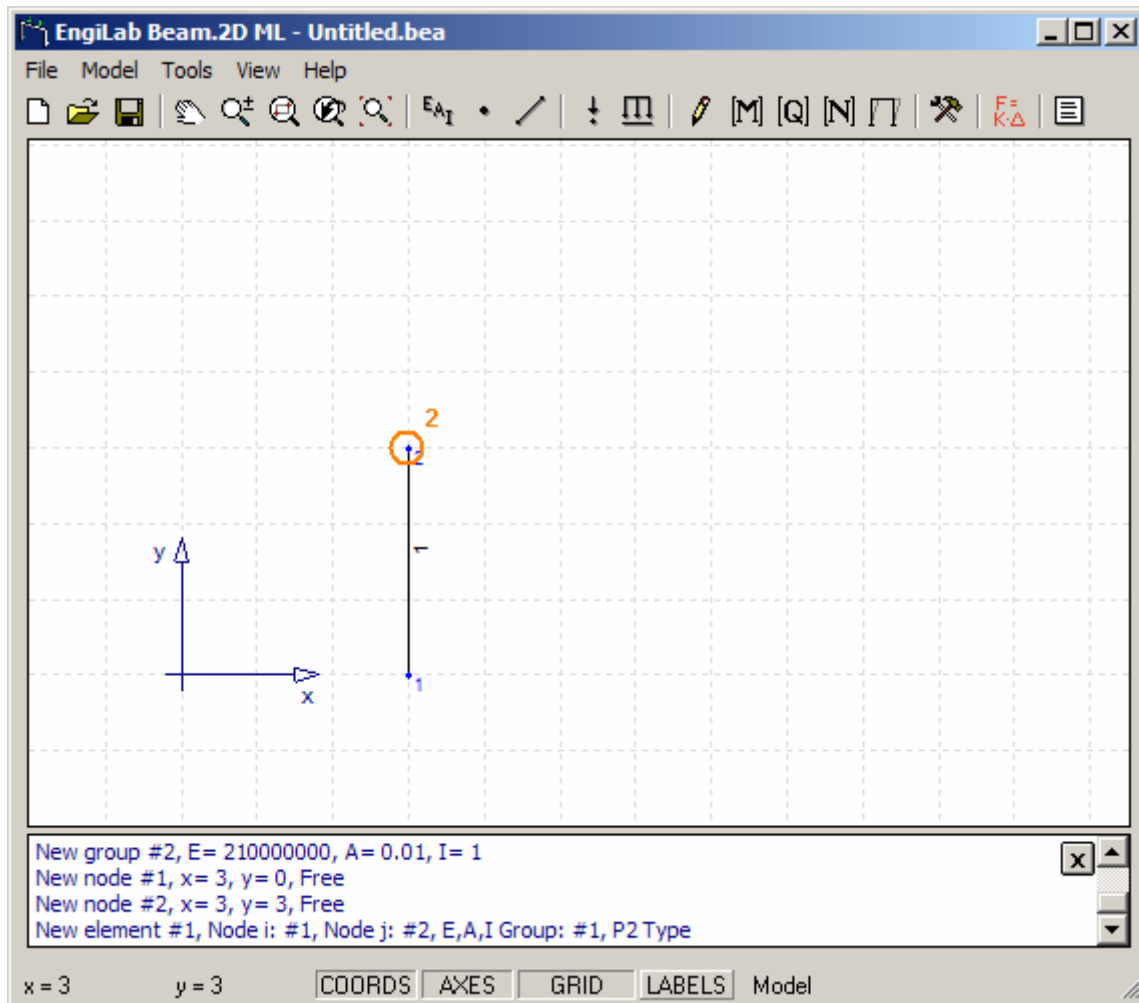


Figure 7. Nodes #1, #2 and Element #1 have been defined

Note that **every element that is created graphically** (on screen), as described above is a **P2 element** (with no rotation releases) that belongs to **E,A,I Element Group #1** (which is considered as the **main E,A,I Element Group**).

Now, for the definition of the second element, **hold the left mouse button at Node #2** and move the pointer to the right until you read **@4, 0**. The picture should be as shown below.

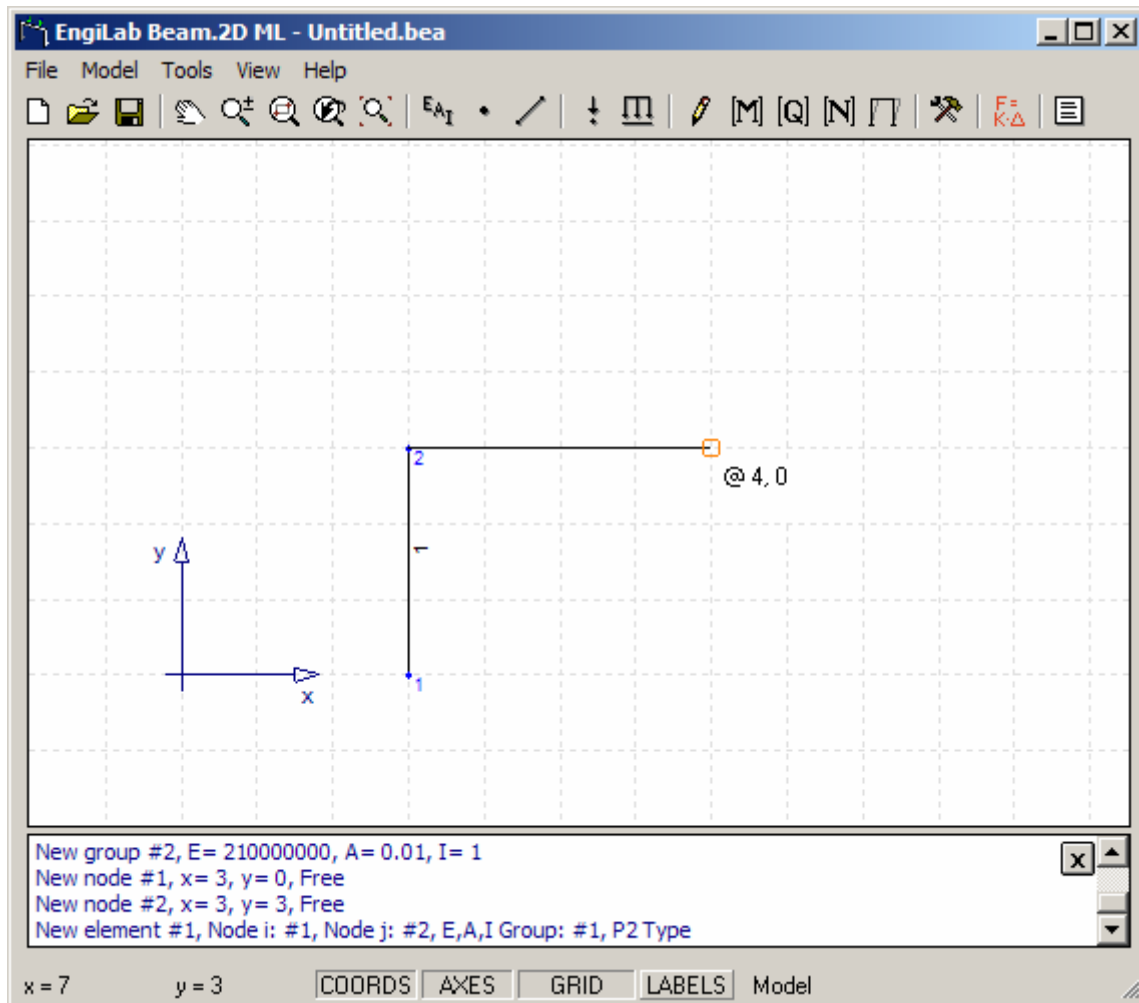


Figure 8. Defining the second element

Now **release the left mouse button**. This creates **Node #3** at $\{x=7, y=3\}$ and **Element #2** connecting nodes **#2** and **#3**.

Again, this new element is a **P2 element** that belongs to the **main E,A,I Element Group (#1)**. In our example **this is not the case**, as this should be a **P1 element** (with hinges at both ends) belonging to **E,A,I Element Group #2**. **We will fix this mistake later on**, after we have defined all the horizontal and vertical beam elements.

Using the same technique:

- Create **Element #3**, connecting nodes **#3** and **#4**.
- Create **Element #4**, connecting nodes **#3** and **#5**.
- Create **Element #5**, connecting nodes **#5** and **#6**.

The picture should be as shown below.

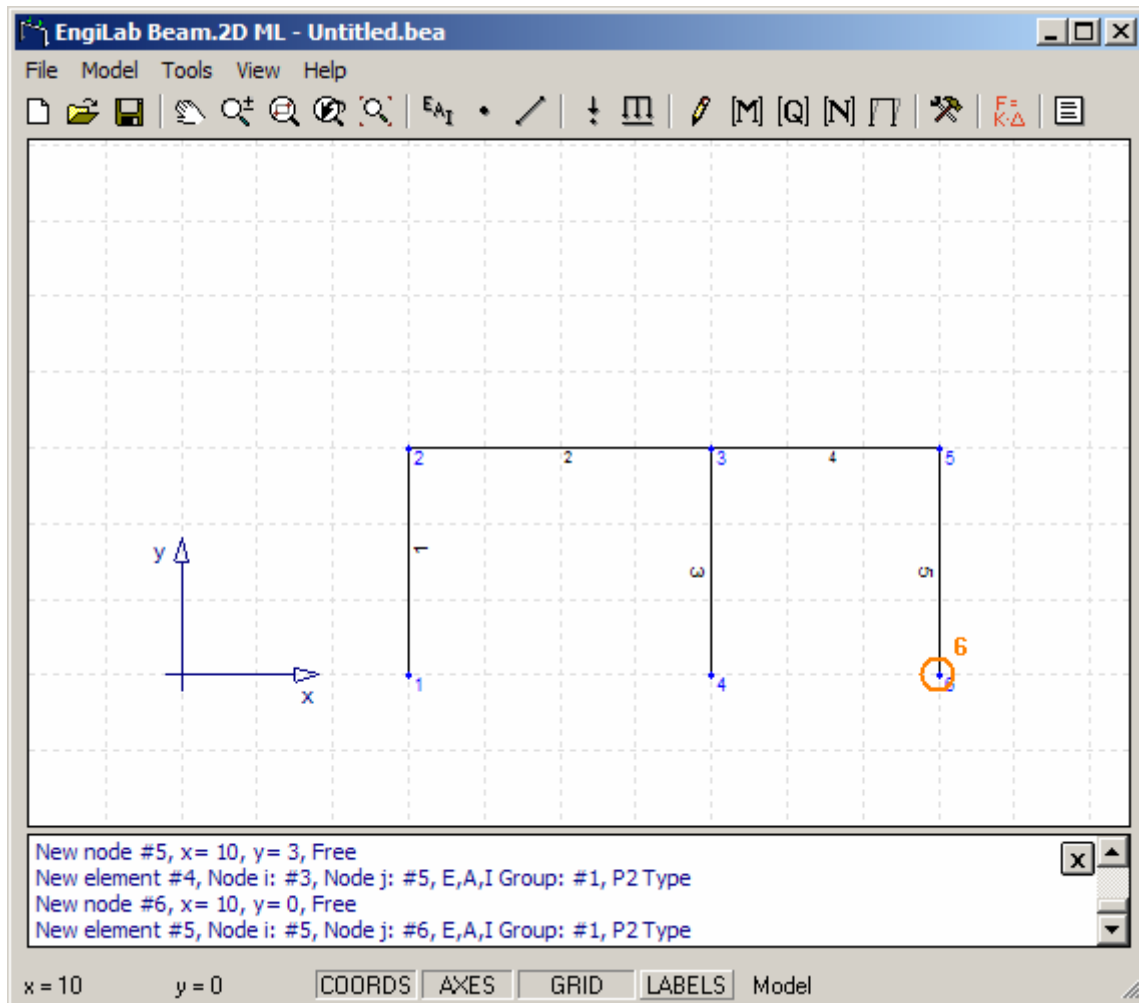


Figure 9. Elements #1 to #5 have been defined

5. Modifying Element #2 properties

Now, its time to fix the mistake: **Element #2** should be changed to **P1 type** (with hinges at both ends) and should belong to **E,A,I Element Group #2**, instead on #1.

Move the pointer **near Element #2** until the element becomes **red** and then press the **right mouse button**. A pop-up menu should appear, as shown below.

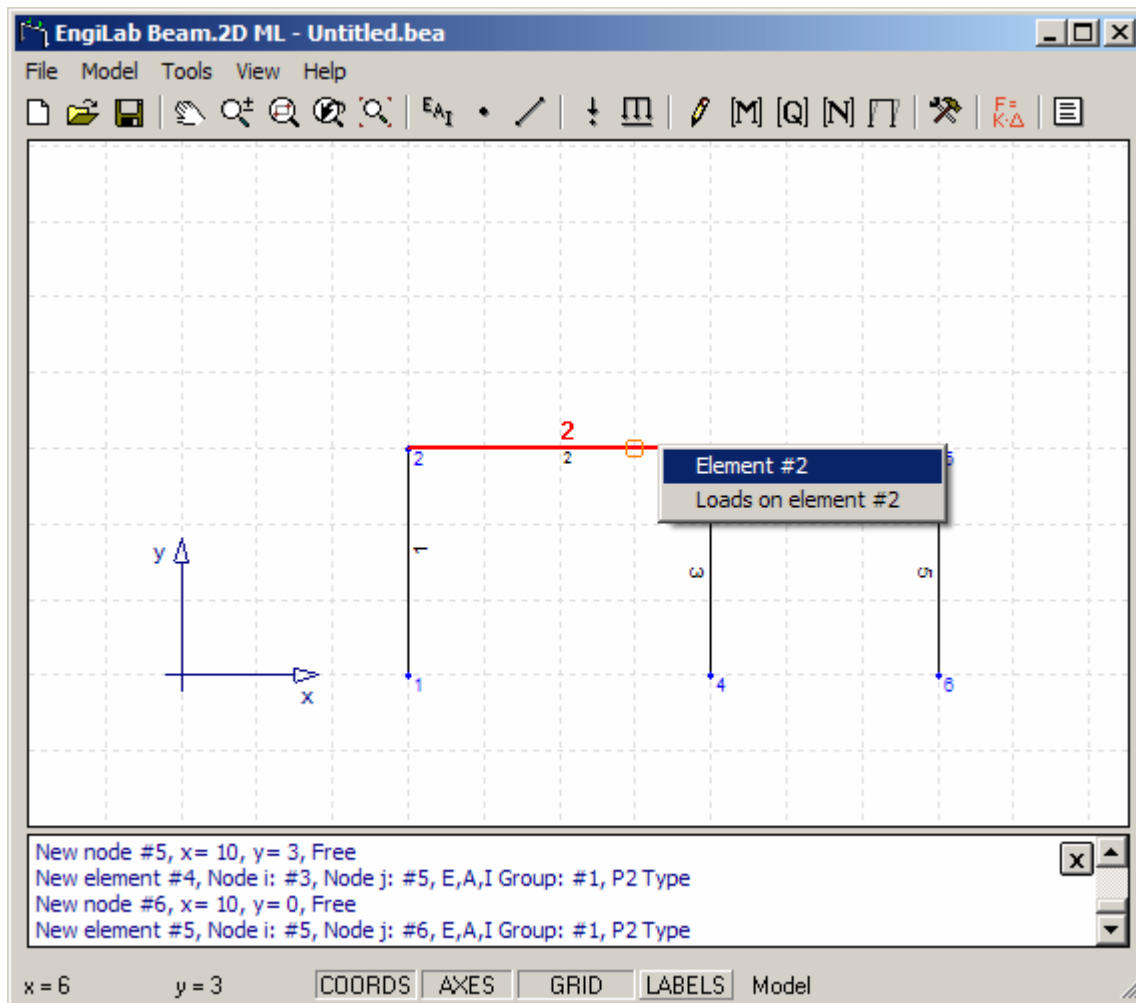


Figure 10. Selecting Element #2 from the pop-up menu

Select **Element #2** and press the **left mouse button**. This brings up the Elements window with **Element #2** selected.

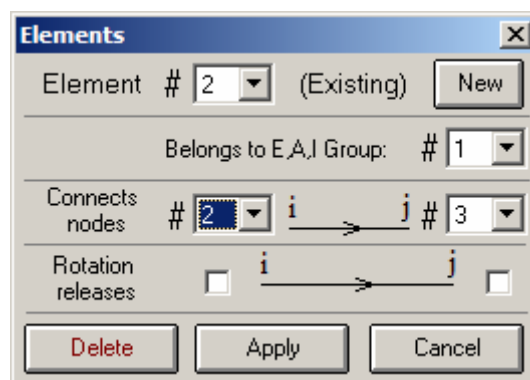


Figure 11. Element #2 properties

As **Element #2** was defined on screen, it belongs to **E,A,I Group #1** and has **no rotation releases**. Make it belong to **E,A,I Group #2** and have **rotation releases at both ends i and j**, as shown below.

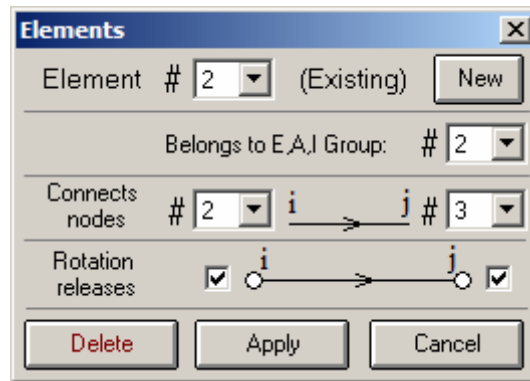


Figure 12. Modifying element #2 properties

Press **Apply** and then **Cancel**. **Element #2** has been modified as shown below.

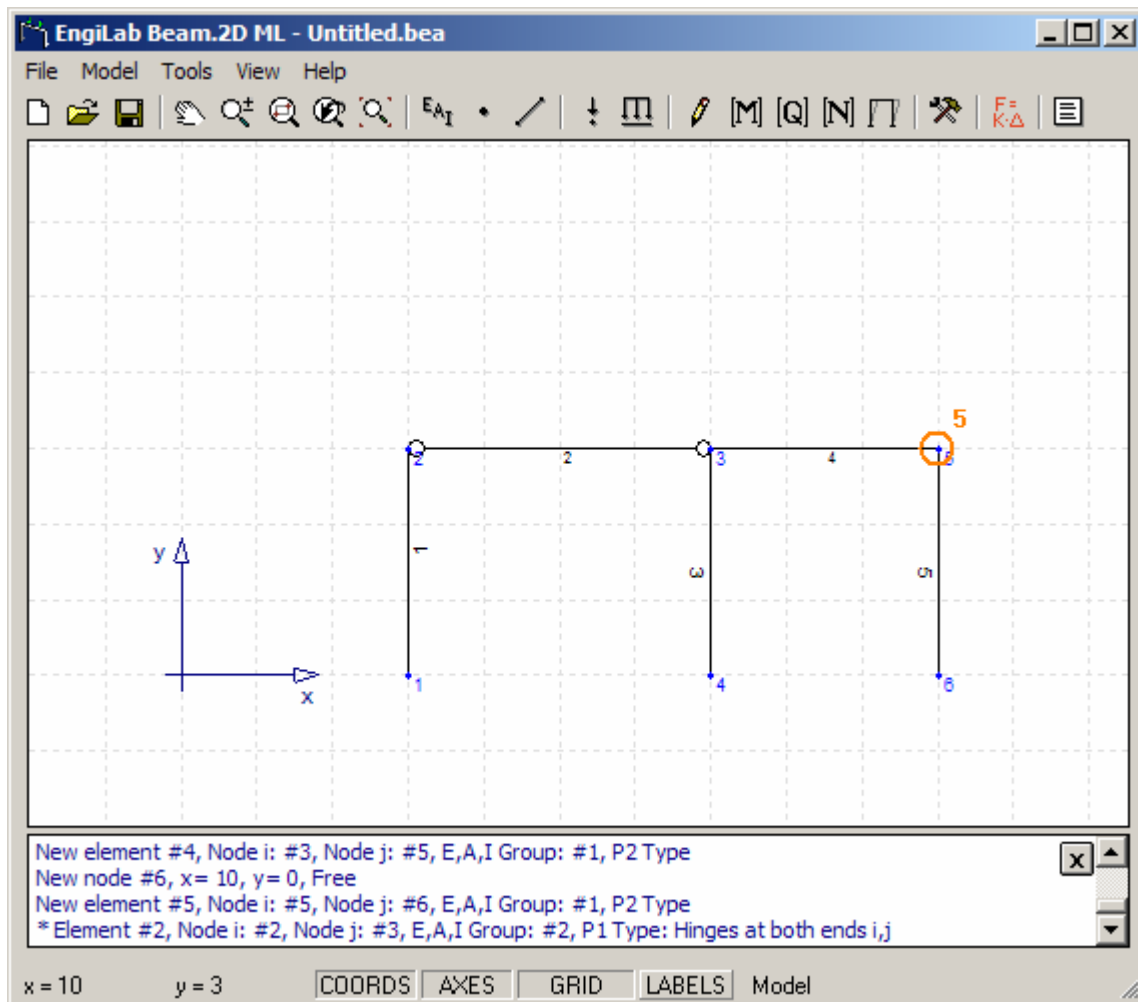


Figure 13. Element #2 properties have been modified

6. Adding the constraints

6.1 Add a pinned constraint with a rotational spring for node #1

Move the pointer near **Node #1** until you see an **orange ring** around it. Press the **right mouse button**. A pop-up menu should appear, as shown below.

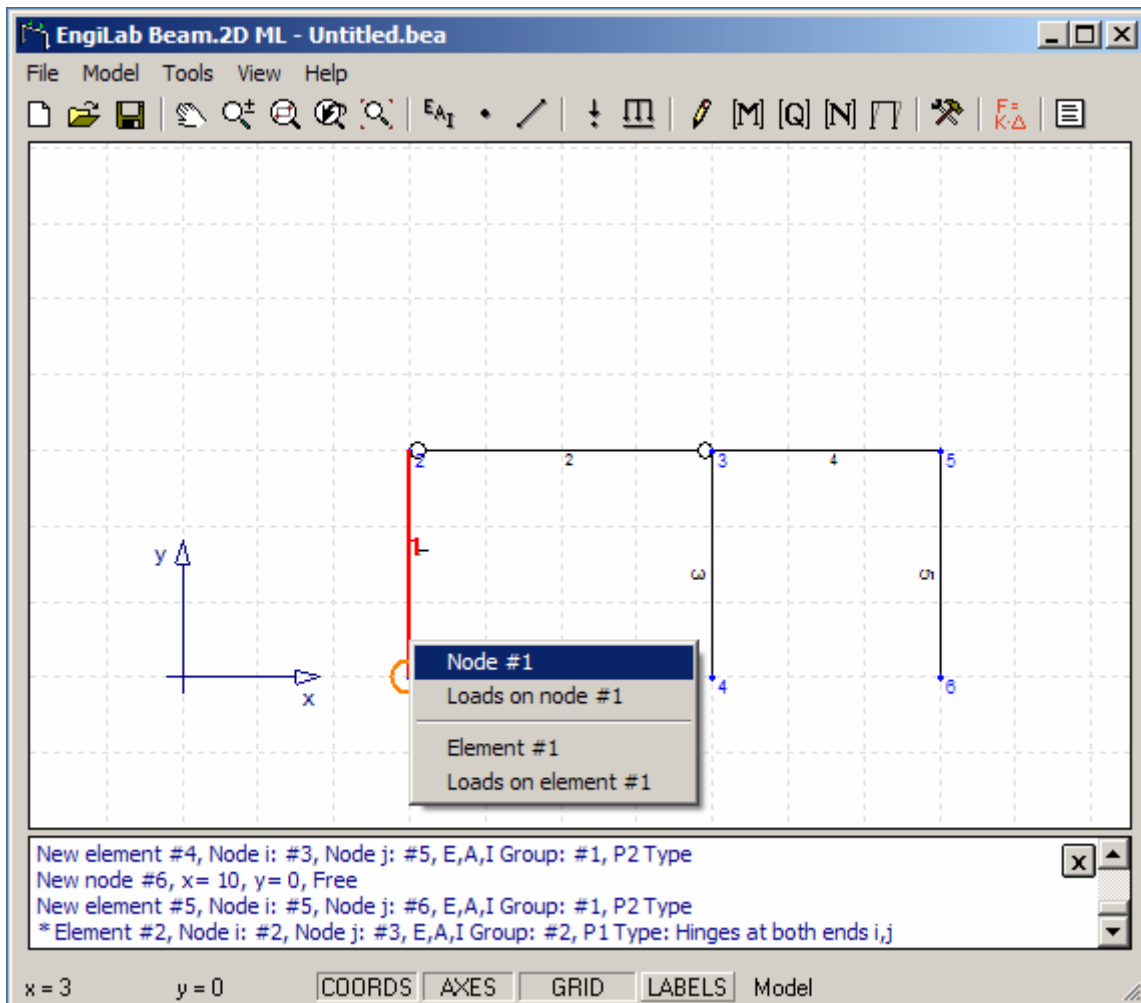
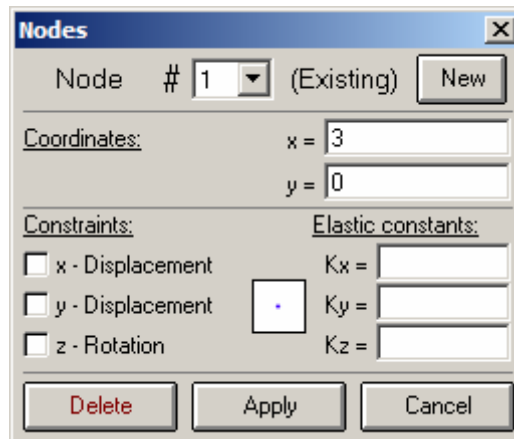


Figure 14. Selecting Node #1 from the pop-up menu

Select **Node #1** and press the **left mouse button**. This brings up the Nodes window with **Node #1** selected, as shown below.

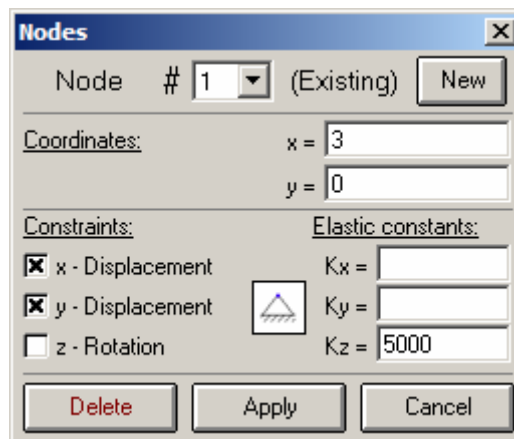


The 'Nodes' dialog box for Node #1 (Existing) shows the following settings:

- Node # 1 (Existing) [New]
- Coordinates: x = 3, y = 0
- Constraints: ☐ x - Displacement, ☐ y - Displacement, ☐ z - Rotation
- Elastic constants: Kx = , Ky = , Kz =
- Buttons: Delete, Apply, Cancel

Figure 15. Node #1 properties

Modify the constraints and add the **stiffness of the rotational spring ($K_z=5000$)**, as shown below.



The 'Nodes' dialog box for Node #1 (Existing) shows the following modified settings:

- Node # 1 (Existing) [New]
- Coordinates: x = 3, y = 0
- Constraints: ☒ x - Displacement, ☒ y - Displacement, ☐ z - Rotation
- Elastic constants: Kx = , Ky = , Kz = 5000
- Buttons: Delete, Apply, Cancel

Figure 16. Modifying Node #1 properties

Press **Apply** and then **Cancel**. **Node #1** has been modified as shown in the figure below.

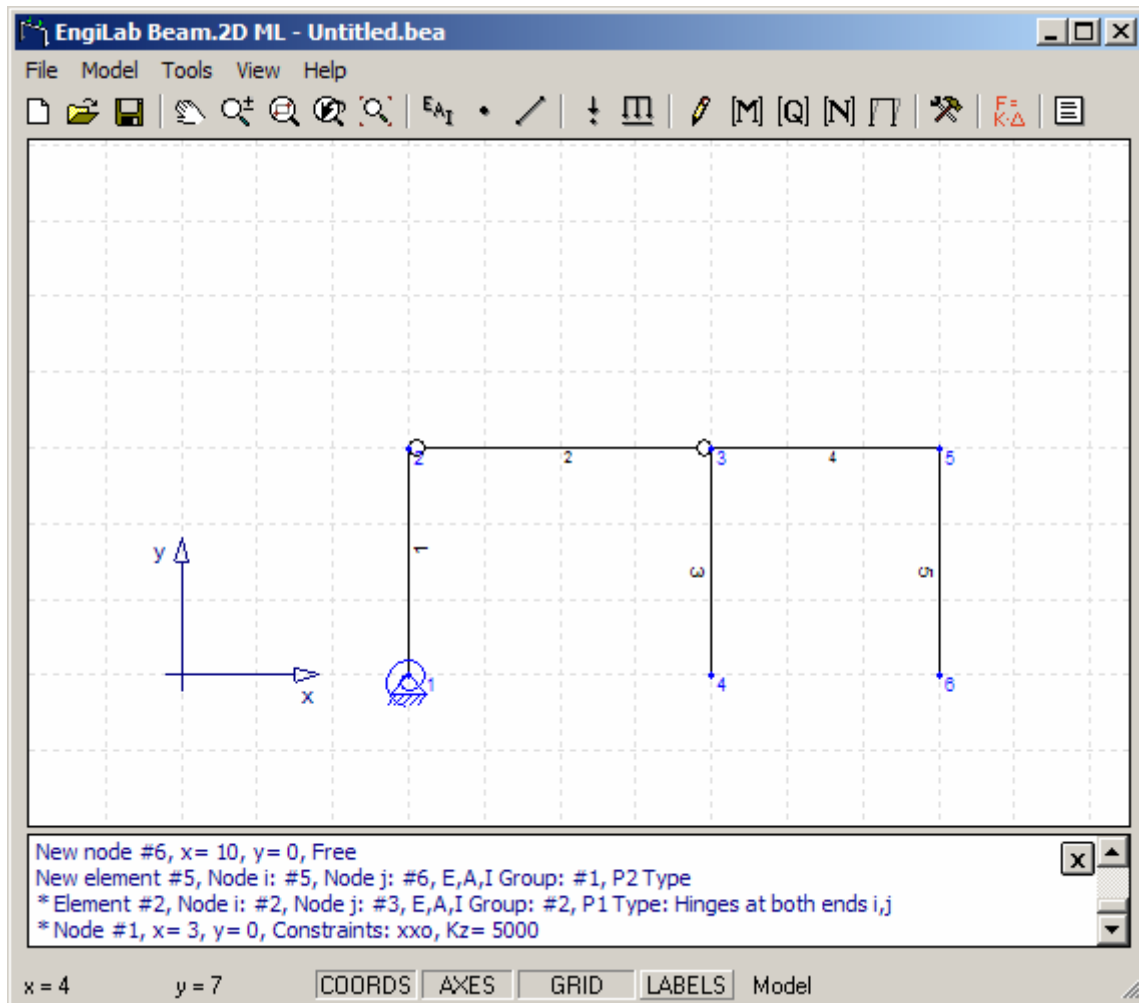


Figure 17. Node #1 properties have been modified

6.2 Add a fixed constraint for node #4

Move the pointer near Node #4 until you see an **orange ring** around it. Press the **right mouse button**. A pop-up menu should appear. Select **Node #4** and press the **left mouse button**. This brings up the Nodes window with **Node #4** selected. Modify the constraints by **checking all the Degrees Of Freedom (DOFs)**, as shown below.

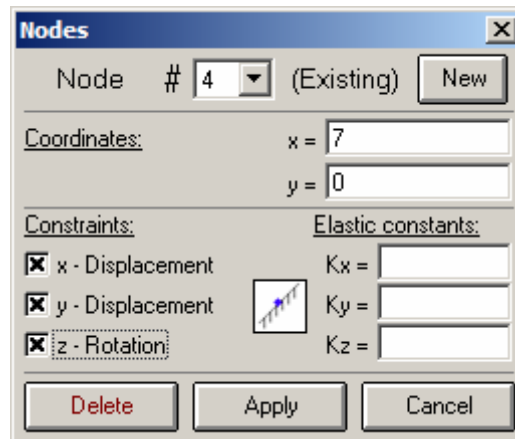


Figure 18. Modifying Node #4 properties

Press **Apply** and then **Cancel**. **Node #4** has been modified as shown below.

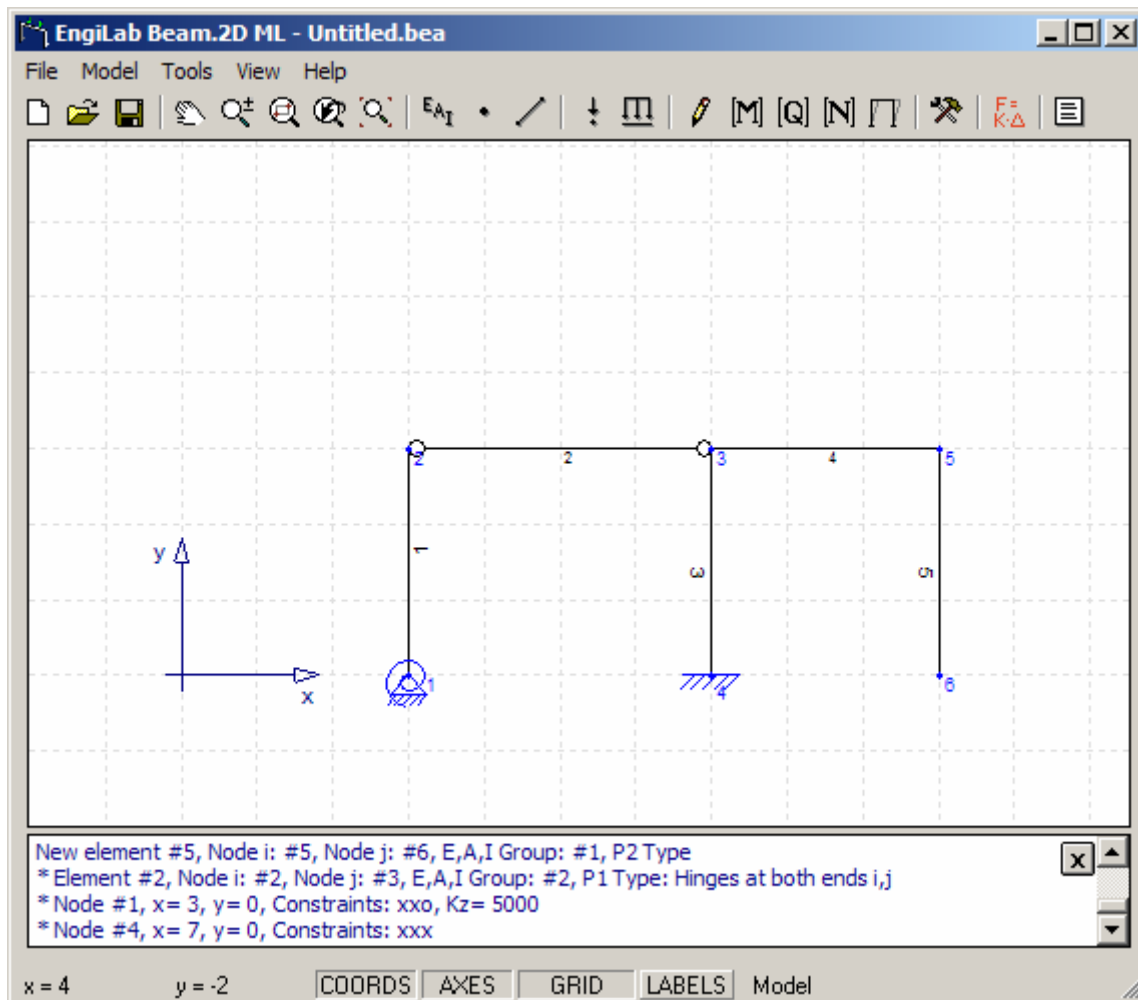


Figure 19. Node #4 properties have been modified

6.3 Add a roller constraint with an axial spring for node #6

Move the pointer near **Node #6** until you see an **orange ring** around it. Press the **right mouse button**. A pop-up menu should appear. Select **Node #6** and press the **left mouse button**. This brings up the Nodes window with **Node #6** selected. Modify the constraints by **checking the y - Displacement** and add the stiffness of the axial spring (**$K_x=20000$**), as shown below.

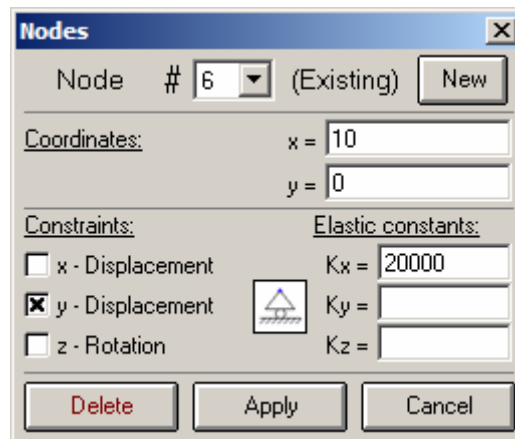


Figure 20. Modifying Node #6 properties

Press **Apply** and then **Cancel**. **Node #6** has been modified as shown in the figure below.

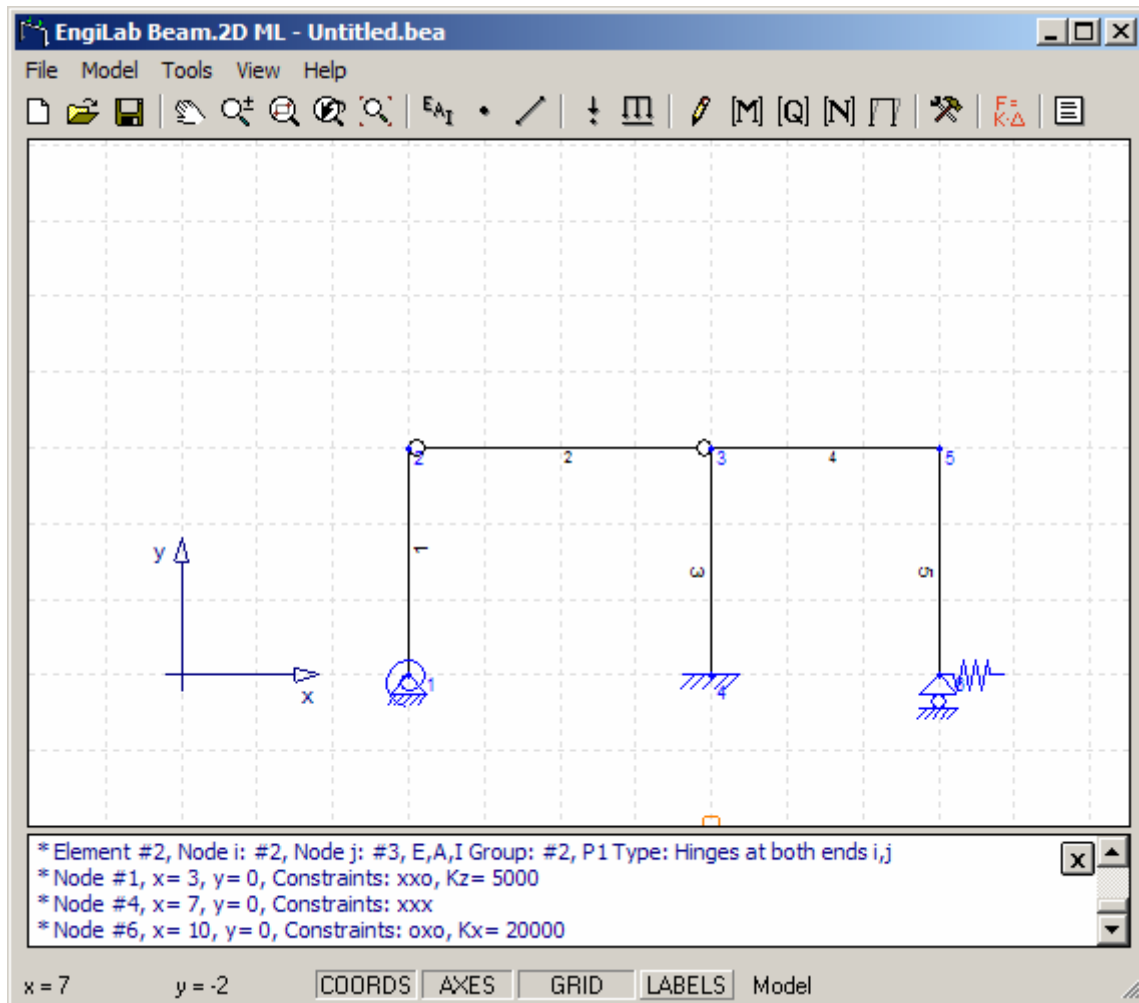


Figure 21. Node #6 properties have been modified

7. Adding loads

7.1 Adding the nodal loads

Move the pointer near **Node #2** until you see an **orange ring** around it. Press the **right mouse button**. A pop-up menu should appear, as shown below.

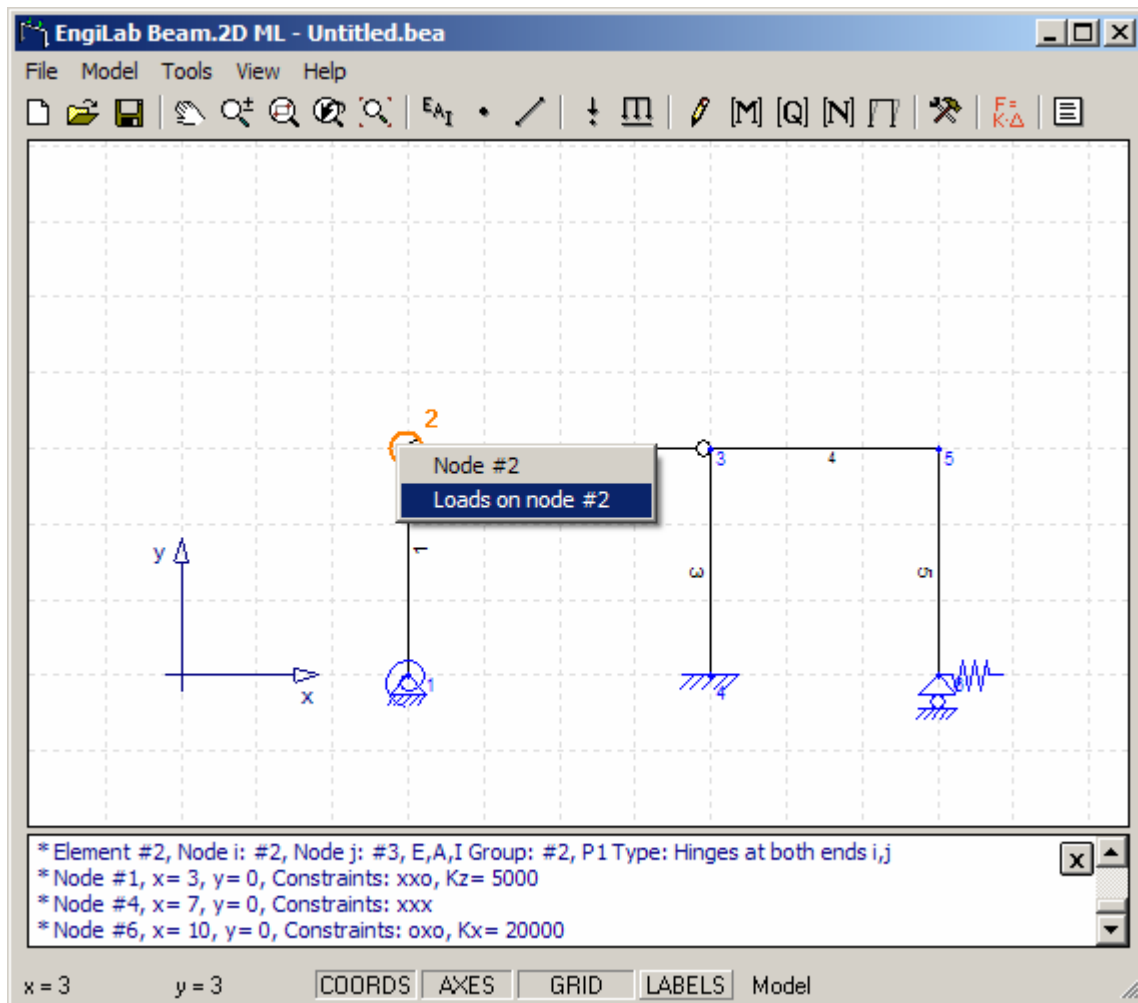


Figure 22. Selecting Loads on Node #2 from the pop-up menu

Select **Loads on node #2** and press the **left mouse button**. This brings up the Nodal loads window with **Node #2** selected, as shown below.

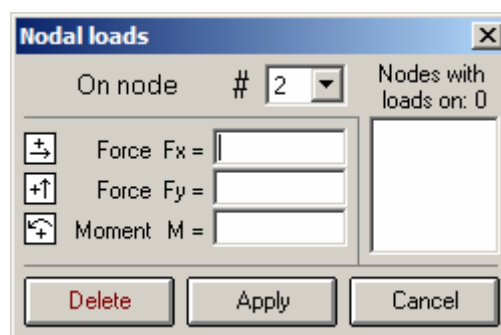


Figure 23. Loads on Node #2

No loads have been specified yet. Add a **horizontal load $F_x=40$** , as shown below.

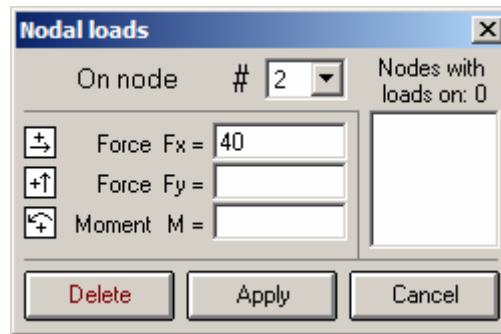


Figure 24. Loads on Node #2 have been specified

Press **Apply** and then **Cancel**. A horizontal load has been added on **Node #2**.

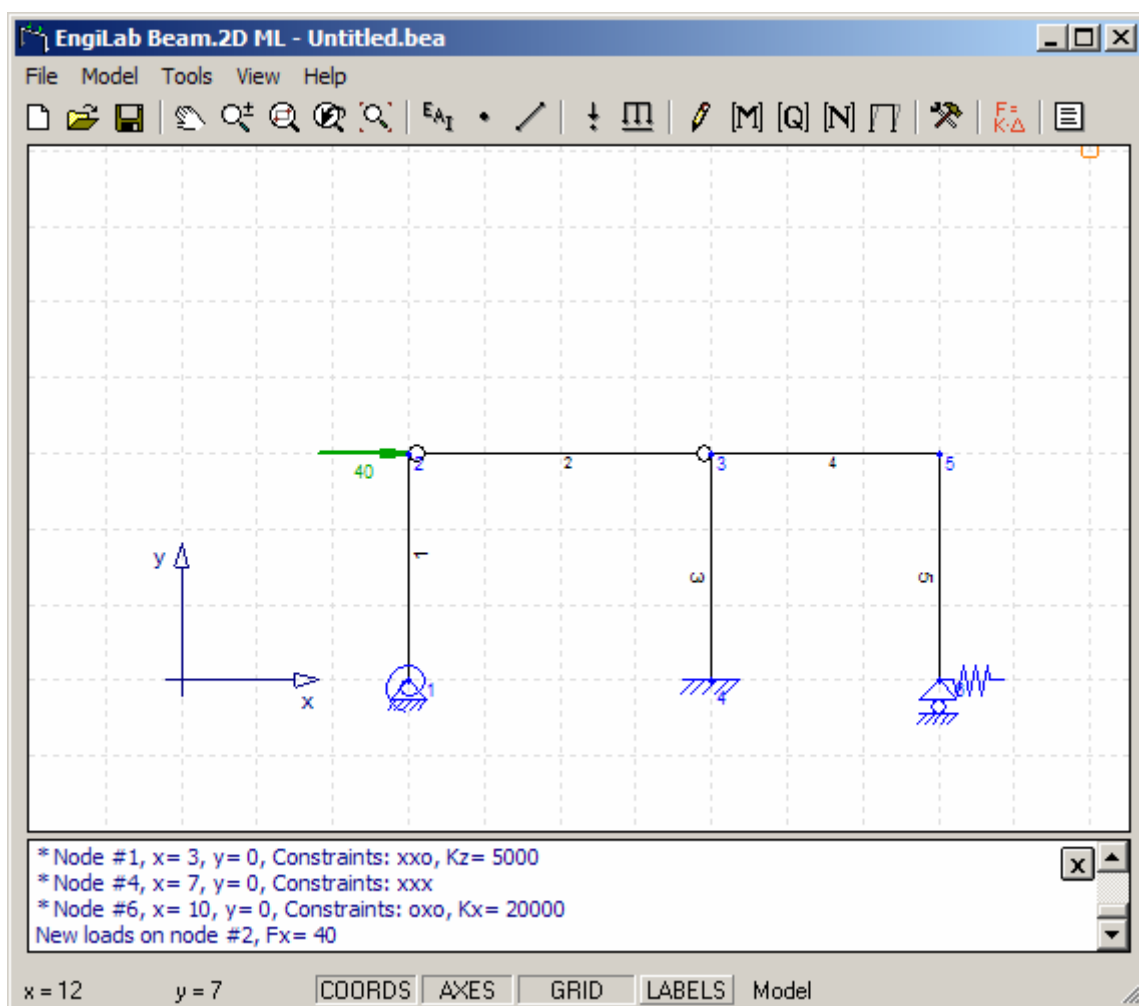


Figure 25. A horizontal load on Node #2 has been added to the model

Do the same for **Node #5**, where a moment has to be added. Move the pointer near **Node #5** until you see an **orange ring** around it. Press the **right mouse button**. A pop-up menu appears. Select **Loads on node #5** and press the **left mouse button**. This brings up the Nodal loads window with **Node #5** selected. Add a **moment M=20**, as shown below.

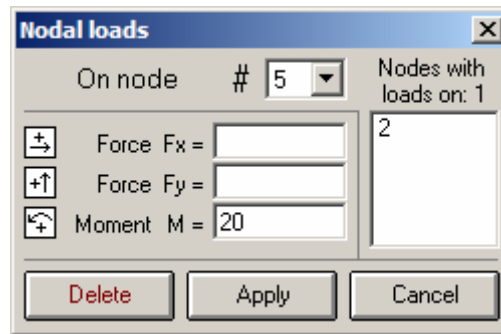


Figure 26. Loads on Node #5 have been specified

Press **Apply** and then **Cancel**. A moment has been added on **Node #5**.

7.2 Adding the elemental load

Move the pointer near Element #4 until it becomes **red** and press the **right mouse button**. A pop-up menu should appear, as shown below.

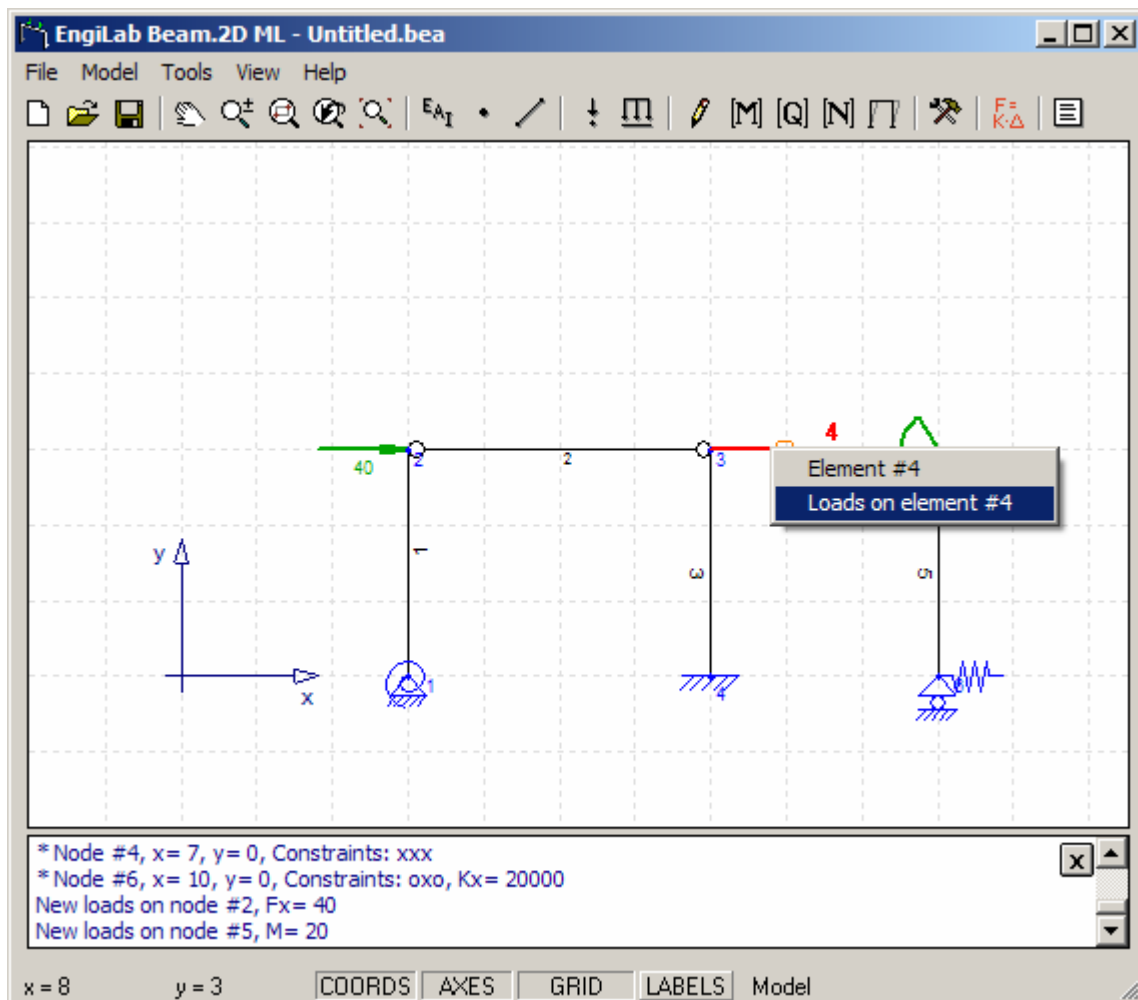


Figure 27. Selecting Loads on element #4 from the pop-up menu

Select **Loads on element #4** and press the **left mouse button**. This brings up the Elemental loads window with **Element #4** selected.

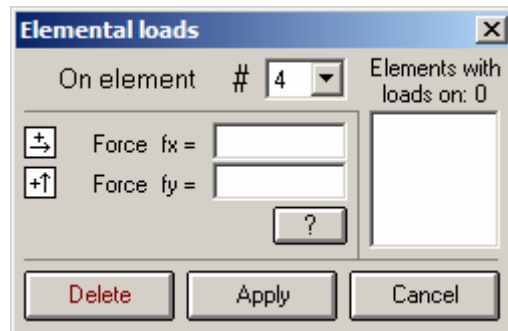


Figure 28. Loads on Element #4

Add a **vertical load $f_y = -10$** , as shown below.

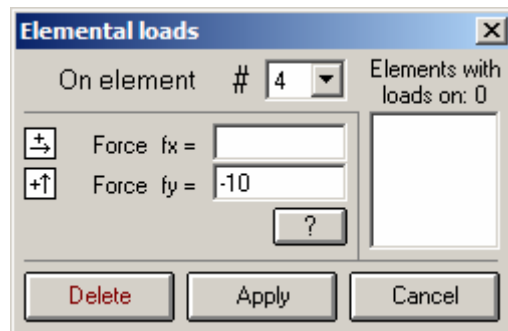


Figure 29. Loads on Element #4 have been specified

Press **Apply** and then **Cancel**. A uniform load has been added on **Element #4**.

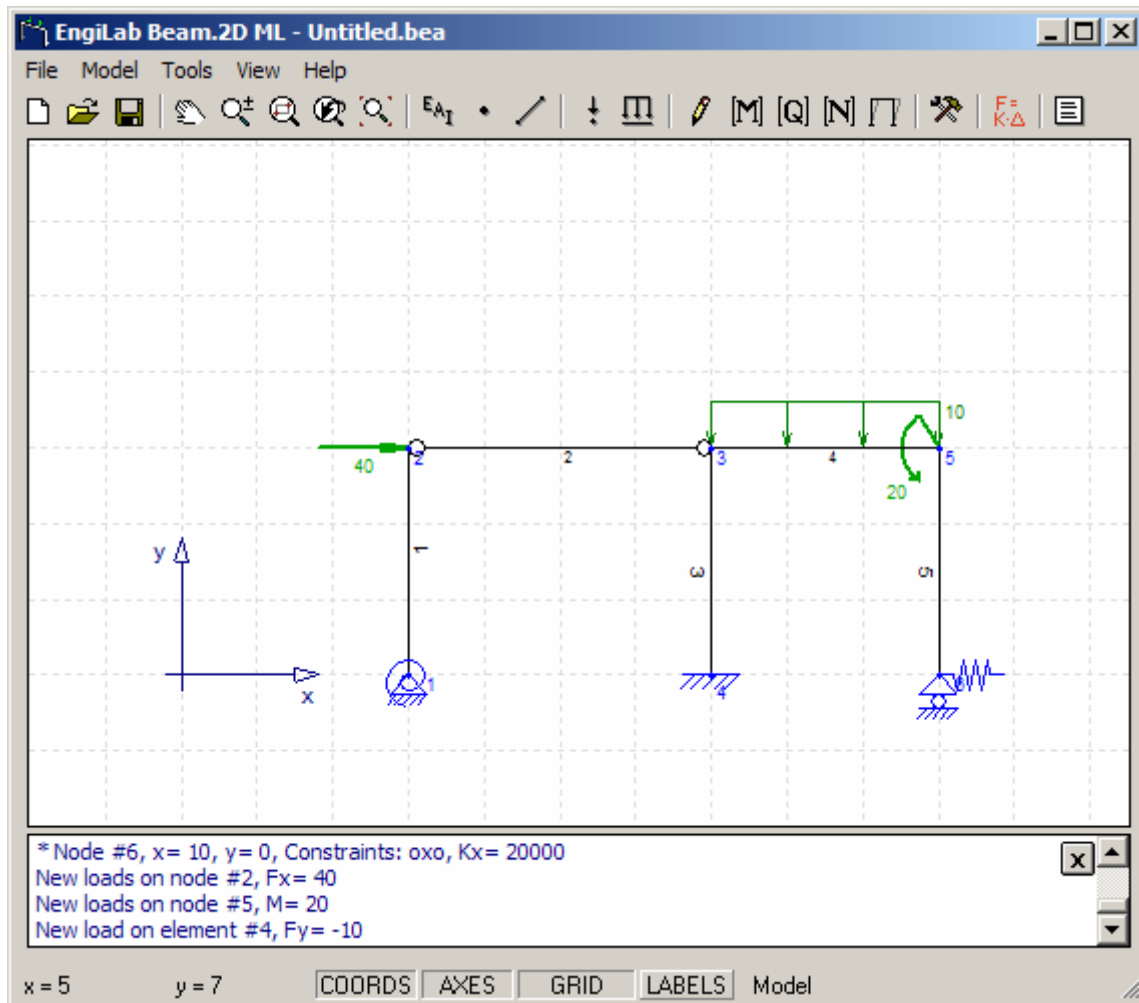


Figure 30. A uniform load has been added on Element #4

8. Creating the arc

Now it is time to create the arc. Select '**Tools**' from the menu at the top of the program's window and then '**Create arc**', as shown below.

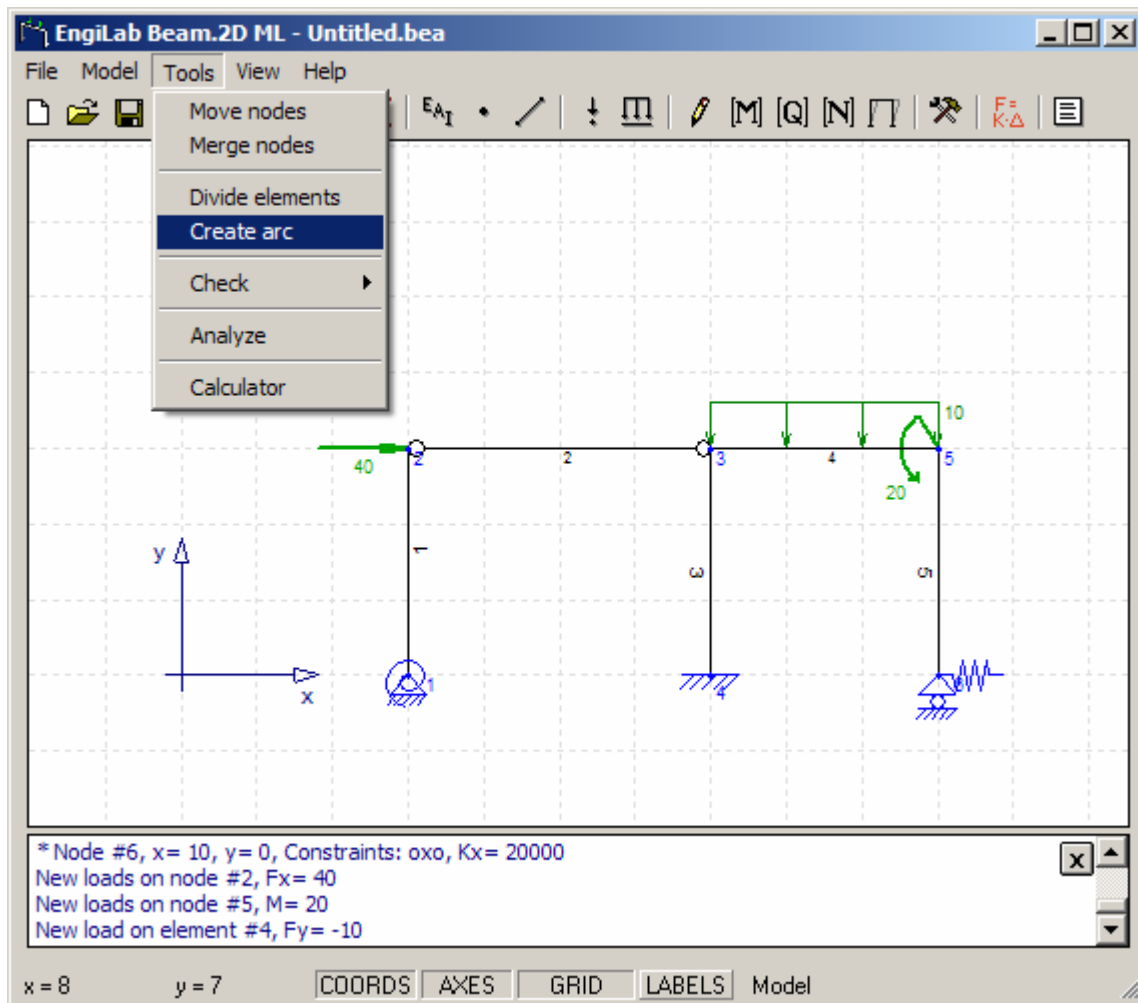


Figure 31. Selecting 'Create arc' from the Tools menu of the program

This brings up the **Create arc window**, as shown below.

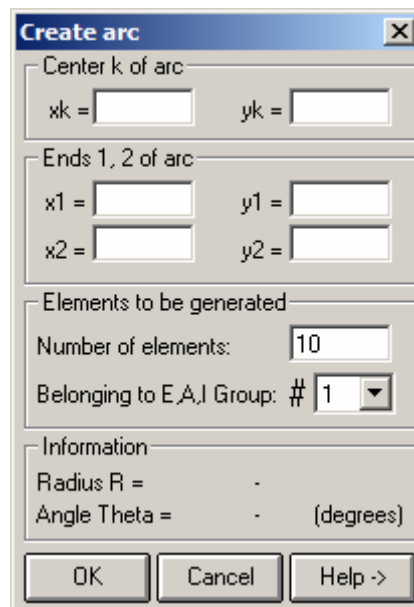


Figure 32. Create arc window

Press the **Help** button that will make the window expand in order to show helpful information for the definition of the arc, as shown in the figure below.

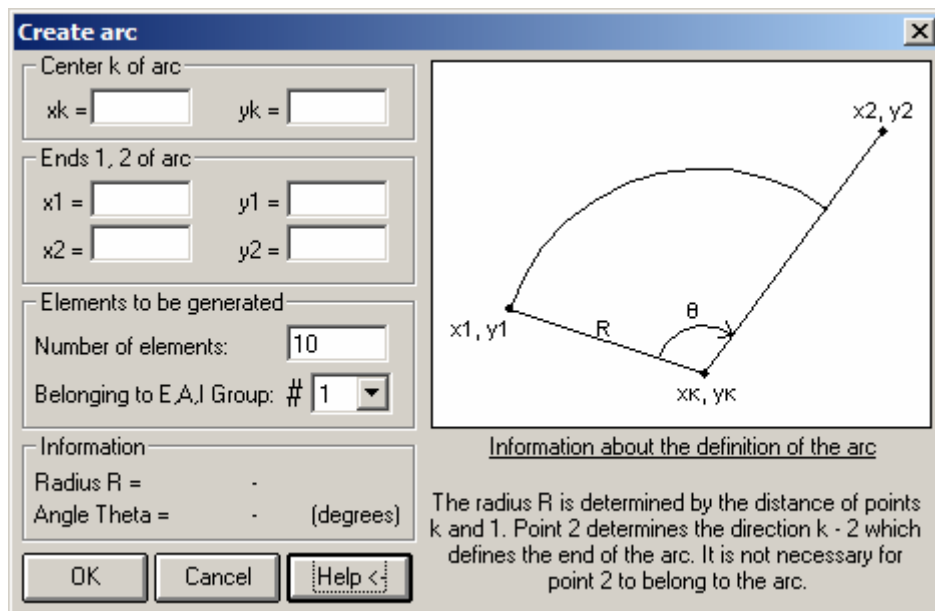


Figure 33. Create arc expanded window

The arc will be modelled with **10 linear beam elements**. Type in the data, as seen in the figure below.

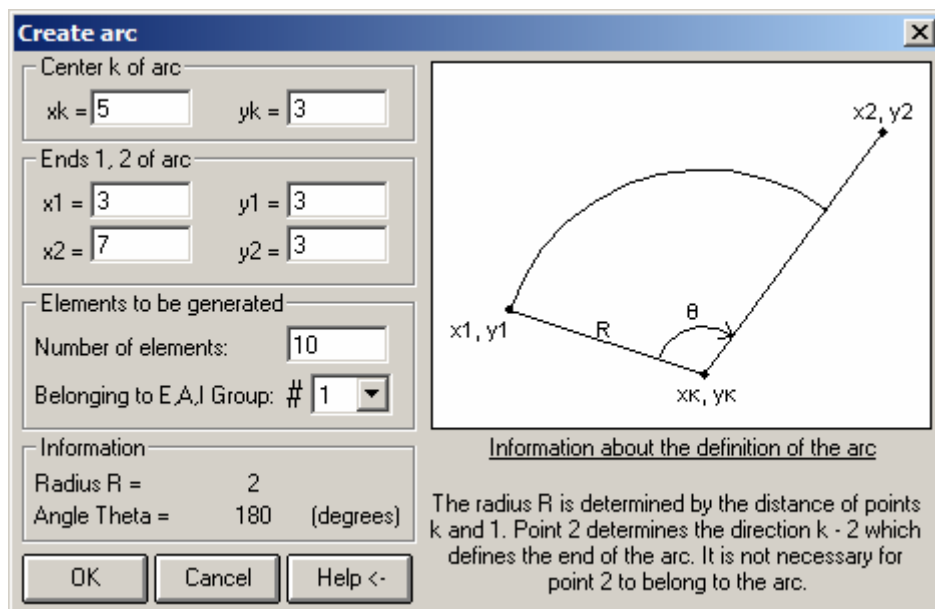


Figure 34. Specifying the arc properties

Now press **OK**. The arc has been created as shown below.

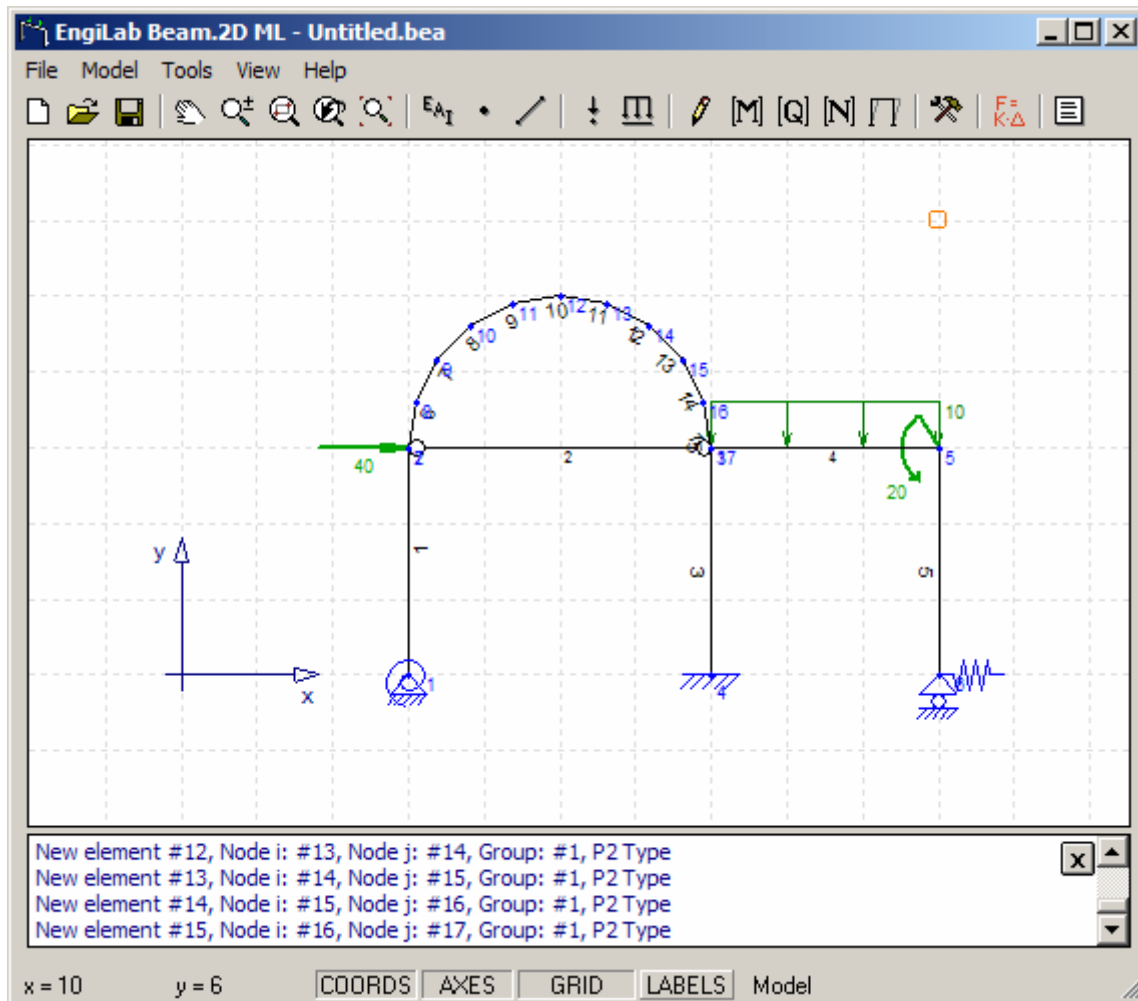


Figure 35. The arc has been created

At this point, **the arc is not connected to the main frame**. Two new nodes, **Node #7** and **Node #17** (start node and end node of the arc) have been defined at points $\{x=3, y=3\}$ and $\{x=7, y=3\}$ respectively, at locations where there were already **Node #2** and **Node #3**. These two new nodes have to be merged into Nodes **#2** and **#3** in order for the arc to be connected to the main frame.

9. *Checking for coincident nodes*

Let's have the program check if there are coincident nodes in the model indeed, as we suspected before. Select '**Tools**' from the menu at the top of the program's window, then '**Check**' and then '**Coincident nodes**', as shown below.

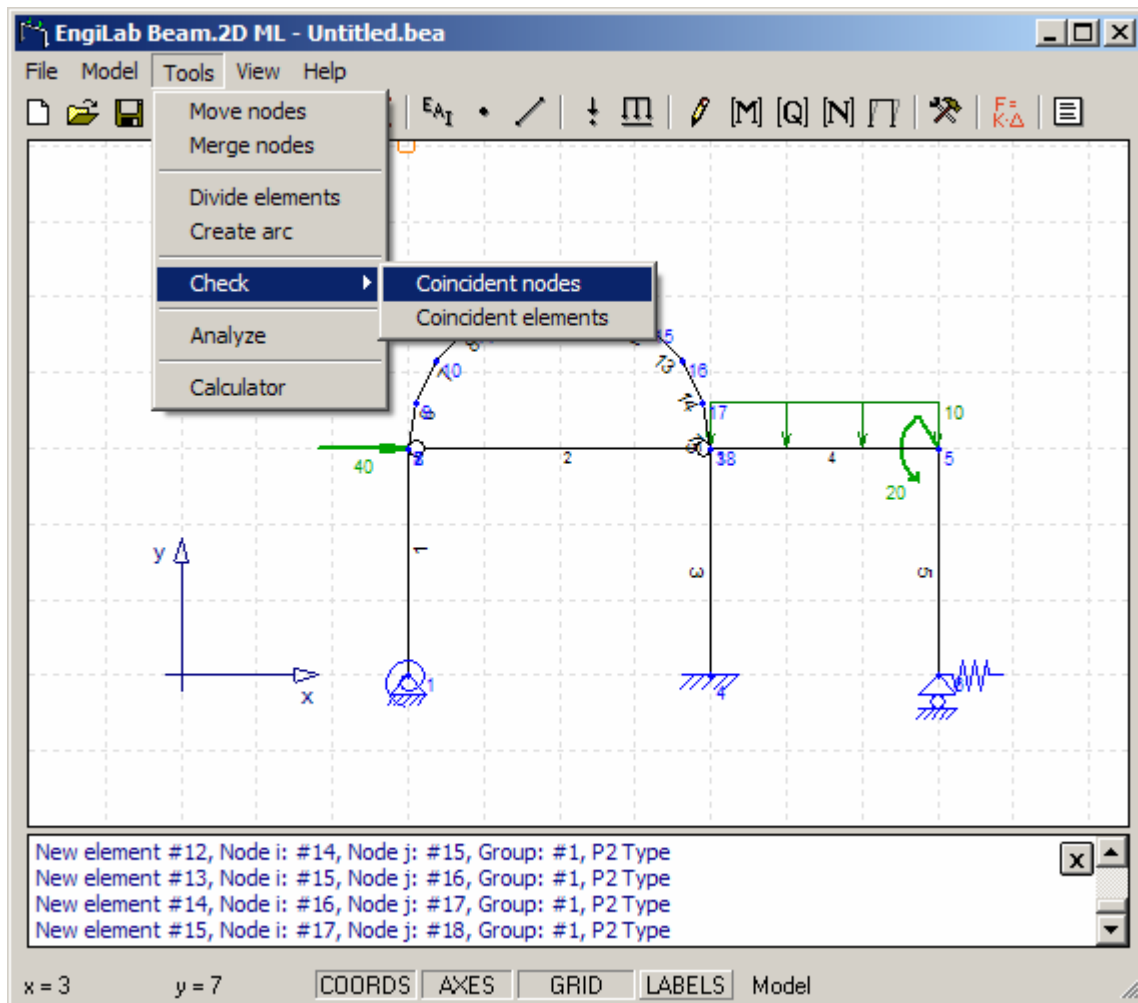


Figure 36. Selecting 'Check for coincident nodes' from the Tools menu of the program

This brings up the **Tolerance control input box**, as shown below.

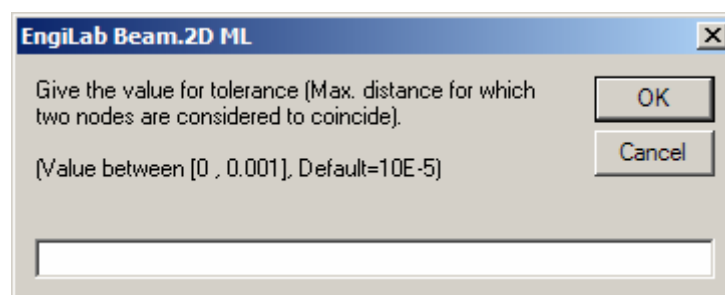


Figure 37. Tolerance control input box

Click **OK**, which makes the program use the **default value of 10^{-5} for tolerance**. The program informs us, as shown in the figure below.

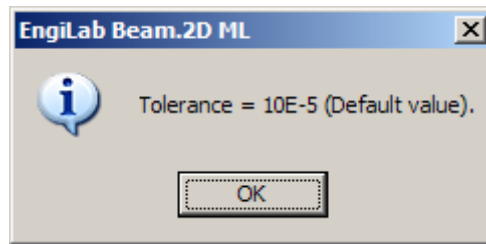


Figure 38. Tolerance value message box

Click **OK**. A new message box appears which informs us that **Nodes #2 and #7 coincide**, as shown below.

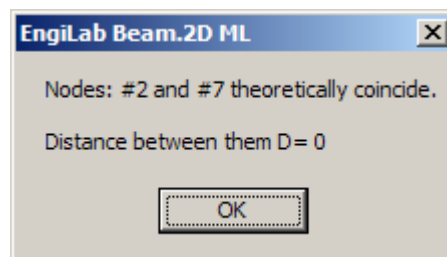


Figure 39. Coincident nodes #2 and #7

Click **OK**. The program now informs us about the **coincidence of Nodes #3 and #17**, as shown below.

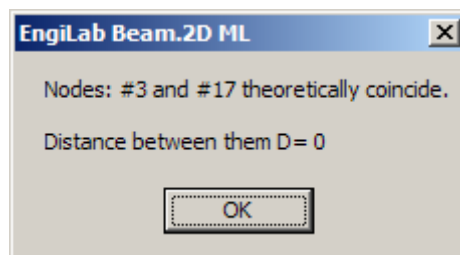


Figure 40. Coincident nodes #3 and #17

Click **OK**. This brings up a new message box with the results of the '**Check for coincident nodes**' procedure, as shown below.

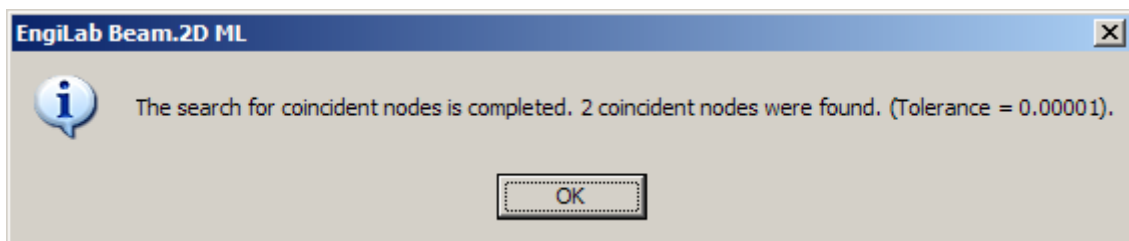


Figure 41. Check for coincident nodes result

2 coincident nodes were found, as we suspected before. Click **OK**. Now, the nodes that were found to coincide have to be merged.

10. Merging the coincident nodes

Select '**Tools**' from the menu at the top of the program's window and then '**Merge nodes**', as shown below.

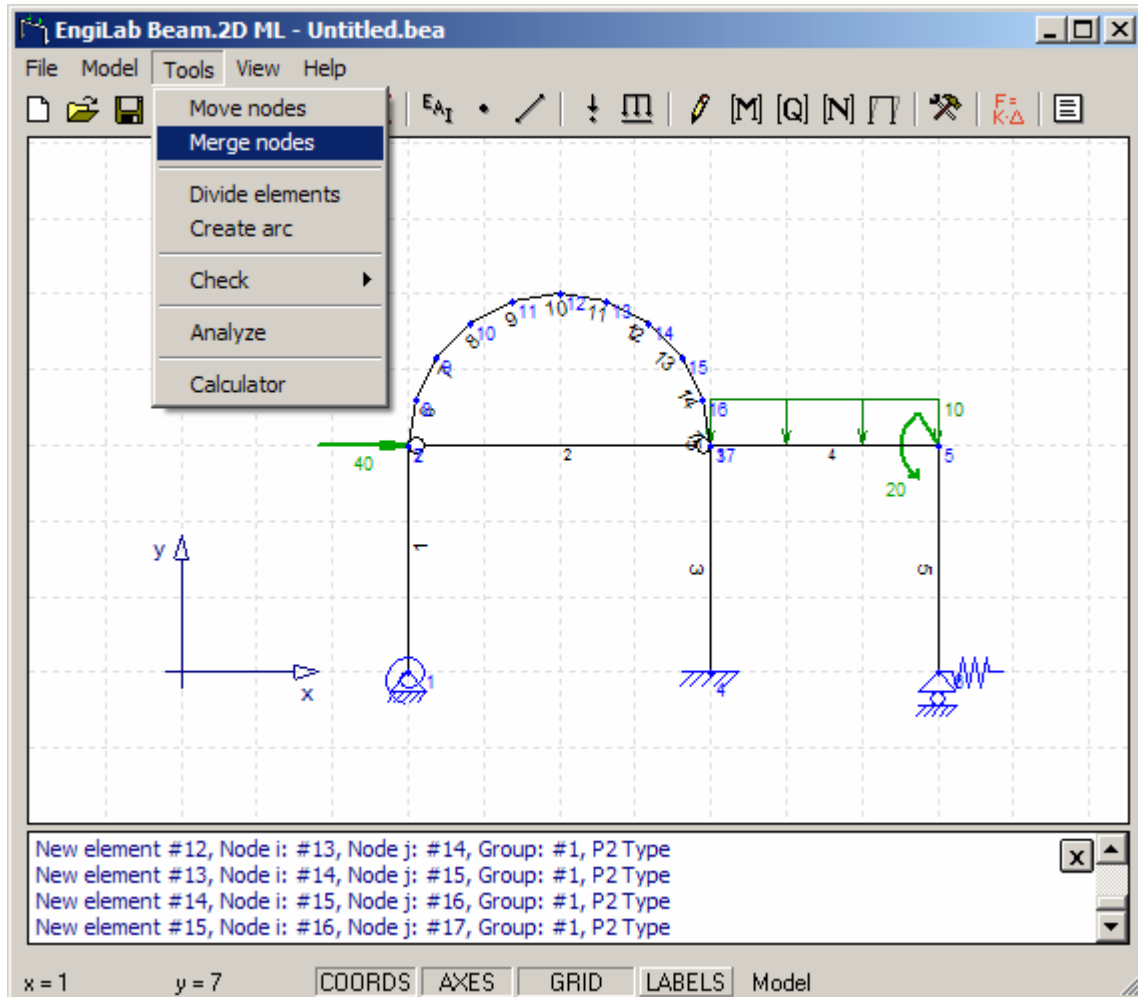


Figure 42. Selecting 'Merge nodes' from the Tools menu

This brings up the **Merge nodes** window.

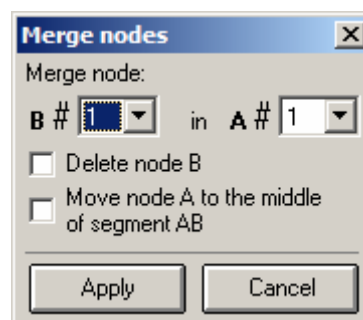


Figure 43. Merge nodes window

Select **Node #7** as node **B** and **Node #2** as node **A**. Check the '**Delete node B**' checkbox and press **Apply**.

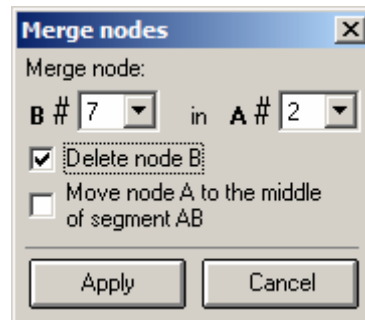


Figure 44. Specifying the node to be merged

Node #7 has been merged into **Node #2** and has been deleted. Note that the program automatically renumbers **Nodes #8 to #17** (as **#7 to #16**), as shown below.

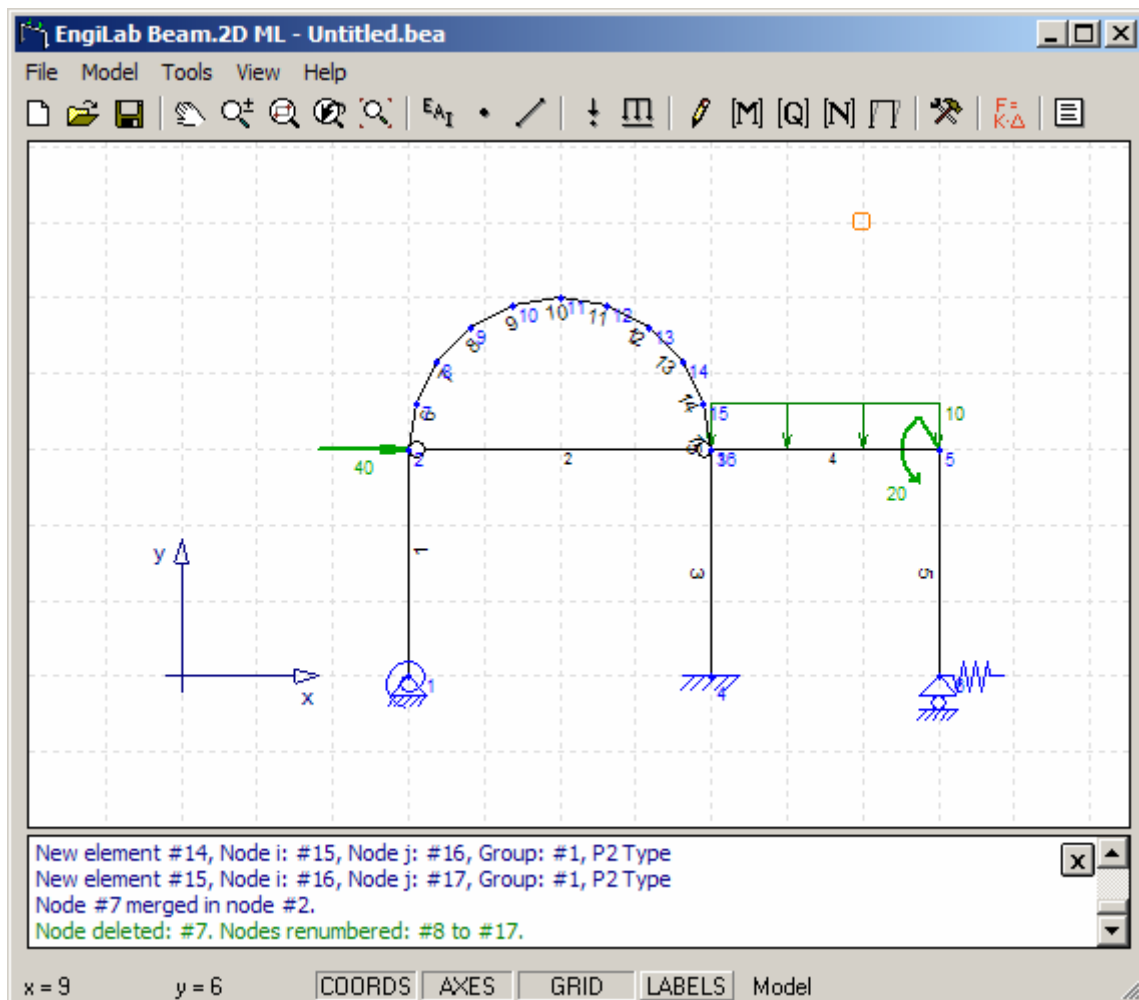


Figure 45. Node #7 has been merged in Node #2

Now **Node #16** (previously #17, before the renumbering) has to be merged into **Node #3**. as shown below.

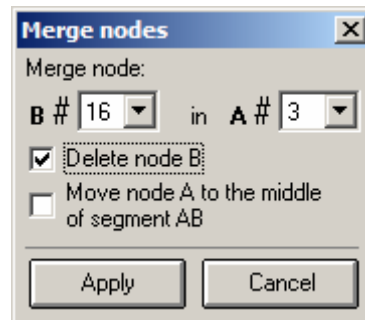


Figure 46. Specifying the second node to be merged

Press **Apply**. Both nodes were merged and now **the arc is connected to the main frame**.

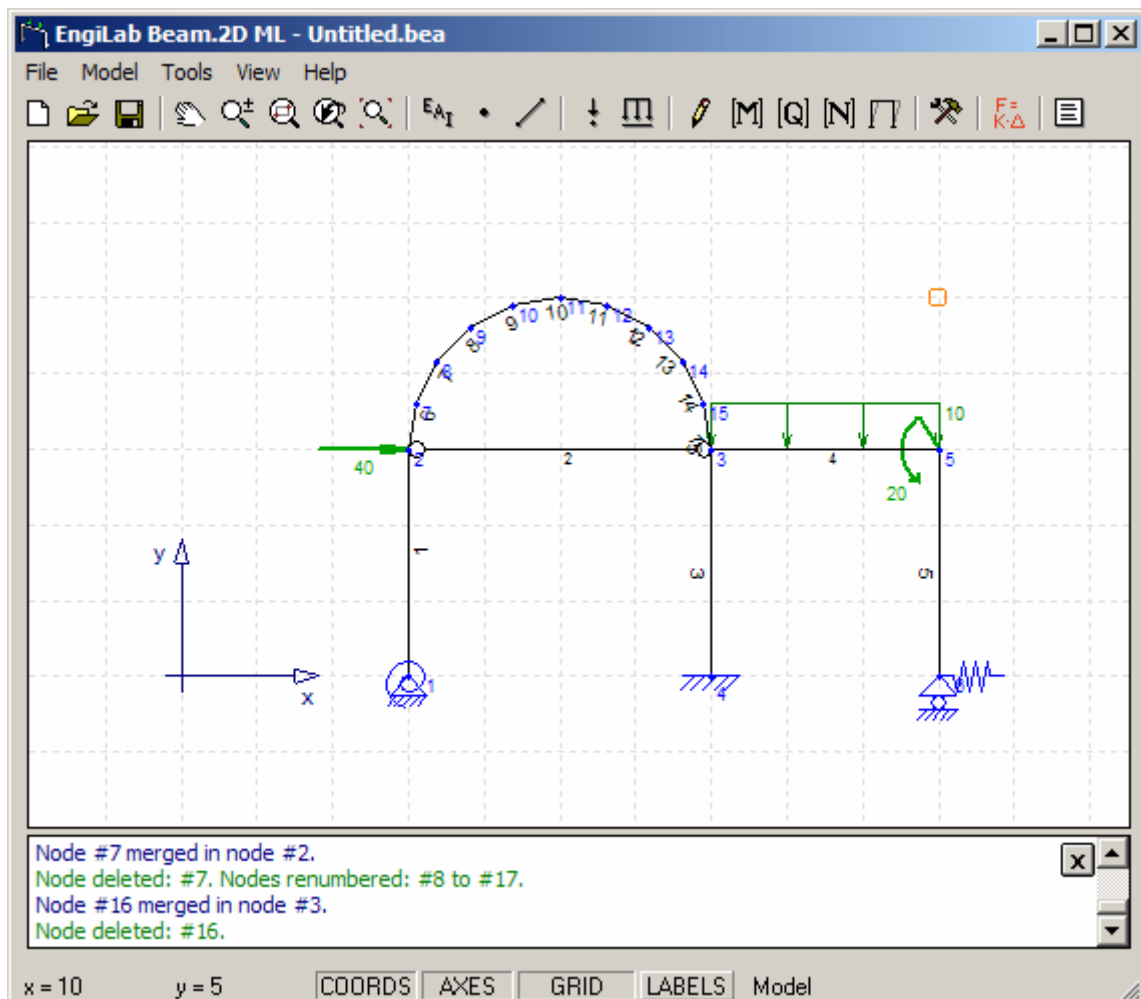



Figure 47. The second node has been merged

11. Analysis

Now that the model is ready, it is time to perform the **Finite Elements Analysis**. Press the 'Analyze' button .

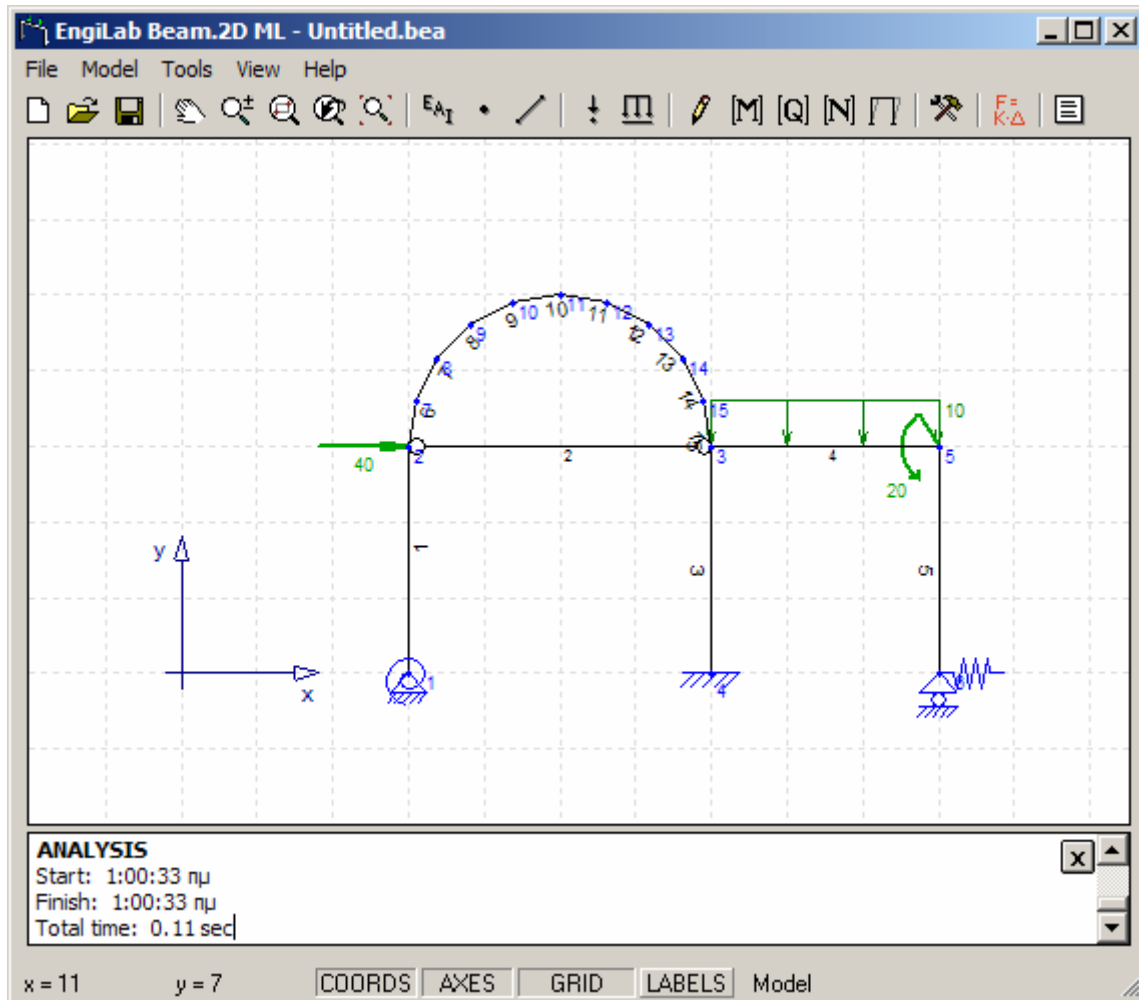



Figure 48. The model is ready

The total time for the analysis was **0.11 sec**, as shown in the information window above.

12. Post-processing

12.1 Draw the deformed model

After the analysis has been performed, press the 'Draw deformed model' button .

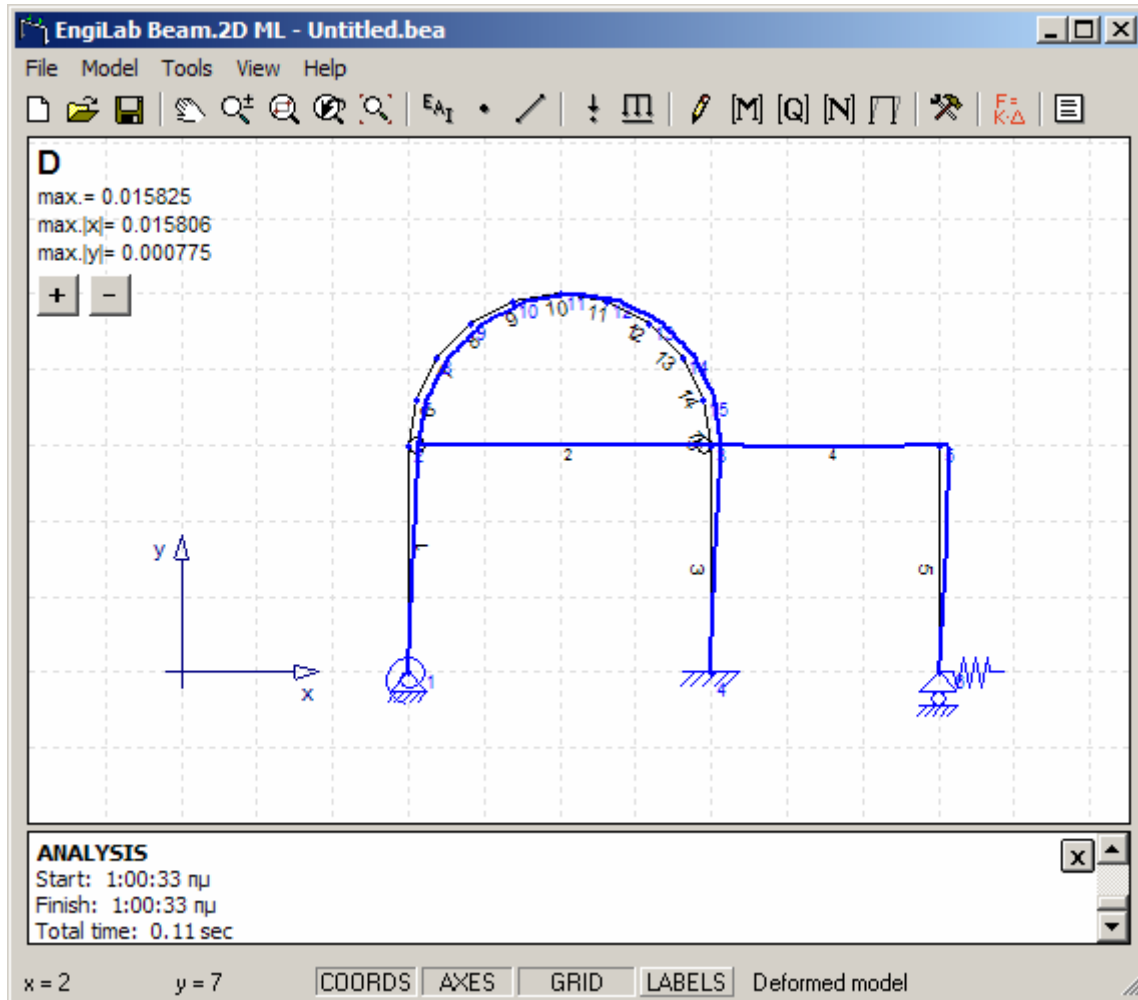


Figure 49. Deformed state of the model

Press the **[+]** button to adjust the deformation scale.

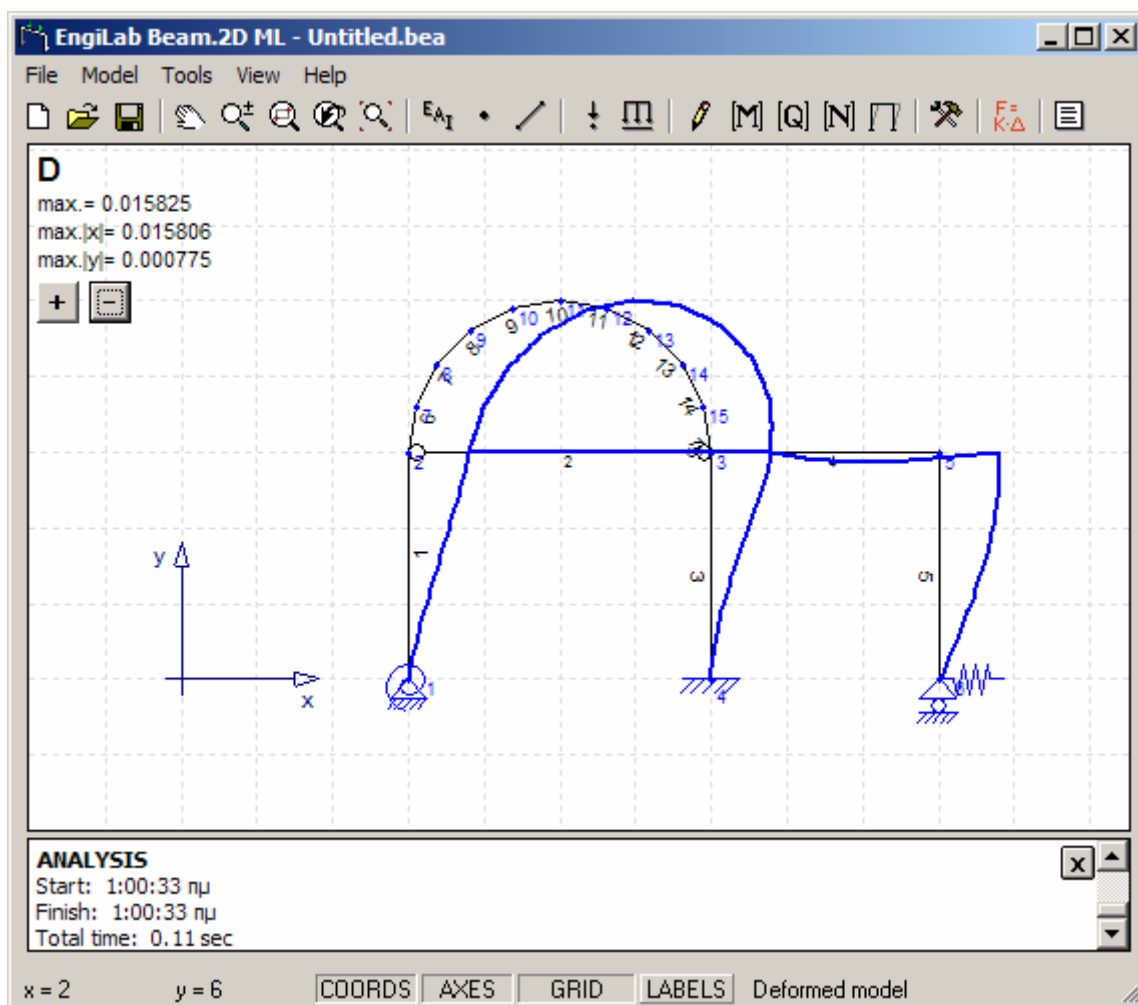


Figure 50. Deformed state of the model (adjusted scale)

12.2 Drawing the bending moment diagram

Press the 'Draw [M] diagram' button  that draws the bending moment diagram.

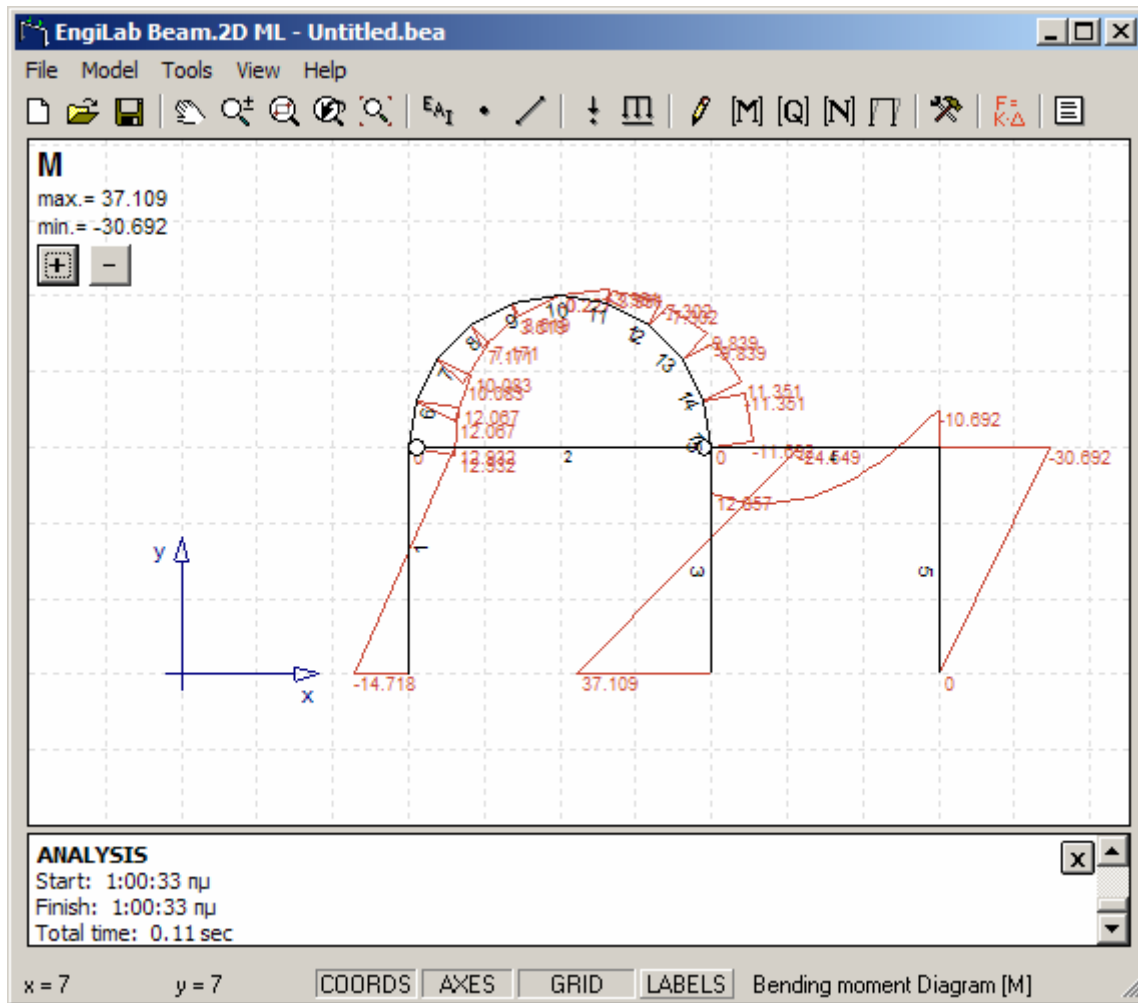


Figure 51. Bending moment diagram

You can adjust the scaling by using the [+] / [-] buttons.

12.3 Drawing the shear force diagram

Press the 'Draw [Q] diagram' button **[Q]** that draws the shear force diagram.

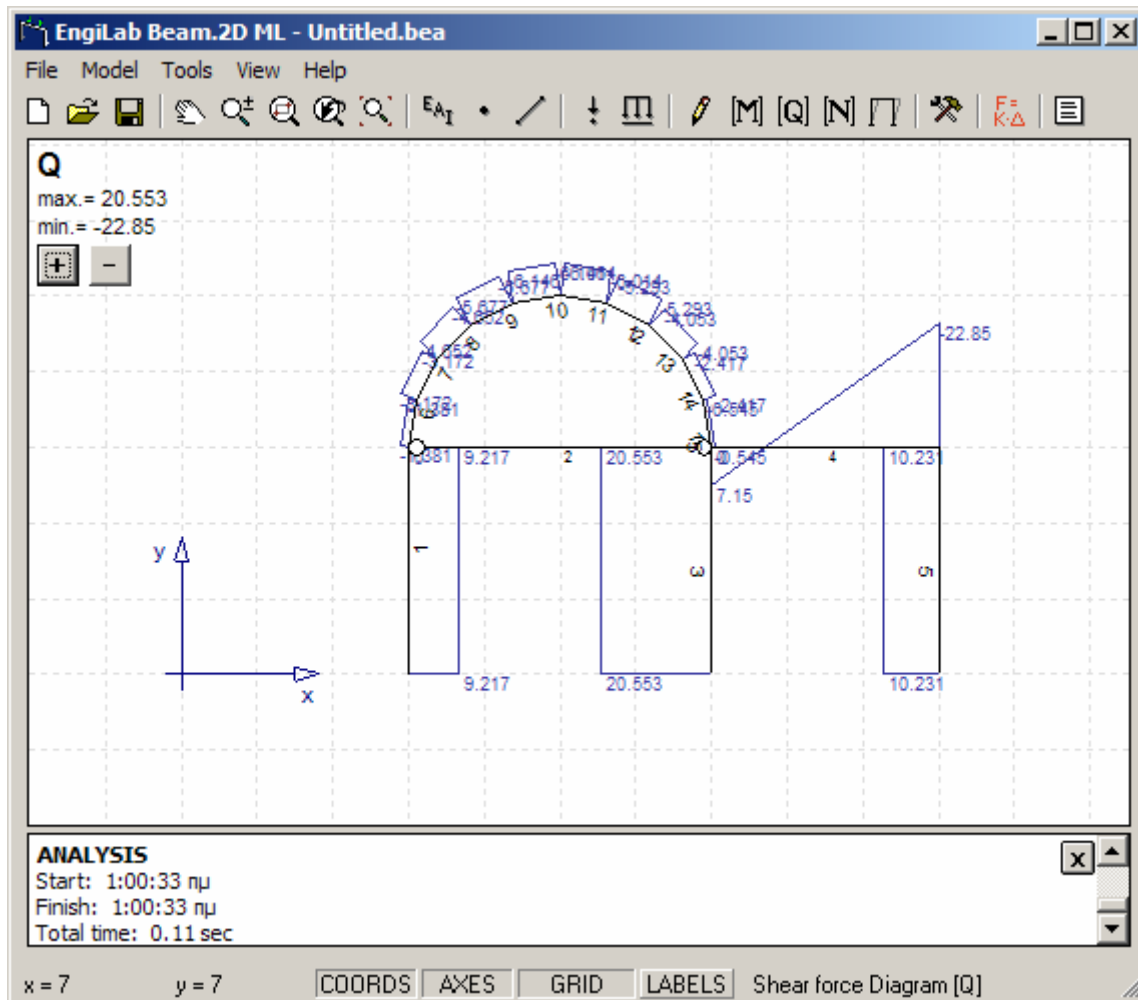



Figure 52. Shear force diagram

You can adjust the scaling by using the [+] / [-] buttons.

12.4 Drawing the axial force diagram

Press the 'Draw [N] diagram' button  that draws the axial force diagram.

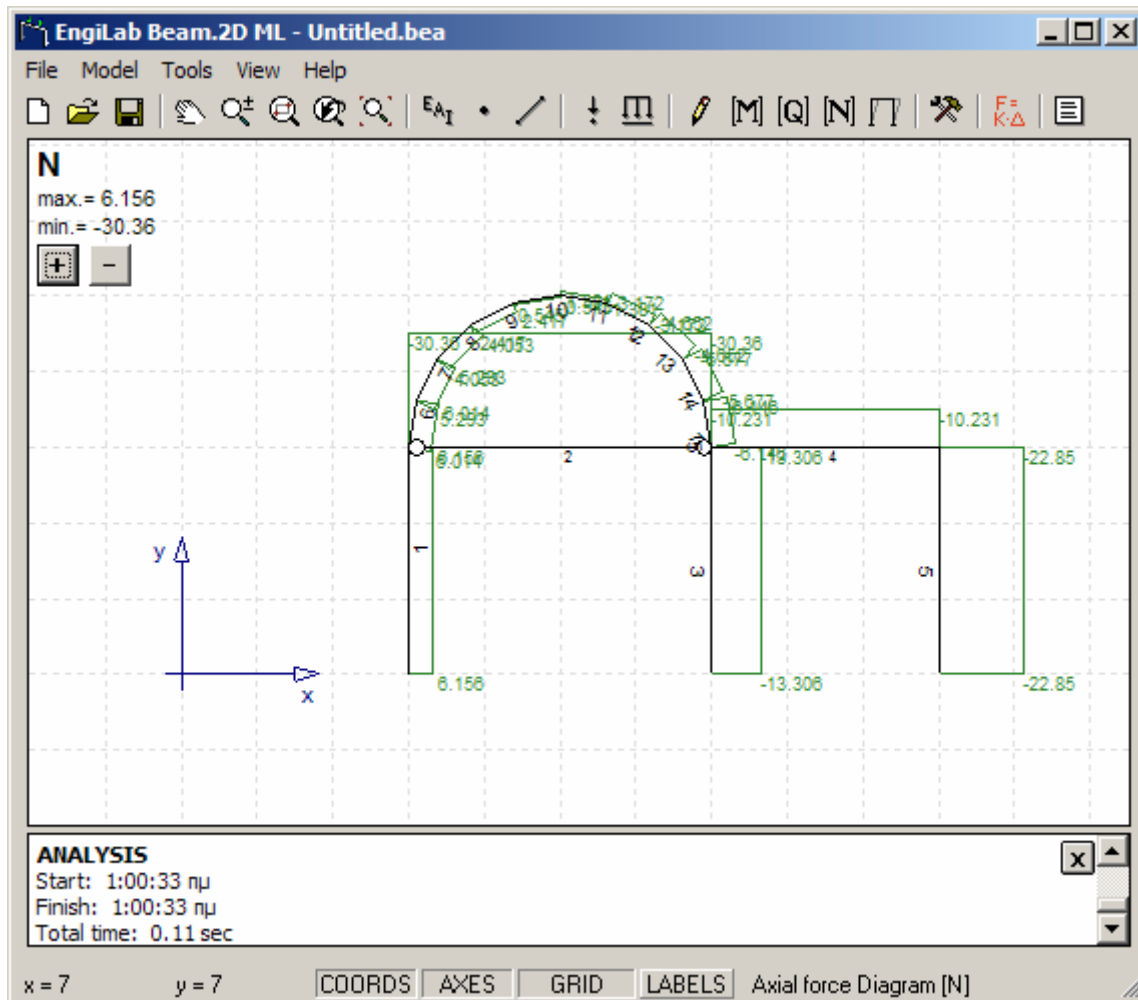



Figure 53. Axial force diagram

You can adjust the scaling by using the [+] / [-] buttons.

12.5 Back to normal model view

Press the '**Draw model**' button  to return back to normal model view.

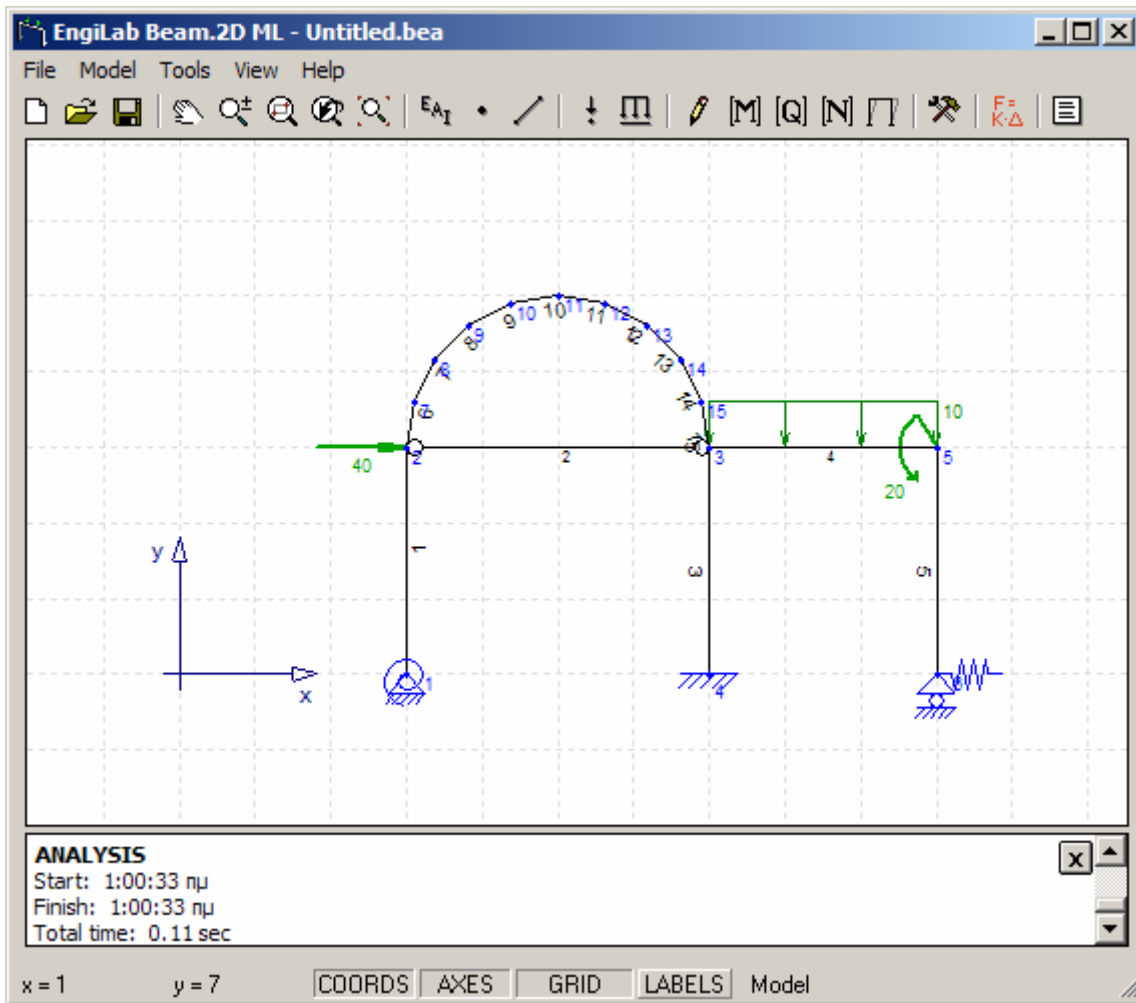



Figure 54. Return to the normal model view

13. *Displaying the analytical results*

Press the '**Data - Results**' button  to see the model data and the numerical analysis results.

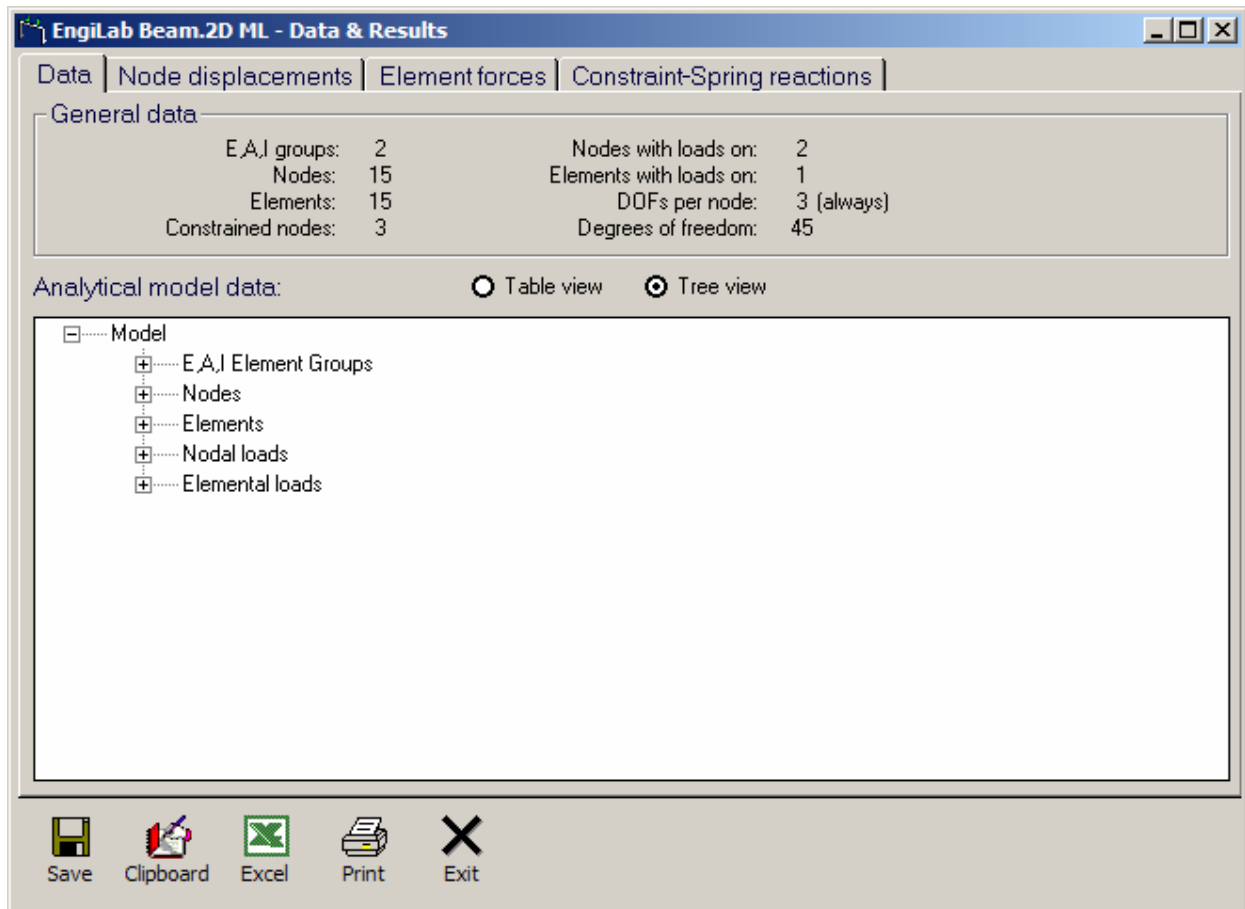


Figure 55. The Data & Results window

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
- Constraint - spring reactions.

13.1 Node displacements

Click on the 'Node displacements' pane.

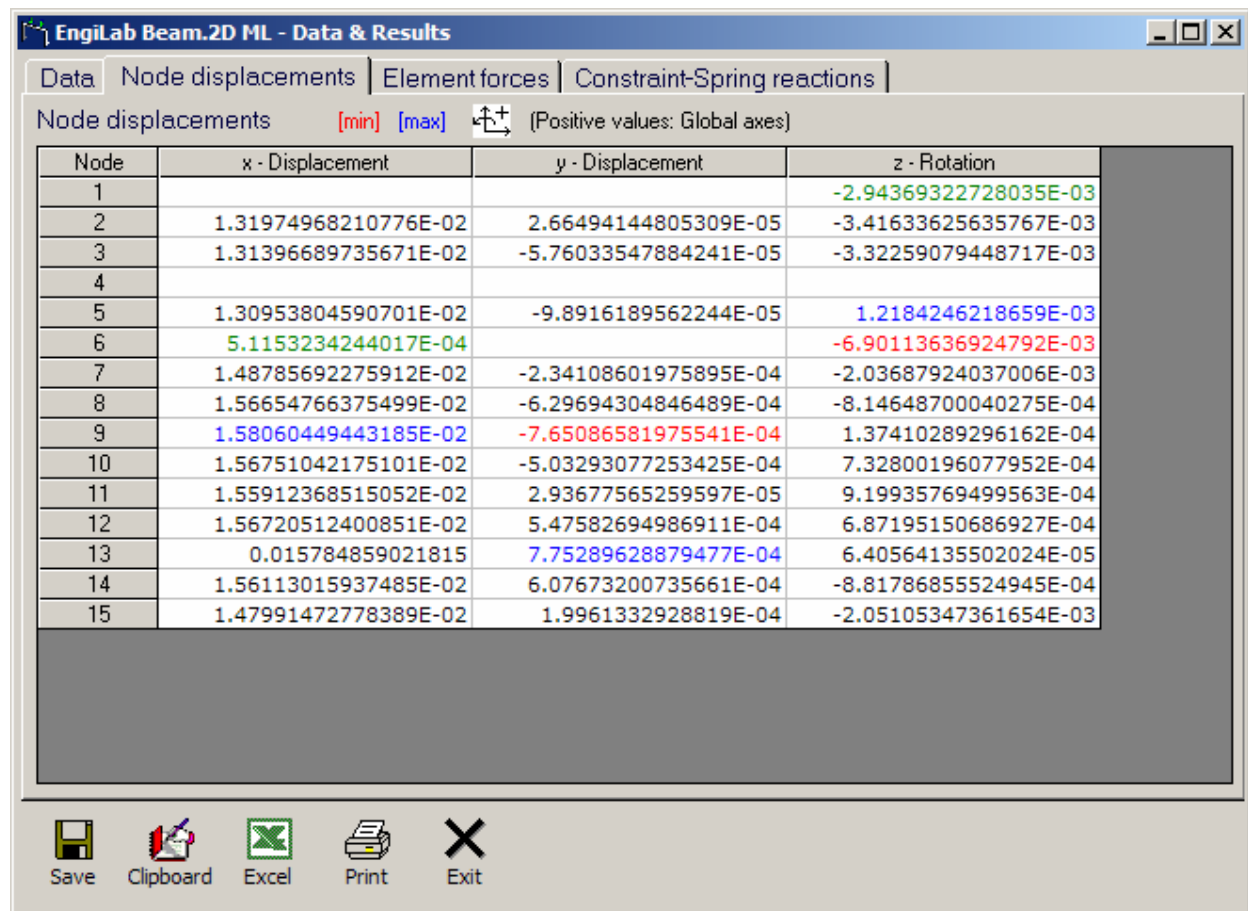


Figure 56. Node displacements

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 (nodal) displacement.

13.2 Element forces

Click on the 'Element forces' pane.

Element	i, j	Axial force	Shear force	Moment
1	1	-6.15601474500263	9.21678054096375	14.7184661364018
	2	6.15601474500263	-9.21678054096375	12.9318754864895
2	2	30.3596199429949	0	0
	3	-30.3596199429949	0	0
3	3	13.306374956126	20.5525726102276	24.5491623137606
	4	-13.306374956126	-20.5525726102276	37.1085555169221
4	3	10.2306468488027	7.15036021112164	-12.8569788202248
	5	-10.2306468488027	22.8496397888784	-10.6919405464102
5	5	22.8496397888784	10.2306468488034	30.6919405464103
	6	-22.8496397888784	-10.2306468488034	0
6	2	-6.01395823248595	-1.38139801237651	-12.9318754864892
	7	6.01395823248595	1.38139801237651	12.0674825749356
7	7	-5.29273920306423	-3.17220204574546	-12.0674825749349
	8	5.29273920306423	3.17220204574546	10.0825153665613
8	8	-4.05342968098012	-4.65248986157572	-10.0825153665612
	9	4.05342968098012	4.65248986157572	7.17127688322371

Figure 57. Element forces

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

13.3 Constraint - Spring reactions

Click on the 'Constraint – Spring reactions' pane.





Node	Force Fx	Force Fy	Moment M
1	-9.21678054096375	-6.15601474500263	14.7184661364018
4	-20.5525726102276	13.306374956126	37.1085555169221
6	-10.2306468488034	22.8496397888784	

Figure 58. Constraint – spring reactions

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

13.4 Exporting data and the analysis results

You 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).