

**EngiLab Beam.2D 2015**  
**v2.2**

(v2.2.5602.0)

**User Manual**



[www.engilab.com](http://www.engilab.com)

*This page intentionally left blank.*

# EngiLab Beam.2D 2015 v2.2 User Manual

(c) 2017 EngiLab SMPC

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

## **Publisher**

EngiLab SMPC

## **Document type**

User Manual

## **Program name**

EngiLab Beam.2D

## **Program version**

v2.2.5602.0

## **Document version**

v2.0

## **Document release date**

October 3, 2017

*"I was originally supposed to become an engineer, but the thought of having to expend my creative energy on things that make practical everyday life even more refined, with a loathsome capital gain as the goal, was unbearable to me"*

**Albert Einstein**



*This page intentionally left blank.*

# Table of Contents

<b>Chapter 1 Introduction to EngiLab Beam.2D</b>	<b>2</b>
1 Overview .....	2
2 Lite and Pro Editions .....	3
<b>Chapter 2 Structural Modeling</b>	<b>6</b>
1 System of Units .....	6
2 Setting up a model .....	10
3 Model Properties .....	12
Materials .....	12
Sections .....	15
Nodes .....	19
Elements .....	23
Nodal Loads .....	26
Elemental Loads .....	28
Body (Acceleration) Loads .....	31
<b>Chapter 3 Tools</b>	<b>34</b>
1 Import Materials/Sections .....	34
2 Convert Model to Truss .....	34
<b>Chapter 4 Analysis and Analysis results</b>	<b>40</b>
1 Analysis .....	40
2 N, V, M Diagrams .....	44
3 Deformation .....	49
4 Free Body Diagram (F) .....	51
5 Analysis results .....	53
Node Displacements .....	54
Element End Forces .....	55
Support Reactions .....	56
Full Report (RTF) .....	57
Analysis Validation .....	58
<b>Chapter 5 Settings</b>	<b>62</b>
1 General (Settings) .....	62
2 Model (Settings) .....	63
3 NVM Diagrams (Settings) .....	64
4 Colors (Settings) .....	66
5 Fonts (Settings) .....	69
6 Decimals (Settings) .....	70
7 Results (Settings) .....	70
<b>Chapter 6 Useful information</b>	<b>74</b>

---

1 Import DXF file .....	74
2 Ready to-analyze Examples .....	75
3 Tips on Modeling hinges .....	76
4 Tips on Modeling symmetric structures .....	78
<b>Chapter 7 Example Problems</b>	<b>82</b>
1 Example Problem 1 .....	82
Overview - Example 1 .....	82
Step 1. Preparation of the input data .....	84
Step 2. Define Materials .....	84
Step 3. Define Sections .....	86
Step 4. Draw the Model on screen .....	94
Step 5. Edit Nodes .....	98
Step 6. Edit Elements .....	101
Step 7. Define Nodal Loads .....	103
Step 8. Define Elemental Loads .....	108
Step 9. Define Body (Acceleration) Loads .....	113
Step 10. Run the Analysis .....	115
Step 11. View N, V, M Diagrams, Model Deformation and Free Body Diagram .....	116
Step 12. View the analytical results .....	125
<b>Chapter 8 License Agreement</b>	<b>132</b>
1 EULA (Lite Edition) .....	132
2 EULA (Pro Edition) .....	135

# Chapter

---

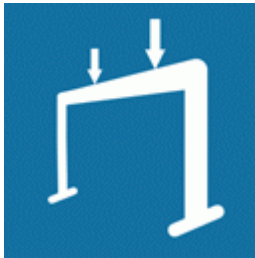


**Introduction to EngiLab Beam.2D**

# 1 Introduction to EngiLab Beam.2D

- [Overview](#)
- [Lite and Pro Editions](#)

## 1.1 Overview



### EngiLab Beam.2D 2015

**v2.2** (v2.2.5602.0)

Copyright (c) 2015 EngiLab SMPC.

All rights reserved



[www.engilab.com](http://www.engilab.com)

EngiLab Beam.2D is an easy-to-use yet powerful engineering tool for the linear static analysis of plane (2D) structures for Windows. It can analyze **continuous beams**, **plane frames** and **plane trusses**. The program features a full Graphical User Interface (GUI) for pre-processing or post-processing and uses the **Finite Element Method (FEM)** for its analysis purposes.

### Communication

For general inquiries, please contact us at [info@engilab.com](mailto:info@engilab.com)

### Technical support

EngiLab offers technical support via email at [support@engilab.com](mailto:support@engilab.com). Email questions are normally answered within 24 hours for registered users. Considering our customer support team's busy work schedule, EngiLab strives to provide technical support, via email, to non-registered (Lite Edition) users as well.

### Required Information for Support

When sending us a question via email, please make sure to include the following information:

- Operating system (Windows XP, Windows Vista, Windows 7, Windows 8, etc.)
- Name and exact version of the product being used.
- Brief description of the problem.
- Detailed instructions on how to duplicate the problem, and/or an input file as an attachment.

You may find additional information, new releases, etc on the web at [www.engilab.com](http://www.engilab.com). We take customer satisfaction very seriously and always welcome your feedback.



## Compatibility

EngiLab Beam.2D is compatible with the following versions of Windows (32 bit and 64 bit):

- Windows XP, Windows 2000, Windows Vista, Windows 7, Windows 8, Windows 8.1
- Windows Server 2003, Windows Server 2008, Windows Server 2008 R2, Windows Server 2012



## Limited Warranty - Disclaimer of Damages

See the [License Agreement](#).

## 1.2 Lite and Pro Editions

EngiLab Beam.2D comes in two Editions: **Lite** and **Pro**. The Lite Edition is completely free, but it has limitations compared to the full Pro Edition, as described in detail below.

### Lite Edition

- **Free of charge**, available for download at [www.engilab.com](http://www.engilab.com)
- Only for **non-profit use** (personal, academic or educational purposes). It may *not* be used for any commercial purposes.
- **Analysis limitations:** The Lite Edition can open, save and modify any structural Model. Its input files are exactly the same as the ones of the Pro Edition. Yet, the Lite Edition can only analyze small Models of up to **10 Nodes, 3 Materials and 3 Sections**.
- **No missing features:** You get the real experience of the Pro Edition. All the functions of the Pro Edition are present also in the Lite Edition. No trial version banners printed anywhere, no annoying pop-ups, no hidden fees.
- Ideal for: Students and Hobbyists who need to analyze small models, or Professional Engineers who would like to test the Program before purchasing the Pro Edition.
- Technical Support for the Lite Edition is only provided via email at [support@engilab.com](mailto:support@engilab.com)

## Pro Edition

- The Pro Edition is available for purchase at [www.engilab.com](http://www.engilab.com).
- It can be used for **any purpose** (personal, academic, educational or **commercial use**)
- **No modeling limitations:** The Pro Edition can analyze any structural model (Unlimited number of Nodes, Materials and Sections).
- Ideal for: Professional Engineers, Universities, Students, Researchers, Hobbyists who wish to analyze models without any limitations.
- **Priority Technical Support** is provided within 24 hours via email at [support@engilab.com](mailto:support@engilab.com), or by phone.

# Chapter

---



## Structural Modeling

## 2 Structural Modeling

- [System of Units](#)
- [Setting up a Model](#)
- [Model Properties](#)
  - [Materials](#)
  - [Sections](#)
  - [Nodes](#)
  - [Elements](#)
  - [Nodal Loads](#)
  - [Elemental Loads](#)
  - [Body \(Acceleration\) Loads](#)

### 2.1 System of Units

EngiLab Beam.2D has no default system of units. This is not a limitation of the program, but a deliberate choice in order for the program to work globally, no matter what system of units is used. This way there is no limitation in the system of units that can be used. **Any consistent system of units can be used.**

Before starting to define any model, you need to decide which system of units you will use. All input data must be specified in **consistent units**. As a result, the analysis results will also comply to that system. The important point about using consistent units is the necessity to stick with units that work correctly together - not to mix units that do not have a correct relationship with each other.

In order to define a consistent system of units, you have to define first the **primary (basic) units** and then the **derived units** which are dependent on the primary units. We propose two different approaches for defining a consistent system of units, as described in detail below:

### A. Consistent system of units based on Force

Define the three primary (basic) units for **Force (F)**, **Length (L)**, **Time (T)**. For example you can choose to use kN, m, s (sec). The **derived units** are then the following:

Derived Unit	Formula	Formula explanation	In our example (kN, m, s)
<b>Acceleration</b>	$L/T^2$	(1 Length unit) / (1 time unit) <sup>2</sup>	$m/s^2$
<b>Mass *</b>	$F \cdot T^2 / L$	(1 force unit) / (1 acceleration unit)	$kN / (m/s^2) = Mg = t *$
<b>Density</b>	$F \cdot T^2 / L^4$	(1 mass unit) / (1 length unit) <sup>3</sup>	$t/m^3$
<b>Stress</b>	$F/L^2$	(1 force unit) / (1 length unit) <sup>2</sup>	$kN/m^2$

\* The mass unit (in our example 1 t) is the mass that accelerates by the acceleration unit rate (in our example  $1 m/s^2$ ) when the unit force (in our example 1 kN) is exerted on it.

### B. Consistent system of units based on Mass

Define the three primary (basic) units for **Mass (M)**, **Length (L)**, **Time (T)**. For example you can choose to use kg, m, s (sec). The **derived units** are then the following:

Derived Unit	Formula	Formula explanation	In our example (kg, m, s)
<b>Acceleration</b>	$L/T^2$	(1 Length unit) / (1 time unit) <sup>2</sup>	$m/s^2$
<b>Force *</b>	$M \cdot L / T^2$	(1 mass unit) · (1 acceleration unit)	$kg \cdot m/s^2 = N *$
<b>Density</b>	$M/L^3$	(1 mass unit) / (1 length unit) <sup>3</sup>	$kg/m^3$
<b>Stress</b>	$M/L/T^2$	(1 force unit) / (1 length unit) <sup>2</sup>	$N/m^2$

\* The force unit (in our example 1 N) is the force required to accelerate the unit mass (in our example 1 kg) at the acceleration unit rate (in our example  $1 m/s^2$ ).

## Common consistent systems of units

Some common systems of consistent units are shown in the table below.

Quantity	SI (MKS)	MTS	mmNS	US Unit (ft)	US Unit (in)
Length	m	m	mm	ft	in
Force	N	kN	N	lbf	lbf
Mass	kg	t (tonne)	t	slug	lbf·s <sup>2</sup> /in
Time	s	s	s	s	s
Stress	Pa (N/m <sup>2</sup> )	kPa (kN/m <sup>2</sup> )	MPa	lbf/ft <sup>2</sup>	psi (lbf/in <sup>2</sup> )
Density	kg/m <sup>3</sup>	t/m <sup>3</sup>	t/mm <sup>3</sup>	slug/ft <sup>3</sup>	lbf·s <sup>2</sup> /in <sup>4</sup>
Acceleration	m/s <sup>2</sup>	m/s <sup>2</sup>	mm/s <sup>2</sup>	ft/s <sup>2</sup>	in/s <sup>2</sup>

As points of reference, the mass density of steel, the Young's Modulus of steel and the standard earth gravitational acceleration are given for each system in the table below.

Quantity	SI (MKS)	MTS	mmNS	US Unit (ft)	US Unit (in)
Steel density	7850 kg/m <sup>3</sup>	7.85 t/m <sup>3</sup>	7.85×10 <sup>-9</sup> t/mm <sup>3</sup>	15.23151461 slug/ft <sup>3</sup> (lbf·s <sup>2</sup> /ft <sup>4</sup> )	7.34544493×10 <sup>-4</sup> blob/in <sup>3</sup> (lbf·s <sup>2</sup> /in <sup>4</sup> )
Steel E	210×10 <sup>9</sup> N/m <sup>2</sup>	210×10 <sup>6</sup> kN/m <sup>2</sup>	210×10 <sup>3</sup> MPa	4,385,941,188.96153 lbf/ft <sup>2</sup>	30,457,924.9233 psi
Earth Gravity	9.80665 m/s <sup>2</sup>	9.80665 m/s <sup>2</sup>	9806.65 mm/s <sup>2</sup>	32.17404856 ft/s <sup>2</sup>	386.088583 in/s <sup>2</sup>

### Notes:

- 1 t (tonne) = 10<sup>3</sup> kg = 1 Mg. It is a mass that accelerates by 1 m/s<sup>2</sup> when a force of 1 kN is exerted on it.
- 1 slug = 1 lbf·s<sup>2</sup>/ft. It is a mass that accelerates by 1 ft/s<sup>2</sup> when a force of 1 pound-force (lbf) is exerted on it.
- 1 blob = 1 lbf·s<sup>2</sup>/in. It is a mass that accelerates by 1 in/s<sup>2</sup> when a force of 1 pound-force (lbf) is exerted on it.
- 1 MPa = 1 MN/m<sup>2</sup> = 1 N/mm<sup>2</sup>

### Practical example

The user chooses to use the **MTS system**, a common choice for structural engineering applications:

- Length: **m**
- Force: **kN**
- Time: **s**

EngiLab Beam.2D data have to be given as shown below:

<b>Quantity</b>	<b>Unit used</b>
Node Coordinates X, Y	m
Material Elastic Modulus E	kPa = kN/m <sup>2</sup>
Material Density d	t/m <sup>3</sup>
Section Area A	m <sup>2</sup>
Section Moment of Inertia I	m <sup>4</sup>
Nodal Force F	kN
Nodal Moment M	kN·m
Elemental load f	kN/m
Spring Elastic Stiffness KX, KY	kN/m
Spring Elastic constant KZ	kN·m (/RAD) *
Acceleration	m/s <sup>2</sup>

The results will also comply to that system, thus they will be given as:

<b>Quantity</b>	<b>Unit used</b>
Node X, Y Displacement	m
Node Z Rotation	RAD *
Axial, Shear Force at Element end i, j	kN
Moment at Element end i, j	kN·m
Support reaction FX, FY	kN
Support reaction MZ	kN·m

\* Rotations are ALWAYS given in RADIANS.

**Note:** In the above example, if one wants to apply self-weight to the structure, he can add the standard earth gravitational acceleration at the -y direction: **ay = -9.80665**

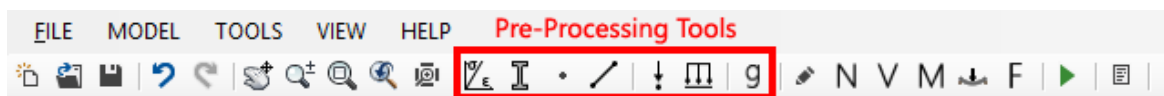
## 2.2 Setting up a model


The user can set-up, analyze and view the results of a model using the following **12 simple steps**:


### A. Preparation of the input data

**1.** Define the **System of Units**. This is a very important step for the model. This step has nothing to do with the program itself. Instead, the user has to define the preferred system of units and then all the program data have to be consistent with this system. Then the results will also comply to that system. For details, see [System of Units](#).

### B. Pre-Processing: Building the Model





**2.** Click  to define **Material(s)**. You need to define the Material **Name** (optional), Material **Density d** (optional) and Material **Elastic Modulus E** for each Material. For details, see [Materials](#).

**3.** Click  to define **Sections(s)**. You need to define the Section **Name** (optional), **Section Area A** and Section **Moment of Inertia I** for each Section. For details, see [Sections](#).

**4.** After you have defined at least one Material and one Section, you can **start drawing your Model on screen**, as follows:

- Left-click on screen to define a new free Node (no constraints).
- Left-click on screen, hold down the left button and then release it at another location to define a new Element and two nodes at ends i, j.
- Double-click on a Node or close to it to define a new Constraint (Support) set. Each time you double-click a Node, a new Nodal Constraint set is applied to the Node.
- If **SNAP** is activated (Default=True), then you can only draw Nodes and Elements at increments defined by the Snap Size setting (Default=0.1).
- If **SNAPNODE** is activated (Default=True), then you can "catch" Nodes so that new elements can be connected to existing Nodes.
- **Note:** All Elements that are defined on screen are assigned Material 1 and Section 1.

If you do not want to use on-screen drawing, you can still click  to define Nodes manually and  to define Elements manually.

**5.** Click  to:



- Move **Nodes** to their exact positions, if needed. For example, a Node with X-Coordinate 5.8 defined on screen with Snap Size = 0.1 should be moved to the exact position 5.85.
- Define or change Nodal Constraints.
- Define Springs.

For details, see [Nodes](#).

**6.** Click  to:

- Assign the right Material and Section to every **Element**, if needed. All Elements that are defined on screen are assigned Material 1 and Section 1. For example, an Element that has been defined on screen has to be assigned Material 2 or Section 2.
- Define Hinges at Element ends. Each Element can have a hinge at either end (Start-i or End-j or both).

For details, see [Elements](#).

**7.** Click  to define **Nodal Loads**. For details, see [Nodal Loads](#).

**8.** Click  to define **Elemental Loads**. For details, see [Elemental Loads](#).

**9.** Click  to define **Body (Acceleration) Loads** (if needed). For details, see [Body \(Acceleration\) Loads](#).

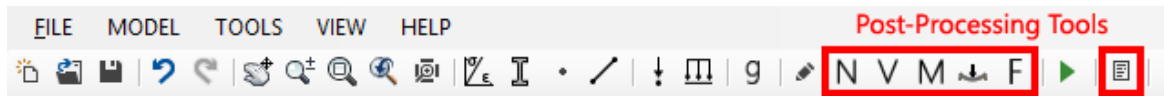
- If you want to take into account the self-weight of Elements as an additional elemental load for each Element, then you have to provide the Material Density for the Material of each Element and also to define a Linear Acceleration Vector equal to the standard earth gravitational acceleration. A common practice is to put the earth gravitational acceleration with a minus (-) sign at the Y direction - this means gravity acting towards -Y global axis.
- **Example: If you are using kN for forces, m for length and s for time, then the Material Density has to be given in t/m<sup>3</sup> and you have to enter -9.80665 (or simply -9.81) at the aY component of the Linear Acceleration Vector.**

## C. Analyze the Model



**10.** Click  (or press F5) to **run the Finite Element Analysis**. For details on the Analysis, see [Analysis](#).

## D. Post-Processing: See the Analysis Results



**11.** Click N, V, M, ↓ or F to see the **Axial Force Diagram**, **Shear Force Diagram**, **Bending Moment Diagram**, **Model Deformation** or the **Free Body Diagram**.

- N: [Axial Force Diagram](#)
- V: [Shear Force Diagram](#)
- M: [Bending Moment Diagram](#)
- ↓: [Deformation](#) (deformed shape of the Model)
- F: [Free Body Diagram](#) (external forces together with support reactions)

**12.** Click  to see the **analytical results**. The analytical results include:

- [Node displacements](#)
- [Element forces](#)
- [Support reactions](#)
- [Full Analysis Report in Rich Text Format \(RTF\)](#)
- [Analysis validation](#)

## 2.3 Model Properties

- [Materials](#)
- [Sections](#)
- [Nodes](#)
- [Elements](#)
- [Nodal Loads](#)
- [Elemental Loads](#)
- [Body \(Acceleration\) Loads](#)

### 2.3.1 Materials



Each element is assigned a **Material**. The Material properties are the following:

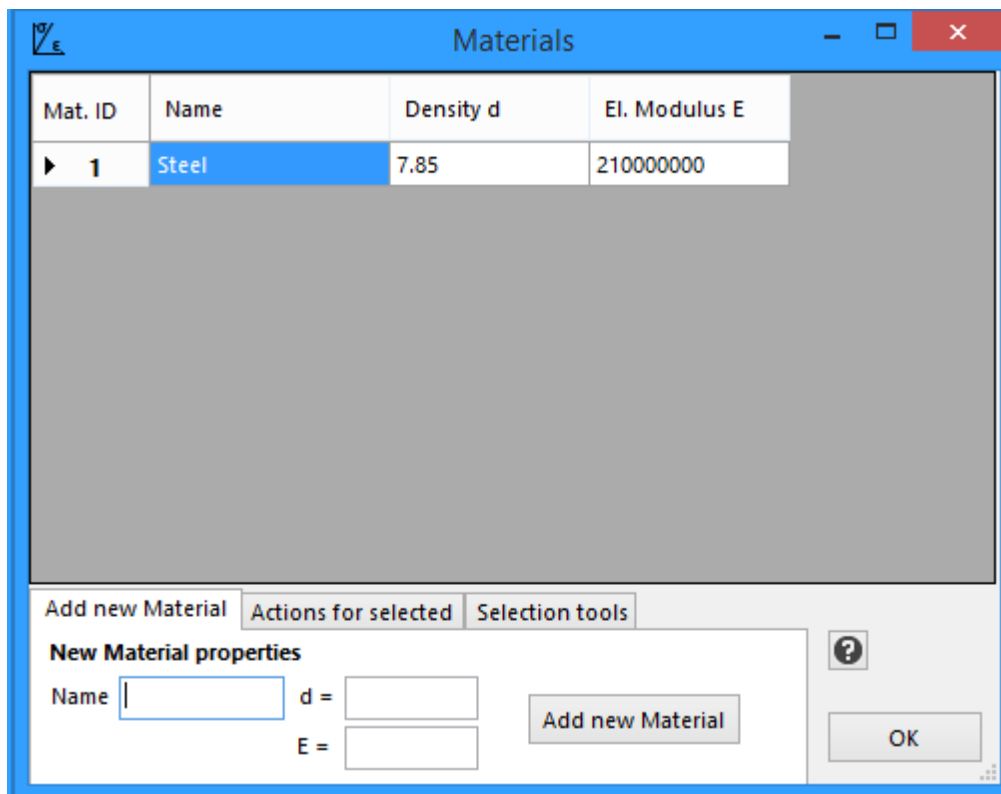
- **Name** (optional, max. number of characters in Name: 20)
- **Density d** (optional, can be omitted, see below)

- **Elastic Modulus E**

The **Elastic Modulus E** of the Material is an important parameter affecting the Model Stiffness Matrix.

The **Density** is used in order for the program to calculate the Body Loads due to Linear Acceleration. For example, if you want to take into account the self-weight of Elements as an additional elemental load for each Element, then you have to provide the Material Density for the Material of each Element and also to define a Linear Acceleration Vector equal to the standard earth gravitational acceleration. If Density is omitted or it is equal to zero for a Material, then Elements that are assigned this Material will not take any Body (Acceleration) Loads even if a Linear Acceleration Vector is defined. By setting Density equal to zero for a Material, you can model Weightless Elements.

**See also:** [Body \(Acceleration\) Loads](#).



### Add new Material

To add a new Material, type the Name (optional), Density (optional) and Elastic Modulus E of the Material and click the "Add new Material" button.

### Edit an existing Material

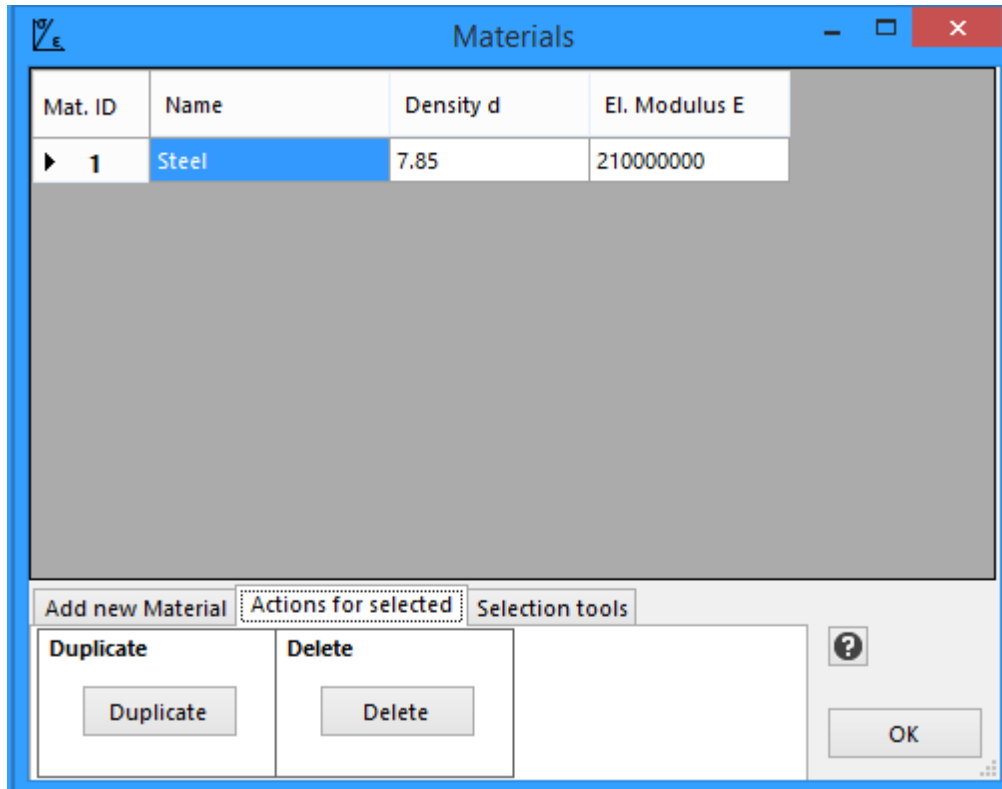
You can click on the table and you can easily edit the properties of an already existing

Material. Any change you make is automatically reflected to the Model.

### Actions for selected Materials

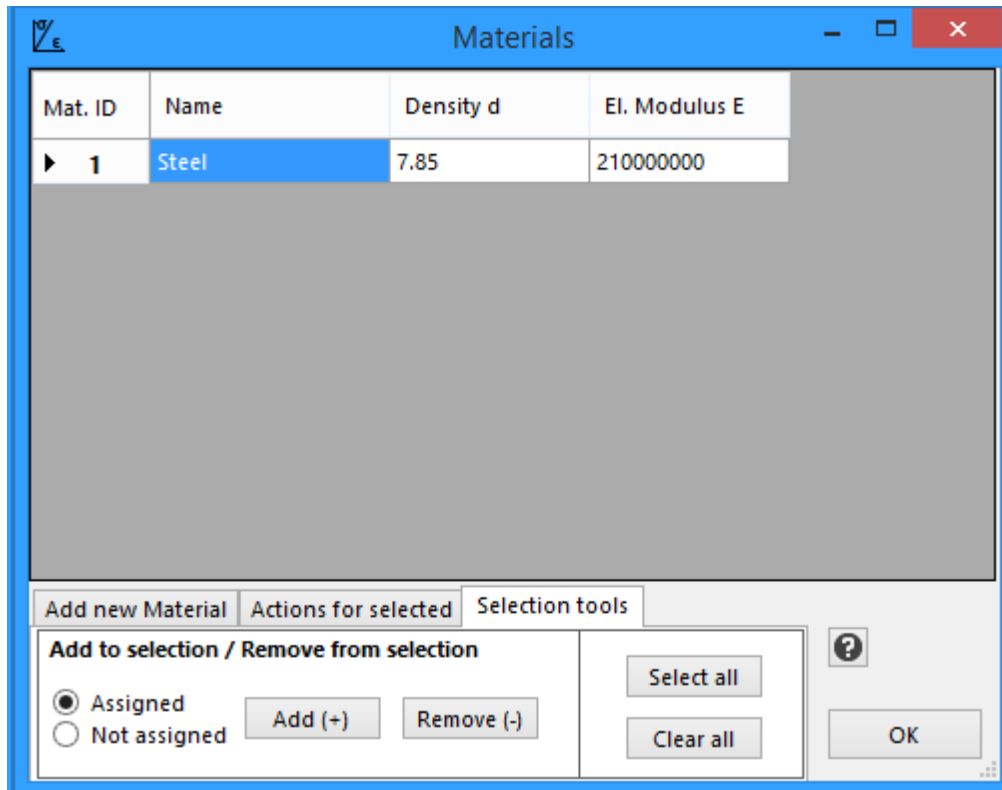
In order to perform these actions, **you have to first select the entire row(s) of the corresponding Material(s)** and then apply the action.

- **Duplicate Material(s)**. Click the "Duplicate" button. You can Duplicate more than one Materials at a time.
- **Delete Material(s)**. Click the "Delete" button. You can Delete more than one Materials at a time.

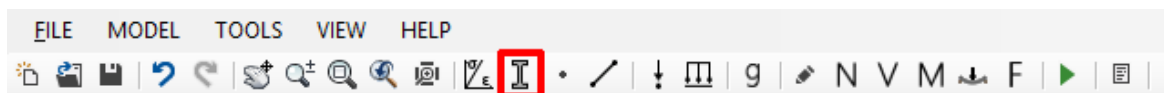


## Materials selection tools

This tab provides tools for the selection of Materials. For example, you can select (add to selection) or deselect (remove from selection) all the Materials that are assigned to Elements, or all the Materials that are NOT assigned to Elements.



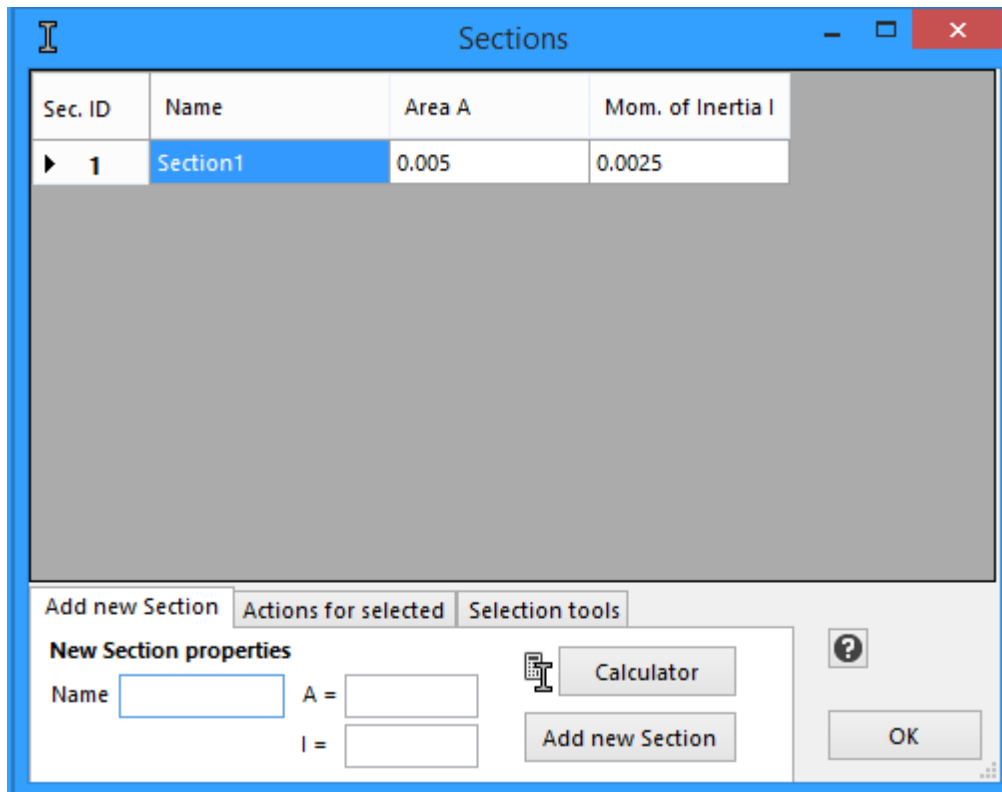
### 2.3.2 Sections



Each element is assigned a **Section**. The Section properties are the following:

- **Name** (optional, max. number of characters in Name: 20)
- **Area A**
- **Moment of Inertia I**

The **Area A** and the **Moment of Inertia I** of the Section are important parameters affecting the Model Stiffness Matrix.



### Add new Section

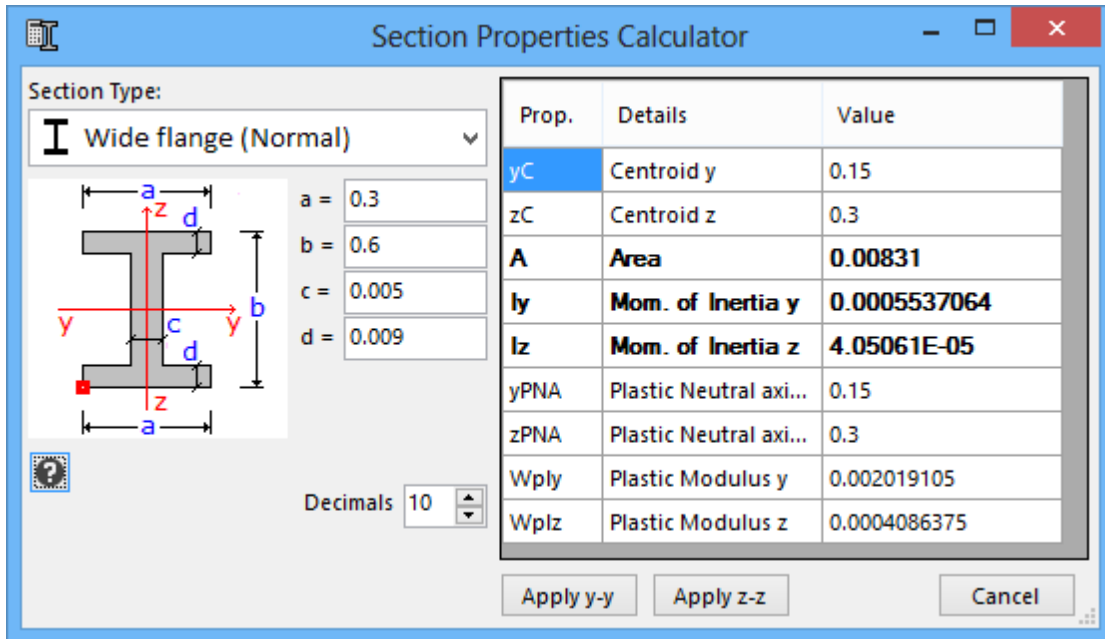
To add a new Section, type the Name (optional), Area A and Moment of Inertia I of the Section and click the "Add new Section" button.

### Edit an existing Section

You can click on the table and you can easily edit the properties of an already existing Section. Any change you make is automatically reflected to the Model.

## Section Properties Calculator

By clicking the "Calculator" button, a new form appears, as shown below. You can use the Section Properties Calculator to calculate the properties of various section shapes. The various section shapes are shown in the right Picture below. The Calculator calculates various properties. The program needs only to take the Area A and Moment of Inertia I property. Click "Apply y-y" to apply the Area A and the  $I_y$  Moment of Inertia. Click "Apply z-z" to apply the Area A and the  $I_z$  Moment of Inertia.



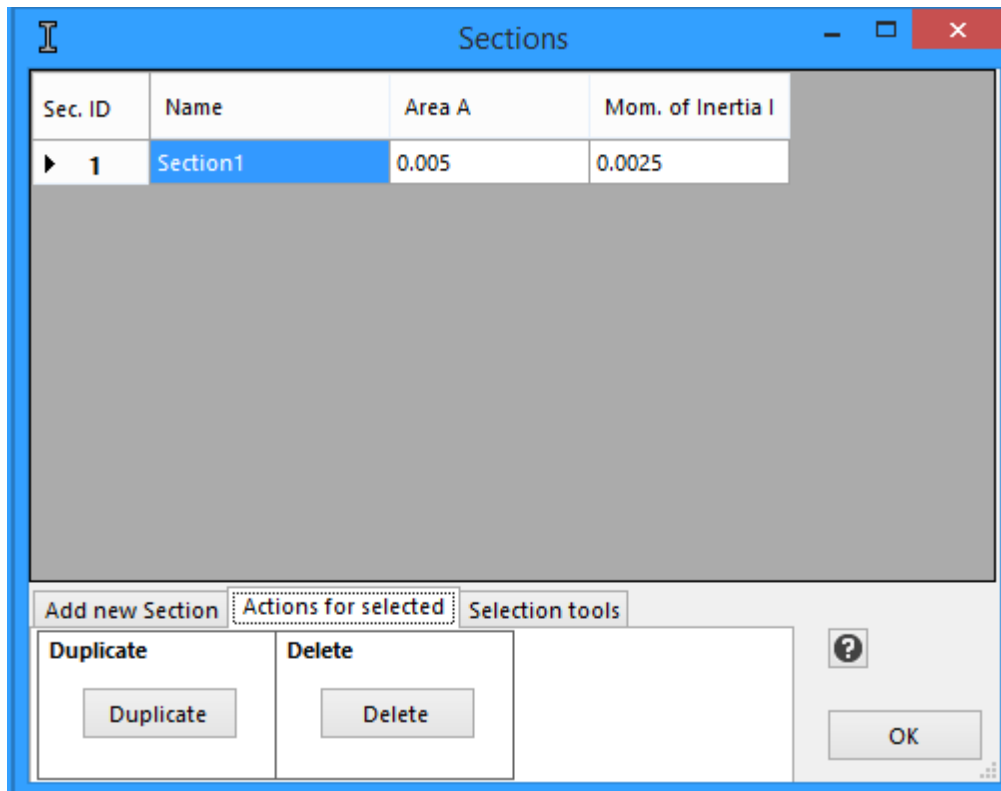
The available sections are the following:

- Square bar
- Square tube
- Rectangular bar
- Rectangular tube
- Circular bar
- Circular tube
- + Cross (Normal)
- + Cross (General)
- I Wide flange (Normal)
- I Wide flange (General)
- ⊥ T-Shape
- C C-Shape (Normal)
- C C-Shape (General)
- L L-Shape (Normal)
- L L-Shape (General)

## Actions for selected Sections

In order to perform these actions, **you have to first select the entire row(s) of the corresponding Section(s)** and then apply the action.

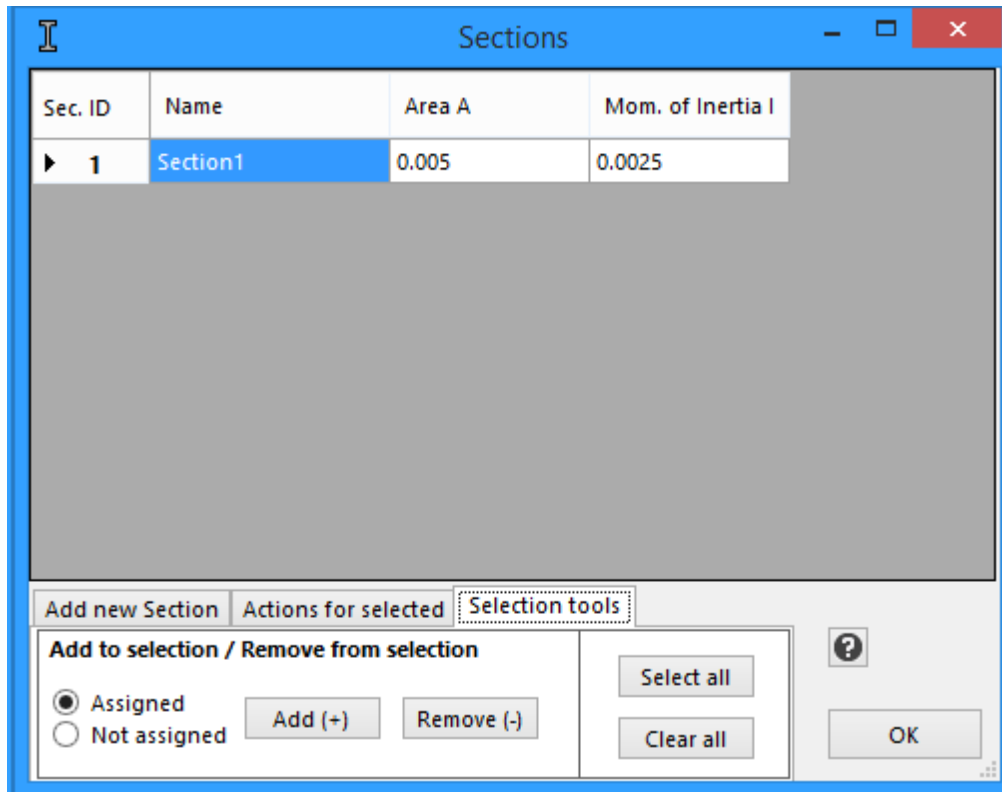
- **Duplicate Section(s)**. Click the "Duplicate" button. You can Duplicate more than one Sections at a time.
- **Delete Section(s)**. Click the "Delete" button. You can Delete more than one Sections at a time.





## Sections selection tools




This tab provides tools for the selection of Sections. For example, you can select (add to selection) or deselect (remove from selection) all the Sections that are assigned to Elements, or all the Sections that are NOT assigned to Elements.



### 2.3.3 Nodes



**Nodes** connect Elements. Each Element has a Start Node (i) and an End Node (j). Each node is located at a (**X, Y**) Point (Global Axes) in the 2D plane and it has 3 Degrees Of Freedom (DOFs):

-  X-Displacement
-  Y-Displacement
-  Z-Rotation

#### Each DOF can be:

- Free to move (No support = No , No spring)
- Fixed (Constrained, not able to move at all)
- With spring on (Given the spring stiffness, the spring provides a reaction force that is proportional to the deflection of the corresponding DOF)

## Node properties

The Node properties are the following:

- **Coordinates: X, Y**
- **Constraints: DX-Con, DY-Con, RZ-Con** (Each checked or Not)
- **Springs stiffness: DX-Stiff, DY-Stiff, RZ-Stiff**

Node ID	X	Y	DX-Con	DY-Con	RZ-Con	DX-Stiff	DY-Stiff	RZ-Stiff
▶ 1	0	0	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0	0	0
2	0	3	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
3	5	3	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
4	5	0	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0	0	0
5	10	3	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
6	10	0	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0	0	0
7	0	6	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
8	5	6	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
9	10	6	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0

**New Node coordinates**

X=

Y=

### Coordinates

Each node is located at a **(X, Y)** Point (Global Axes) in the 2D plane.






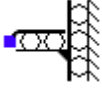


### Constraints

The properties **DX-Con**, **DY-Con**, **RZ-Con** determine whether the corresponding DOF of the Node is constrained or not.

Each DOF can be:

- Free (0 - Unchecked), or
- Fully constrained (1 - Checked).

As a result, there are in total 8 types of Nodes:

<b>Picture</b>	<b><u>DX-Constraint</u></b>	<b><u>DY-Constraint</u></b>	<b><u>RZ-Constraint</u></b>	<b><u>Description</u></b>
	NO	NO	NO	Free Node (000)
	✓	NO	NO	y-Roller (100)
	NO	✓	NO	x-Roller (010)
	✓	✓	NO	Pinned (110)
	✓	✓	✓	Fixed (111)
	NO	NO	✓	(001)
	✓	NO	✓	(101)
	NO	✓	✓	(011)

### Springs

The properties **DX-Stiff**, **DY-Stiff**, **RZ-Stiff** determine the Stiffness of the Spring (Elastic constant) of the corresponding DOF. A spring provides a spring reaction force that is proportional to the deflection at the corresponding DOF.

**Note:** A DOF that has a spring should not be constrained, as there is no point in that. In any case, if for a given DOF there is a spring and it is also constrained, then the spring is ignored and the DOF is handled as constrained.

### Add new Node

To add a new Node, type the X and Y Coordinates of the Node and click the "Add new Node" button.

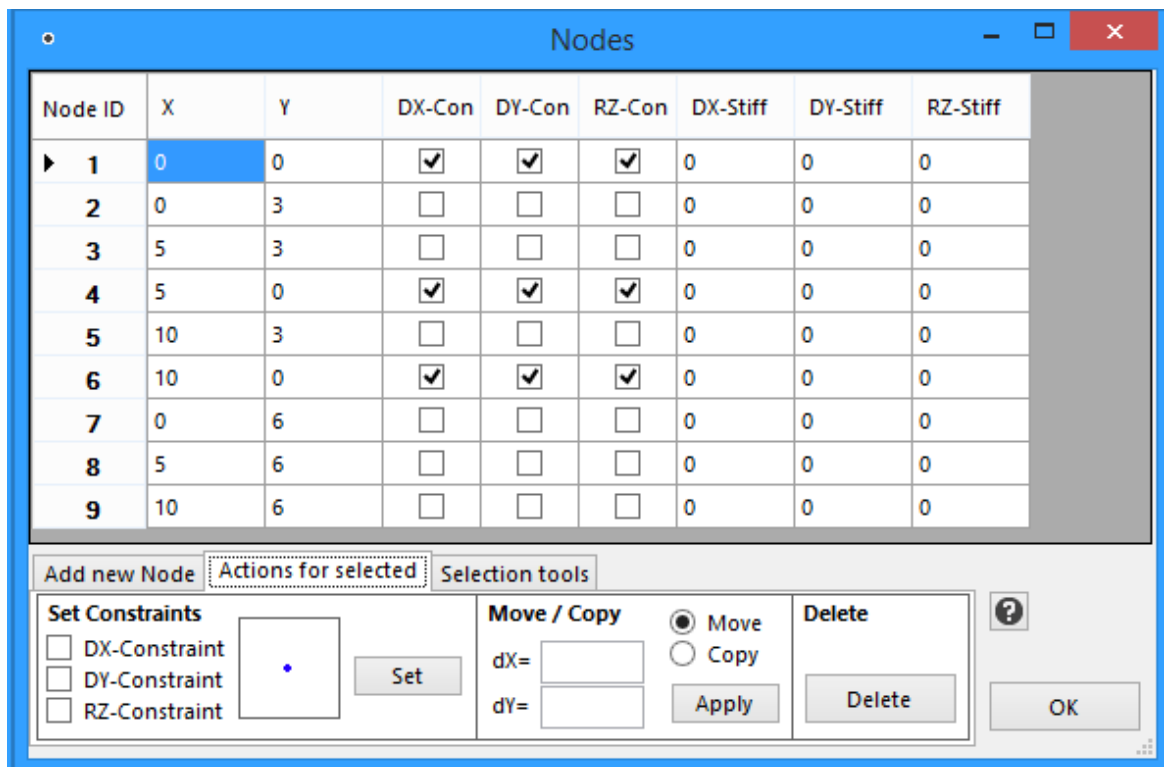
### Edit an existing Node

You can click on the table and you can easily edit the properties of an already existing Node. Any change you make is automatically reflected to the Model.

## Actions for selected Nodes

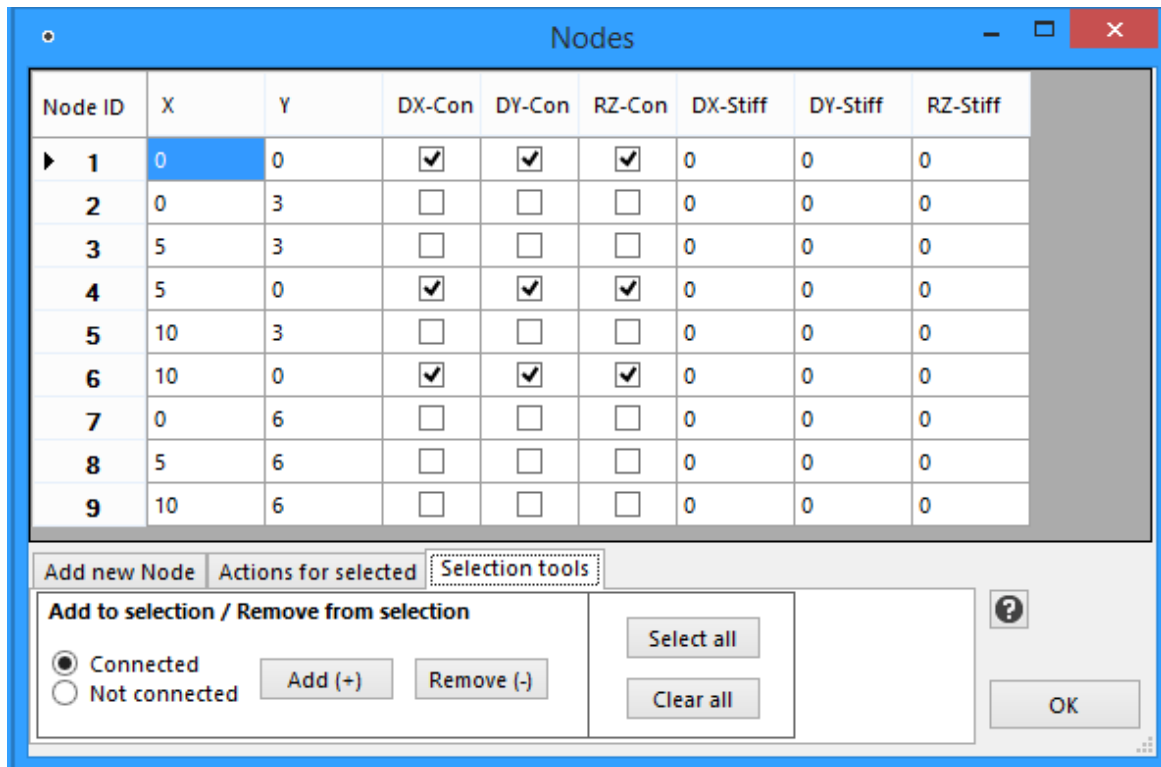
In order to perform these actions, **you have to first select the entire row(s) of the corresponding Node(s)** and then apply the action.

- **Set Constraints:** Select the proper constraints and then click the "Set" button. You can Set Constraints for more than one Nodes at a time.
- **Move or Copy Node(s):** Give the Move/Copy Vector  $\{V\}=\{dX, dY\}$ , select "Move" or "Copy" and then click the "Apply" button. You can Move or Copy more than one Nodes at a time.
- **Delete Node(s):** Click the "Delete" button. You can Delete more than one Nodes at a time.



## Nodes selection tools

This tab provides tools for the selection of Nodes. For example, you can select (add to selection) or deselect (remove from selection) all the Nodes that are connected to Elements, or all the Nodes that are NOT connected to Elements. This is useful when for example you need to Delete all the Nodes of the Model that are not connected to Elements.



### 2.3.4 Elements



An Element connects two Nodes with each other. It can have rotation releases (hinges) at each one of its ends (start  $i$  and/or end  $j$ ) and it has to be assigned a Material and a Section.

The **Element** properties are the following:

- **Material**
- **Section**
- **Nodes: Node  $i$**  (Start Node), **Node  $j$**  (End Node)
- **Hinges: Hinge  $i$**  (Hinge at Start Node), **Hinge  $j$**  (Hinge at End Node), each Checked or Not Checked
- **Length:** Length of the Element (**Read-only property**, calculated automatically by the program based on the locations of the Start and End Nodes)

Elem. ID	Material	Section	Node i	Node j	Hinge i	Hinge j	Length
▶ 1	1. Concrete	1. Columns	1	2	<input type="checkbox"/>	<input type="checkbox"/>	3.00
2	1. Concrete	2. Beams	2	3	<input type="checkbox"/>	<input type="checkbox"/>	5.00
3	1. Concrete	1. Columns	3	4	<input type="checkbox"/>	<input type="checkbox"/>	3.00
4	1. Concrete	2. Beams	3	5	<input type="checkbox"/>	<input type="checkbox"/>	5.00
5	1. Concrete	1. Columns	5	6	<input type="checkbox"/>	<input type="checkbox"/>	3.00
6	1. Concrete	1. Columns	2	7	<input type="checkbox"/>	<input type="checkbox"/>	3.00
7	1. Concrete	2. Beams	7	8	<input type="checkbox"/>	<input type="checkbox"/>	5.00
8	1. Concrete	1. Columns	8	3	<input type="checkbox"/>	<input type="checkbox"/>	3.00
9	1. Concrete	2. Beams	8	9	<input type="checkbox"/>	<input type="checkbox"/>	5.00
10	1. Concrete	1. Columns	9	5	<input type="checkbox"/>	<input type="checkbox"/>	3.00

Add new Element    Actions for selected    Selection tools

**New Element properties**

Material: 1. Concrete    Node i:     Node j:

Section: 1. Columns     i    Hinges     j

Add new Element    OK

### Add new Element

To add a new Element, select the Material and the Section and type the Start and End Nodes of the new Element. You can optionally add hinges to the new Element. Then click the "Add new Element" button.

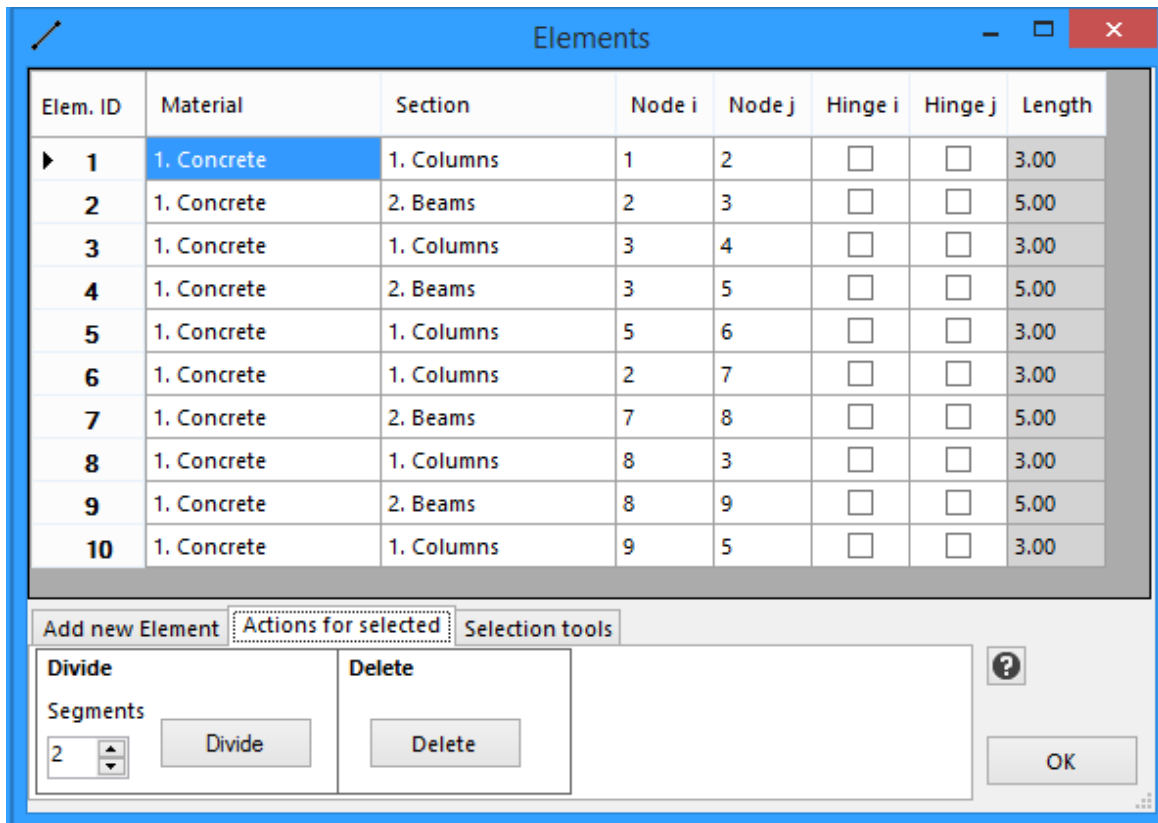
### Edit an existing Element

You can click on the table and you can easily edit the properties of an already existing Element. Any change you make is automatically reflected to the Model.

## Actions for selected Elements

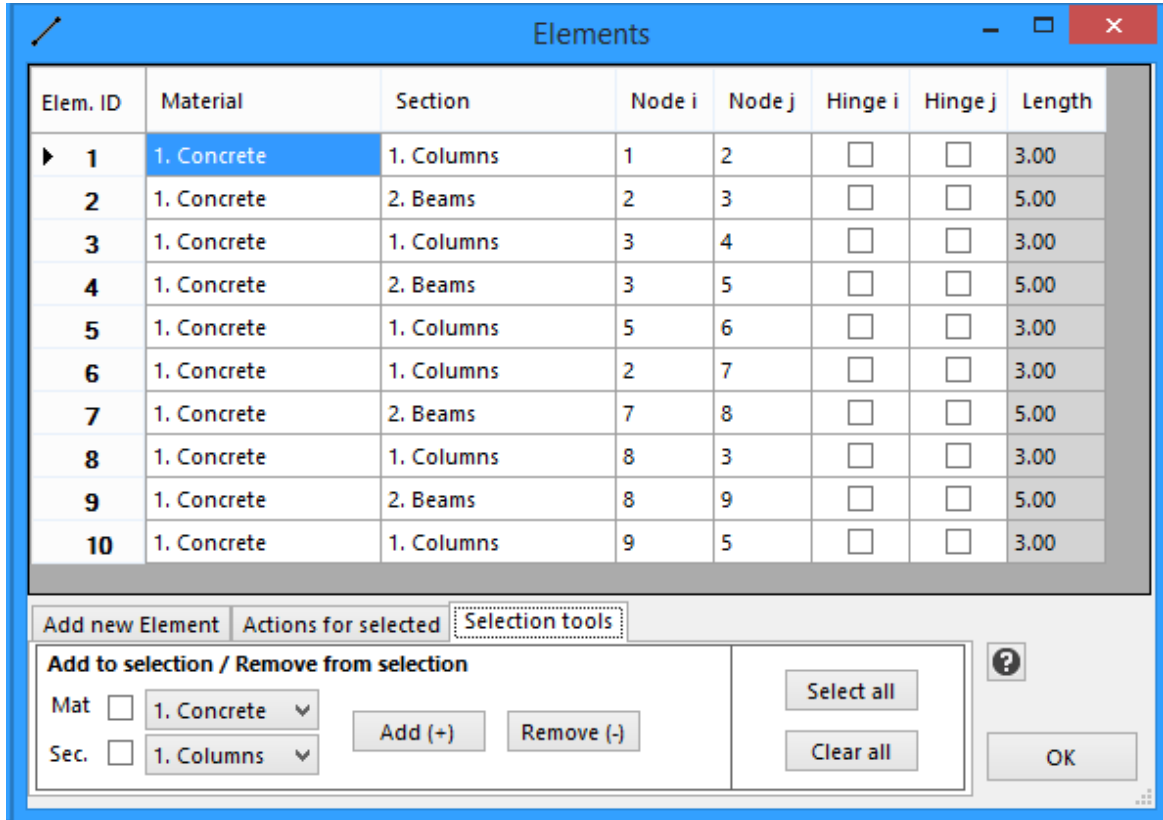
In order to perform these actions, **you have to first select the entire row(s) of the corresponding Element(s)** and then apply the action.

- **Divide Element(s):** Type the division segments and then click the "Divide" button. You can Divide more than one Elements at a time.
- **Delete Element(s):** Click the "Delete" button. You can Delete more than one Elements at a time.



## Elements selection tools

This tab provides tools for the selection of Elements. For example, you can select (add to selection) or deselect (remove from selection) all the Elements that are assigned a specific Material and/or Section.



### 2.3.5 Nodal Loads

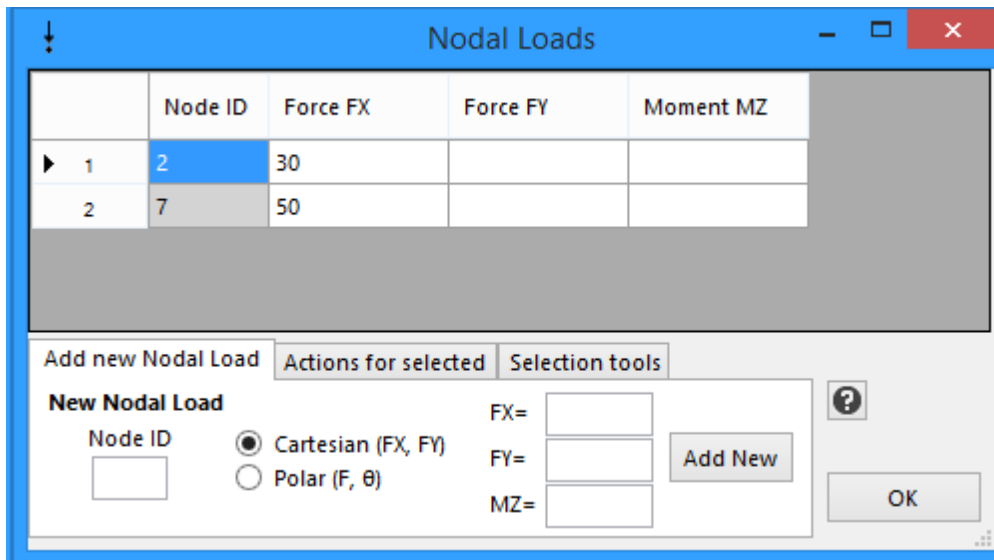


Point loads can only be NODAL loads (acting on Nodes). To define a load (FX, FY and/or MZ) at a specific point, first a Node must have been defined at that location.

The **Nodal Load** properties are the Loads in each Degree Of Freedom (DOF) of the Node (Sign convention: Global axes):

- **Force FX**
- **Force FY**
- **Bending Moment MZ**





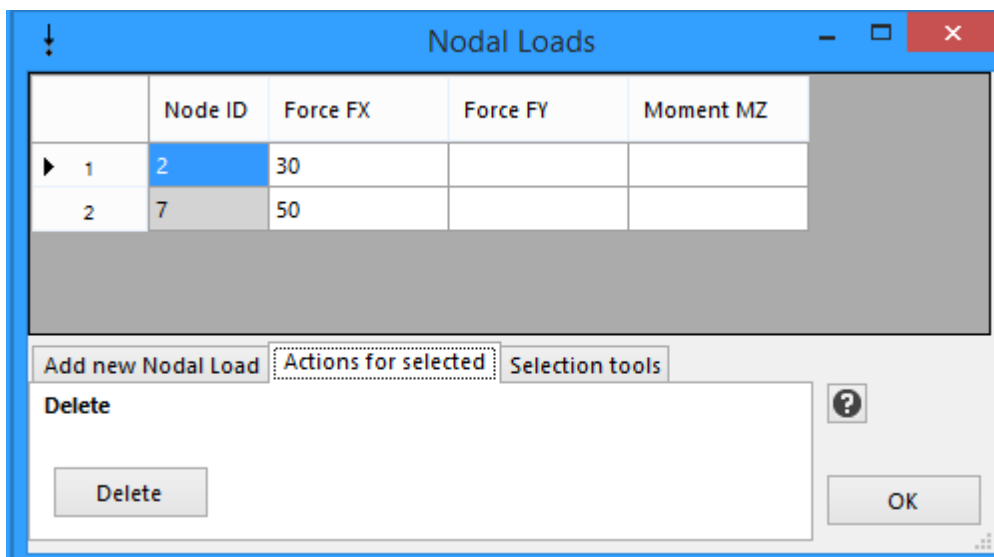
### Add new Nodal Load

To add a new Nodal Load, specify the Node ID and then type the FX, FY and MZ values of the Nodal Load set. Then click the "Add Nodal Load" button.

### Actions for selected Nodal Loads

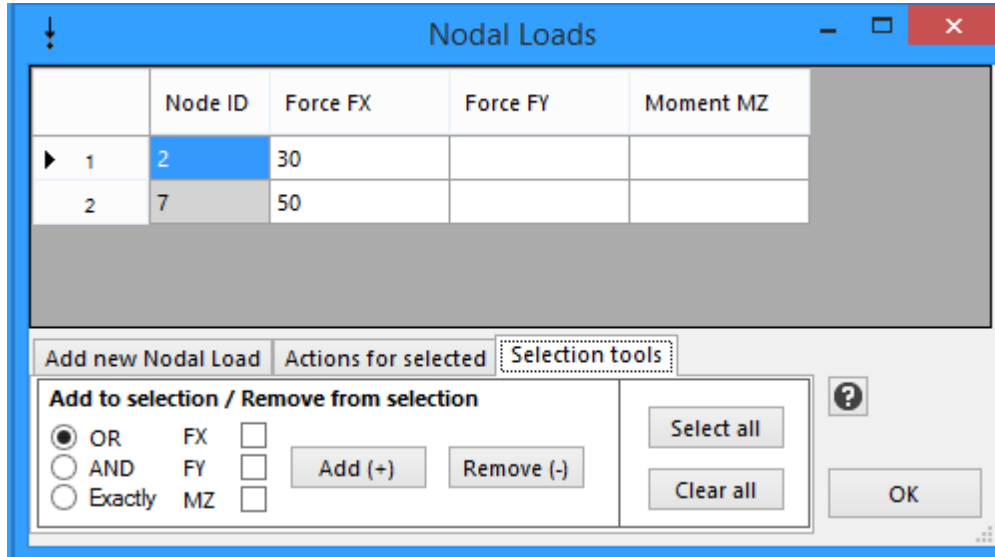
In order to perform these actions, **you have to first select the entire row(s) of the corresponding Nodal Load(s)** and then apply the action.

- **Delete Nodal Load(s)**. Click the "Delete" button. You can Delete more than one Nodal Loads at a time.

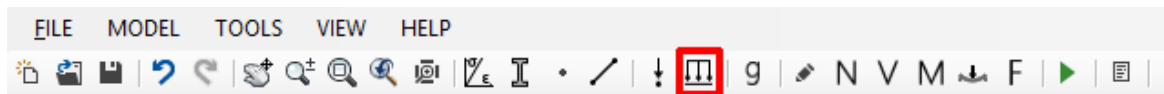


## Nodal Loads selection tools

This tab provides tools for the selection of Nodal Loads. For example, you can select (add to selection) or deselect (remove from selection) all the Nodal Loads that have FX, FY and/or MZ values.





### 2.3.6 Elemental Loads





The **Elemental Load** properties are the Loads acting at the X and/or Y directions (in Global axes system) along an Element. EngiLab Beam.2D supports linear varying loads along Elements. Uniform and triangular loads can be considered as special cases of the more general linear varying load case. The Elemental Loads are applied along the Element and must be given as Force per Unit Length of the element. The user specifies the start and end value of the Elemental force, per unit length of the element.

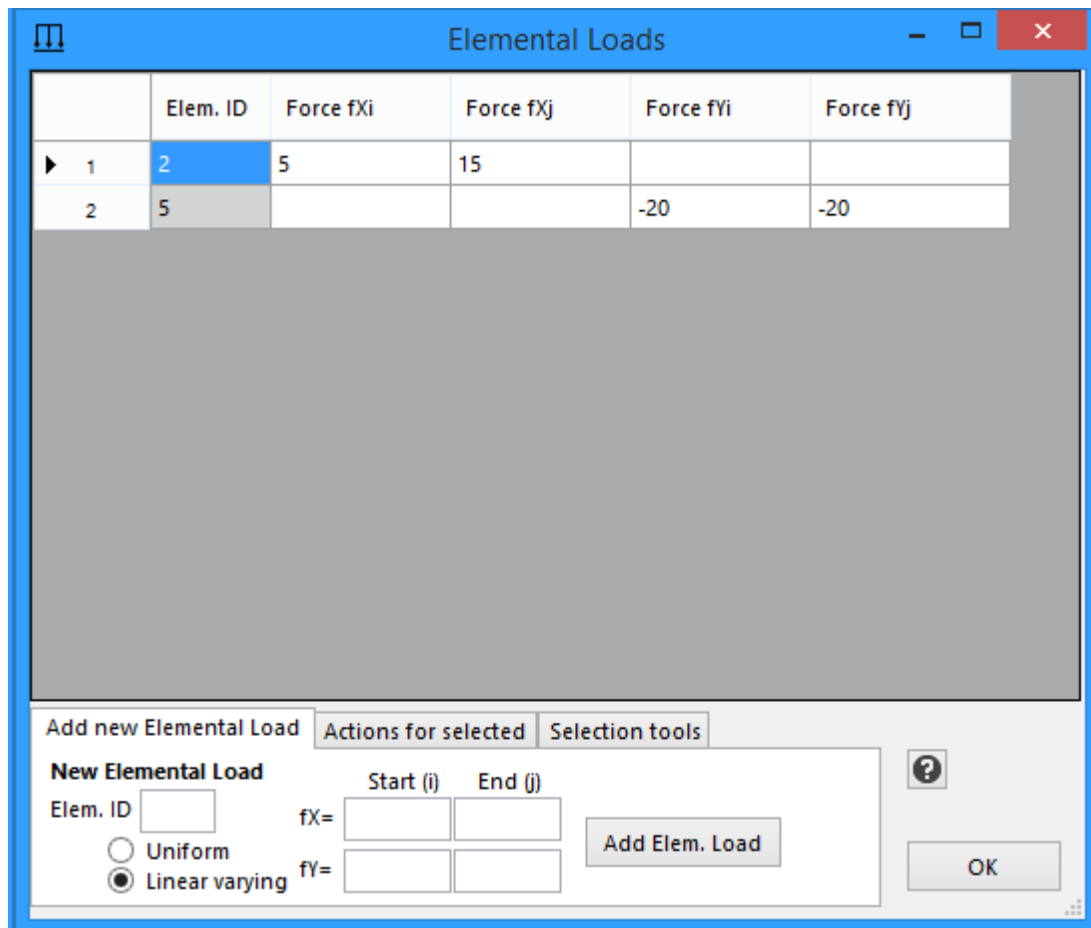
The Elemental Load properties are the following:

#### Elemental Load in X-Direction

-  **Force  $f_{Xi}$** : Start value (at start  $i$ ) of the X-Linear varying Load acting along the element
-  **Force  $f_{Xj}$** : End value (at end  $j$ ) of the X-Linear varying Load acting along the element

#### Elemental Load in Y-Direction

-  **Force  $f_{Yi}$** : Start value (at start  $i$ ) of the Y-Linear varying Load acting along the element
-  **Force  $f_{Yj}$** : End value (at end  $j$ ) of the Y-Linear varying Load acting along the element



#### Add new Elemental Load

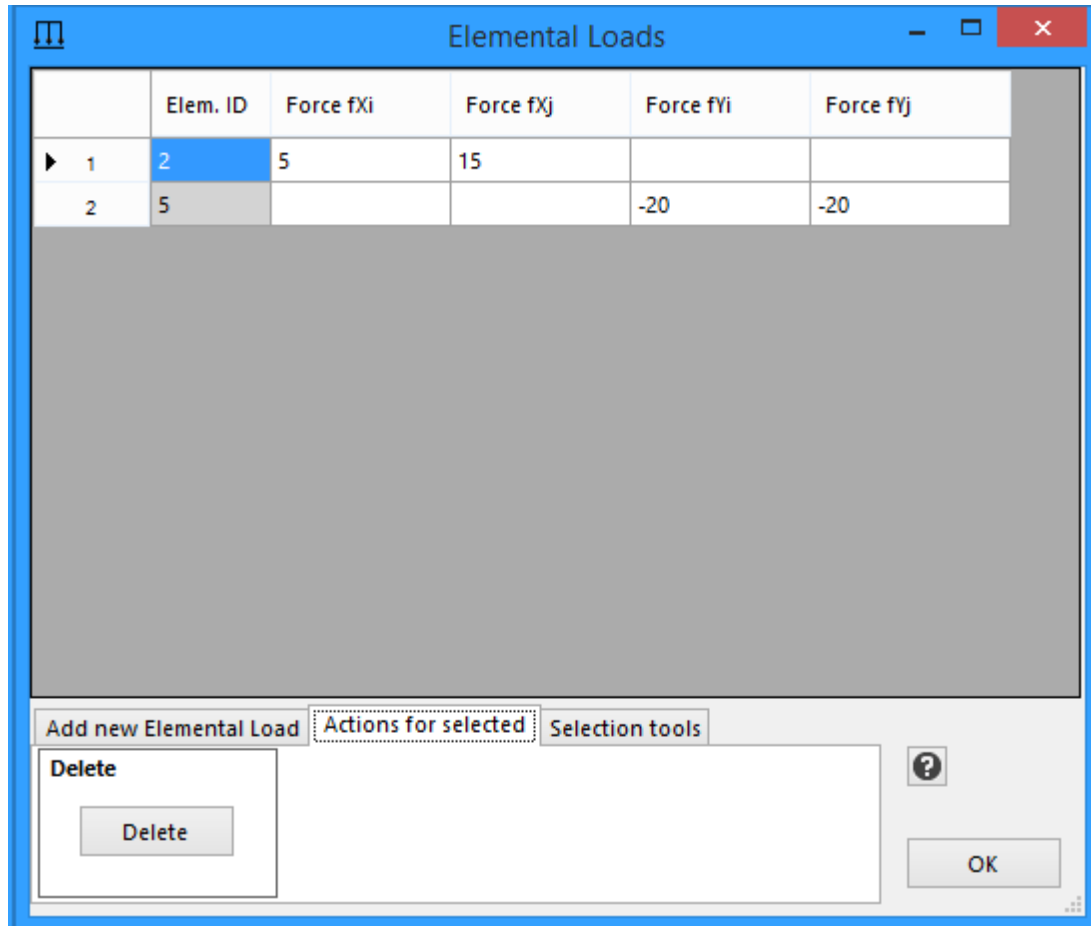
To add a new Elemental Load, first specify the Element ID. Then select Uniform or Linear varying Load. In general, all Elemental Loads are considered as linear varying loads by the program, but if you select a uniform load then the program will ask only for two values  $f_X=f_{Xi}=f_{Xj}$  and  $f_Y=f_{Yi}=f_{Yj}$ . If you select Linear varying load, then the

program will ask for all four values,  $f_{Xi}$ ,  $f_{Xj}$ ,  $f_{Yi}$  and  $f_{Yj}$ . After you have finished with the loads, then click the "Add Elemental Load" button.

### Actions for selected Elemental Loads

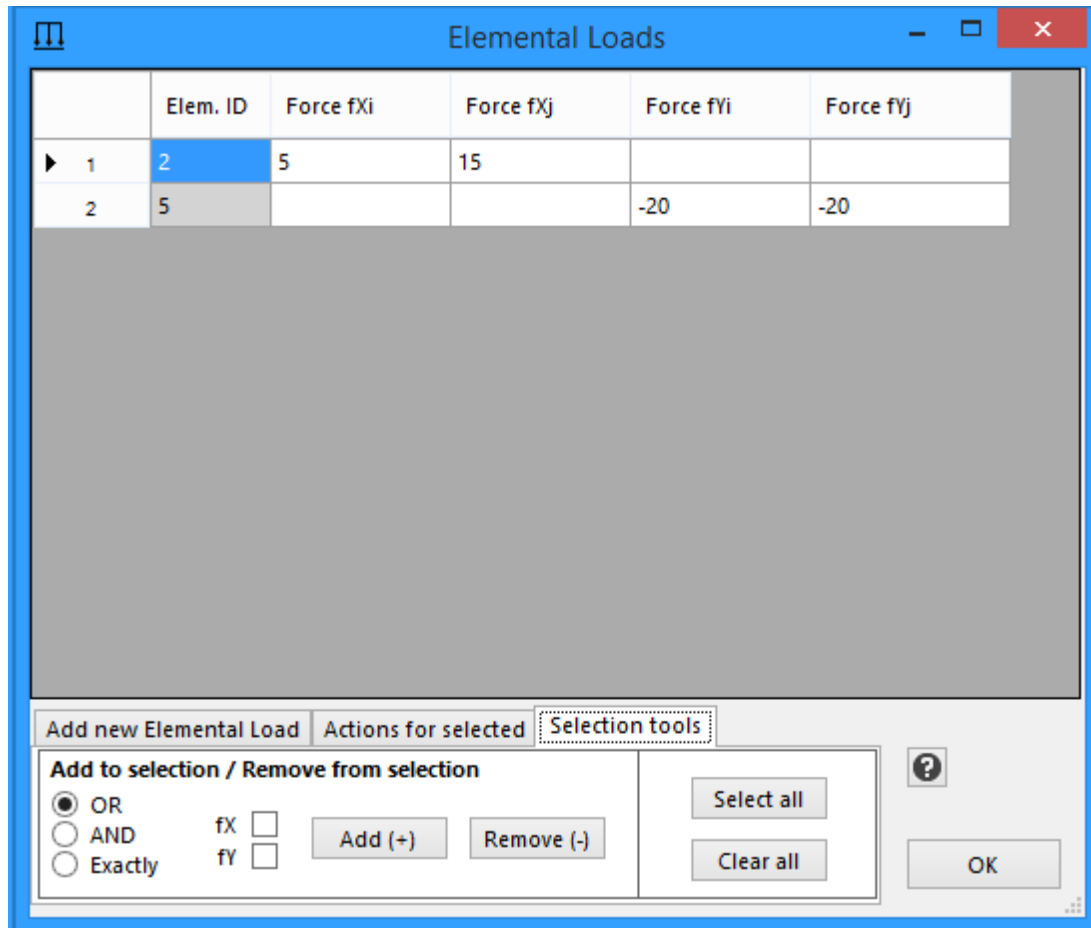
In order to perform these actions, **you have to first select the entire row(s) of the corresponding Elemental Load(s)** and then apply the action.

- **Delete Elemental Load(s)**. Click the "Delete" button. You can Delete more than one Elemental Loads at a time.



## Elemental Loads selection tools

This tab provides tools for the selection of Elemental Loads. For example, you can select (add to selection) or deselect (remove from selection) all the Elemental Loads that have  $f_x$  and/or  $f_y$  values.

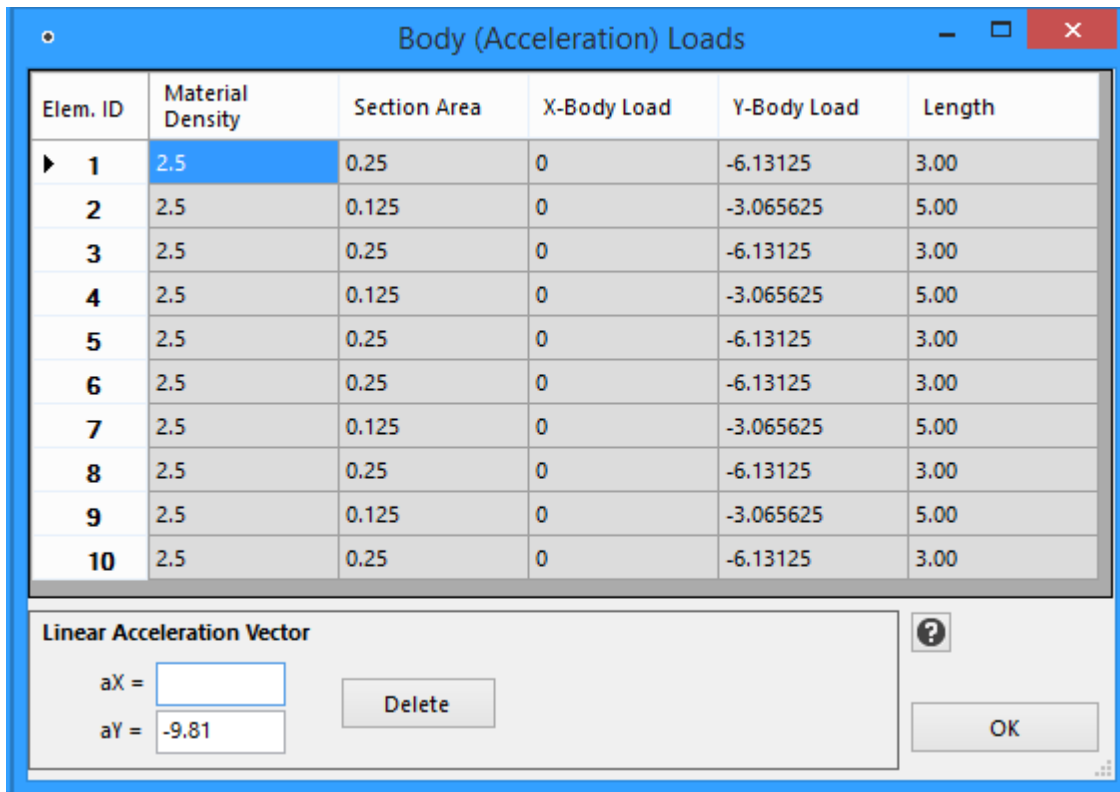


### 2.3.7 Body (Acceleration) Loads



You can specify the X and Y components of an Acceleration vector acting on the Model. If the Model has Mass (Non-zero Density is defined for the Model's Material), then the acceleration will cause inertial body (Elemental) forces on the Elements of the Model, according to Newton's Second Law of Motion. These elemental forces are calculated automatically by the program and the results are shown on the form.

To apply an acceleration vector, type the values for the  $a_X$  and  $a_Y$  acceleration components and **press the ENTER on the keyboard**. The model is updated automatically.



## Applications

### 1. Define the self-weight of the Model or the self-weight of specific Elements

If you want to take into account the self-weight of Elements as an additional elemental load for each Element, then you have to provide the Material Density for the Material of each Element (See [Materials](#)) and also to define a Linear Acceleration Vector equal to the standard earth gravitational acceleration. A common practice is to put the earth gravitational acceleration with a minus (-) sign at the Y direction - this means gravity acting towards -Y global axis. **See also** [System of Units](#).

**Example:** If you are using kN for forces, m for length and s for time, then the Material Density has to be given in  $t/m^3$  and you have to input -9.80665 (or simply -9.81) at the aY component of the Linear Acceleration Vector. This way, the self-weight will be applied to all Elements of the Model that have Mass (i.e. are assigned a Material which has Density).

### 2. Define forces acting on a moving (accelerating) object

Acceleration is not only Gravity. If an object accelerates, then inertial forces are applied on it, provided that the object has Mass. For example, you can analyze a structure that is fixed on a moving (accelerating) vehicle. Even if there are no other loads, the acceleration of the vehicle will cause inertial forces on the object. By providing the X and Y components of the acceleration vector, you can calculate these forces and analyze the Model.

# Chapter

---

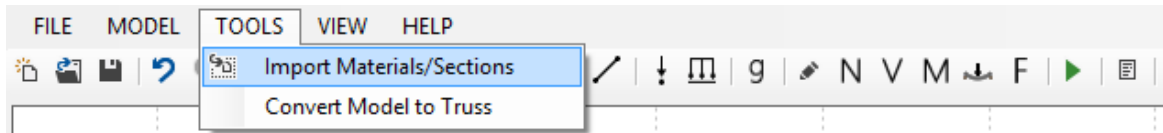


**Tools**

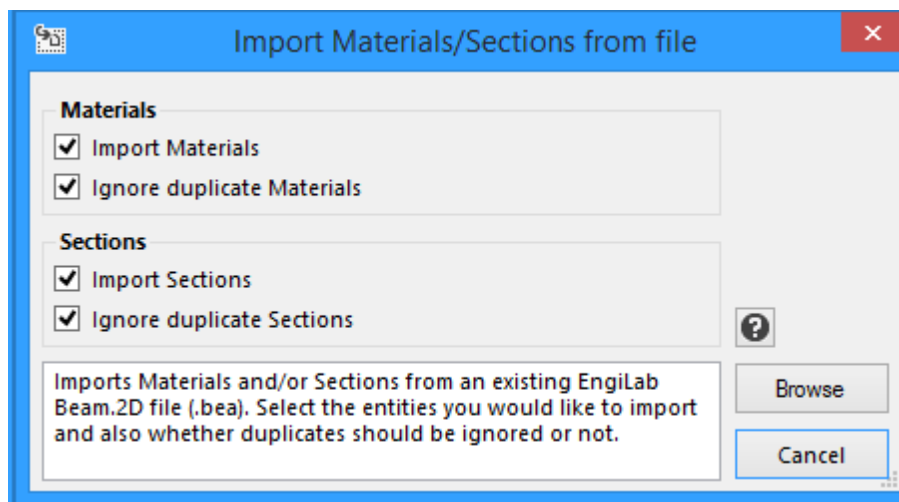
## 3 Tools

- [Import Materials/Sections](#)
- [Convert Model to Truss](#)

### 3.1 Import Materials/Sections

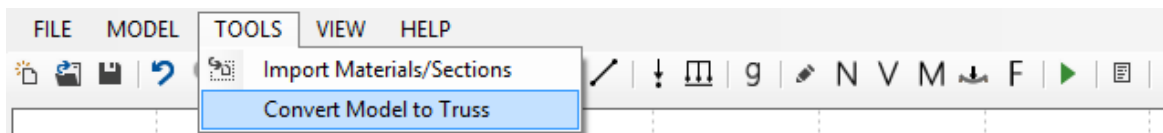


The user can import **Materials** and/or **Sections** from an existing EngiLab Beam.2D file (.bea file).



The user can choose to import Materials and/or Sections and also whether duplicate Materials and/or Sections should be ignored.

### 3.2 Convert Model to Truss



This command converts a **Frame** model to a **Truss**. A Truss model has only axial tension (no shear, no bending moment).

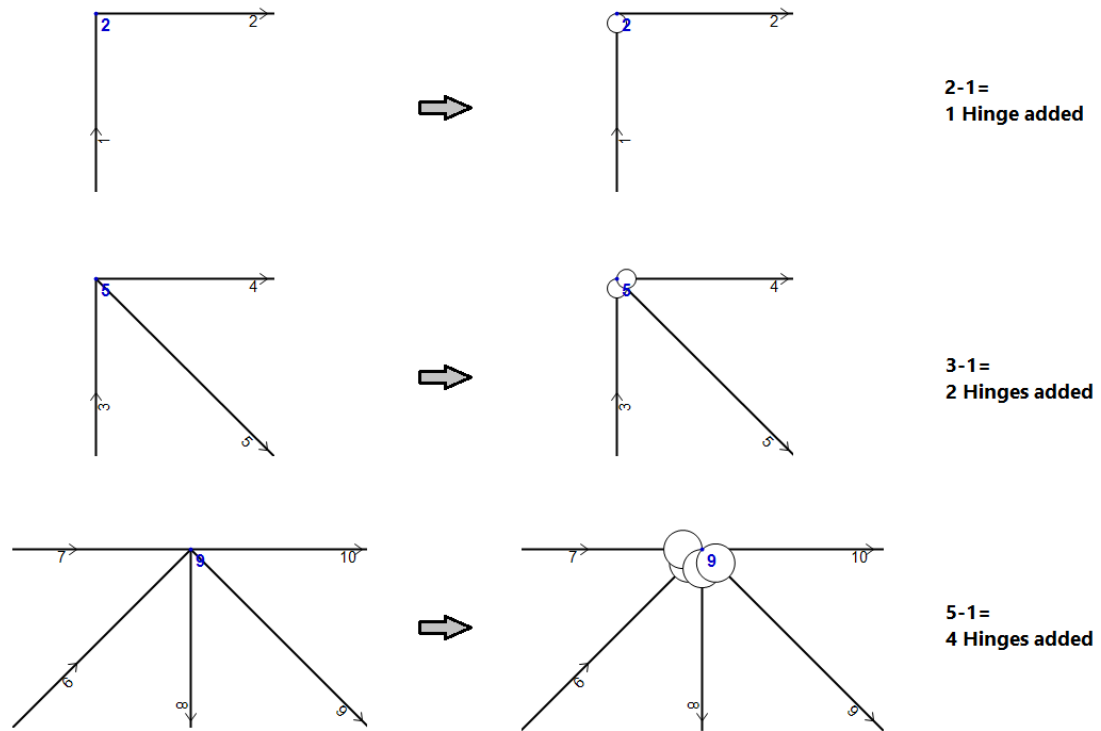
**Important:** Please note that if a Body (Acceleration) Load is present, then a Truss model can have also shear and bending, for example bending of a truss element due to its self weight. In this case, each element acts as a simply supported beam with uniform load on.

In order to convert a Frame model to a Truss, the program automatically makes the

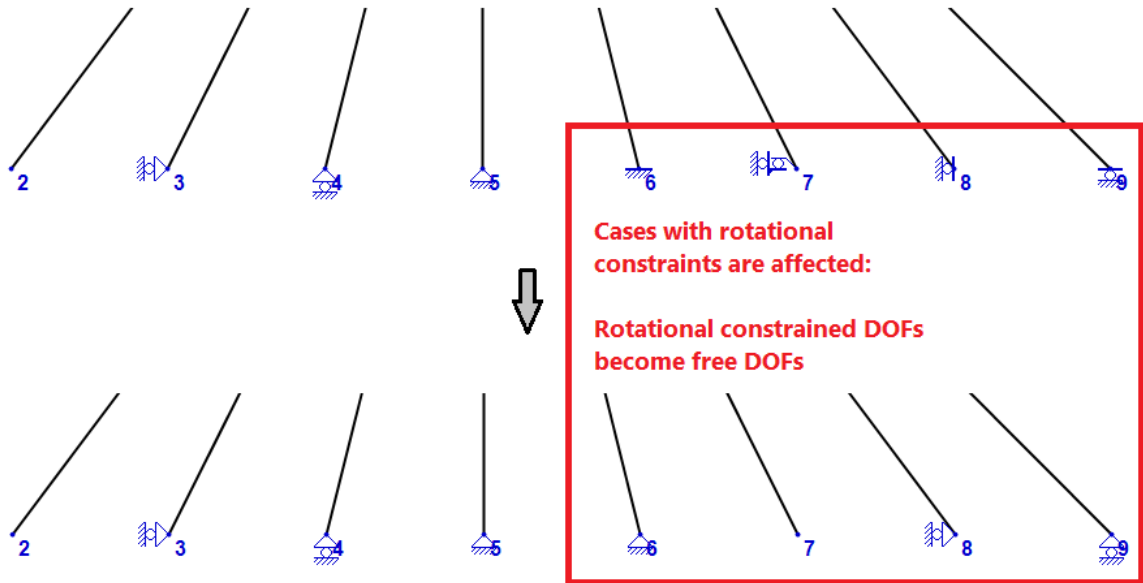


following changes to the model:

1. **Every node where elements are connected is converted to a Hinge.** When **N** elements are connected to a node, **N-1 hinges** are added to the N elements, as explained in detail in the following picture.

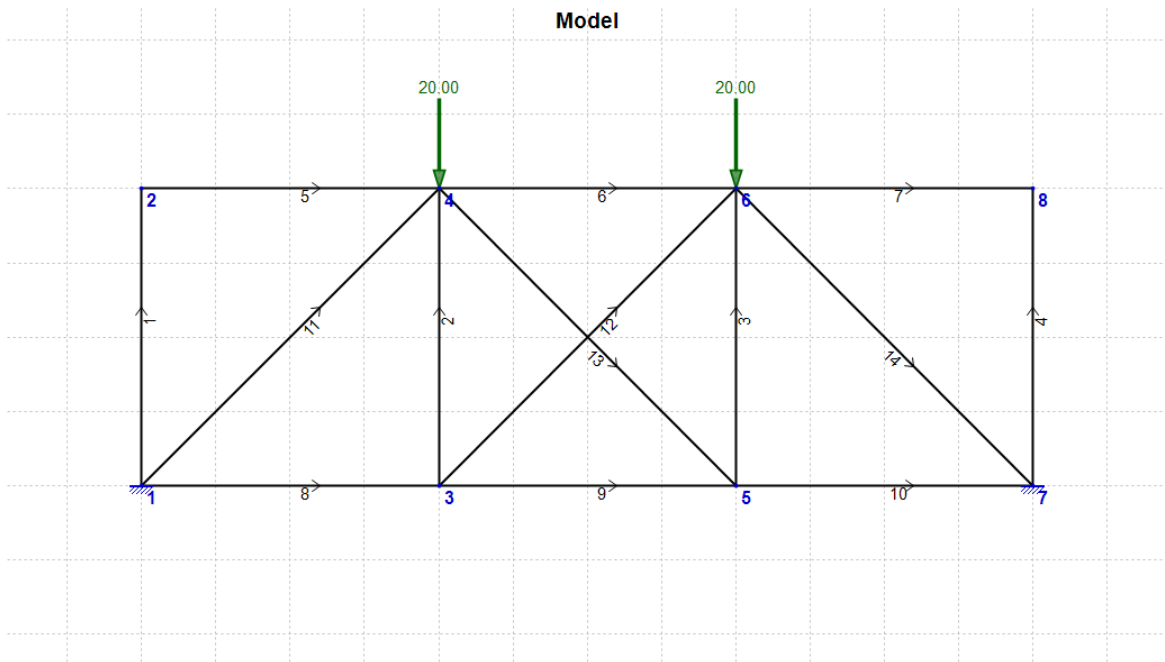


2. **Supports are converted to Truss-Model supports**, by releasing any constraints on the rotational Degree of Freedom (**RZ-Con**). The other two DOFs of each node are not affected. The picture below explains this conversion for every support case in EngiLab Beam.2D.



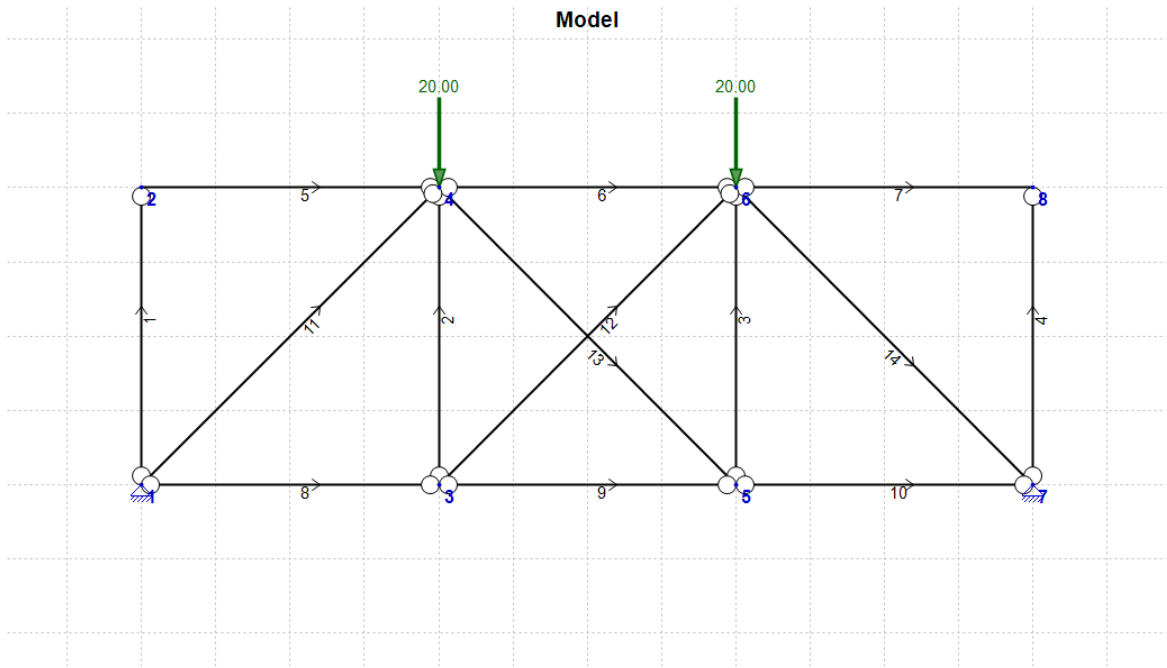
See the example below.

The picture below shows the initial Frame Model



The picture below shows the Model after it has been converted to a Truss

Model



*This page intentionally left blank.*

# Chapter

---

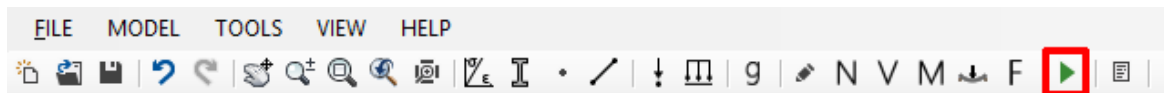


**Analysis and Analysis results**

## 4 Analysis and Analysis results

- [Analysis](#)
- [N, V, M Diagrams](#)
- [Deformation](#)
- [Free Body Diagram \(F\)](#)
- [Analysis results](#)
  - [Node Displacements](#)
  - [Element End Forces](#)
  - [Support Reactions](#)
  - [Full Report \(RTF\)](#)
  - [Analysis Validation](#)

### 4.1 Analysis



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:

$$\{D\} = [K]^{-1} \cdot \{F\}$$

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.

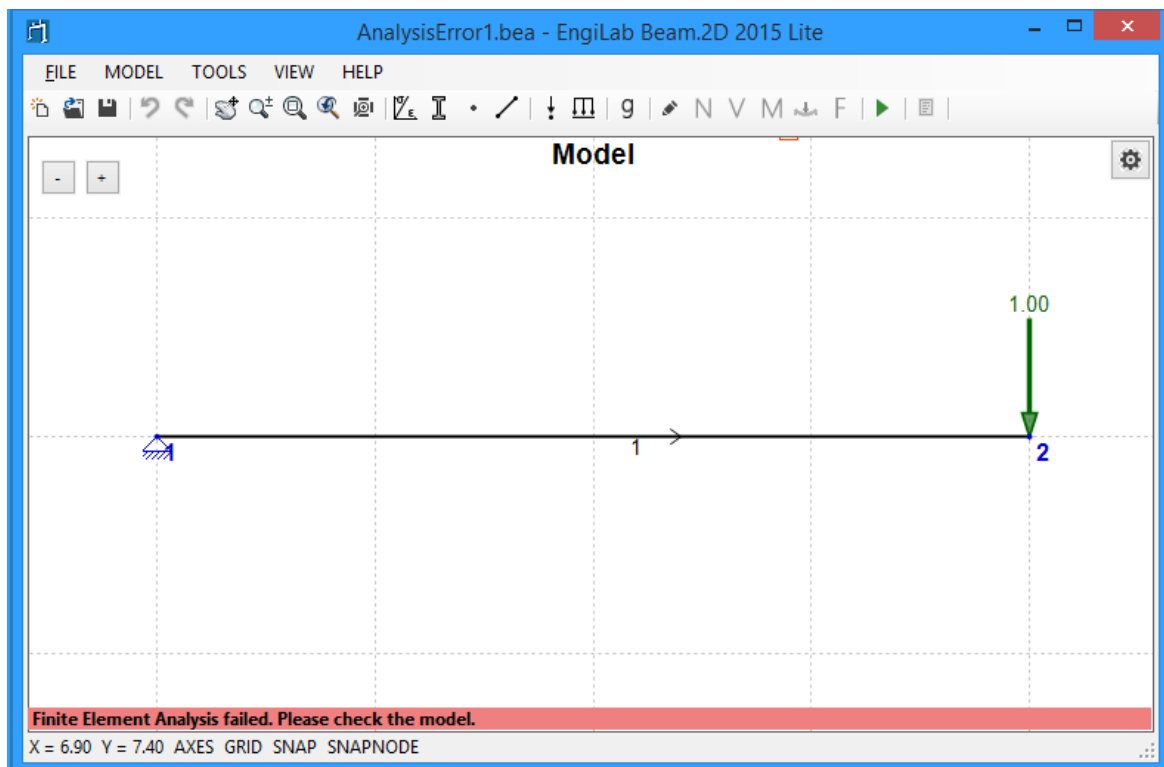
#### Errors during the analysis process

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

##### 1. 'Finite Element Analysis failed. Please check the Model.'

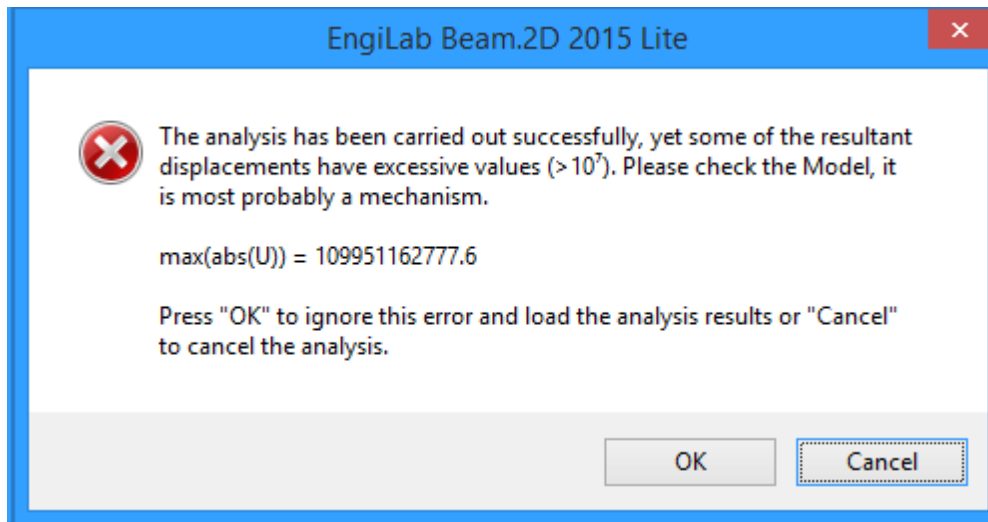
This error occurs due to the model stiffness matrix being non-reversible. There is a zero-element appearing at the matrix diagonal, the determinant of the model stiffness matrix equals to zero, the matrix is non-reversible thus the linear equations system cannot be solved for displacements.

**Example:** Under-constrained model, such as a Cantilever beam where instead of a fixed support, there is a Pinned support as shown below. The model can freely rotate and Equilibrium cannot be reached, by no means. The model cannot be analyzed and the programs gives us an error message, as shown below.



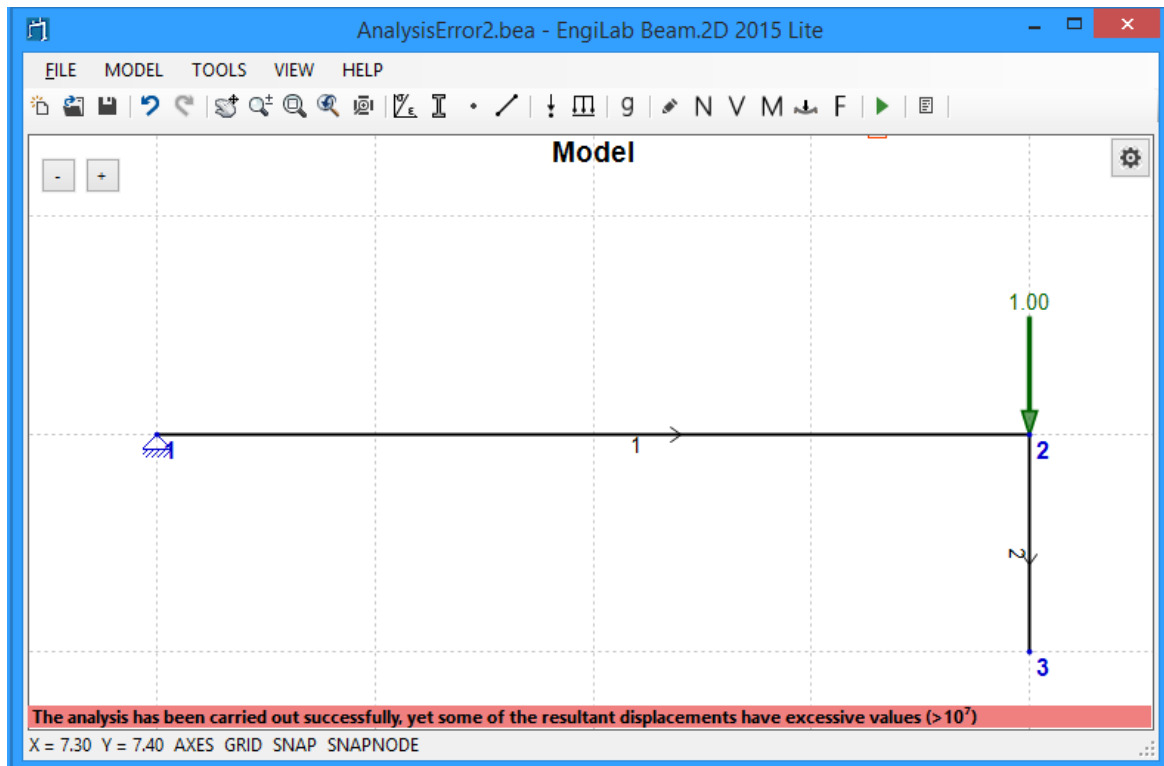
**2. 'The analysis has been carried out successfully, yet some of the resultant displacements have excessive values ( $>10^7$ ). Please check the Model, it is most probably a mechanism.'**

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.





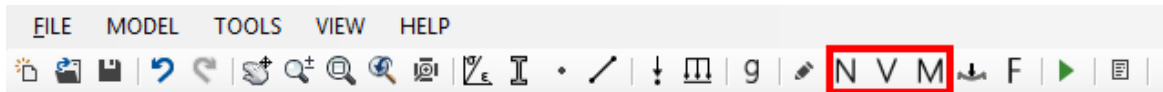
This for example can happen in the Model shown below.




### Possible reasons for such error messages:

- There are nodes that are not connected to the model via elements.
- The constraints are inadequate (under-constrained Model), for example the first model shown above.
- Some parts of the structure are not connected to each other and as a result the forces cannot be transferred from one part of the structure to another.
- The forces cannot be transferred from one part of a structure to another due to the type of the connecting elements (Hinges).
- Due to some other reason, the structure is a mechanism.

## 4.2 N, V, M Diagrams



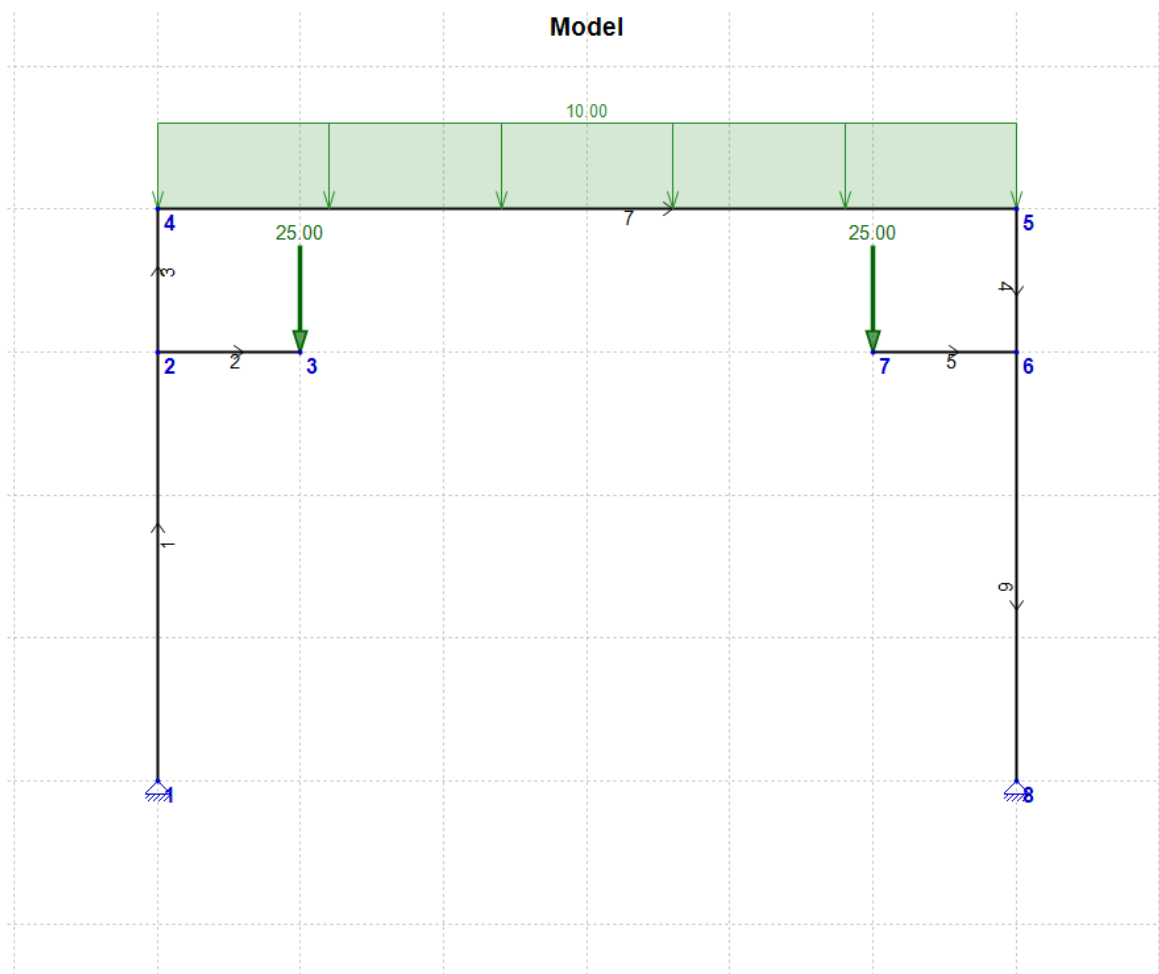
After setting up the model and analyzing it (Clicking the 'Analyze' button ) you can click **N**, **V** or **M** to see the **Axial Force** Diagram, **Shear Force** Diagram or **Bending Moment** Diagram.

- **N**: Axial Force Diagram
- **V**: Shear Force Diagram
- **M**: Bending Moment Diagram

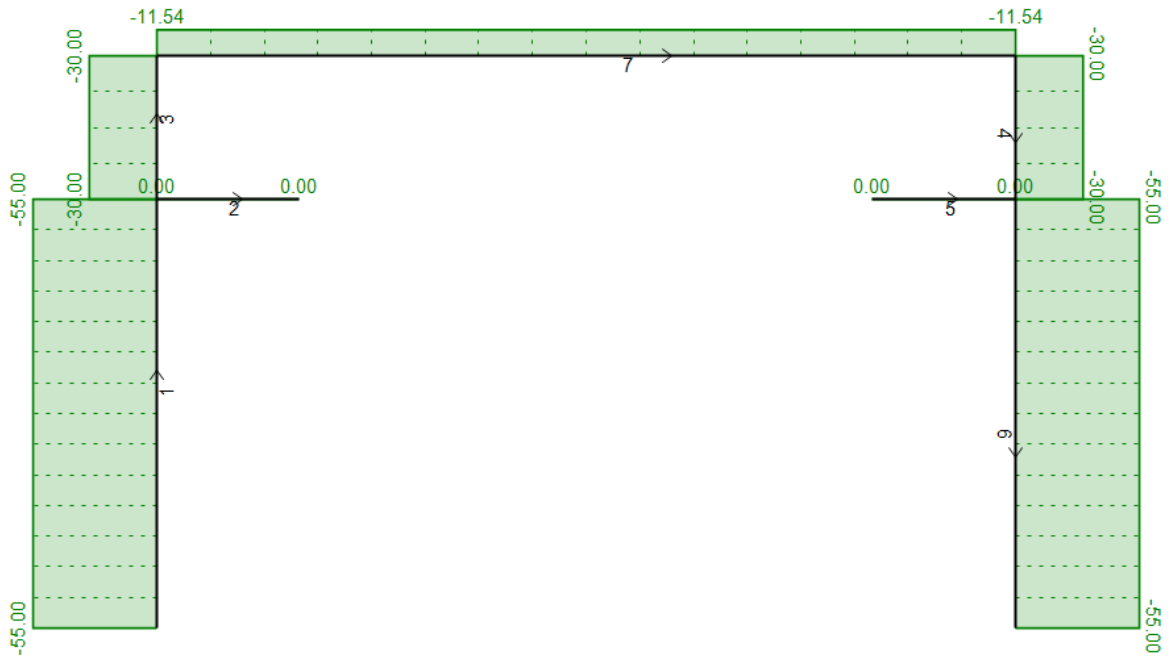
**Note:** Diagram values are given also on screen, if the mouse pointer moves over an element.

**See also:** [NVM Diagrams \(Settings\)](#).

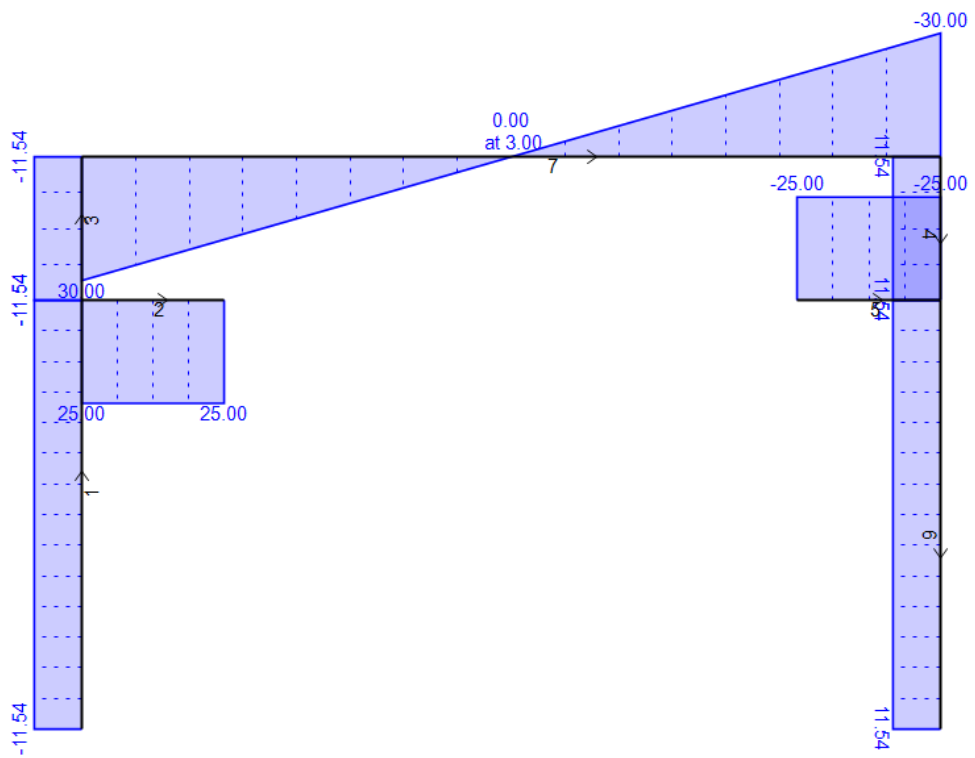
### Example



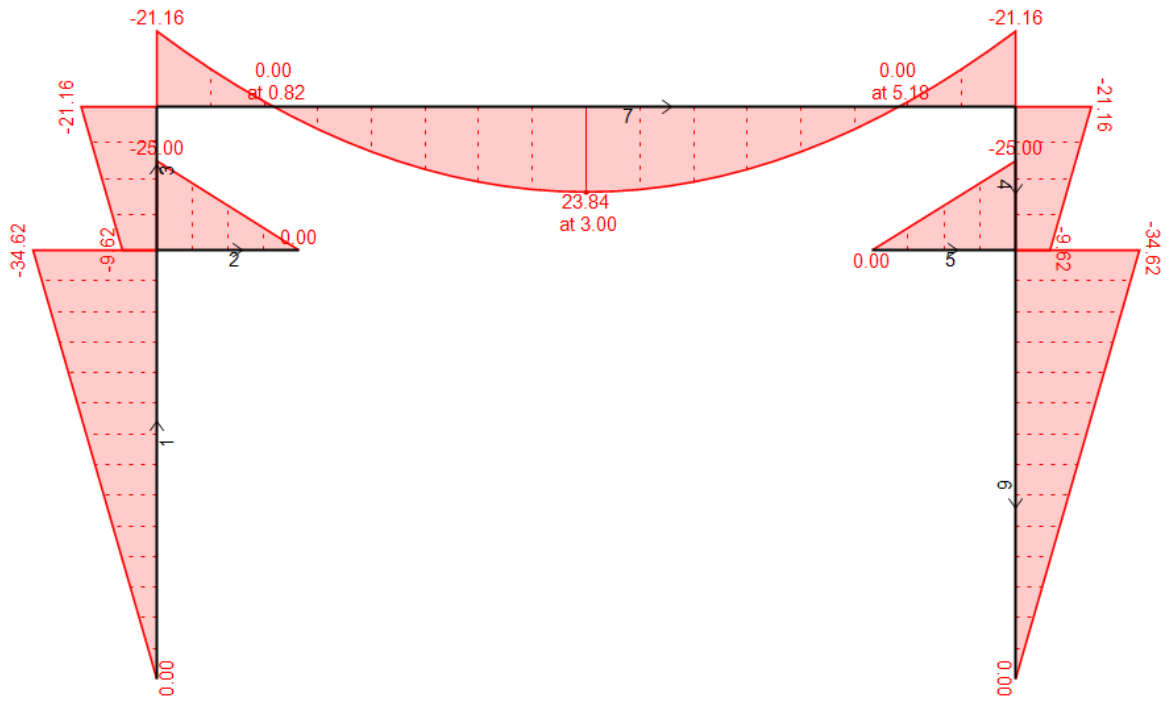
Axial Force Diagram [N]



Shear Force Diagram [V]



Bending Moment Diagram [M]



## 4.3 Deformation



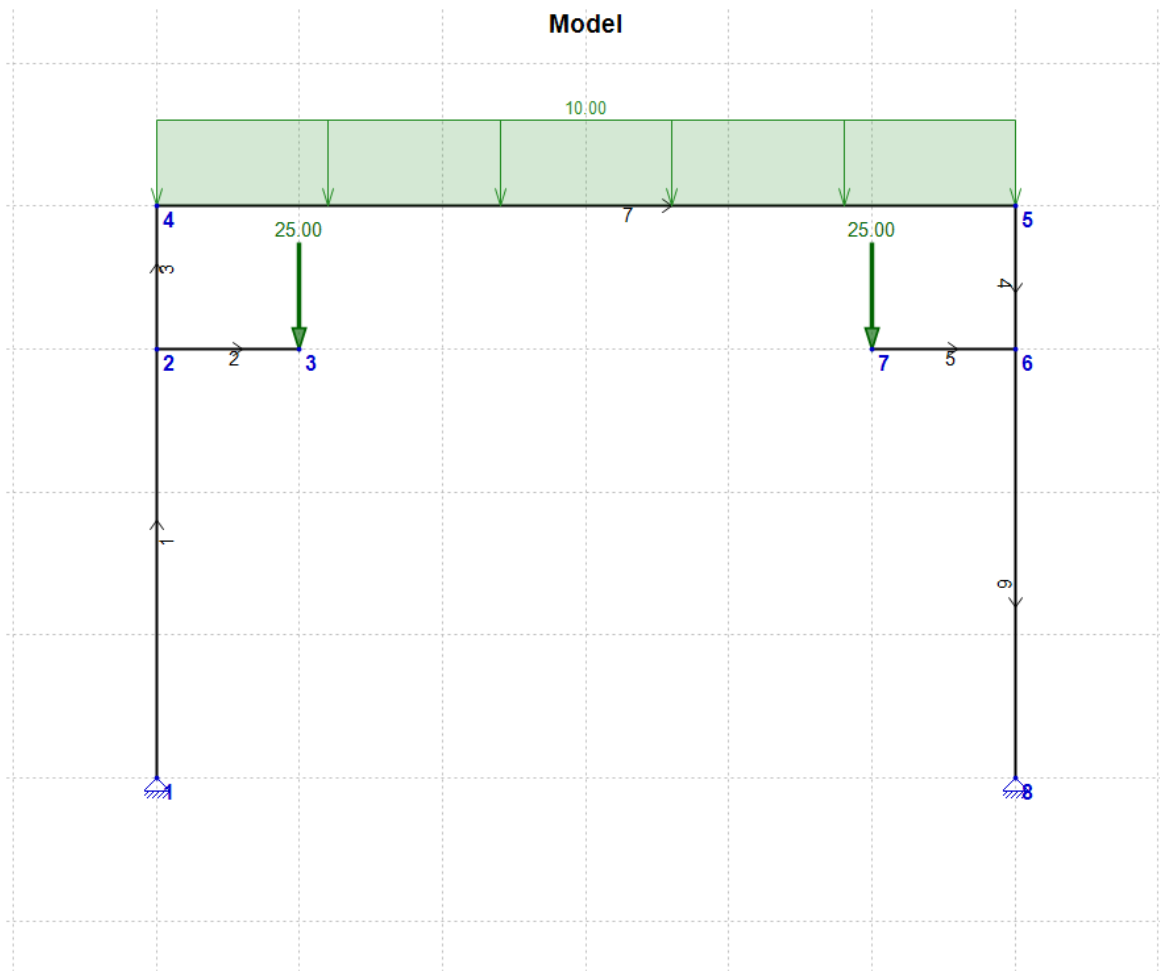
EngiLab Beam.2D uses the Finite Element Method (FEM) to analyze the Model. According to FEM, displacements are computed only for Nodes. Loads within elements (elemental loads) are distributed to the connecting Nodes and the analysis results give the Node Displacements vector (displacements at the positions of the Nodes). Calculating the intermediate displacements within elements and drawing accurately the deformed shape of the model is not an easy task, especially for cases of Elements with Hinges and linear varying loads on. EngiLab Beam.2D uses special computational techniques that give 100% accurate results without any approximations, even for the displacements along an element.

After setting up the model and analyzing it (Clicking the 'Analyze' button ►) you can click ↴ to see the deformed shape of the Model.

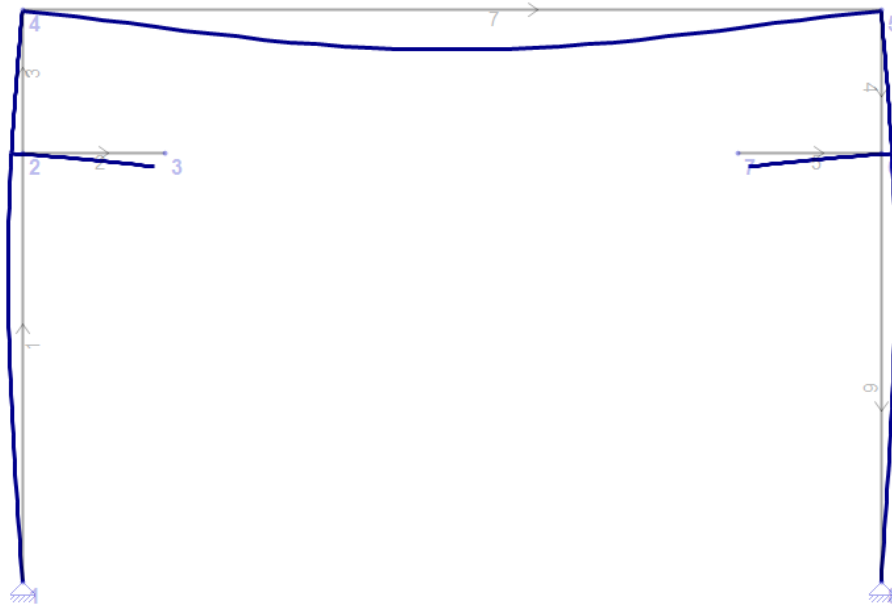
- ↴: Deformation

**Note:** Deformation values are given also on screen, if the mouse pointer moves over an element. The values that are given on screen are the x and y displacements of the corresponding point of each element. These are given in **Local Element Axes**, not Global axes.

### Example






Deformation  
(x15.57)

#### 4.4 Free Body Diagram (F)



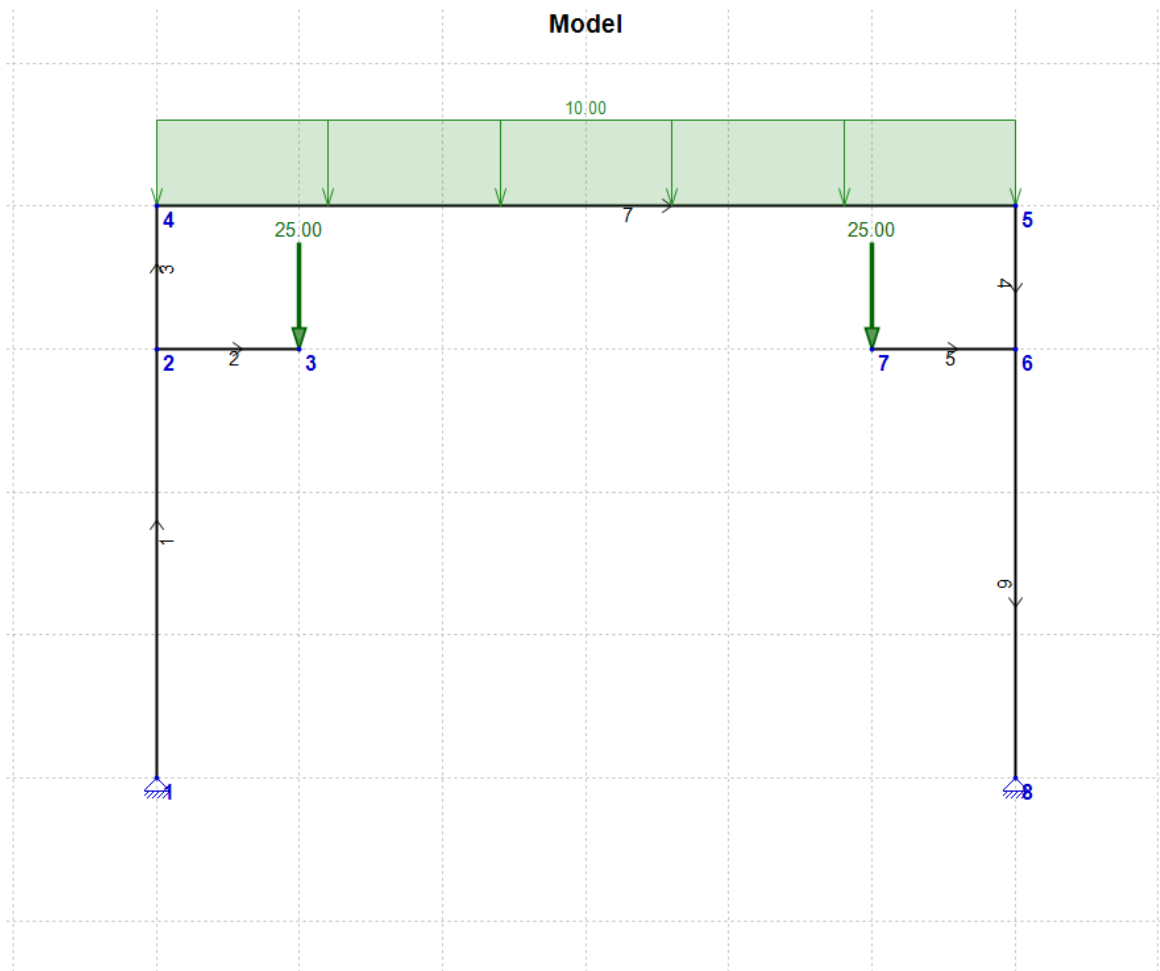
After setting up the model and analyzing it (Clicking the 'Analyze' button ) you can click **F** to see the Free Body Diagram of the Model.

- **F** : Free Body Diagram

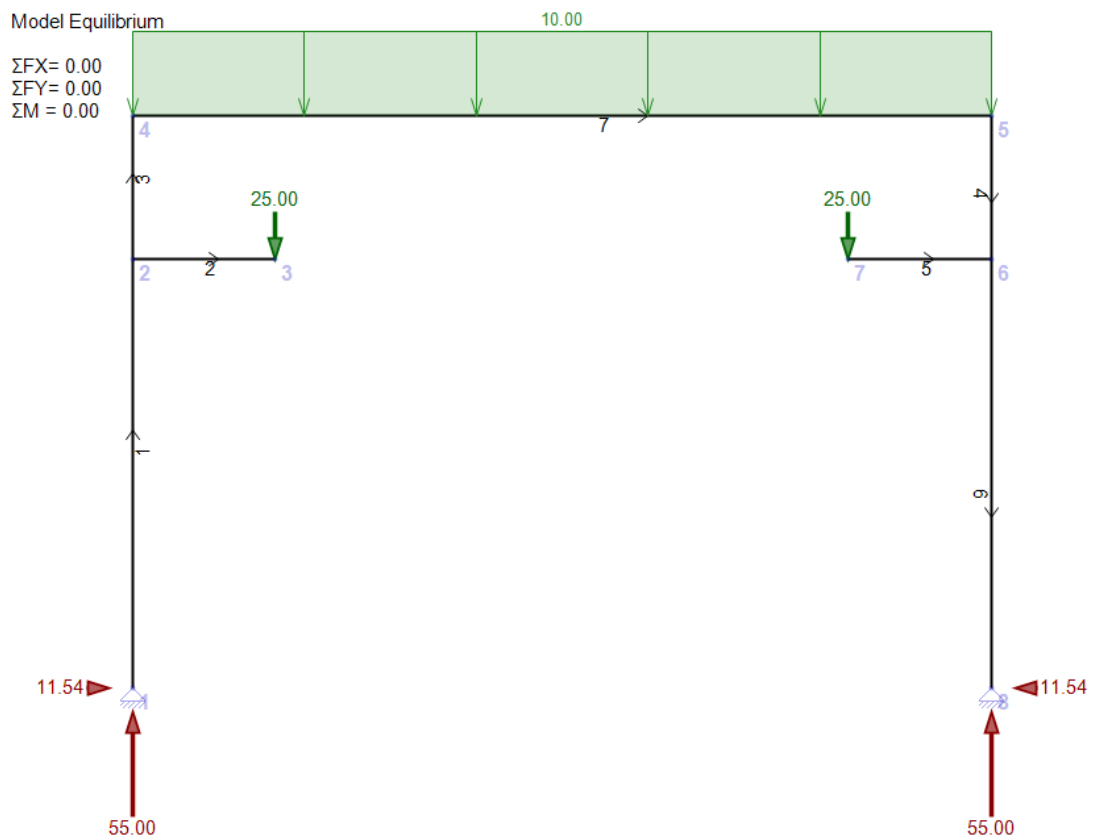
The Free Body Diagram shows the support reactions on screen and also the calculations of the equilibrium of the Model.

**Note:** If the mouse pointer moves over an element, the program shows the corresponding N, V and M values at the corresponding point along the element. The program shows also the Element End Forces and the calculations for the equilibrium of the specific Element.

### Example



Free Body Diagram



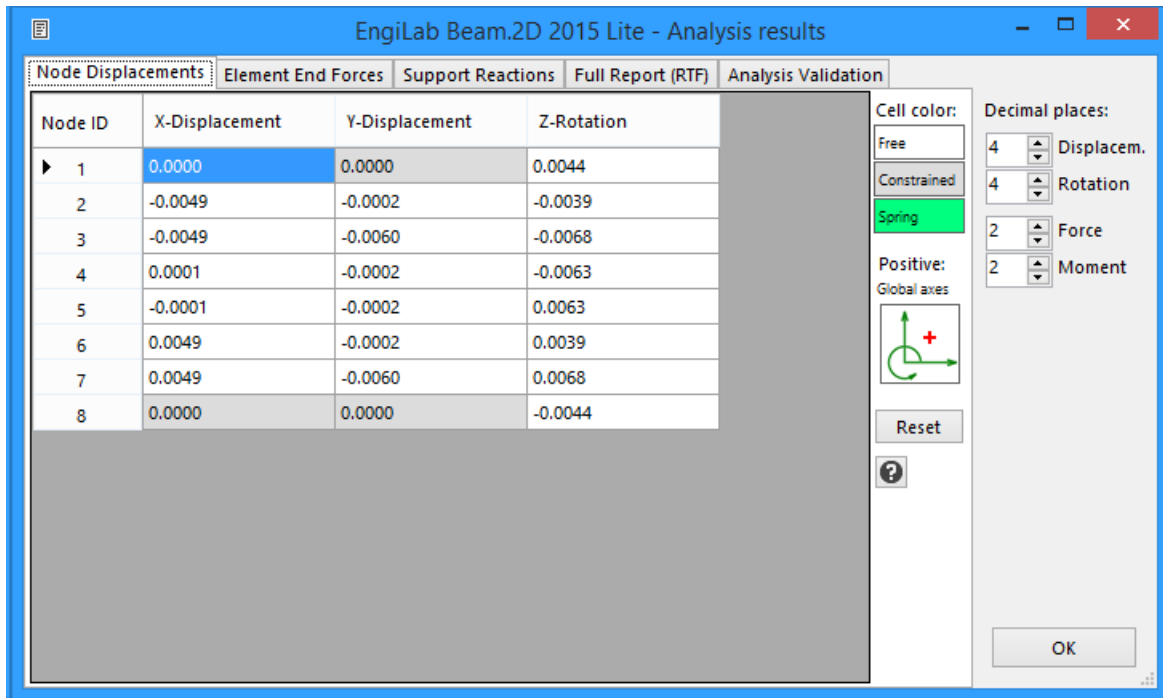
4.5 Analysis results



- [Node Displacements](#)
- [Element End Forces](#)
- [Support Reactions](#)
- [Full Report \(RTF\)](#)
- [Analysis Validation](#)

### 4.5.1 Node Displacements

Shows the Node Displacements of the Model. The sign convention is according to the Global Axes.



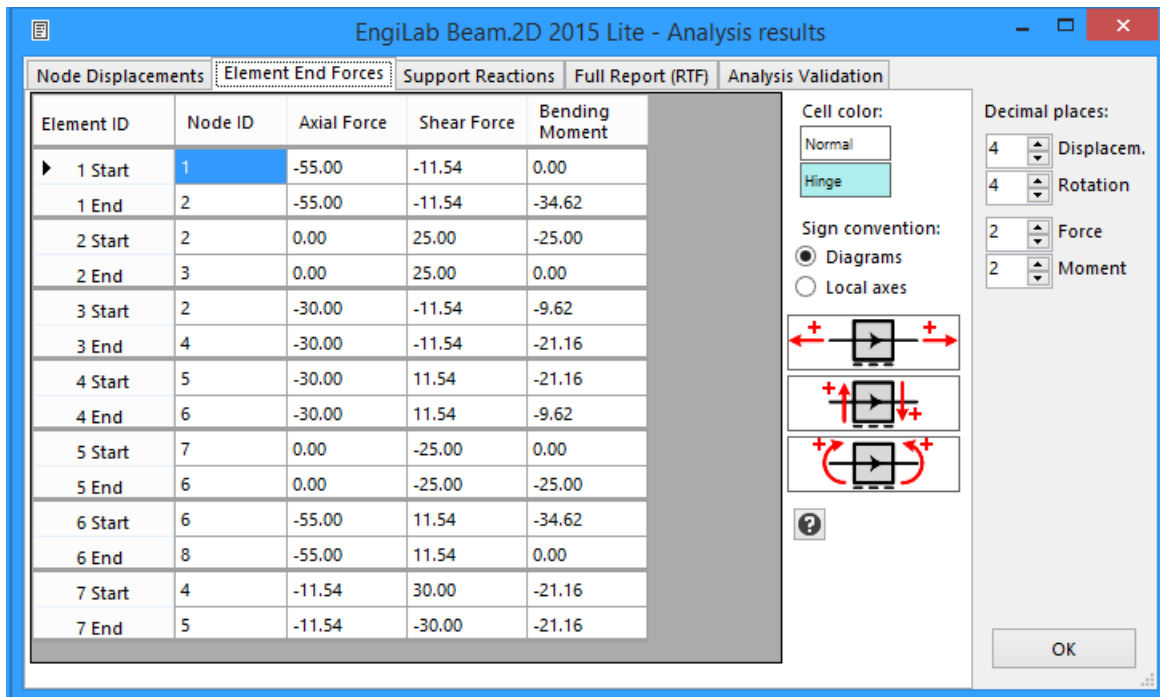
The screenshot displays the 'Node Displacements' window in EngiLab Beam.2D 2015 Lite. The window title is 'EngiLab Beam.2D 2015 Lite - Analysis results'. It features a tabbed interface with 'Node Displacements' selected. The main area contains a table with the following data:

Node ID	X-Displacement	Y-Displacement	Z-Rotation
1	0.0000	0.0000	0.0044
2	-0.0049	-0.0002	-0.0039
3	-0.0049	-0.0060	-0.0068
4	0.0001	-0.0002	-0.0063
5	-0.0001	-0.0002	0.0063
6	0.0049	-0.0002	0.0039
7	0.0049	-0.0060	0.0068
8	0.0000	0.0000	-0.0044

On the right side of the window, there are controls for 'Cell color' (Free, Constrained, Spring), 'Decimal places' (4 for Displacement, 4 for Rotation, 2 for Force, 2 for Moment), and a 'Positive: Global axes' diagram. A 'Reset' button and an 'OK' button are also present.

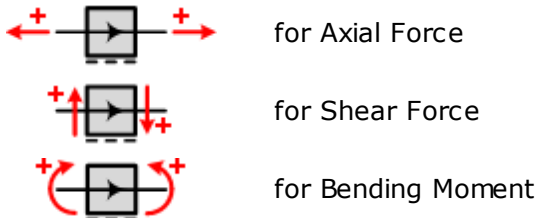
### 4.5.2 Element End Forces

Shows the End Forces of the Elements.



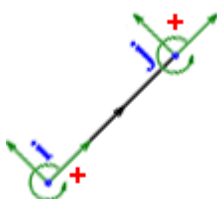
**There are two options for the sign convention:**

1. Use the sign convention of the N, V, M diagrams. For example, the convention could be as shown below.



The above is only an example of these settings, as the sign convention for the diagrams can be changed by the user. See [NVM Diagrams \(Settings\)](#).

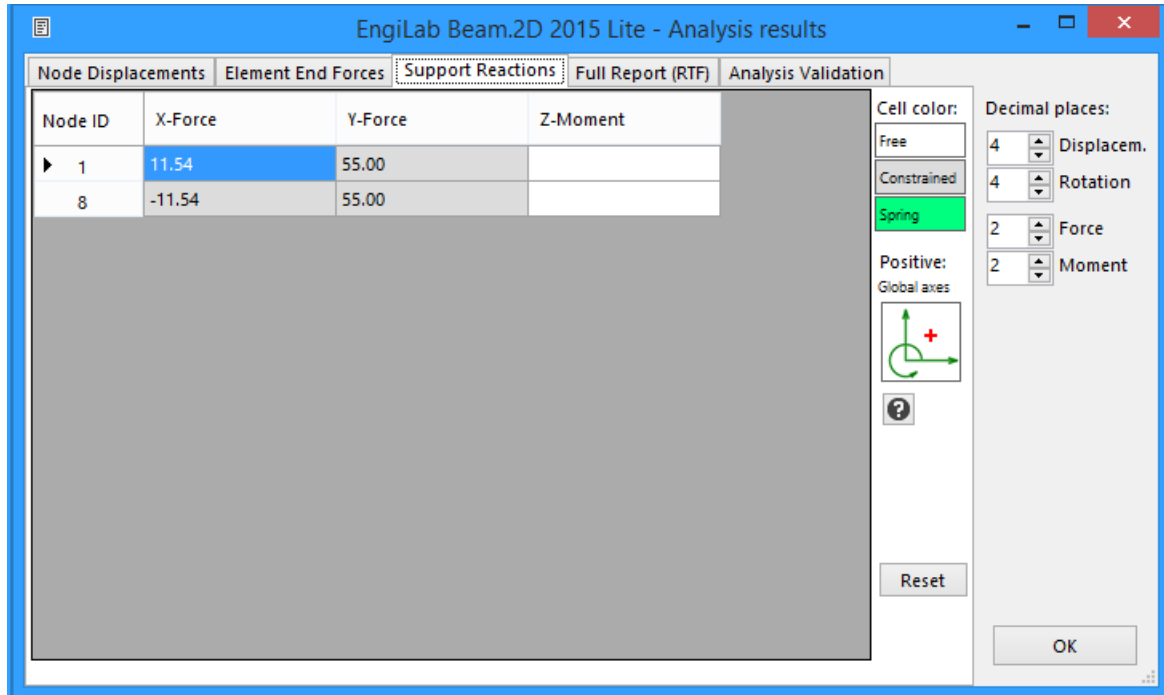
2. Use the sign convention of the Finite Element Analysis (Element local axes), as shown below. This sign convention cannot change.



### 4.5.3 Support Reactions

Shows the Support Reactions (Constraints or Springs) of the Model. The sign convention is in accordance with the Global Axes.

Special colors are used for every Degree Of Freedom (Free, Constrained or Spring), as shown below.



EngiLab Beam.2D 2015 Lite - Analysis results

Node ID	X-Force	Y-Force	Z-Moment
1	11.54	55.00	
8	-11.54	55.00	

Cell color: Free, Constrained, Spring (highlighted)

Positive: Global axes

Decimal places: 4 Displacement, 4 Rotation, 2 Force, 2 Moment

Reset, OK

### 4.5.4 Full Report (RTF)

Shows all the Analysis results in Rich Text Format (RTF). You can easily copy the document and paste it anywhere, or you can click the "Save" button and save the document in RTF format.

In the latest version of the program, all the diagrams have also been included in the Full Report. The following pictures are added:

- Model
- Axial Force Diagram (N)
- Shear Force Diagram (V)
- Bending Moment Diagram (M)
- Deformation (D)
- Free Body Diagram (F)

**EngiLab Beam.2D 2015 Analysis Report**  
 Model: "Frame 1.bea"  
 Report created: Sunday, 10 May 2015, 16:20:22

**Node Displacements**

Node ID	X-Displacement	Y-Displacement	Z-Rotation
1	0.0000	0.0000	0.0044
2	-0.0049	-0.0002	-0.0039
3	-0.0049	-0.0060	-0.0068
4	0.0001	-0.0002	-0.0063
5	-0.0001	-0.0002	0.0063
6	0.0049	-0.0002	0.0039
7	0.0049	-0.0060	0.0068
8	0.0000	0.0000	-0.0044

Sign convention: Positive according to Global Axes  
 Displacements reported with 4 decimal places  
 Rotations (in Radians) reported with 4 decimal places  
 Background color: Free DOF Constrained DOF **Spring DOF**

**Element End Forces**

Element ID	Axial Force	Shear Force	Bending Moment
1 Start	-55.00	-11.54	0.00
1 End	-55.00	-11.54	-34.62
2 Start	0.00	25.00	-25.00
2 End	0.00	25.00	0.00
3 Start	-30.00	-11.54	-9.62
3 End	-30.00	-11.54	-21.16
4 Start	-30.00	11.54	-21.16
4 End	-30.00	11.54	-9.62
5 Start	0.00	-25.00	0.00
5 End	0.00	-25.00	-25.00
6 Start	-55.00	11.54	-34.62
6 End	-55.00	11.54	0.00
7 Start	-11.54	30.00	-21.16
7 End	-11.54	-30.00	-21.16

Sign convention: According to the diagrams' sign convention  
 Axial force: Positive when member is in tension  
 Shear force: Positive when it rotates the member clockwise  
 Bending moment: Positive when the "bottom" fiber is in tension ("top" fiber in compression)  
 Forces reported with 2 decimal places  
 Bending moments reported with 2 decimal places  
 Background color: Normal DOF **Hinge DOF**

**Support Reactions**

Node ID	X-Force	Y-Force	Z-Moment
1	11.54	55.00	
8	-11.54	55.00	

Sign convention: Positive according to Global Axes  
 Forces reported with 2 decimal places  
 Bending moments reported with 2 decimal places  
 Background color: Free DOF Constrained DOF **Spring DOF**

**Model**

10.00

25.00 25.00 25.00 25.00

### 4.5.5 Analysis Validation

Shows the Validation of the Analysis results. If the Analysis results are correct, then the Model should be in equilibrium. The program calculates the Sum of Forces (X and Y direction) and the Sum of Bending Moments, for the entire Model (Global Equilibrium) and also for each Element separately. There are two available tabs:

#### 1. Global (Model) Equilibrium

The program calculates the Sum of X-Forces, the Sum of Y-Forces and the Sum of Bending Moments (at Point X=0 and Y=0) for the whole Model, for the Nodal Loads, Elemental Loads, Acceleration (Body) Loads and Support Reactions. All three Sums (last row of the table, in bold) have to be zero if the Model is in Equilibrium.

Load/Reaction type	$\Sigma(FX)$	$\Sigma(FY)$	$\Sigma(MZ \text{ at } 0,0)$
▶ Nodal Loads	0.00	-50.00	-350.00
Elemental Loads	0.00	-60.00	-420.00
Acceleration (Body) Loads	0.00	0.00	0.00
Support Reactions	0.00	110.00	770.00
<b>SUM</b>	<b>0.00</b>	<b>0.00</b>	<b>0.00</b>



## 2. Individual Element Equilibrium

The program calculates the Sum of X-Forces, the Sum of Y-Forces and the Sum of Bending Moments at Start i and at End j of each Element, for the Elemental Forces, Body Forces, End Forces at i and End Forces at j. The Sum of X-Forces, the Sum of Y-Forces and also the Sums of Bending Moments at both Start i and End j (in bold) have to be zero if the Element is in Equilibrium.

The screenshot shows the 'EngiLab Beam.2D 2015 Lite - Analysis results' window. The 'Element End Forces' tab is active, and the 'Individual Element Equilibrium' sub-tab is selected. The table displays the following data:

Element ID	FX Elemental Force	FX Body Force	FX End Force i	FX End Force j	SUM IFX	FY Elemental Force	FY Body Force	FY End Force i	FY End Force j	SUM IFY	SUM IMZ at i	SUM IMZ at j
1	0.00	0.00	11.54	-11.54	<b>0.00</b>	0.00	0.00	55.00	-55.00	<b>0.00</b>	<b>0.00</b>	<b>0.00</b>
2	0.00	0.00	0.00	0.00	<b>0.00</b>	0.00	0.00	25.00	-25.00	<b>0.00</b>	<b>0.00</b>	<b>0.00</b>
3	0.00	0.00	11.54	-11.54	<b>0.00</b>	0.00	0.00	30.00	-30.00	<b>0.00</b>	<b>0.00</b>	<b>0.00</b>
4	0.00	0.00	11.54	-11.54	<b>0.00</b>	0.00	0.00	-30.00	30.00	<b>0.00</b>	<b>0.00</b>	<b>0.00</b>
5	0.00	0.00	0.00	0.00	<b>0.00</b>	0.00	0.00	-25.00	25.00	<b>0.00</b>	<b>0.00</b>	<b>0.00</b>
6	0.00	0.00	11.54	-11.54	<b>0.00</b>	0.00	0.00	-55.00	55.00	<b>0.00</b>	<b>0.00</b>	<b>0.00</b>
7	0.00	0.00	11.54	-11.54	<b>0.00</b>	-60.00	0.00	30.00	30.00	<b>0.00</b>	<b>0.00</b>	<b>0.00</b>

On the right side of the window, there is a 'Decimal places' control panel with the following settings:

- Displacement: 4
- Rotation: 4
- Force: 2
- Moment: 2

An 'OK' button is located at the bottom right of the window.

*This page intentionally left blank.*

# Chapter

---

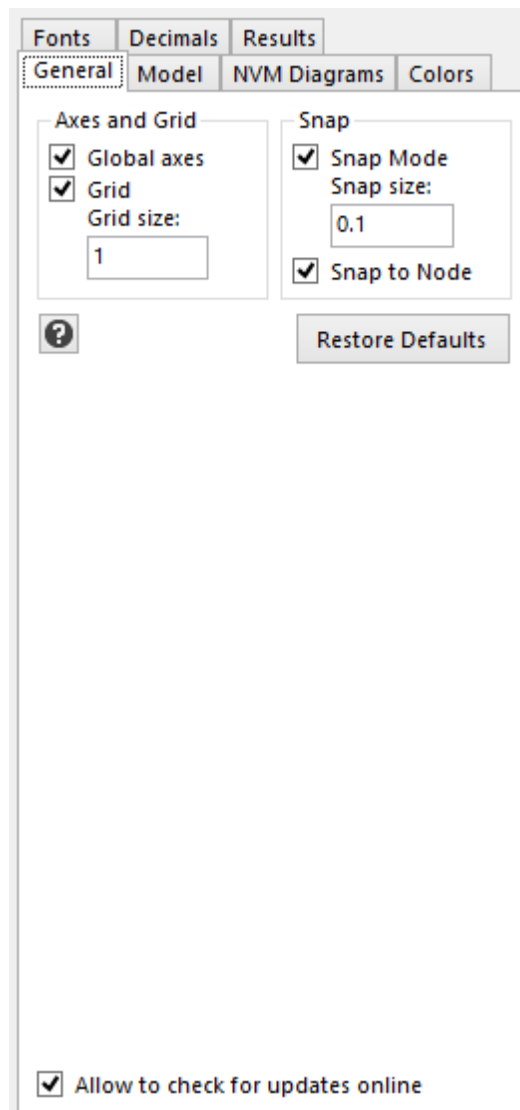


**Settings**

## 5 Settings

- [General](#)
- [Model](#)
- [NVM Diagrams](#)
- [Colors](#)
- [Fonts](#)
- [Decimals](#)
- [Results](#)

### 5.1 General (Settings)



The screenshot shows the 'General' settings tab in the EngiLab Beam.2D 2015 v2.2 software. The dialog box has a title bar with tabs for 'Fonts', 'Decimals', and 'Results'. Below the title bar, there are four sub-tabs: 'General' (selected), 'Model', 'NVM Diagrams', and 'Colors'. The 'General' tab contains two main sections: 'Axes and Grid' and 'Snap'. In the 'Axes and Grid' section, there are two checked checkboxes: 'Global axes' and 'Grid'. Below 'Grid' is a 'Grid size' input field with the value '1'. In the 'Snap' section, there are two checked checkboxes: 'Snap Mode' and 'Snap to Node'. Below 'Snap Mode' is a 'Snap size' input field with the value '0.1'. At the bottom left of the dialog is a help icon (a question mark in a circle). At the bottom right is a 'Restore Defaults' button. At the very bottom of the dialog is a checked checkbox labeled 'Allow to check for updates online'.

### Available settings:

- **Global Axes:** Shows/Hides the Global Axes
- **Grid:** Shows/Hides the Grid
- **Grid size:** Sets the size of the Grid
- **Snap Mode:** Enables/Disables the Snap Mode. If Snap Mode is enabled (Default=True), then you can only draw Nodes and Elements at increments defined by the Snap Size setting
- **Snap Size:** Sets the size of the Snap (Default=0.1)
- **Snap to Node:** Enables/Disables the Snap to Node. If Snap to Node is enabled (Default=True), then the user can "catch" Nodes on screen so that new elements can be connected to existing Nodes.
- **Allow to check for updates online:** Enables/Disables the automatic update feature. If disabled, the program will not check for updates online.

## 5.2 Model (Settings)

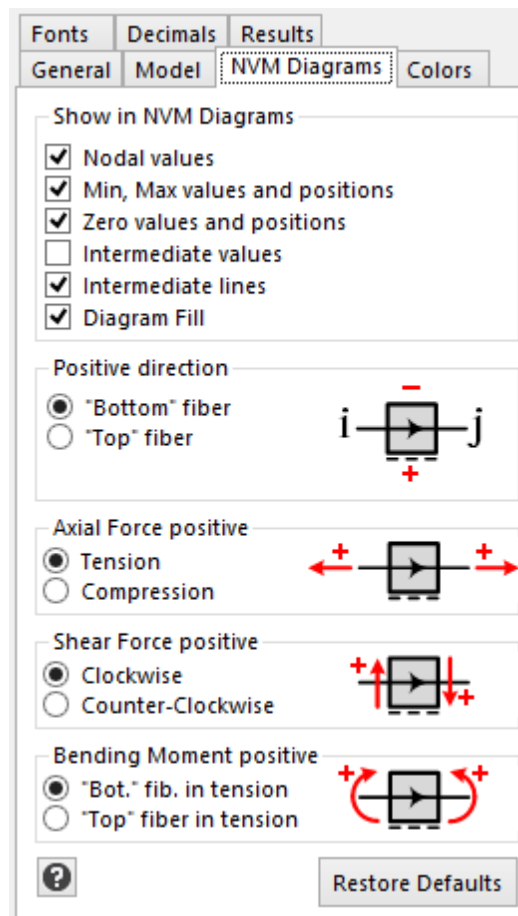
The screenshot shows the 'Model' settings dialog box. It features several tabs: 'Fonts', 'Decimals', 'Results', 'General', 'Model' (selected), 'NVM Diagrams', and 'Colors'. The 'Model' tab contains the following settings:

- Nodes group:**
  - Nodes
  - Numbering
  - Constraints
  - Coordinates
  - Springs
  - Spring values
- Elements group:**
  - Elements
  - Numbering
  - Hinges
  - Orientation
  - Material
  - Section
  - Length
- Loads group:**
  - Nodal Loads
  - Nod. L. values
  - Elem. Loads
  - Elem. L. values
- Nodal Loads representation:**
  - Show the Resultant force
  - Show the X, Y Components
- Acceleration:**
  - Acceleration vector
  - Acceleration values

At the bottom left, there is a help icon (?). At the bottom right, there is a 'Restore Defaults' button.

Each setting Shows/Hides the corresponding object on screen.

## 5.3 NVM Diagrams (Settings)

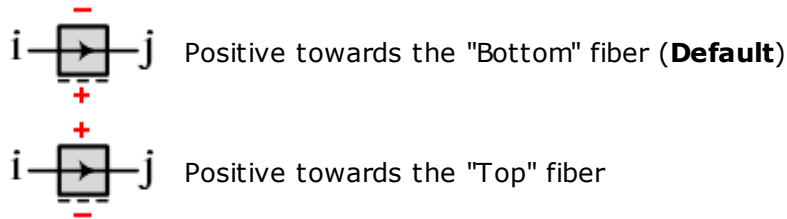


### Available settings:

- **Nodal values:** Shows/Hides the Nodal values of the diagrams (Axial Force, Shear Force or Bending Moment).
- **Min, Max values and positions:** Shows/Hides the (local) Minimum and Maximum values of the diagrams and the corresponding positions along an Element.
- **Zero values and positions:** Shows/Hides the Zero values of the diagrams and the corresponding positions along an Element.
- **Intermediate values:** Shows/Hides the intermediate values of the diagrams.
- **Intermediate lines:** Shows/Hides the intermediate lines of the diagrams.
- **Diagram Fill:** Shows/Hides the diagrams fill.

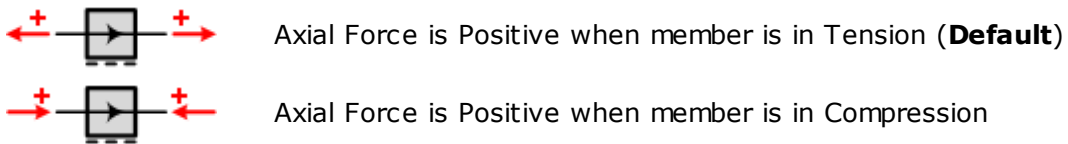
### Positive direction (applies to all diagrams)

Determines the positive direction of the diagrams. Applies to all three diagrams (Axial Force, Shear Force or Bending Moment). There are two options:



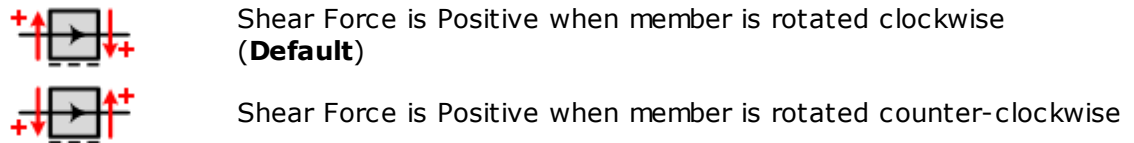
### Axial Force positive

Determines the sign convention for the **Axial Force**. There are two options:



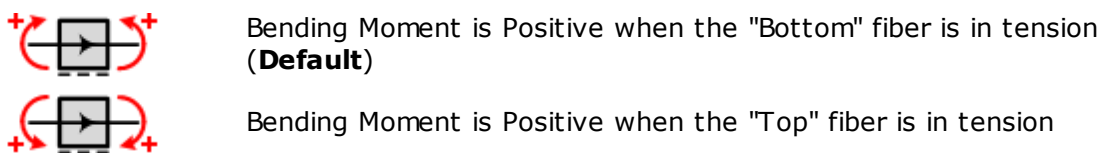
### Shear Force positive

Determines the sign convention for the **Shear Force**. There are two options:

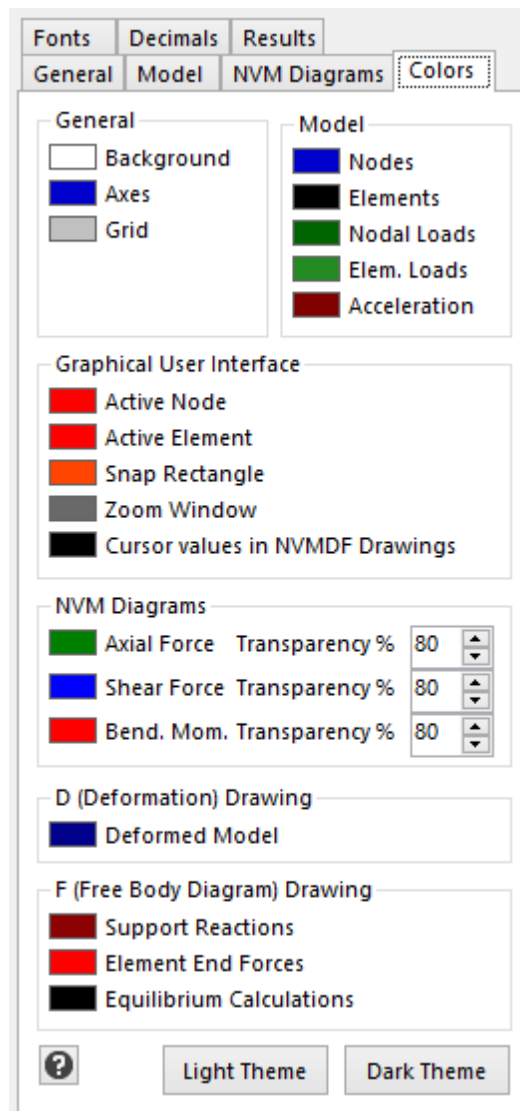


### Bending Moment positive

Determines the sign convention for the **Bending Moment**. There are two options:



## 5.4 Colors (Settings)



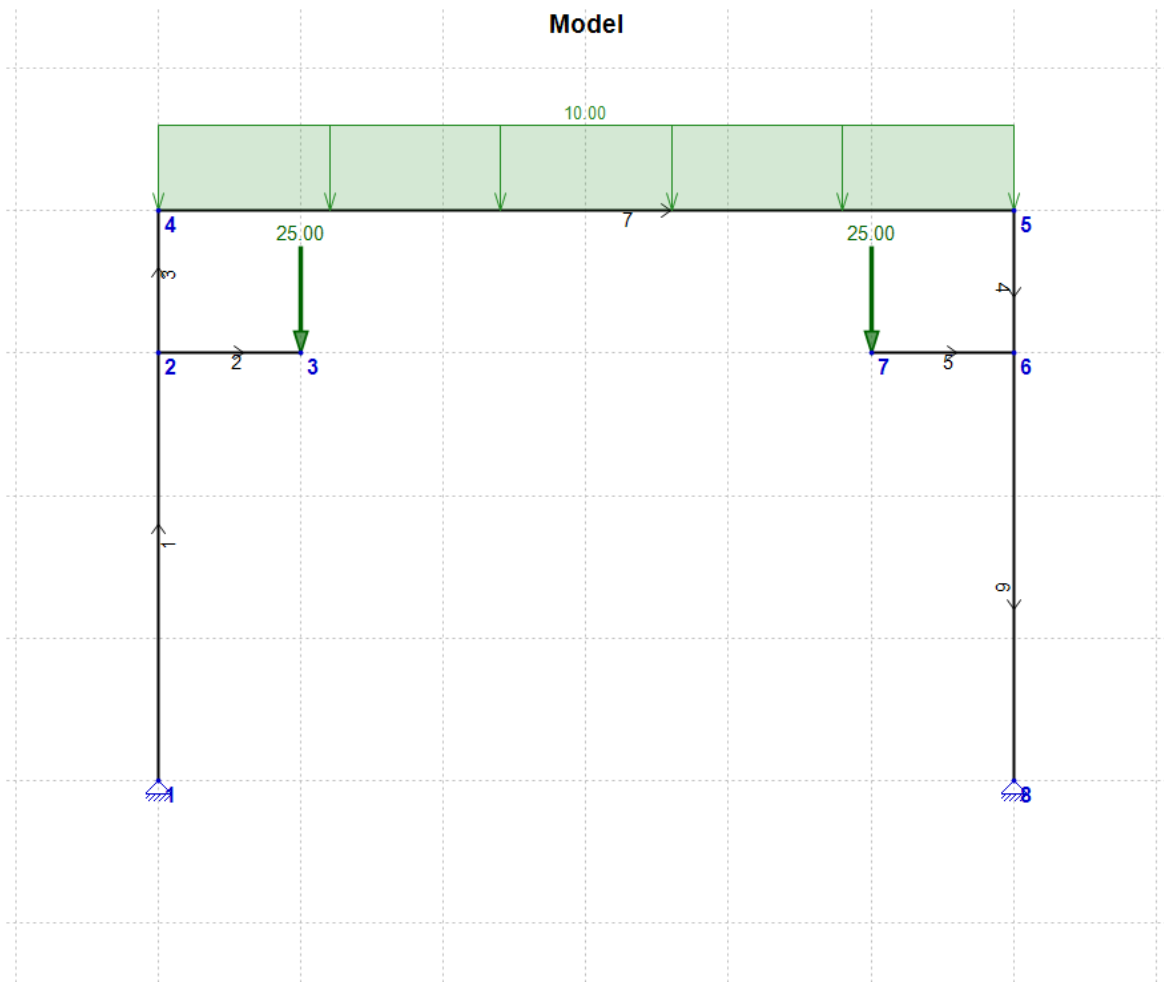
Each setting controls the color of the corresponding object on screen.

You can also set the **transparency** of the filled objects (N, V, M Diagrams fill).

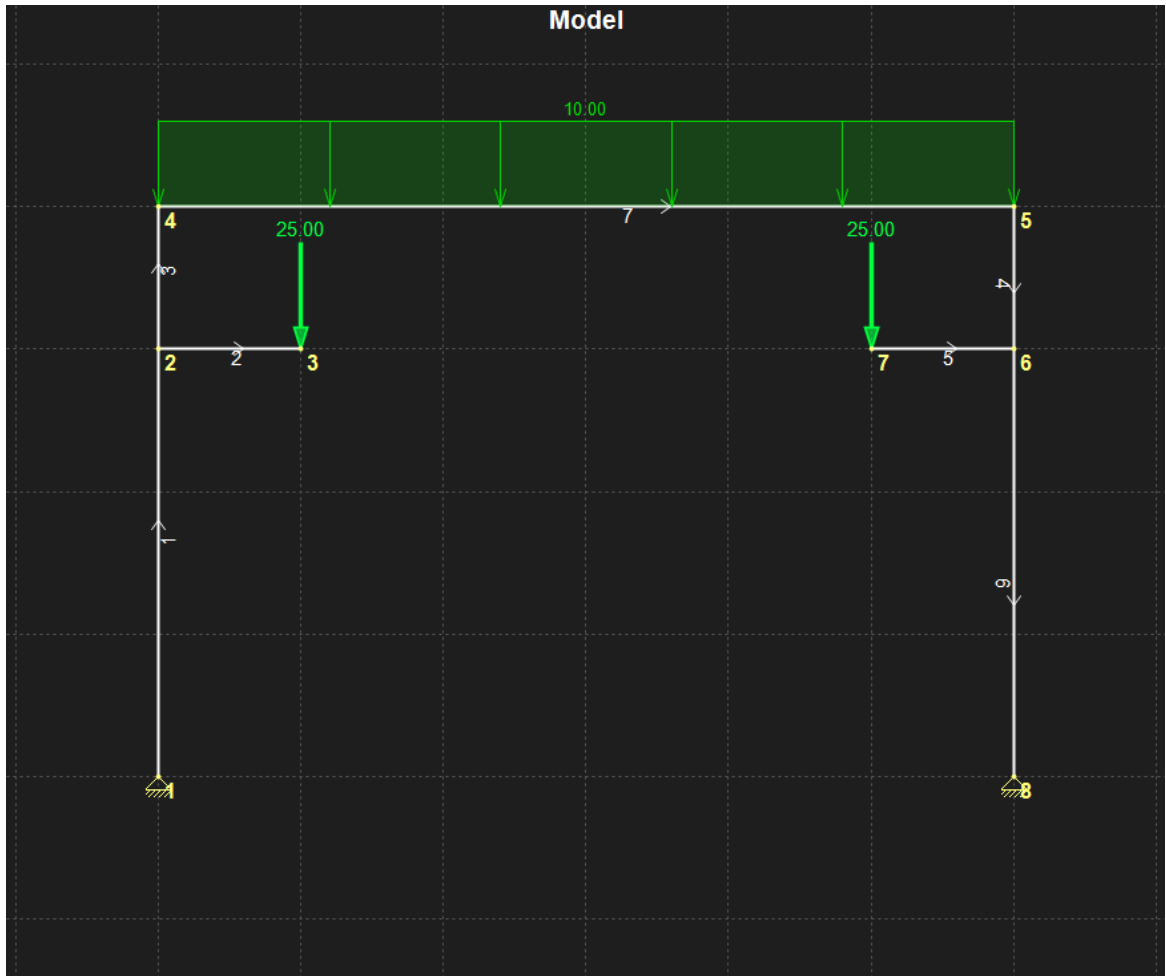
There are two predefined color themes available, the **Light Theme (Default)** and the **Dark Theme**. The two themes are shown below.



### Light Theme (Default)



### Dark Theme



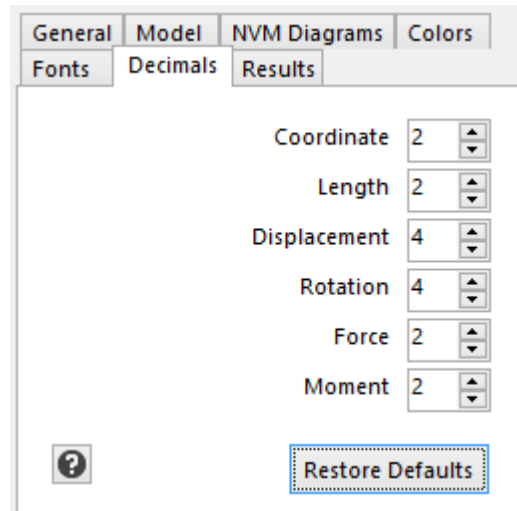
## 5.5 Fonts (Settings)

Category	Setting	Value
General	Picture Title	14
	Axes	13
Model	Nodes	12
	Elements	11
	Nodal Loads	11
	Elemental Loads	10
	Acceleration	11
NVM Diagrams, D and F Drawings	NVM Nodal Values	11
	NVM Intermediate Values	10
	NVMDF Cursor Positions	11
	NVMDF Cursor Values	12
	F Support Reactions	11
	F Element End Forces	12
F Equilibrium Calculations	11	

? Restore Defaults

Each setting sets the Font size for the corresponding object on screen.

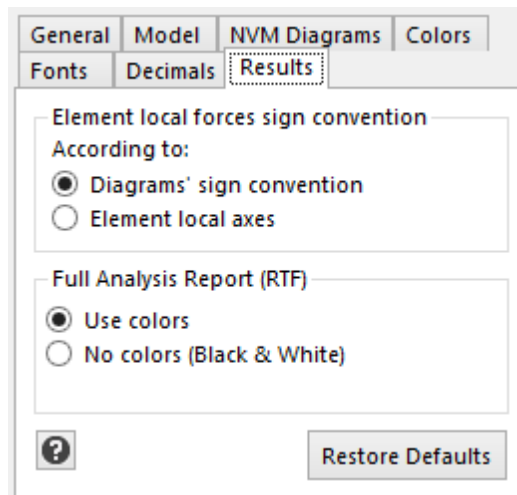
## 5.6 Decimals (Settings)



These settings influence only the appearance of the Model input and the Analysis output.

**Note:** All available digits are taken into consideration for the analysis, at all times. There are no approximations

## 5.7 Results (Settings)



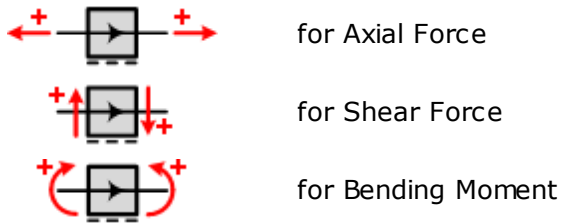
### Available settings:

#### Element local forces sign convention

Determines how the sign of the element local forces will be reported in the Analysis Results. There are two options:

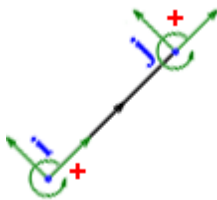
1. Use the sign convention of the N, V, M diagrams. For example, the convention

could be as shown below.



The above is only an example of these settings, as the sign convention for the diagrams can be changed by the user. See [NVM Diagrams \(Settings\)](#).

**2.** Use the sign convention of the Finite Element Analysis (Element local axes), as shown below. This sign convention cannot change.



### **Full Analysis Report (RTF) - Use Color or Black & White**

Applies only for the Full Analysis Report in RTF. You can choose to use color to highlight special items (Constraints, Springs, Hinges) or only Black and White without highlighting special items.

*This page intentionally left blank.*

# Chapter

---



**Useful information**

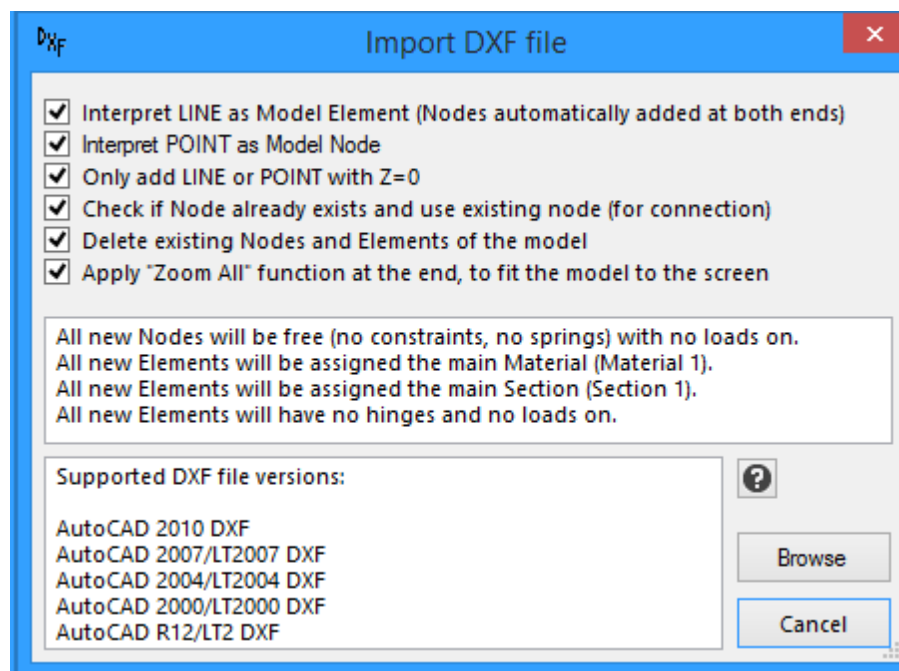
## 6 Useful information

- [Import DXF file](#)
- [Ready-to-analyze Examples](#)
- [Tips on Modeling hinges](#)
- [Tips on Modeling symmetric structures](#)

### 6.1 Import DXF file

An alternative way to build a Model quickly, is by importing a DXF file. You can draw your Model in a CAD program using simple LINES (and optionally POINTS) and you can import your drawing into EngiLab Beam.2D as a structural Model.

In order to open the "Import DXF file" form, **you first have to define at least one Material and one Section**. All the LINES of the DXF file will be converted to Model Elements assigned Material 1 and Section 1.



#### Notes:

- All new Nodes will be free (no constraints, no springs) with no loads on.
- All new Elements will be assigned the main Material (Material 1).
- All new Elements will be assigned the main Section (Section 1).
- All new Elements will have no hinges and no loads on.

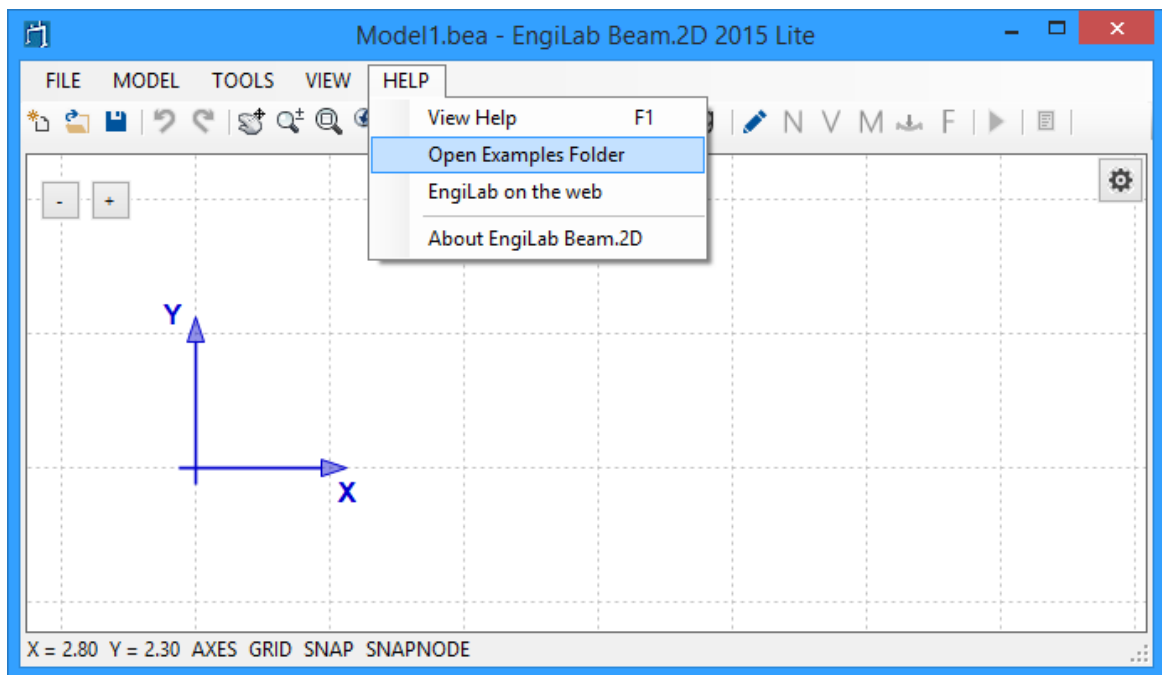


**Supported DXF file versions:**

- AutoCAD 2010 DXF
- AutoCAD 2007/LT2007 DXF
- AutoCAD 2004/LT2004 DXF
- AutoCAD 2000/LT2000 DXF
- AutoCAD R12/LT2 DXF

Most probably the DXF Import feature will work without any problems also with newer versions of AUTOCAD DXF files, yet we have not confirmed it yet.


## 6.2 Ready to-analyze Examples



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.

- There are 18 example files in total

**How to open and Analyze an example:**

- From the FILE menu, select Open.
- Open the Examples folder (Usually **C:\Program Files (x86)\EngiLab\EngiLab Beam.2D 2015 Lite\Examples** for the Lite Edition and **C:\Program Files (x86)\EngiLab\EngiLab Beam.2D 2015 Pro\Examples** for the Pro Edition).
- Select a .bea file to open.
- After the file is opened, click the  (Analyze) button to analyze the model.

**After the analysis is finished:**

Click **N**, **V** or **M** to see the **Axial Force** Diagram, **Shear Force** Diagram or **Bending Moment** Diagram.


- **N**: Axial Force Diagram
- **V**: Shear Force Diagram
- **M**: Bending Moment Diagram

Click  to see the deformed shape of the Model.

- : Deformation

Click **F** to see the Free Body Diagram of the Model.

- **F**: Free Body Diagram

Click  to see the analytical results. The results include the following tabs:

- Node Displacements
- Element End Forces
- Support Reactions
- Full Report (RTF)
- Analysis Validation

## 6.3 Tips on Modeling hinges

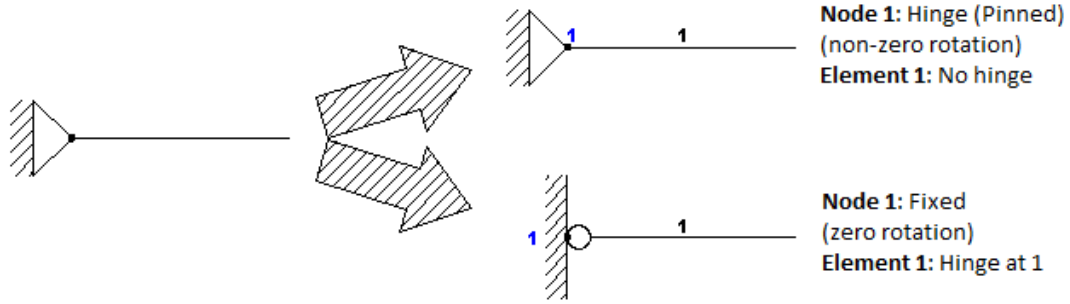
The hinges (rotational releases) of a model is an issue which requires our attention. There are two kinds of hinges: **External hinges (Pinned constraints)** and **Internal hinges**.

### External hinges (Pinned constraints)

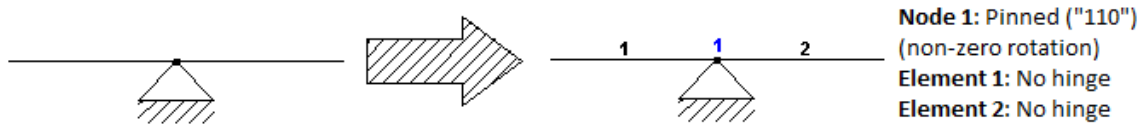
**1.** An external **hinge to which only ONE element is connected** can be given in a model using two possible ways (the result of the analysis should be the same):

- **A.** As a pinned Node (Node 1 - "110") connecting an Element with no hinge at end 1
- **B.** As a fixed Node (Node 1 - "111") connecting an Element with a hinge at end 1.

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



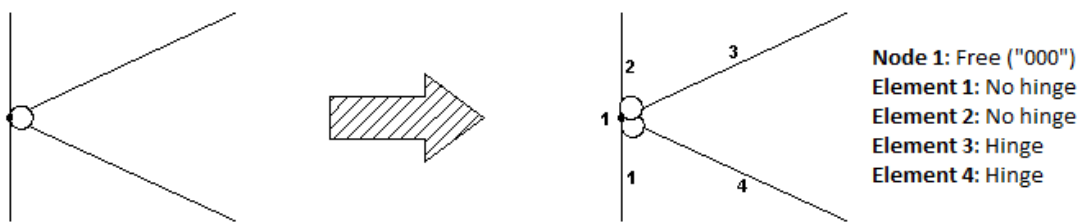
**2.** An **external hinge to which more than one elements are connected** must be given in the Model as follows:



### Internal hinges

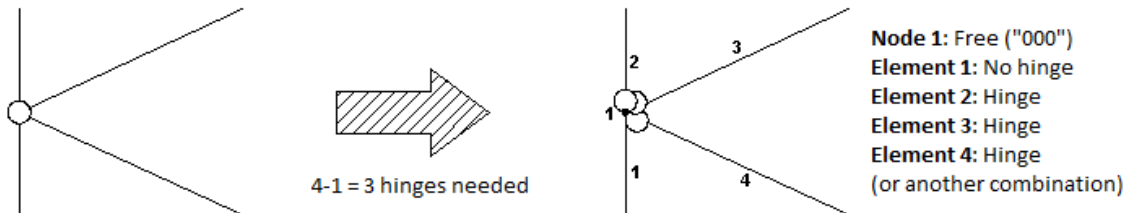
An **internal hinge** must be given always as a hinge of one or more elements as follows:

**1. 'Partial' internal hinge** (Applies to some of the connecting elements)



## 2. 'Full' internal hinge (Applies to all connecting elements)

A 'full' internal hinge which applies to all connecting elements means that for each of the connecting elements the bending moment value at the specific element end is zero. To model that case, you need as many as ('connecting elements' - 1) hinges for the connecting elements. Only one of the connecting elements should have no hinge at the specific end, no matter which of them.

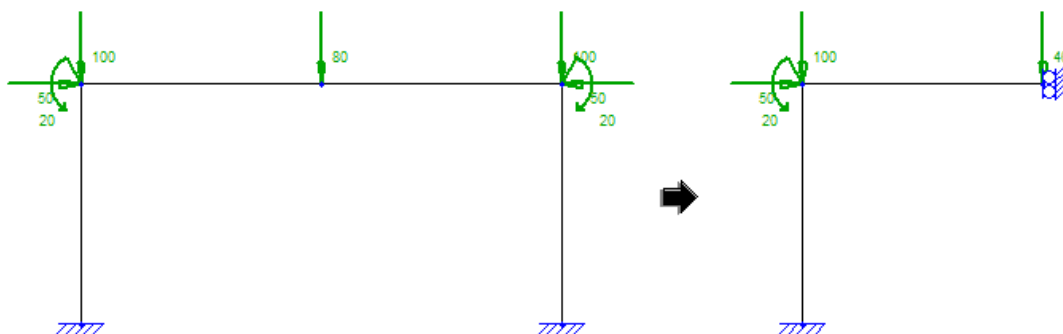


**Note:** In the case of a 'Full' internal hinge, each connecting element has its own rotation at the hinge end. The program only calculates the rotation of the element with no hinge. In the example below, the rotation of element 1 at end 1 will be calculated, while the rotations of the elements 2, 3 and 4 at end 1 will not be given in the analysis results. By using different combinations of releases, one can get the rotations of any connecting element separately.

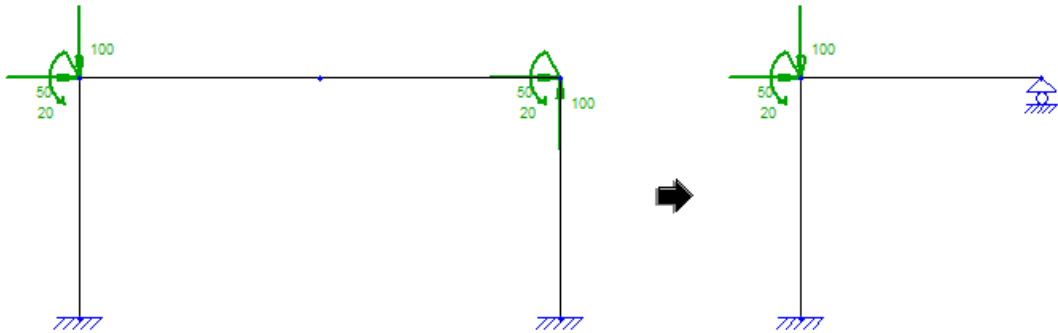
## 6.4 Tips on Modeling symmetric structures

If a structure is symmetric (Symmetric structure with symmetric / anti-symmetric loads), then the size of the finite element model can be reduced, which, in turn, reduces the time and cost of the analysis. For each plane of symmetry in the model, the model size can be reduced by a factor of approximately two. See the examples below.

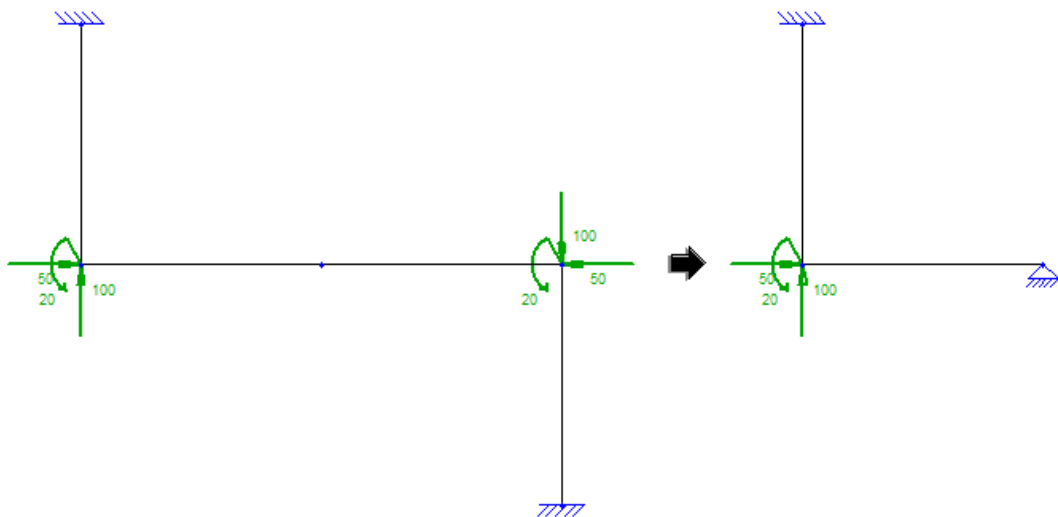
### Case 1: Symmetric structure (axis) + Symmetric loads (axis)



For the above example, half the structure can be analyzed using a "101" constraint for the node on the symmetric 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.**

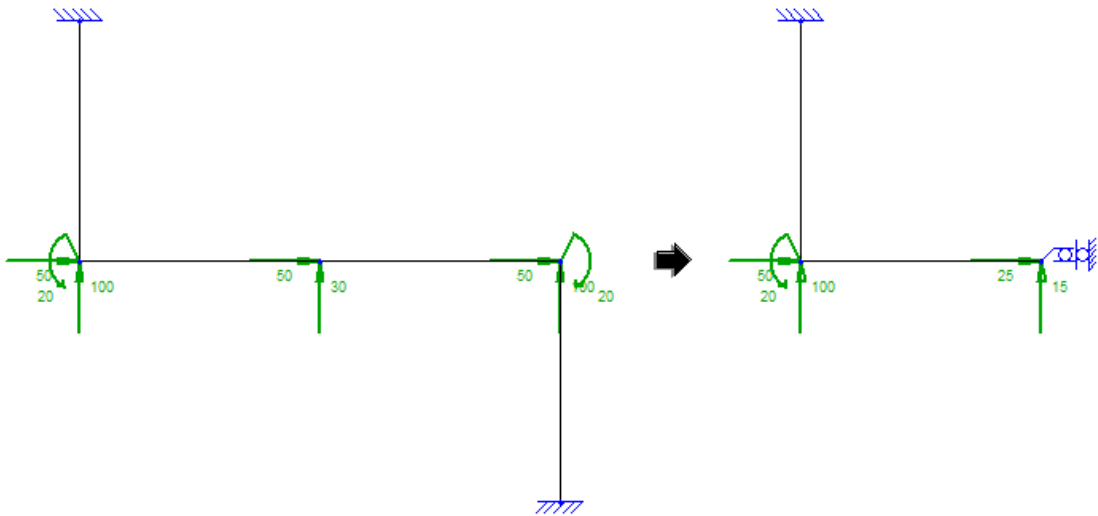
**Case 2: Symmetric structure (axis) + Anti-Symmetric loads (axis)**

For the above example, half the structure can be analyzed using an "010" (x-Roller) constraint for the node on the symmetric axis.

**Case 3: Symmetric structure (point) + Symmetric loads (point)**

For the above example, half the structure can be analyzed using a "110" (Pinned) constraint for the node on the symmetric point.

### Case 4: Symmetric structure (point) + Anti-Symmetric loads (point)



For the above example, half the structure can be analyzed using a "001" constraint for the node on the symmetric 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.**

# Chapter

---



**Example Problems**

## 7 Example Problems

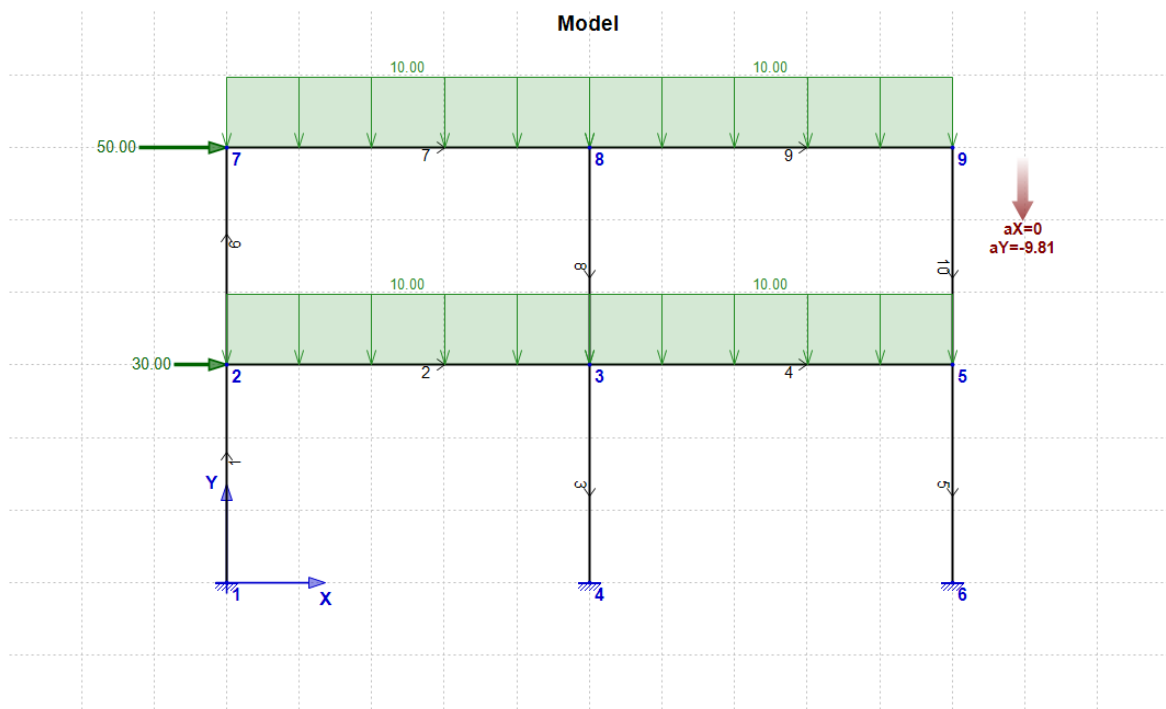
- [Example Problem 1](#)

### 7.1 Example Problem 1

- [Overview - Example 1](#)
- [Step 1. Preparation of the input data](#)
- [Step 2. Define Materials](#)
- [Step 3. Define Sections](#)
- [Step 4. Draw the Model on screen](#)
- [Step 5. Edit Nodes](#)
- [Step 6. Edit Elements](#)
- [Step 7. Define Nodal Loads](#)
- [Step 8. Define Elemental Loads](#)
- [Step 9. Define Body \(Acceleration\) Loads](#)
- [Step 10. Run the Analysis](#)
- [Step 11. View N, V, M Diagrams, Model Deformation and Free Body Diagram](#)
- [Step 12. View the analytical results](#)

#### 7.1.1 Overview - Example 1

The first example is a **two-story, two-bay concrete frame** which is shown in the figure below.





**The properties of the Model are the following:**

### Materials

The material of the Model is **Concrete** with the following properties:

- Elastic Modulus **E = 29 GPa**
- Density **d = 2500 kg/m<sup>3</sup>**

### Sections

There are different sections for the **Columns** and **Beams**:

**Columns:** Square section, **50 cm \* 50 cm**

**Beams:** Rectangular section, **50 cm \* 25 cm** (bending in the major axis - 50 cm is the height of the beam (along the Global Y axis) and 25 cm is its width (perpendicular to the screen))

### Member geometry (Nodes and Elements)

As shown in the Figure above, where **the Grid is 1 m**. So the **height of each floor is 3 m** and each bay has a **span of 5 m**. The total height of the structure is 6 m, while the total width is 10 m.

Elements' orientation is also shown in the figure (It is not very important to define the elements' orientation as shown above).

### Nodal Loads

There is a **FX=30 kN** Load acting on **Node 2** and also a **FX=50 kN** Load acting on **Node 7**, as shown in the figure above.

### Elemental Loads

All the beams have a uniform elemental load of **10 kN/m** at the direction **-Y (Global axes)**, as shown in the figure.

### Body (Acceleration) Loads

The **self-weight** of all the structural elements (columns and beams) has to be taken into consideration in the analysis.

The earth gravitational acceleration is given equal to **g=9.81 m/s<sup>2</sup>** (a typical value).

### 7.1.2 Step 1. Preparation of the input data

The first step is to define the **System of Units**. This is a very important step for the model. This step has nothing to do with the program itself. Instead, the user has to define the preferred system of units and then all the program data have to be consistent with this system. Then the results will also comply to that system. For details, see [System of Units](#).

**We choose a consistent system based on Force. We choose to use:**

- **m** for **Length** (L)
- **kN** for **Force** (F)
- **s** for **Time** (T)

**The derived units are then the following:**

- The **Acceleration unit** is given by:  $L/T^2$  (1 Length unit) / (1 time unit)<sup>2</sup>. In our example: **m/s<sup>2</sup>**
- The **Mass unit** is given by:  $F \cdot T^2 / L$  (1 force unit) / (1 acceleration unit). In our example:  $kN / (m/s^2) = Mg = t$  (**IMPORTANT**: t has to be used instead of kg!)
- The **Density unit** is given by:  $F \cdot T^2 / L^4$  (1 mass unit) / (1 length unit)<sup>3</sup>. In our example: **t/m<sup>3</sup>** (**IMPORTANT**: t/m<sup>3</sup> has to be used instead of kg/m<sup>3</sup>!)
- The **Stress unit** is given by:  $F/L^2$  (1 force unit) / (1 length unit)<sup>2</sup>. In our example: **kN/m<sup>2</sup>**

**Note:** The mass unit (in our example 1 t) is the mass that accelerates by the acceleration unit rate (in our example 1 m/s<sup>2</sup>) when the unit force (in our example 1 kN) is exerted on it.

**The Material properties should be given as:**

- **E** = 29 GPa = **29000000 kN/m<sup>2</sup>**
- **d** = 2500 kg/m<sup>3</sup> = **2.5 t/m<sup>3</sup>**

**The Section properties should be given as:**


- **Columns:** Square section, 50 cm \* 50 cm = **0.50 m \* 0.50 m**
- **Beams:** Rectangular section, 50 cm \* 25 cm = **0.50 m \* 0.25 m**

The **gravity acceleration** should be given as **g = 9.81 m/s<sup>2</sup>**

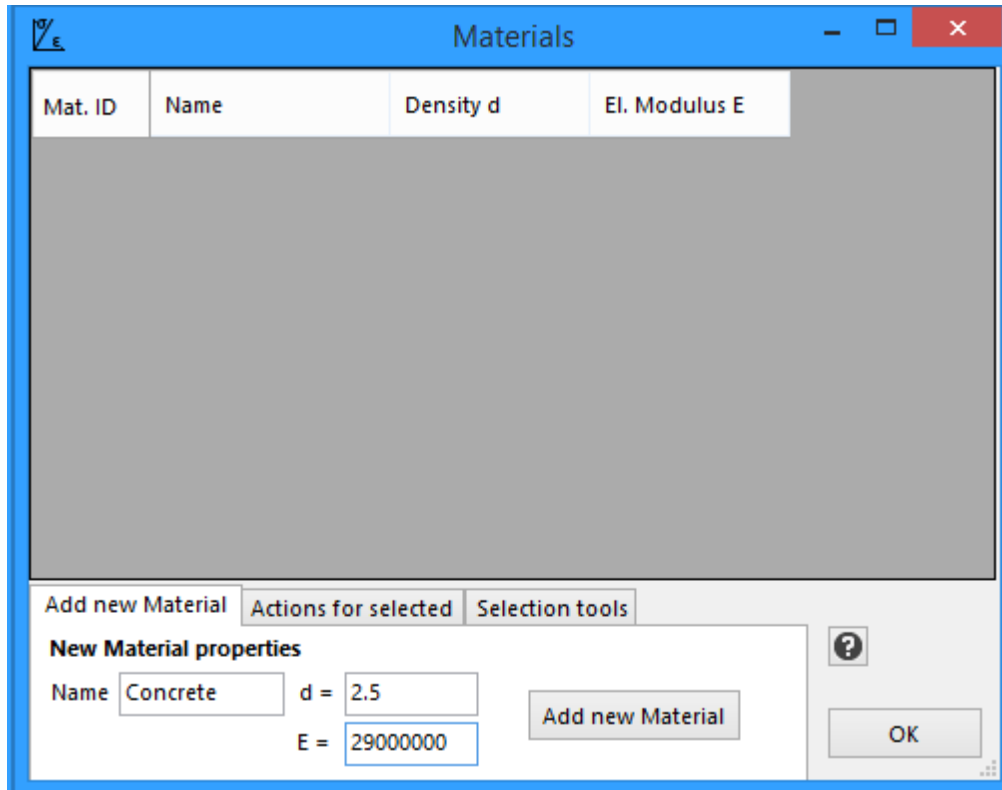
### 7.1.3 Step 2. Define Materials



After the preparation of the input data, it is time to start working with the program.

Click  to define **Material(s)**. For details, see [Materials](#).

You need to define the Material **Name** (optional), **Material Density d** (optional in general, but obligatory if you need to define self-weight loads, as in our case) and Material **Elastic Modulus E** for each Material, as shown below.



Mat. ID	Name	Density d	El. Modulus E
---------	------	-----------	---------------

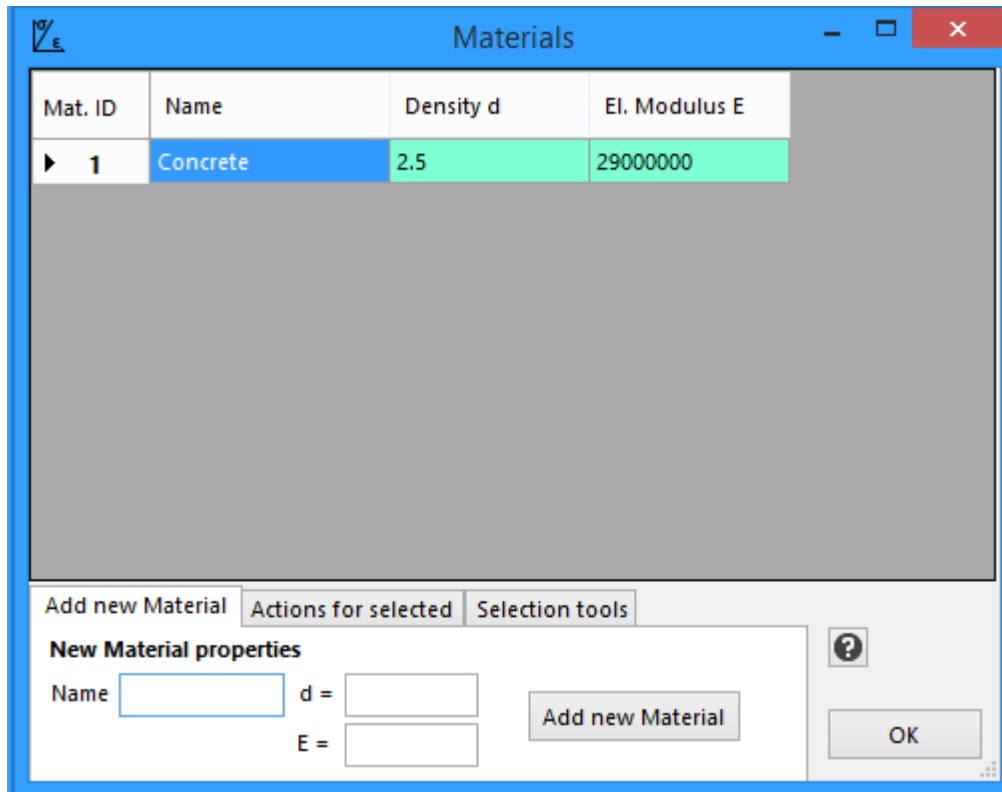
Add new Material | Actions for selected | Selection tools

**New Material properties**

Name  d =

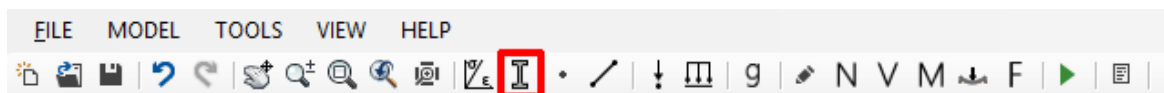
E =

Then **Click the "Add new Material" button**. The new Material will be added to the table that shows the Model Materials.



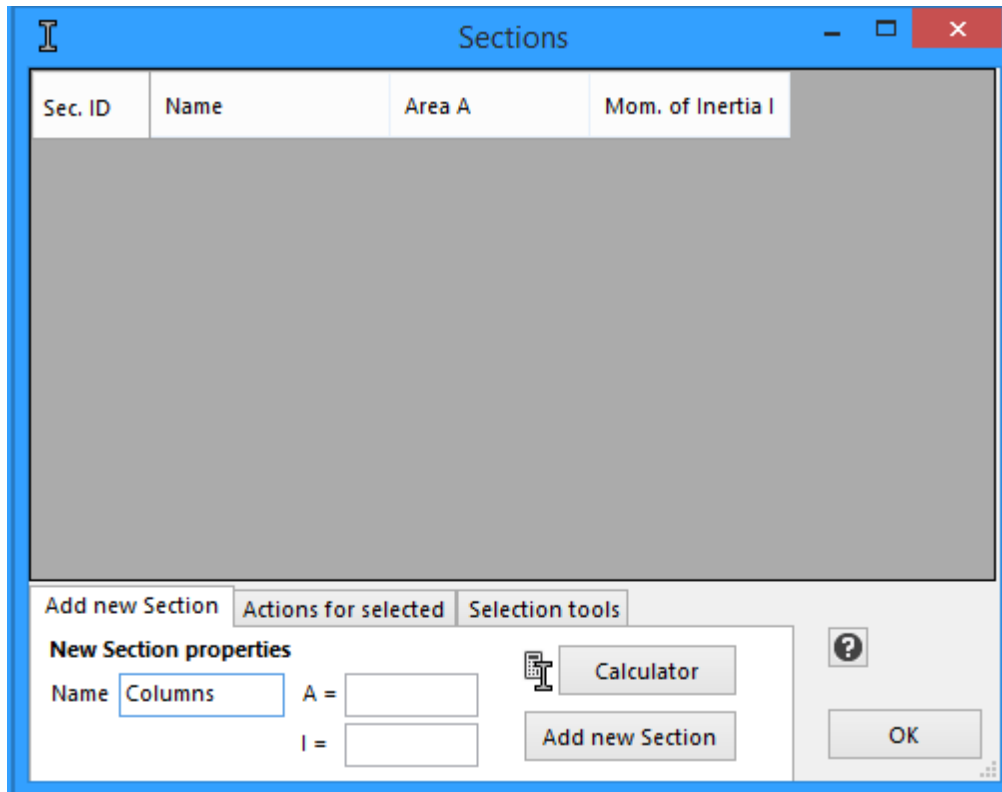
Then **Click OK to exit the Materials form**.

#### 7.1.4 Step 3. Define Sections



After the definition of the Materials, you need to define the Sections of the Model. Click **I** to define **Sections(s)**. For details, see [Sections](#).

You need to define the Section **Name** (optional), Section **Area A** and Section **Moment of Inertia I** for each Section, as shown below. First, type the name of the Section. Let's start with the Columns.



The screenshot shows a software window titled "Sections" with a table and a control panel. The table has four columns: "Sec. ID", "Name", "Area A", and "Mom. of Inertia I". The "Name" column contains the text "Columns". Below the table, there are three tabs: "Add new Section", "Actions for selected", and "Selection tools". Under the "Add new Section" tab, there is a section titled "New Section properties" with three input fields: "Name" (containing "Columns"), "A =", and "I =". To the right of these fields are three buttons: "Calculator", "Add new Section", and "OK".

Sec. ID	Name	Area A	Mom. of Inertia I
	Columns		

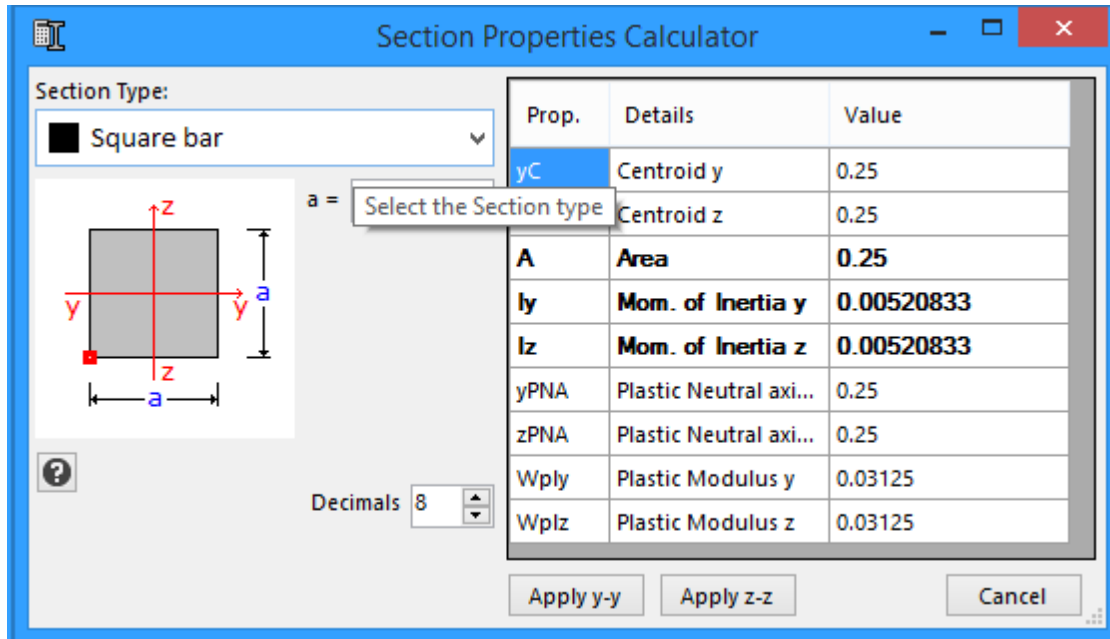
Add new Section    Actions for selected    Selection tools

**New Section properties**

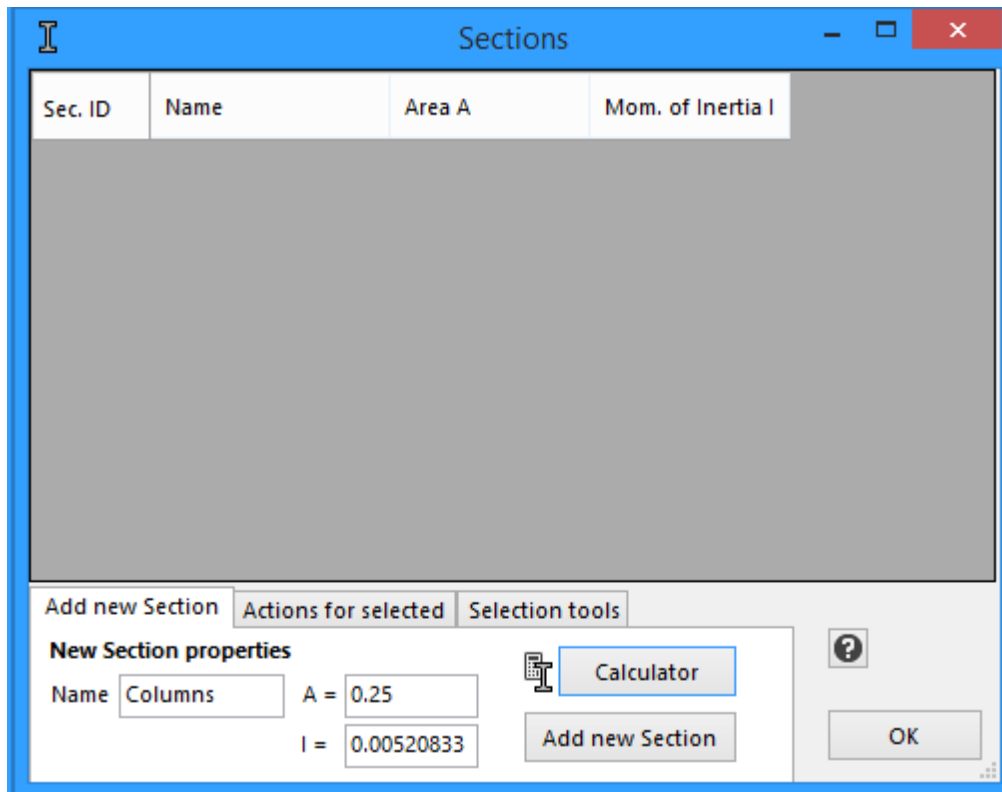
Name:     A =     I =

Calculator    Add new Section    OK

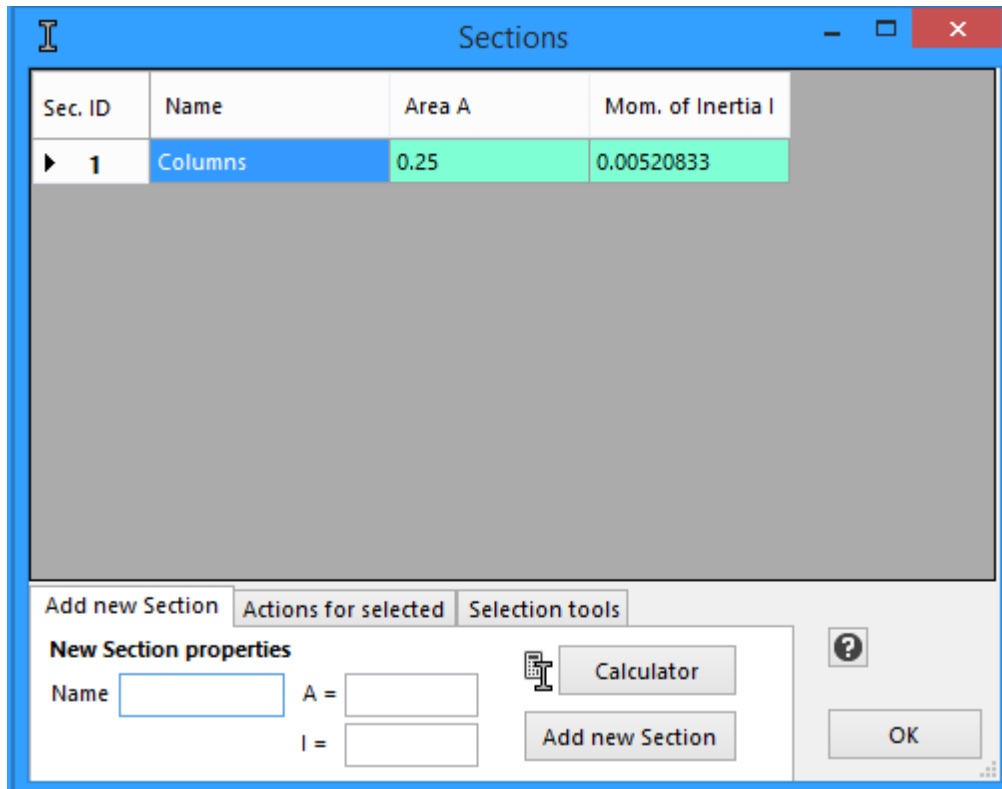
You can calculate the Section properties (A and I) yourself, but the easiest way is to use the "Section Properties Calculator" that is built in EngiLab Beam.2D. **Click the "Calculator" button.** The Section Properties Calculator form appears. Select the Square Bar Section and type "0.5" for the a dimension, as shown below. Press ENTER after you have finished, in order to refresh the table with the Section properties.



Because of the fact that the Section is symmetric, it does not matter if you click "Apply y-y" or "Apply z-z" at this point, as the y and z properties of the section are the same. Let's **Click "Apply y-y"**. Then A and I<sub>y</sub> are transferred to the main Section form, as shown below.

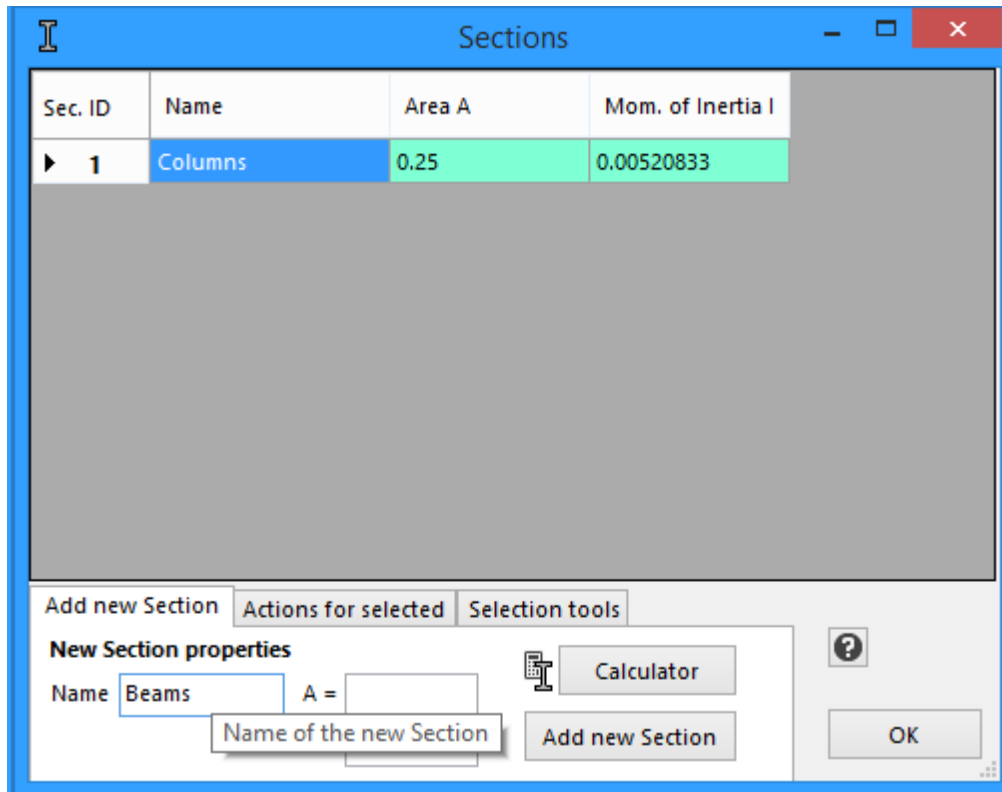


Now **Click the "Add new Section" button**. The new Section will be added to the table that shows the Model Sections.





You now have to continue and add the Section of the Beams. **Type "Beams"** at the Section name, as shown below.



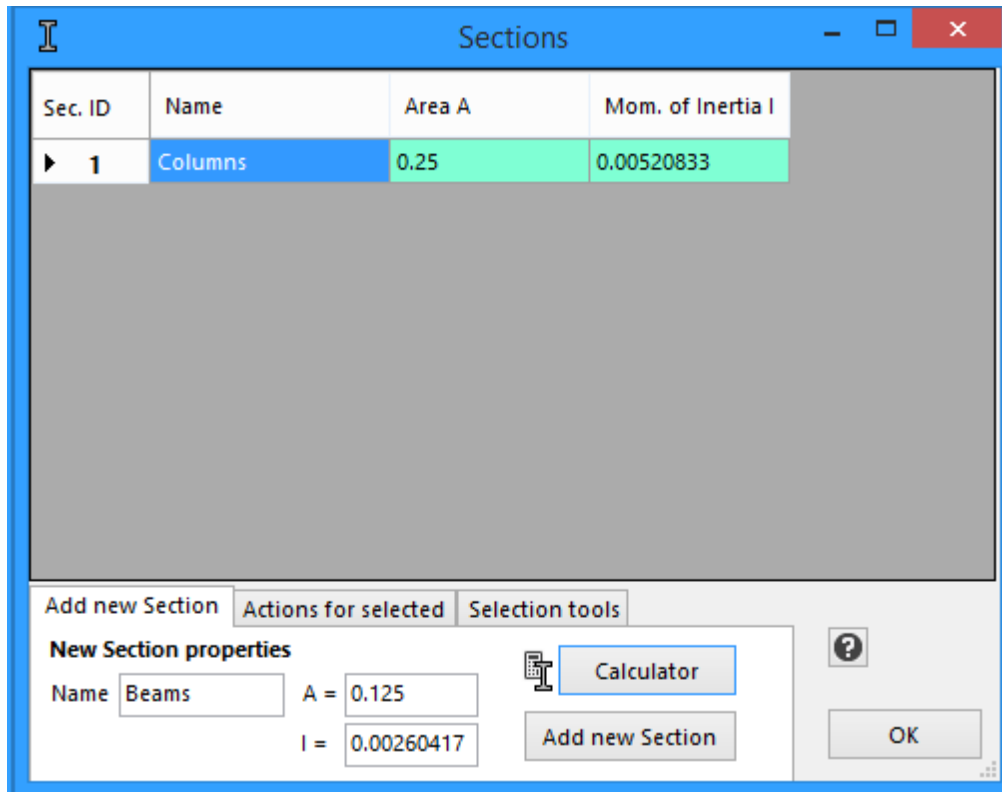
You can use the Section Properties Calculator again. **Click the "Calculator" button.** The Section Properties Calculator form appears. Select the **Rectangular Bar Section** and type "0.25" for the a dimension and "0.50" for the b dimension, as shown below. Press ENTER after you have finished, in order to refresh the table with the Section properties.

The screenshot shows the "Section Properties Calculator" window. The "Section Type" is set to "Rectangular bar". The dimensions are  $a = 0.25$  and  $b = 0.50$ . The diagram shows a rectangular section with a coordinate system where the y-axis is horizontal and the z-axis is vertical. The centroid is at the center. The table below lists the calculated properties:

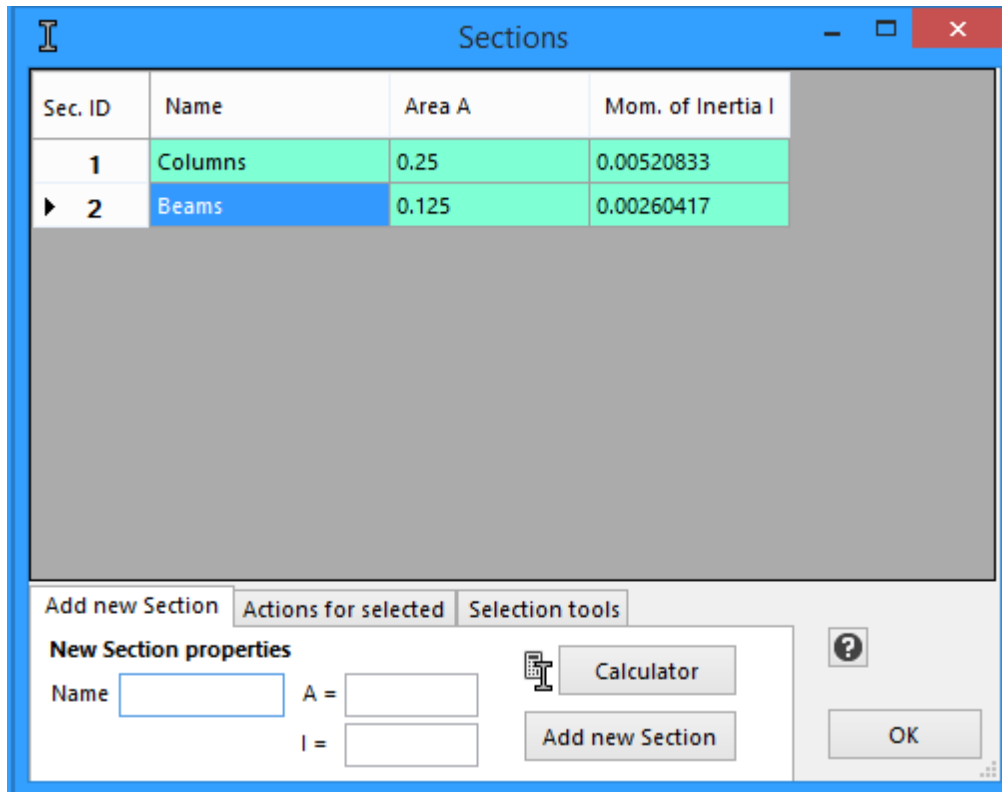
Prop.	Details	Value
yC	Centroid y	0.125
zC	Centroid z	0.25
<b>A</b>	<b>Area</b>	<b>0.125</b>
<b>I<sub>y</sub></b>	<b>Mom. of Inertia y</b>	<b>0.00260417</b>
<b>I<sub>z</sub></b>	<b>Mom. of Inertia z</b>	<b>0.00065104</b>
yPNA	Plastic Neutral axi...	0.125
zPNA	Plastic Neutral axi...	0.25
Wply	Plastic Modulus y	0.015625
Wplz	Plastic Modulus z	0.0078125

At the bottom of the window, there are buttons for "Apply y-y", "Apply z-z", and "Cancel". The "Decimals" field is set to 8.

The bending of the beam is about its strong axis (y-y), so this time you have to **Click "Apply y-y"** button. Then A and Iy properties are transferred to the main Section form, as shown below.



Now **Click the "Add new Section" button**. The new Section will be added to the table that shows the Model Sections.



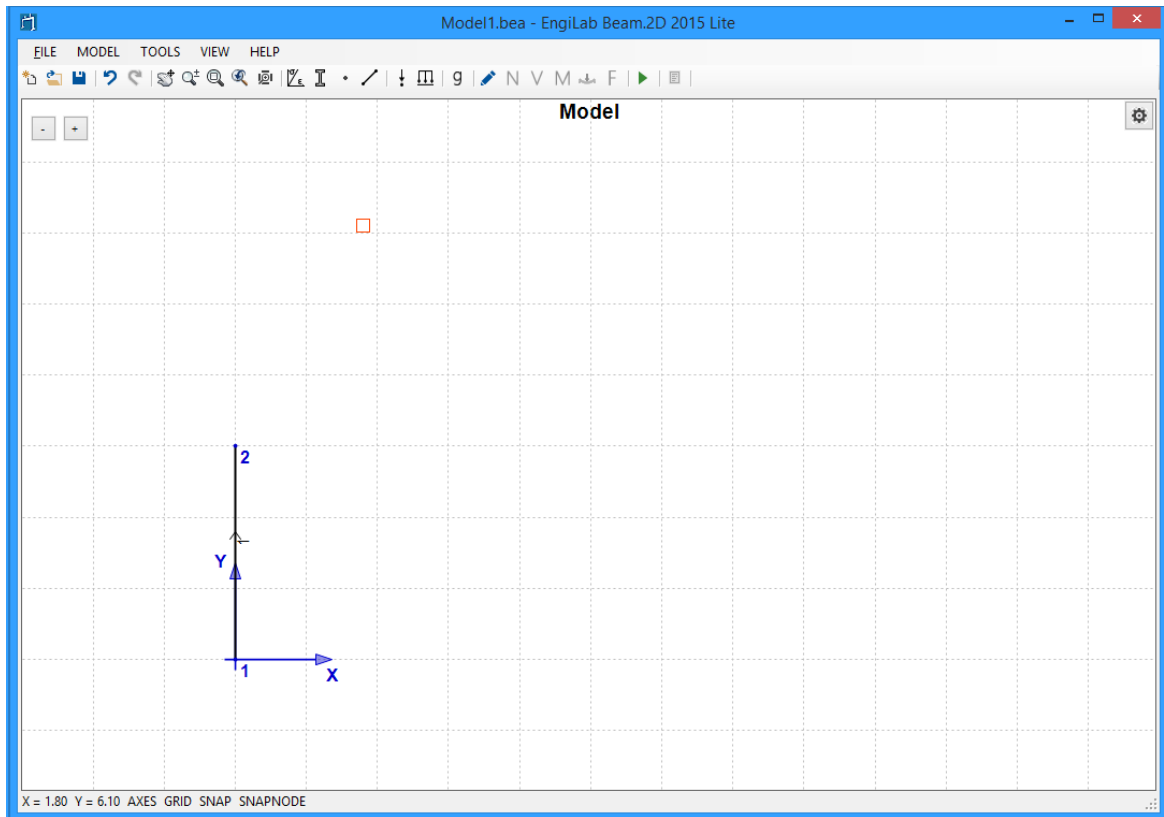
Then **Click OK** to exit the Sections form.

### 7.1.5 Step 4. Draw the Model on screen

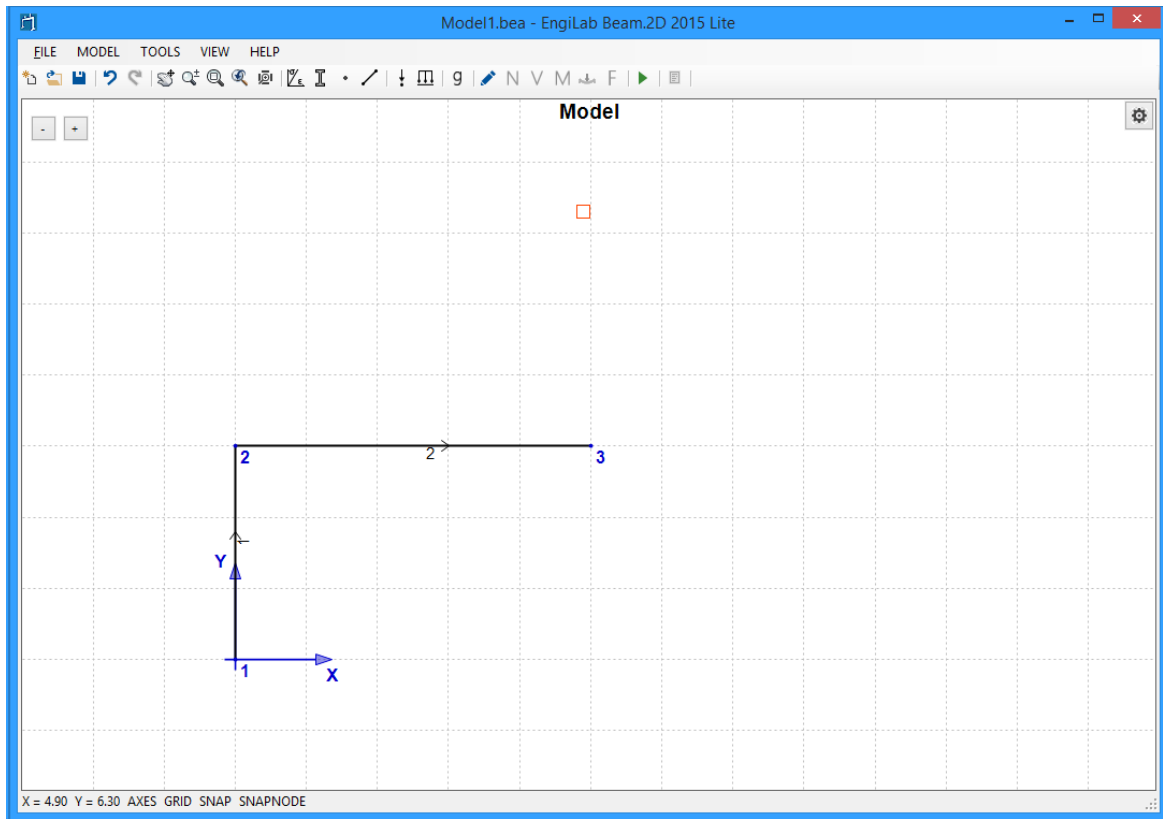
After you have defined the Material and the Sections, you can **start drawing your Model on screen**, as follows:

- First make sure that **SNAP** and **SNAPNODE** are both activated (see bottom right of Picture below).
- If **SNAP** is activated (Default=True), then you can only draw Nodes and Elements at increments defined by the Snap Size setting (Default=0.1). This is fine for our example.
- If **SNAPNODE** is activated (Default=True), then you can "catch" Nodes so that new elements can be connected to existing Nodes. This is essential for building our Model.
- Left-click on screen, hold down the left button and then release it at another location to define a new Element and two nodes at ends  $i, j$ .
- **Note: All Elements that are defined on screen are assigned Material 1 and Section 1.** We will correct this later on.

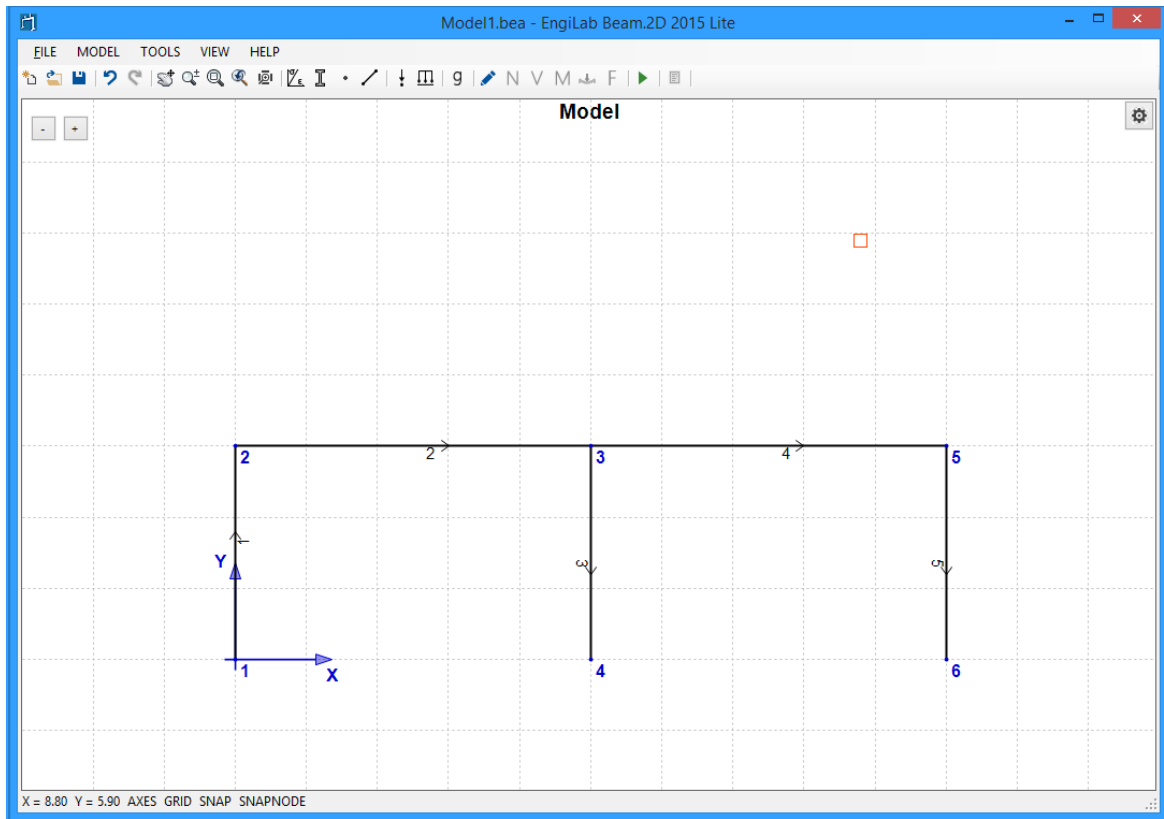
You can start at any point. Let's start from point 0,0 (Origin of axes). Draw the first column as shown below. Release the left button after you have done a distance of 3 in the Y Direction.



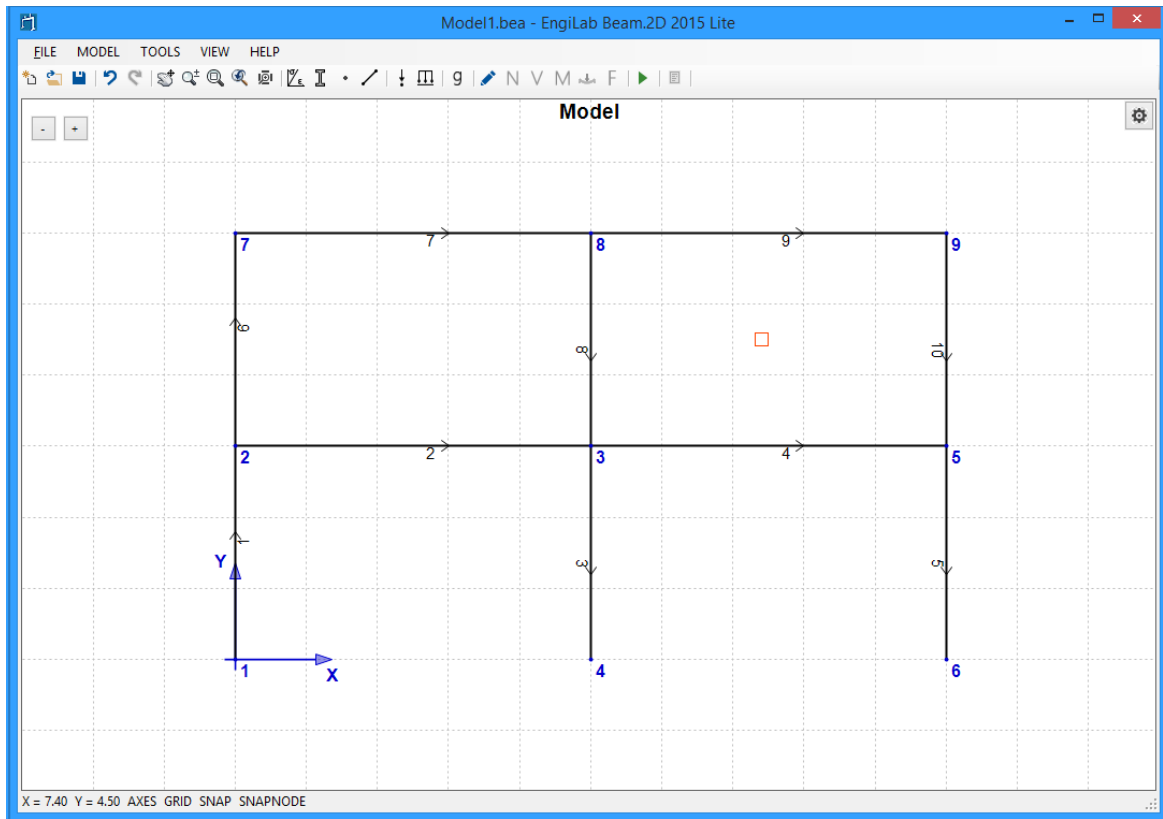
Continue with the Beam. Hold down the left button starting at Node 2 and release it at distance 5 in the X Direction. The picture should be as follows.



Using the same technique, continue with the other Columns and Beams of the first story. The picture should be as follows.



Continue with the Columns and Beams of the second story. Finally the picture should look as follows.



That's it! Now you have built the basic Model. But you need to apply some corrections. For example, now all Elements are assigned Section 1 (Columns), but the horizontal members have to be corrected as they must be assigned Section 2 (Beams). Also, loads (Nodal and Elemental) and also supports have to be added to the Model.

### 7.1.6 Step 5. Edit Nodes



After the preparation of the Model on screen, you will need to do some corrections on the Nodes that have been generated.

For example you may want to:

- Move Nodes to their exact positions, if needed.
- **Define or change Nodal Constraints (Supports).**
- Define Springs.



Click \* to edit **Nodes**. For details, see [Nodes](#).

The screenshot shows a software window titled "Nodes" with a table of node data and a control panel at the bottom.

Node ID	X	Y	DX-Con	DY-Con	RZ-Con	DX-Stiff	DY-Stiff	RZ-Stiff
▶ 1	0	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
2	0	3	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
3	5	3	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
4	5	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
5	10	3	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
6	10	0	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
7	0	6	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
8	5	6	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
9	10	6	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0

Below the table, the control panel includes:

- Buttons: "Add new Node", "Actions for selected", "Selection tools"
- Section: "New Node coordinates" with a help icon (?)
- Input fields: "X=" and "Y="
- Buttons: "Add new Node" and "OK"

In our example, there is no need to move Nodes, as they are already in their correct positions. What we need to do is to add Constraints (Supports) to the Model. **Set the Constraints for Nodes 1, 4 and 6** as follows.

Node ID	X	Y	DX-Con	DY-Con	RZ-Con	DX-Stiff	DY-Stiff	RZ-Stiff
1	0	0	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0	0	0
2	0	3	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
3	5	3	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
4	5	0	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0	0	0
5	10	3	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
▶ 6	10	0	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0	0	0
7	0	6	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
8	5	6	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
9	10	6	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0

Add new Node    Actions for selected    Selection tools

**New Node coordinates** ?

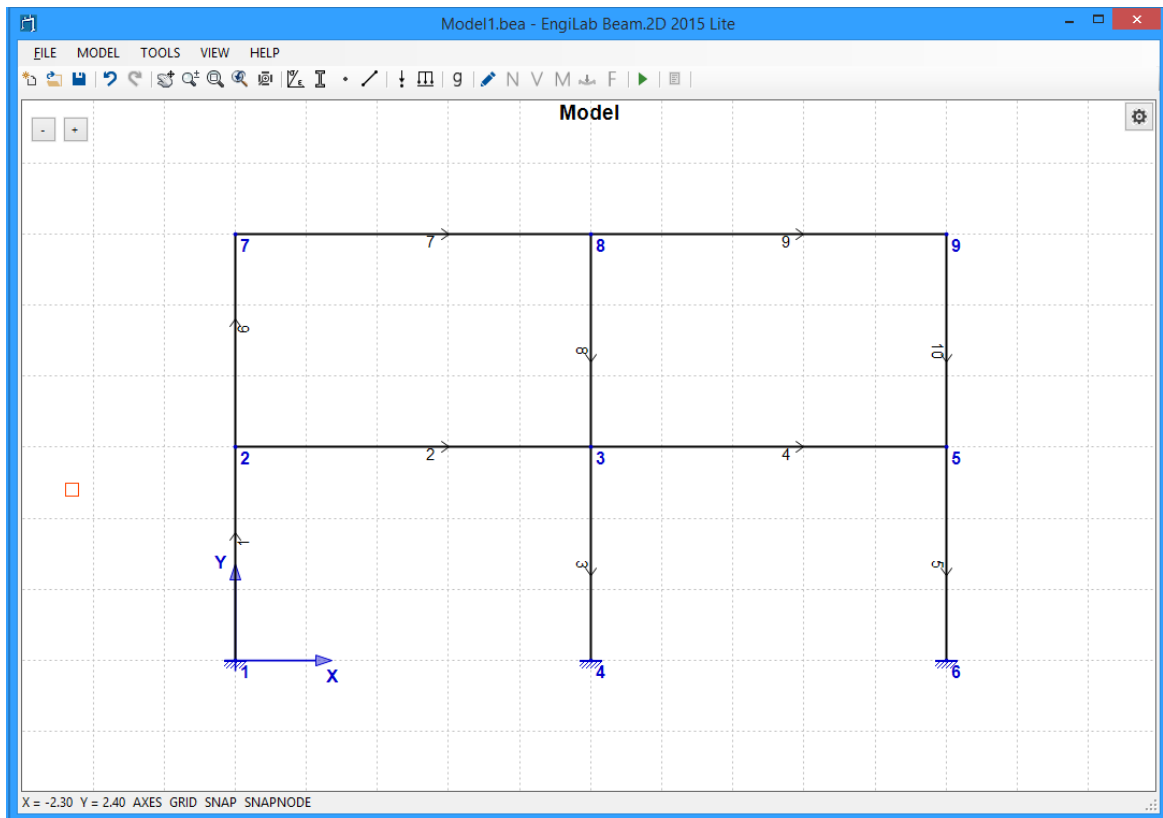
X=

Y=

Add new Node

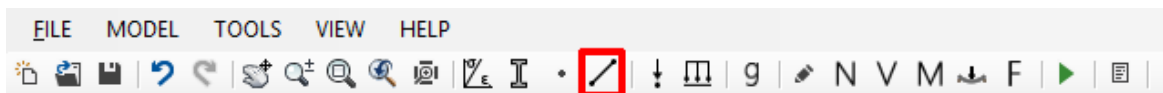
OK

Then **Click OK**. The Model should now look like this:



Fixed supports have been added to Nodes 1, 4, 6, as shown above.


### 7.1.7 Step 6. Edit Elements



Now let's correct the Elements. For example you may want to:

- **Assign the right Material and Section to every Element**, if needed. All Elements that are defined on screen are assigned Material 1 and Section 1.
- Define Hinges at Element ends. Each Element can have a hinge at either end (Start-i or End-j or both).

Click  to edit **Elements**. For details, see [Elements](#).


Elements
- □ ×

Elem. ID	Material	Section	Node i	Node j	Hinge i	Hinge j	Length
▶ 1	1. Concrete	1. Columns	1	2	<input type="checkbox"/>	<input type="checkbox"/>	3.00
2	1. Concrete	1. Columns	2	3	<input type="checkbox"/>	<input type="checkbox"/>	5.00
3	1. Concrete	1. Columns	3	4	<input type="checkbox"/>	<input type="checkbox"/>	3.00
4	1. Concrete	1. Columns	3	5	<input type="checkbox"/>	<input type="checkbox"/>	5.00
5	1. Concrete	1. Columns	5	6	<input type="checkbox"/>	<input type="checkbox"/>	3.00
6	1. Concrete	1. Columns	2	7	<input type="checkbox"/>	<input type="checkbox"/>	3.00
7	1. Concrete	1. Columns	7	8	<input type="checkbox"/>	<input type="checkbox"/>	5.00
8	1. Concrete	1. Columns	8	3	<input type="checkbox"/>	<input type="checkbox"/>	3.00
9	1. Concrete	1. Columns	8	9	<input type="checkbox"/>	<input type="checkbox"/>	5.00
10	1. Concrete	1. Columns	9	5	<input type="checkbox"/>	<input type="checkbox"/>	3.00

Add new Element
Actions for selected
Selection tools

**New Element properties**

Material
1. Concrete ▼

Section
1. Columns ▼

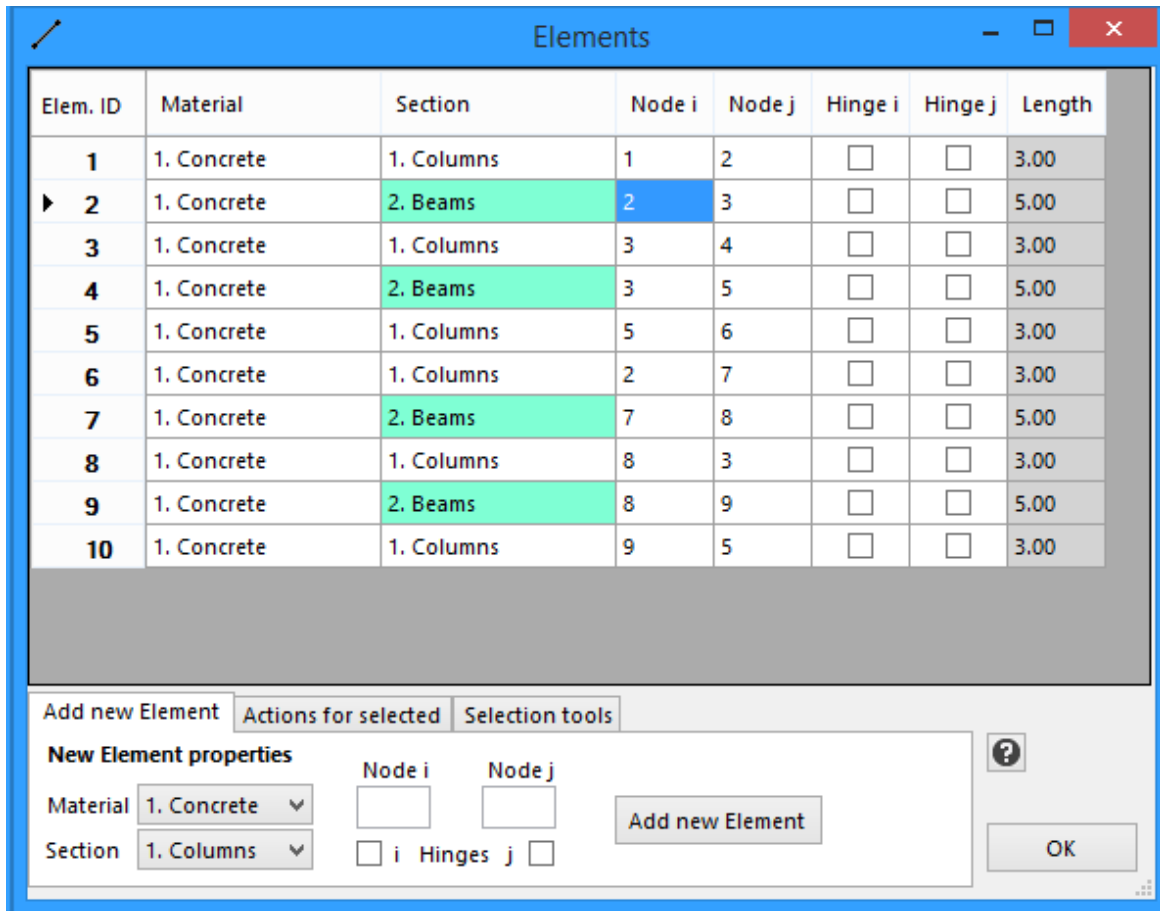
Node i
Node j

i Hinges
 j

Add new Element
OK

?

In our example, we need to assign the right Section (Beams) to the horizontal members. There are no hinges. Edit **Elements 2, 4, 7 and 9** and assign them the second Section (Beams) as follows.



Then **Click OK** to exit Elements.

### 7.1.8 Step 7. Define Nodal Loads



Click ↓ to define **Nodal Loads**. For details, see [Nodal Loads](#).

The Nodal Loads form appears.

The screenshot shows a software window titled "Nodal Loads" with a blue border and standard Windows window controls (minimize, maximize, close). The window contains a table with the following headers: "Node ID", "Force FX", "Force FY", and "Moment MZ". The table body is currently empty. Below the table is a control panel with three tabs: "Add new Nodal Load" (selected), "Actions for selected", and "Selection tools". Under the "Add new Nodal Load" tab, there is a "New Nodal Load" section. It includes a "Node ID" input field, two radio buttons for "Cartesian (FX, FY)" (selected) and "Polar (F,  $\theta$ )", and three input fields for "FX=", "FY=", and "MZ=". To the right of these fields is an "Add New" button. Further right is a help icon (question mark) and an "OK" button.

Add a Nodal Load **FX=30 for Node 2.**

Node ID	Force FX	Force FY	Moment MZ

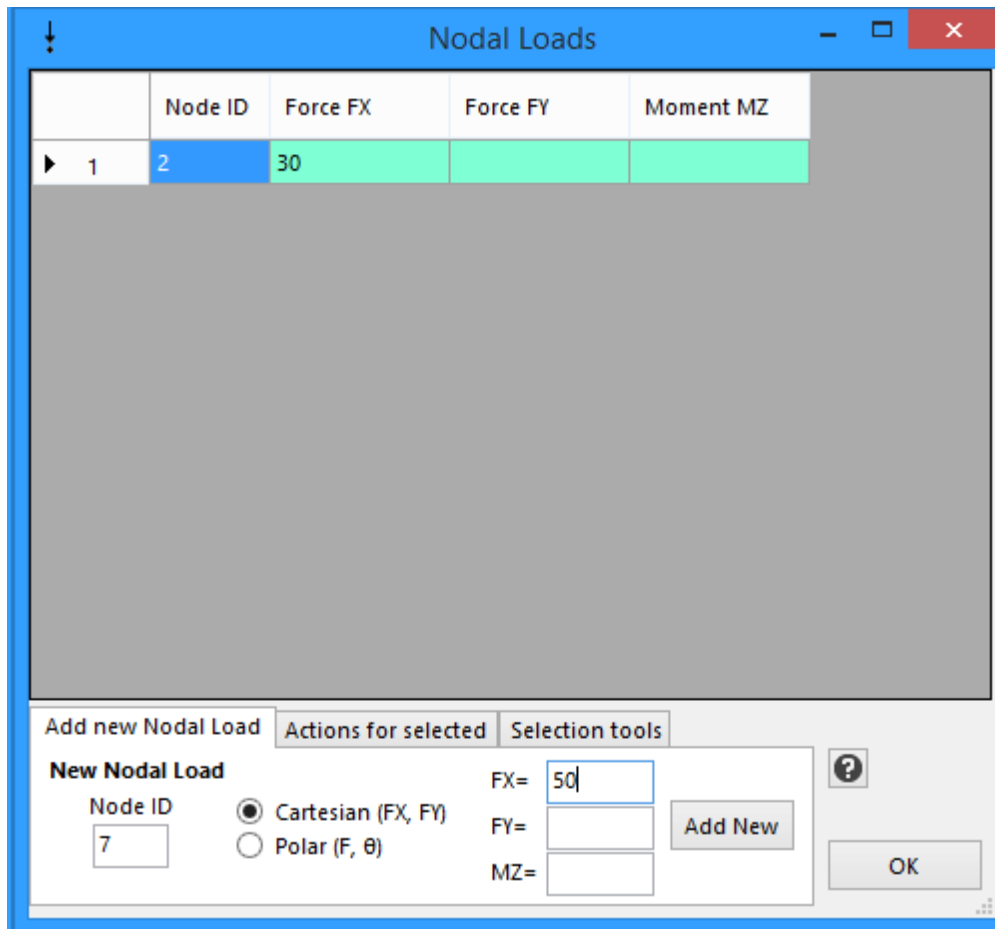
Add new Nodal Load    Actions for selected    Selection tools

**New Nodal Load**  
 Node ID:

Cartesian (FX, FY)     FX=  
 Polar (F, θ)     FY=  
 MZ=

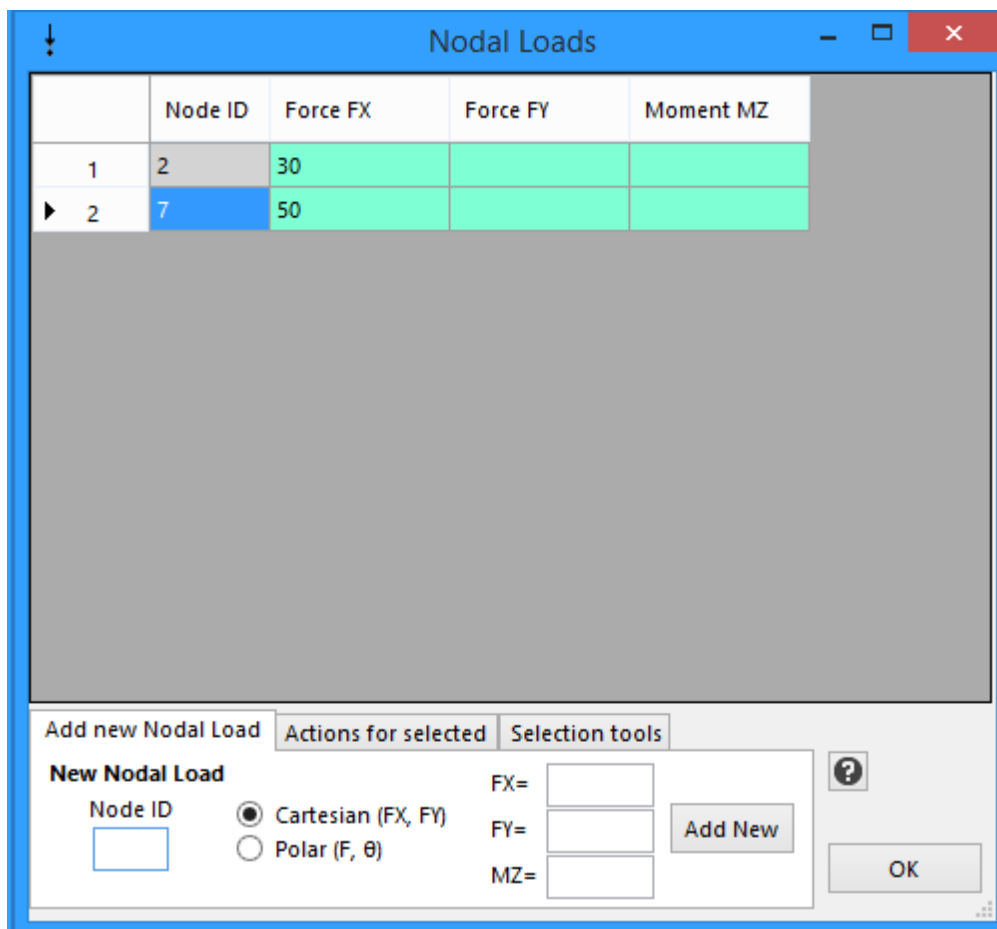
Add New    ?    OK

Click "Add Nodal Load". Then add a Nodal Load **FX=50 for Node 7**.

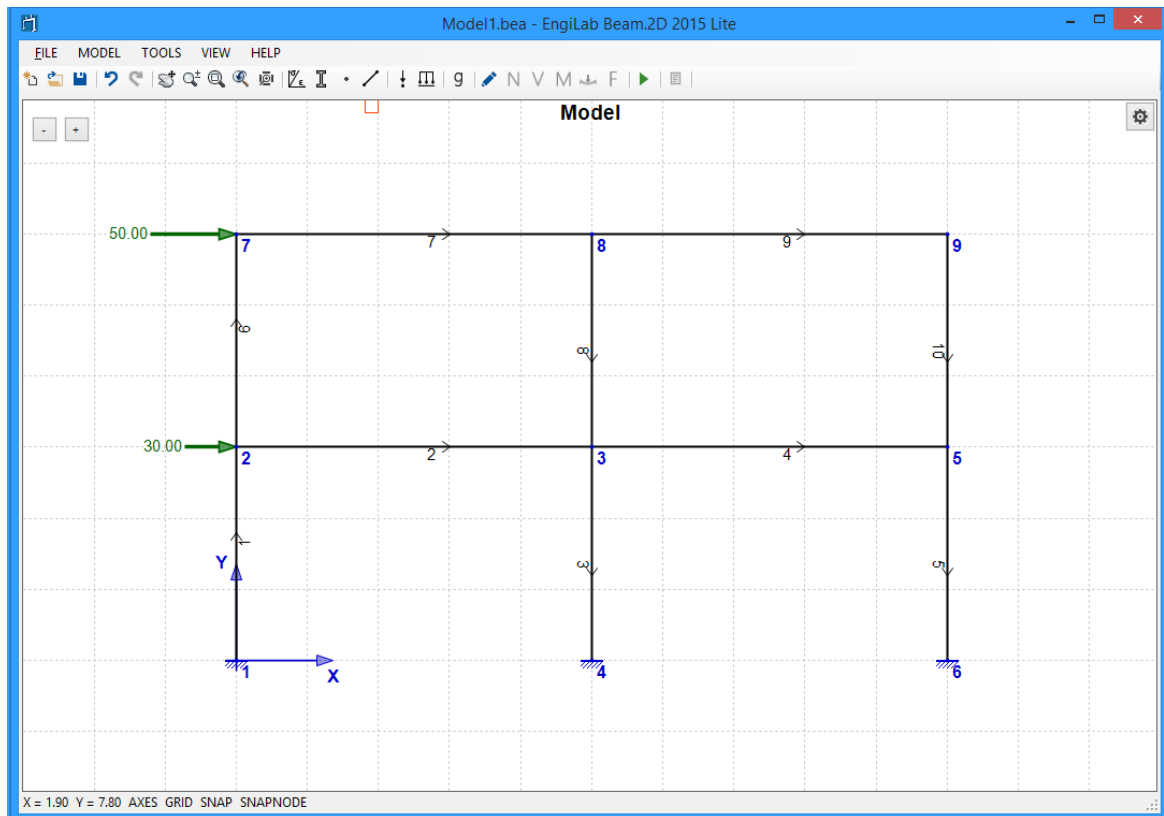




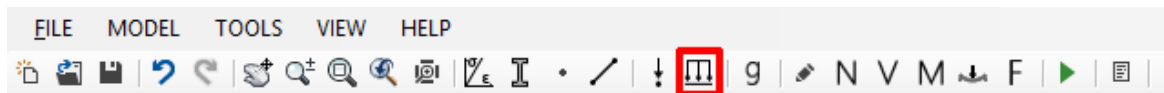
Click "Add Nodal Load". Now the Nodal Loads should look like this.



Click **OK to exit**. Now the Model should look like this.



### 7.1.9 Step 8. Define Elemental Loads



Click  to define **Elemental Loads**. For details, see [Elemental Loads](#).

The Elemental Loads form appears.

Elem. ID	Force fXi	Force fXj	Force fYi	Force fYj

Add new Elemental Load    Actions for selected    Selection tools

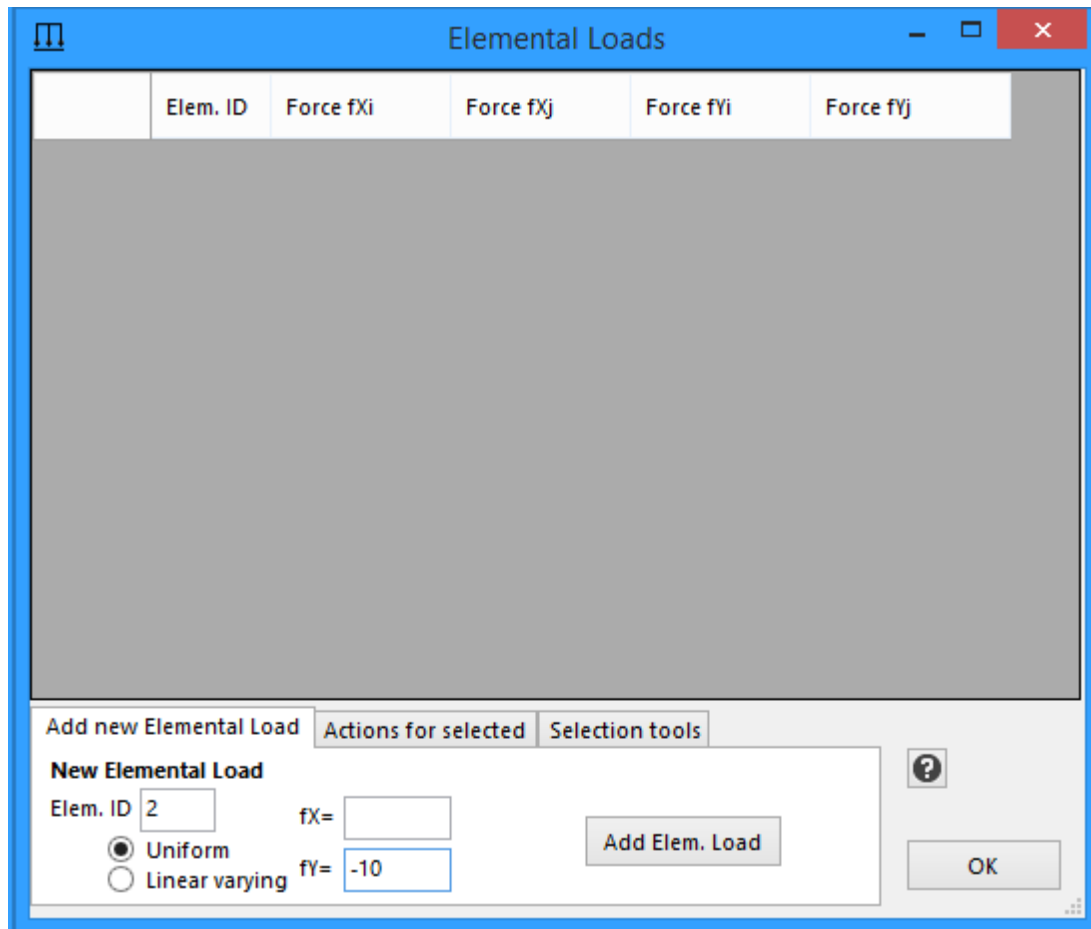
**New Elemental Load**

Elem. ID     fX=    

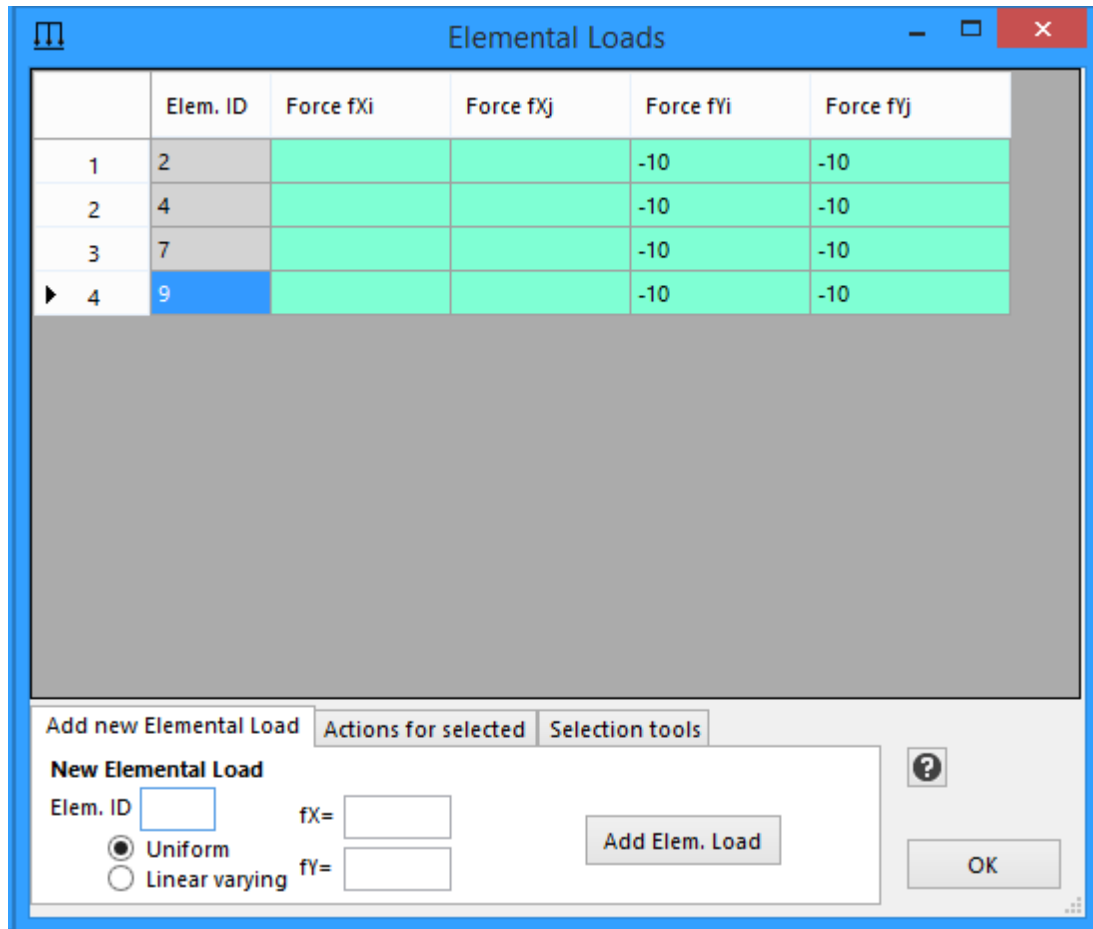
Uniform    fY=

Linear varying

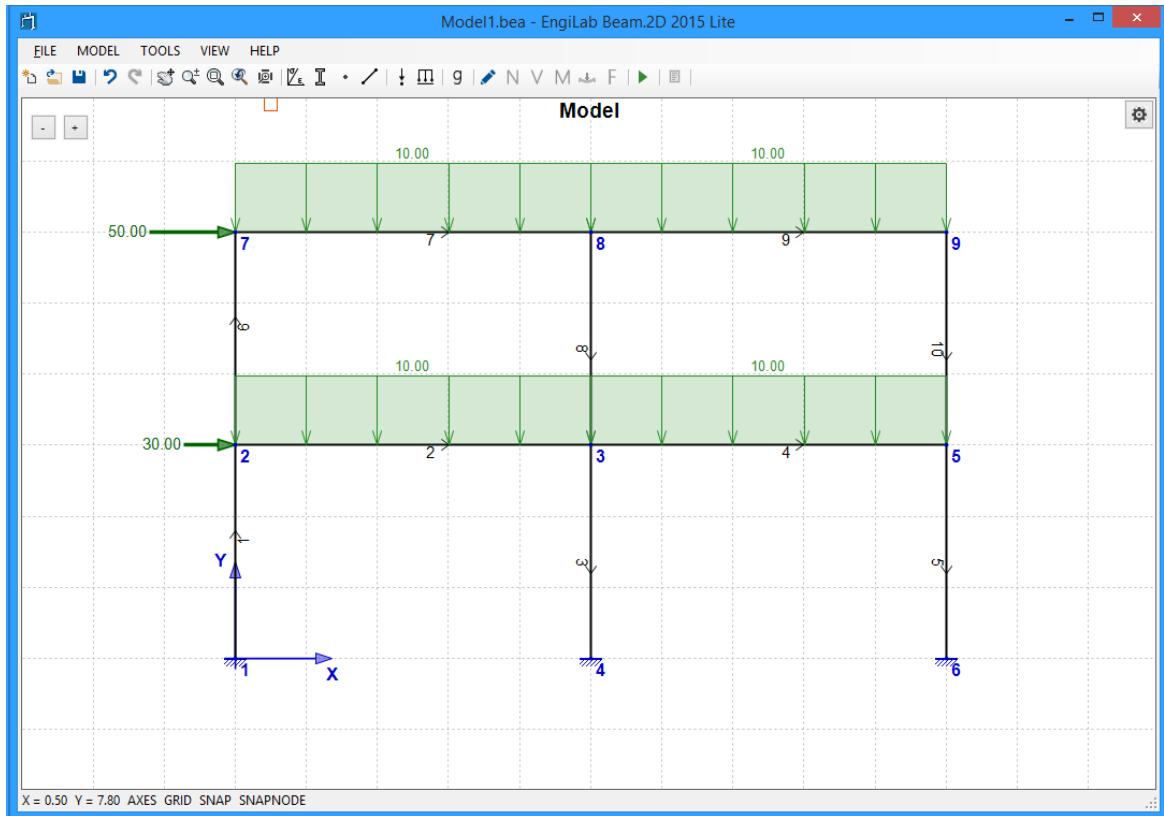
Add a **Uniform Elemental Load  $f_y = -10$  for Element 2.**



Click **"Add Elemental Load"**. Then using the same technique, add Elemental Loads also for **Elements 4, 7, 9**. After you finish, you should see the following picture:



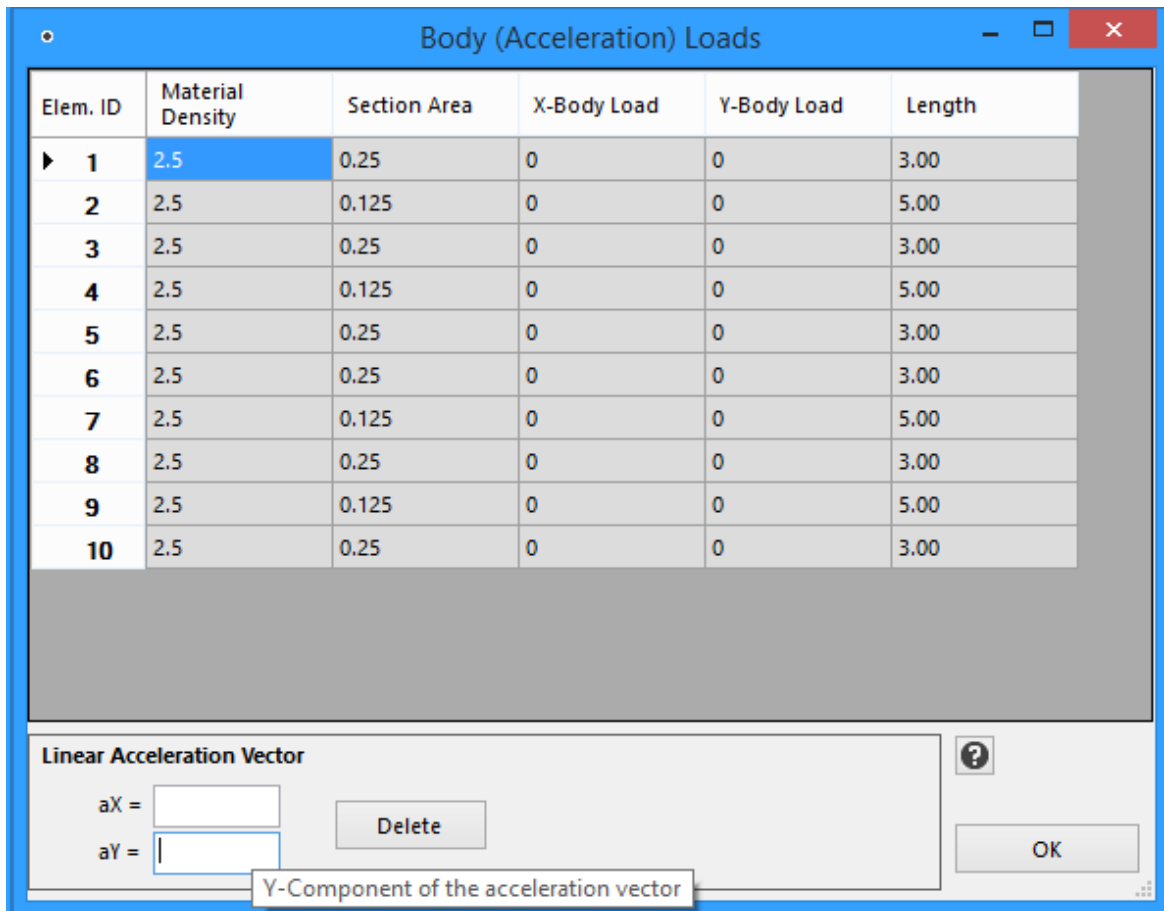
Click OK to exit. Now the Model should look like this.



### 7.1.10 Step 9. Define Body (Acceleration) Loads

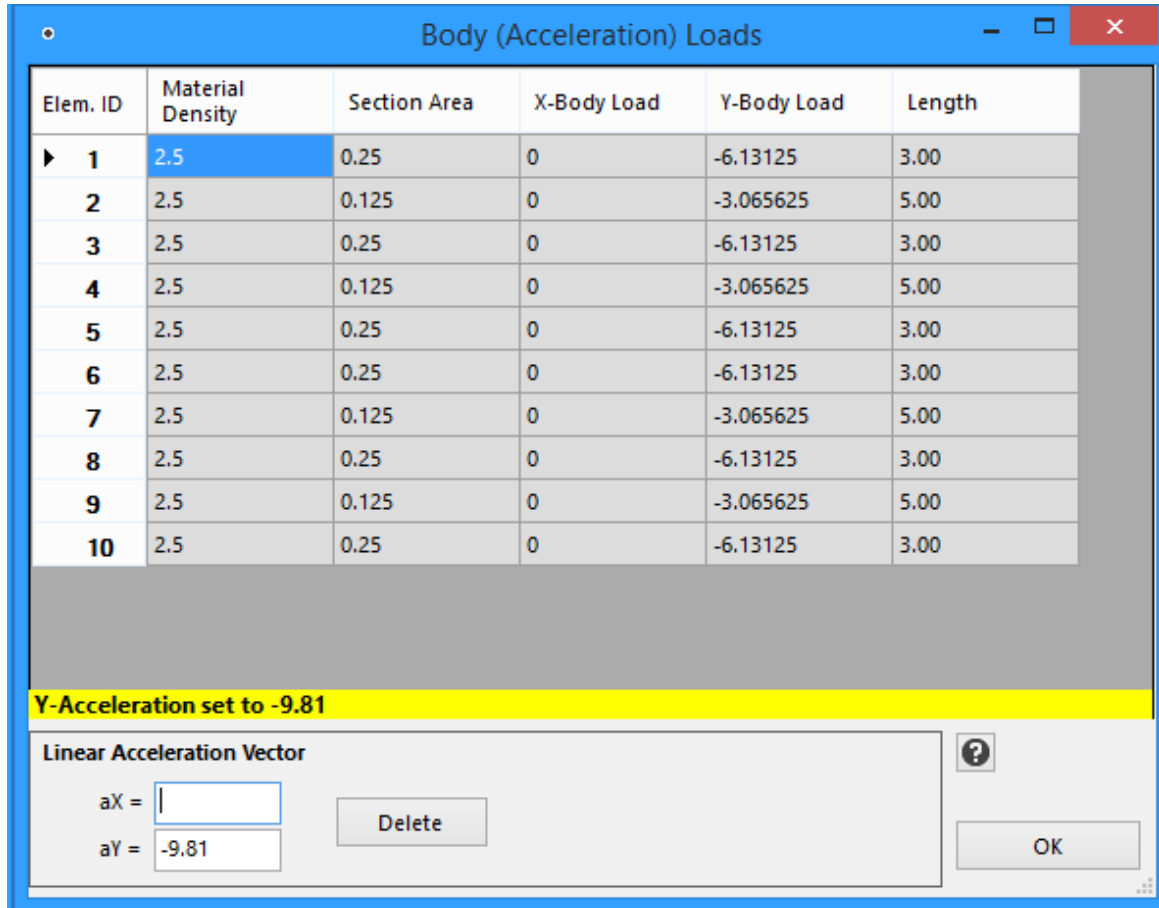


Click **g** to define **Body (Acceleration) Loads**. For details, see [Body \(Acceleration\) Loads](#).



We want to take into account the self-weight of Elements as an additional elemental load for each Element. So we have to provide the **Material Density** for the Material of each Element and also to define a Linear Acceleration Vector equal to the standard earth gravitational acceleration. We should put the earth gravitational acceleration with a minus (-) sign at the Y direction - this means gravity acting towards -Y global axis.

We are using kN for forces, m for length and s for time, so the Material Density has to be given in  $t/m^3$  (we have already done that) and we need to **input -9.81 at the aY component of the Linear Acceleration Vector**. After you type -9.81, make sure that you **press the ENTER key** in order for the changes to take effect. You should see the following picture.



You see that the program has automatically calculated the self weight (in kN/m) for each Element. This is a read-only property. **Click OK to exit the Body (Acceleration) Loads.**

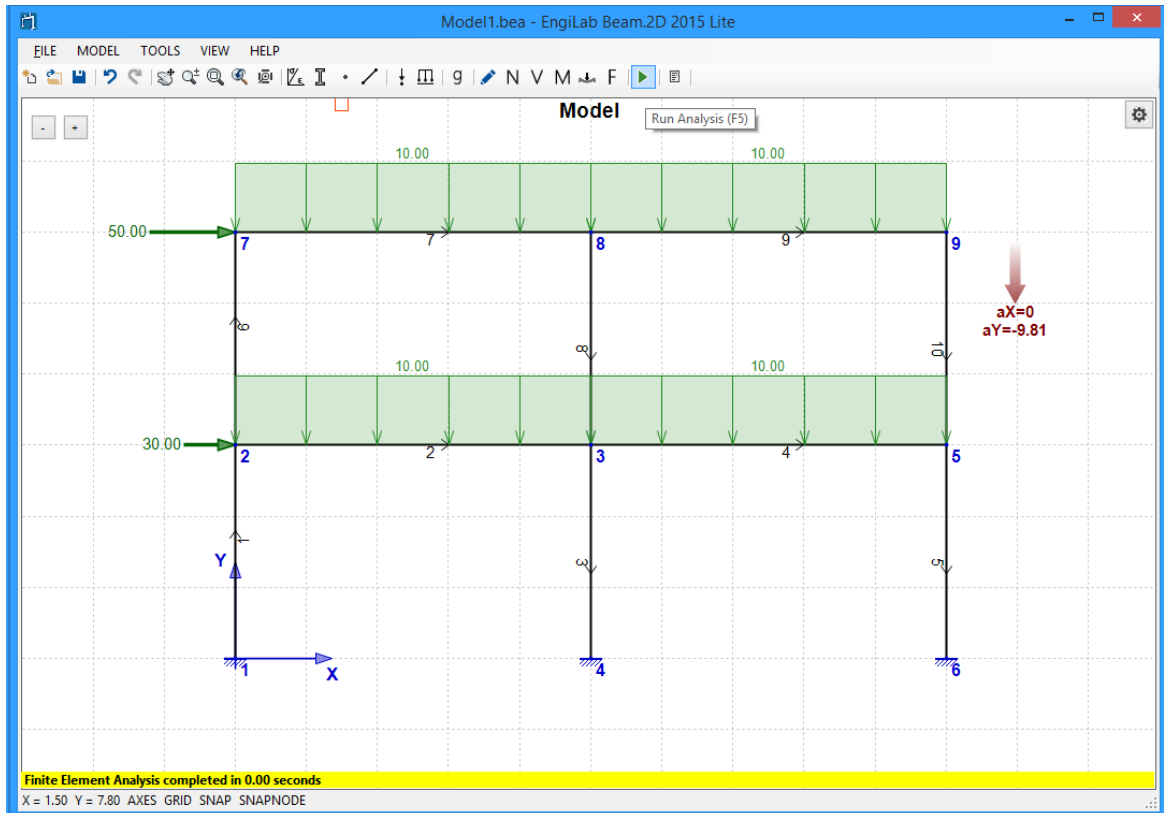


### 7.1.11 Step 10. Run the Analysis



Click  (or press F5) to **run the Finite Element Analysis**. For details on the Analysis, see [Analysis](#).

You should see the following picture.



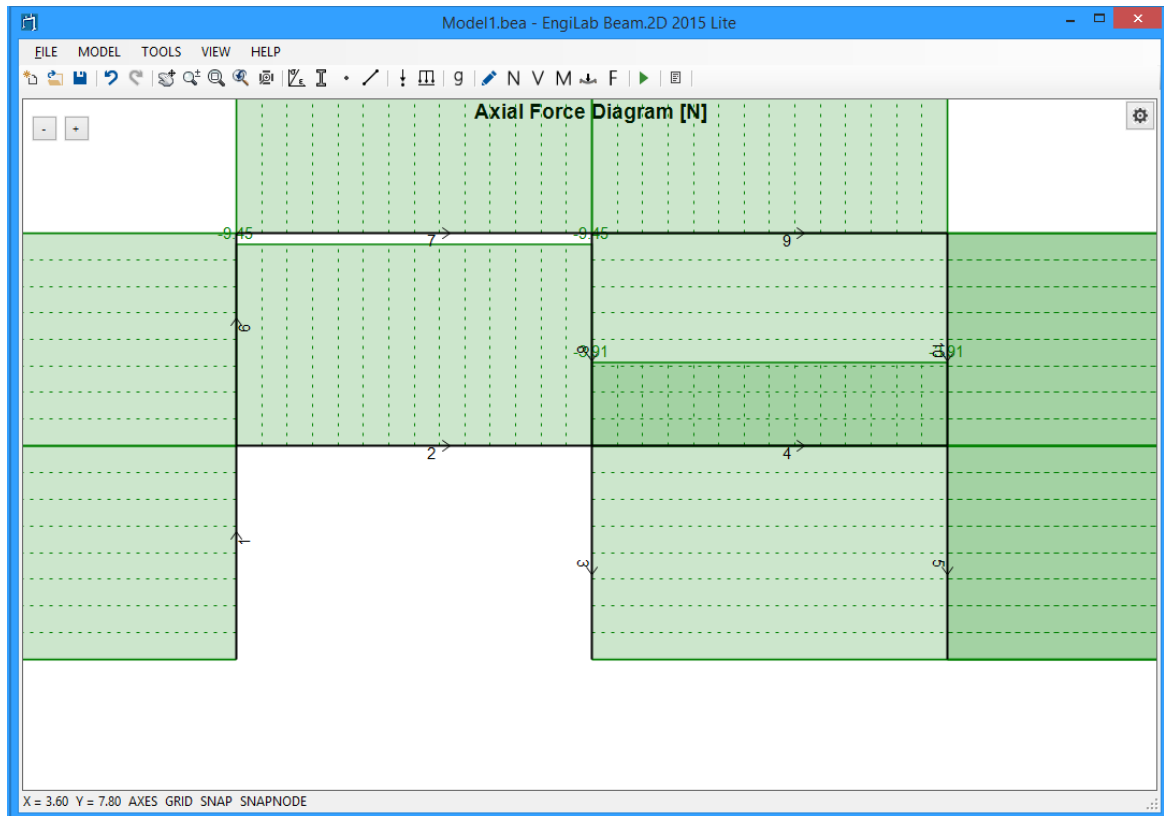
The Finite Element Analysis has been completed successfully.


### 7.1.12 Step 11. View N, V, M Diagrams, Model Deformation and Free Body Diagram



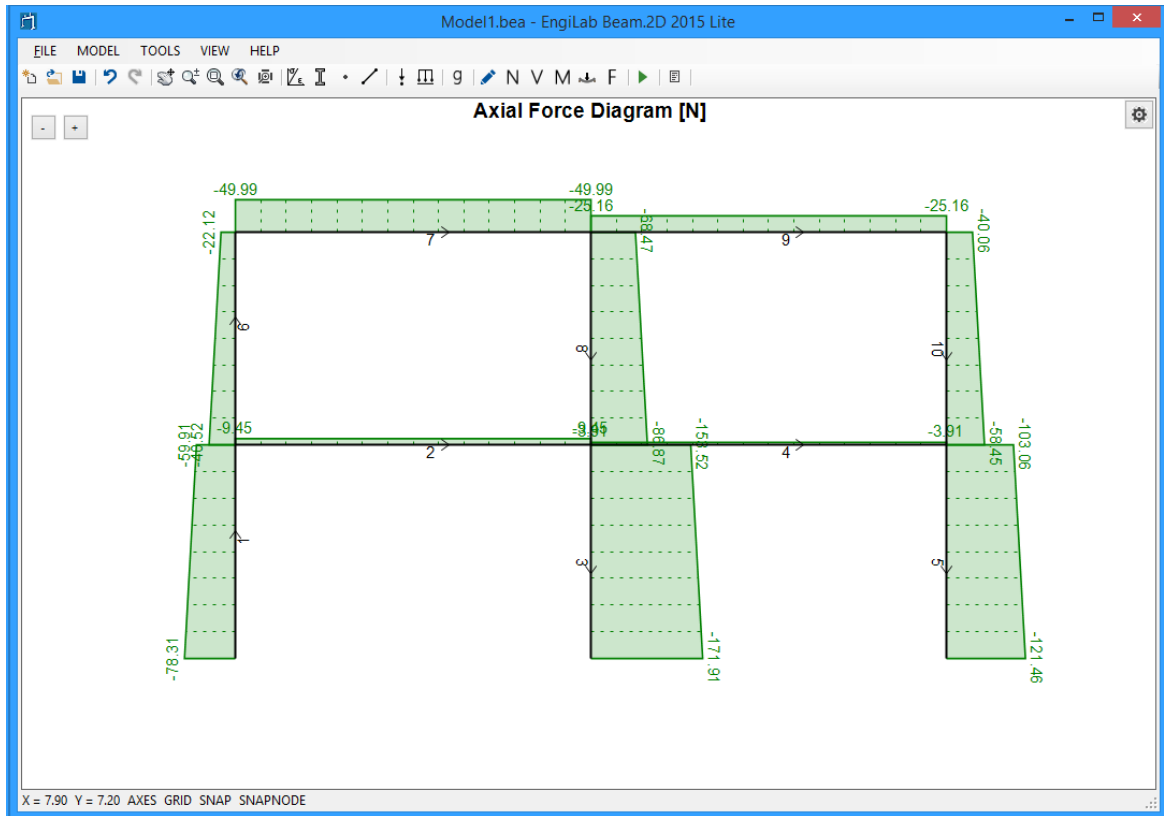
#### 1. Axial Force Diagram

Click **N**, to see the Axial Force Diagram. You may take the picture below.




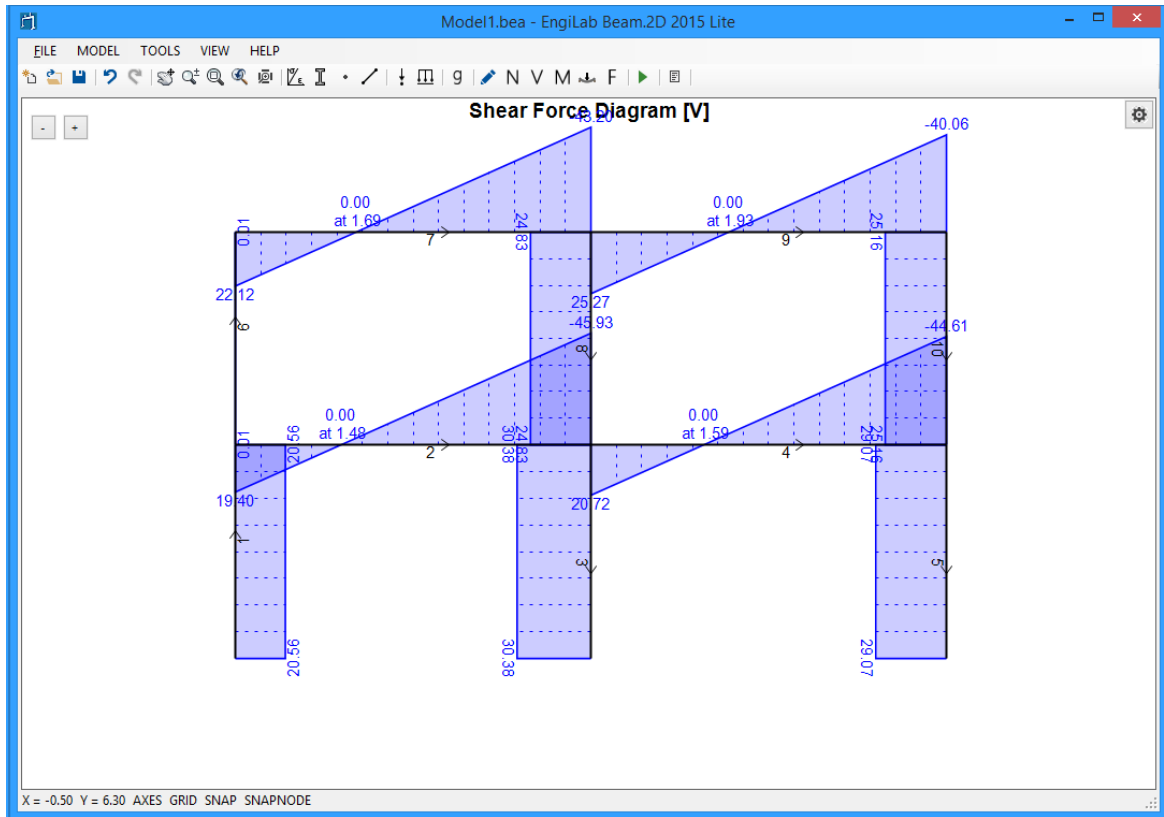
The Axial Force Diagram is out of Scale. Click the "Zoom All" button  to automatically scale the Diagram.

Now the result should look like this, which is much better.




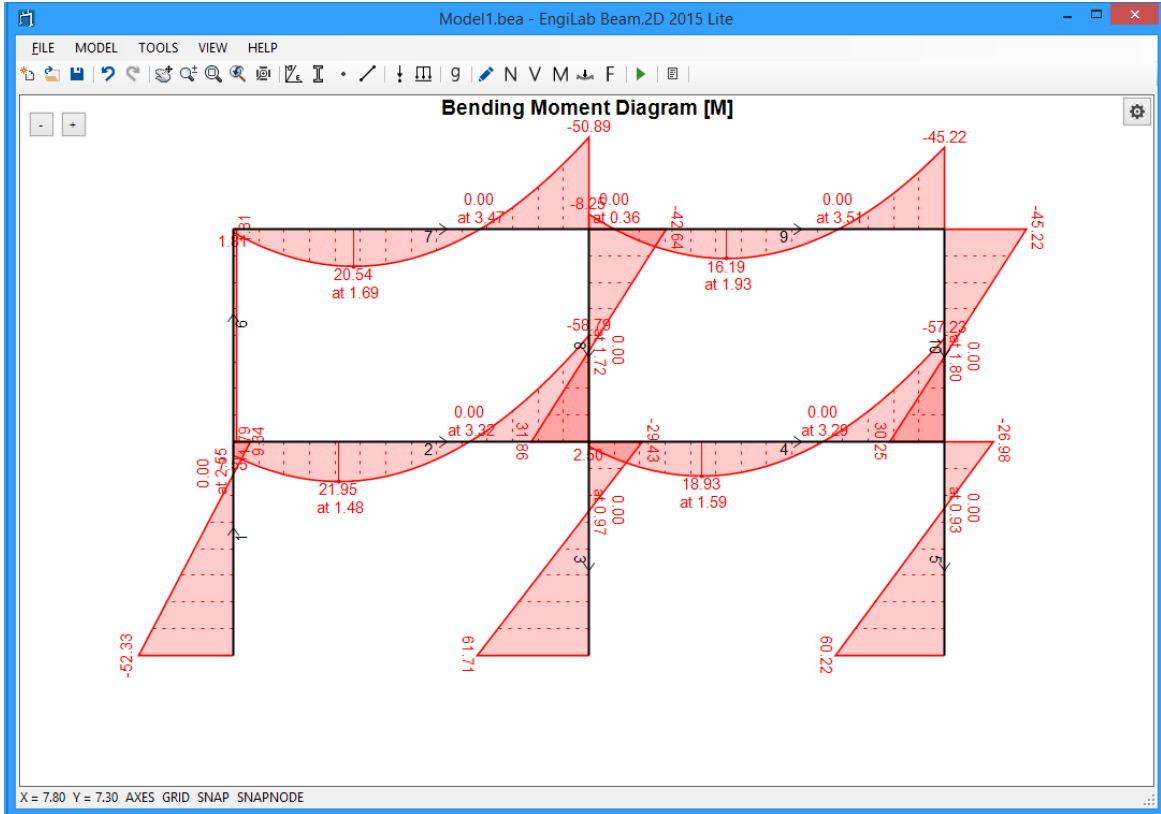
## 2. Shear Force Diagram

Click **V**, to see the Shear Force Diagram. If the Shear Force Diagram is out of Scale, **Click the "Zoom All" button**  to automatically scale the Diagram. You should take the following picture.

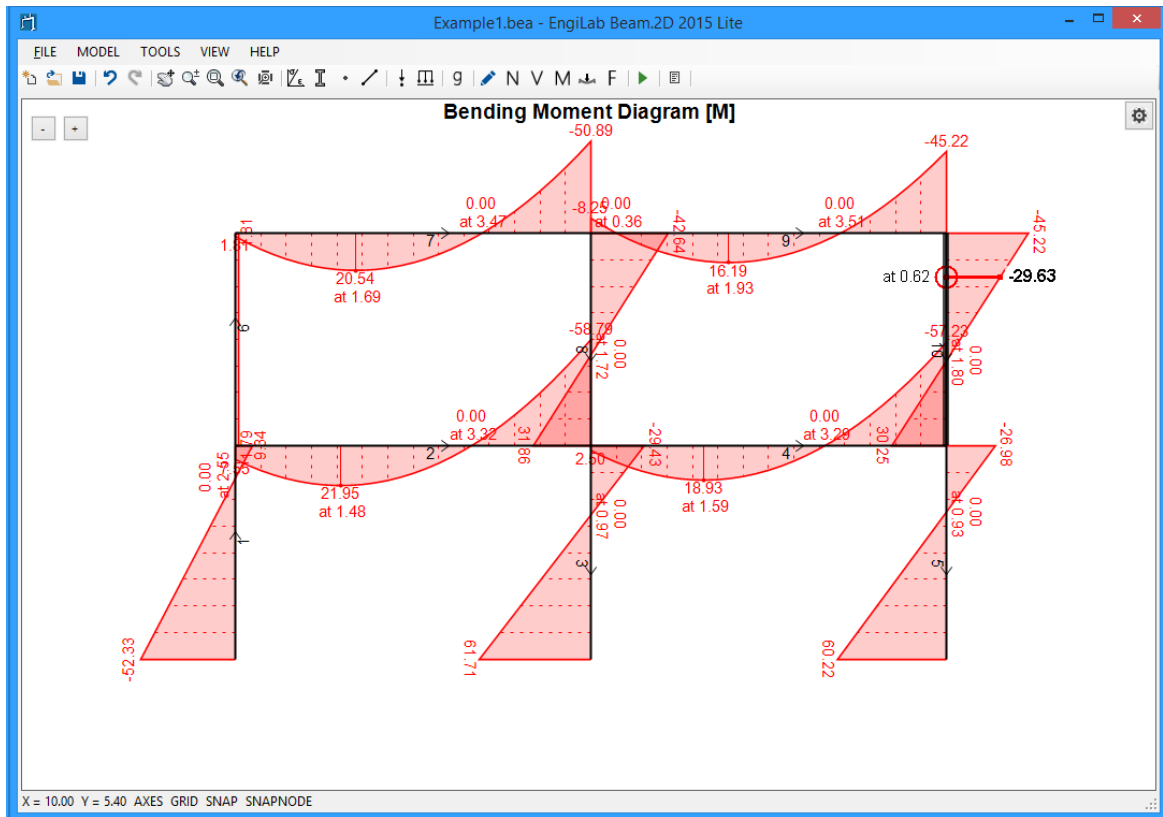


### 3. Bending Moment Diagram


Click **M**, to see the Bending Moment Diagram. If the Bending Moment Diagram is out of Scale, **Click the "Zoom All" button**  to automatically scale the Diagram. You should take the following picture.

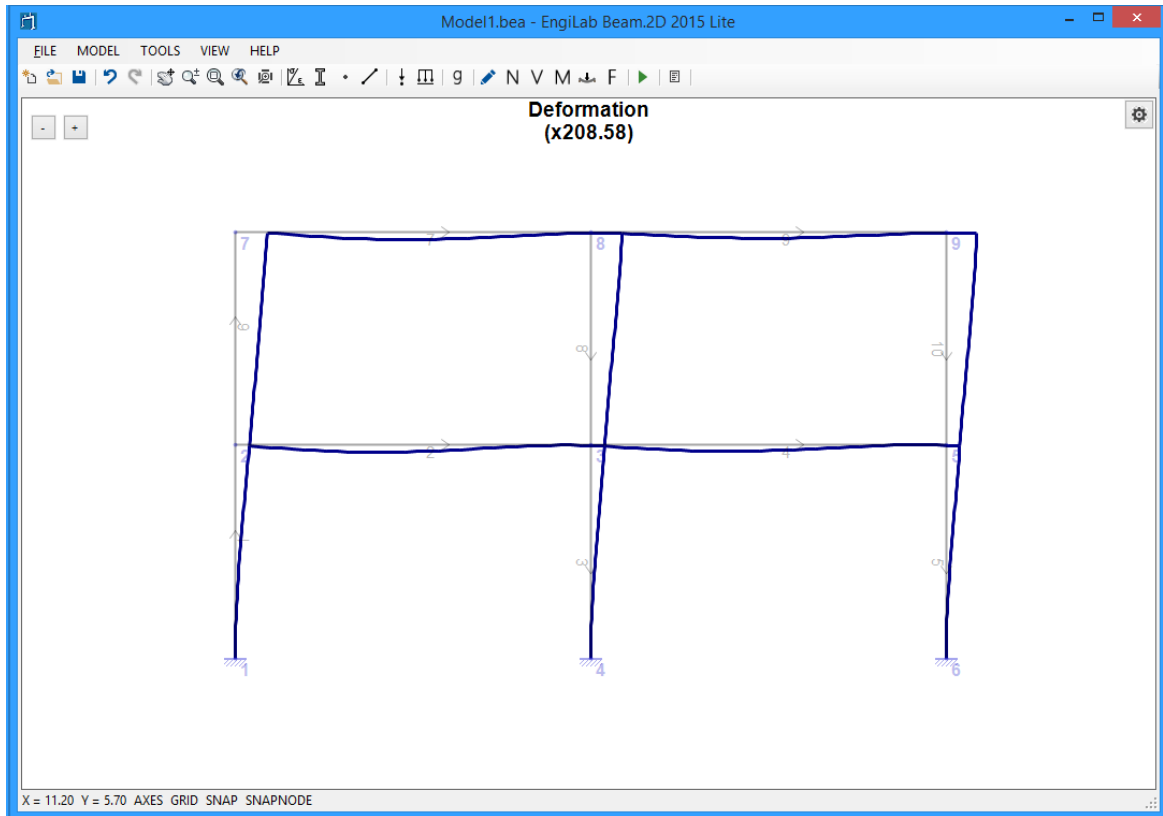


Note that if you move the pointer over an Element, you can read the corresponding value of the Diagram, as shown below for the Bending Moment Diagram case. This happens for all Diagrams and also for the Deformation and the Free Body Diagram.

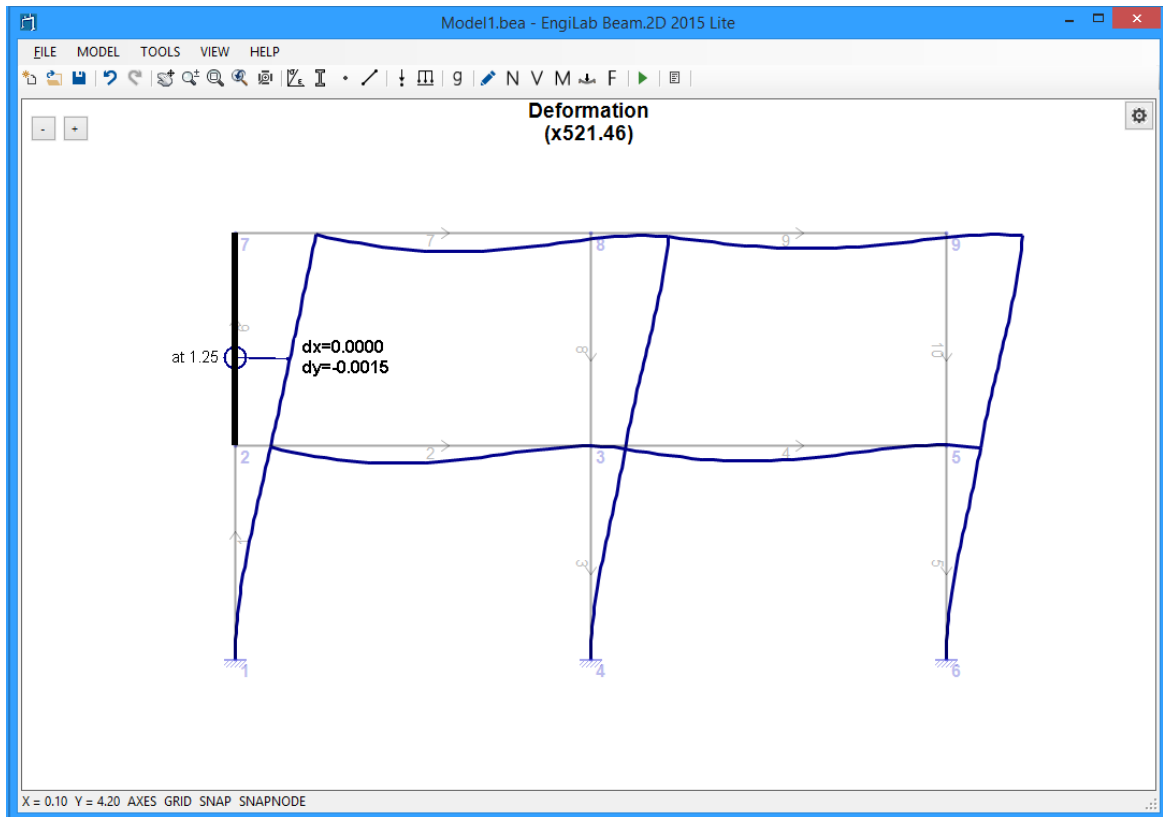


## 4. Deformation

Click , to see the Model Deformation. If the Deformation is out of Scale, **Click the "Zoom All" button** to automatically scale it. You should take the following picture. The program reports also the Deformation magnification, in our example x208.58.



You can adjust scaling yourself by using the +/- buttons at the top left of the picture. If you click the "+" button a few times, you may get a picture like the one below where the magnification factor is now x521.46.



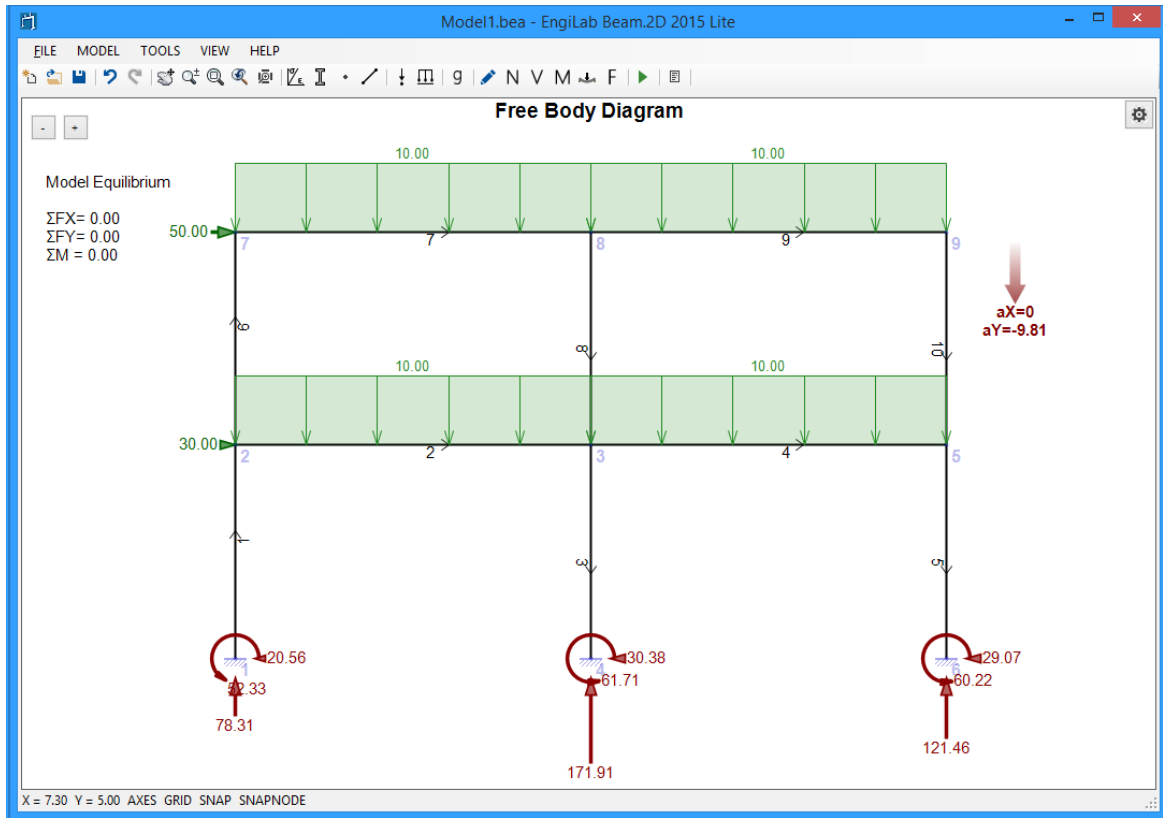
See above that you can get the deformation values on screen, if the mouse pointer moves over an element.

**IMPORTANT:** The values that are given on screen are the x and y displacements of the corresponding point of each element **in Local Element Axes**. For example, in the picture below Element 6 goes upward, which means that the Element x-Axis is pointing upwards and the Element y-Axis is pointing to the left. The point at 1.25 from the Element start (Node 2) has a deformation  $dy = -0.0015$  m towards the Element y-direction (that means 0.0015 m to the right of the picture), which is equivalent to a deformation  $DX = 0.0015$  m in Global Axes (Global X-Axis points towards the right of the picture).

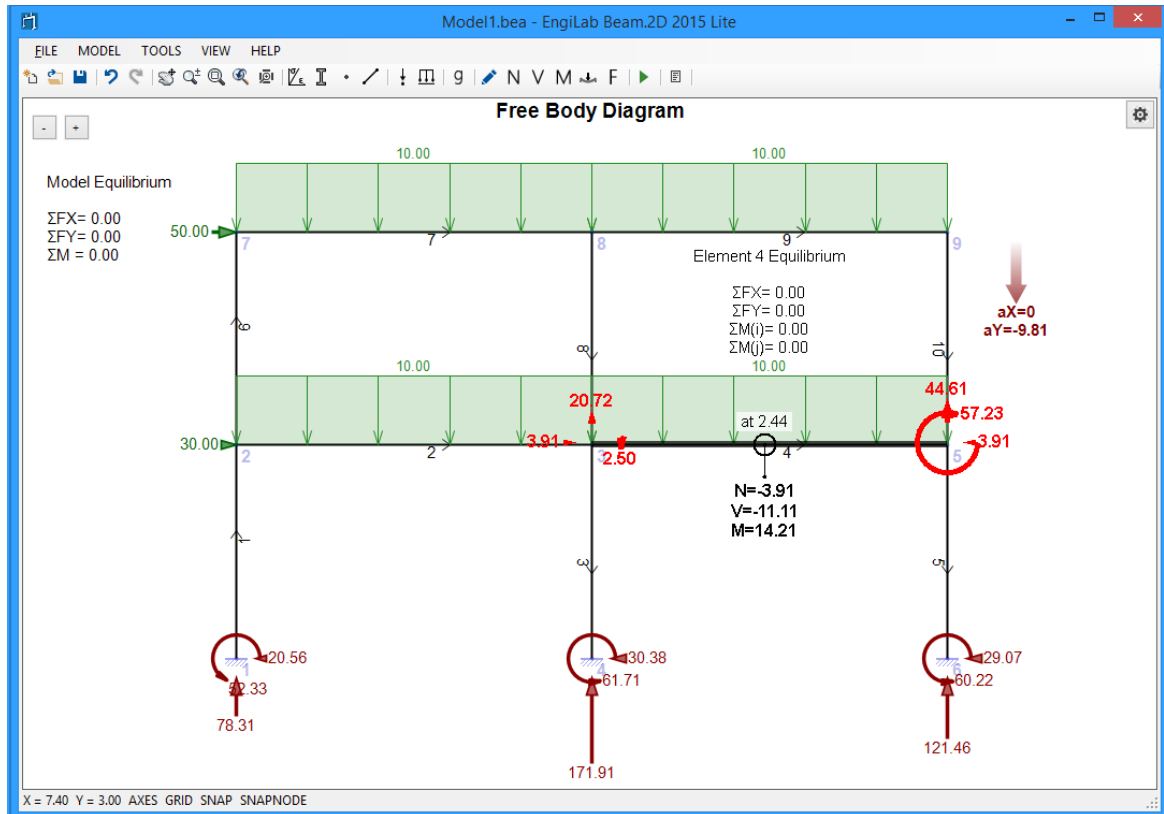


### 5. Free Body Diagram (F)

Click **F**, to see the Free Body Diagram of the Model. The Free Body Diagram shows the support reactions on screen and also the calculations of the equilibrium of the Model.



Note that if you move the pointer over an Element, you can read the corresponding N, V, M values, as shown below. The Element End Forces are also given on screen, for the specific Element and also the calculation of the equilibrium of the specific Element.



**IMPORTANT:** In the example above, we see the calculations of the equilibrium of Element 4 of the Model. All equilibrium calculations ( $\Sigma FX$ ,  $\Sigma FY$ ,  $\Sigma M(i)$ ,  $\Sigma M(j)$ ) are zero, otherwise there would be a problem in our Model or in the analysis results. One may want to calculate the equilibrium of Element 4 on his own, for example the value  $\Sigma FY$ . There is an external uniform load with value 10, acting along Element 4 (Length = 5 m), which gives an external load of  $10 \cdot 5 = 50$  kN acting towards  $-Y$  direction (downwards). The Element end forces sum up to  $20.72 + 44.61 = 65.33$  kN, towards  $Y$  direction (upwards). So why is there this difference of  $65.33 - 50 = 15.33$  kN? Is there a problem with the calculations?


The answer is NO. This is because of the self-weight of Element 4 which results to an additional uniform load (which is not shown on screen) acting towards  $-Y$  direction. Let's calculate this additional load:

**Self weight of Element 5 (in kN)** = Mass \* Acceleration = Volume \* Density \* Acceleration = Area \* Length \* Density \* Acceleration =  $0.125 \cdot 5 \cdot 2.5 \cdot 9.81 = 15.33$  kN.

So, the correct equilibrium calculation should be:  $\Sigma FY = 50 + 15.33 - 65.33 = 0.00$  kN which is reported by the program for Element 4.

### 7.1.13 Step 12. View the analytical results



Click  to see the analytical results.

#### 1. Node Displacements

The screenshot shows the 'EngiLab Beam.2D 2015 Lite - Analysis results' window. The 'Node Displacements' tab is active, displaying a table with the following data:

Node ID	X-Displacement	Y-Displacement	Z-Rotation
1	0.0000	0.0000	0.0000
2	0.0009	0.0000	-0.0004
3	0.0009	-0.0001	-0.0003
4	0.0000	0.0000	0.0000
5	0.0009	0.0000	-0.0003
6	0.0000	0.0000	0.0000
7	0.0022	0.0000	-0.0004
8	0.0021	-0.0001	-0.0002
9	0.0021	-0.0001	-0.0002

On the right side of the window, there are settings for 'Cell color' (Free, Constrained, Spring), 'Decimal places' (4 for Displacement, 4 for Rotation, 2 for Force, 2 for Moment), and 'Positive: Global axes' (indicated by a diagram with a red plus sign). There is also a 'Reset' button and an 'OK' button at the bottom right.

## 2. Element Forces

EngiLab Beam.2D 2015 Lite - Analysis results

Node Displacements | **Element End Forces** | Support Reactions | Full Report (RTF) | Analysis Validation

Element ID	Node ID	Axial Force	Shear Force	Bending Moment
▶ 1 Start	1	-78.31	20.56	-52.33
1 End	2	-59.91	20.56	9.34
2 Start	2	-9.45	19.40	7.55
2 End	3	-9.45	-45.93	-58.79
3 Start	3	-153.52	30.38	-29.43
3 End	4	-171.91	30.38	61.71
4 Start	3	-3.91	20.72	2.50
4 End	5	-3.91	-44.61	-57.23
5 Start	5	-103.06	29.07	-26.98
5 End	6	-121.46	29.07	60.22
6 Start	2	-40.52	0.01	1.79
6 End	7	-22.12	0.01	1.81
7 Start	7	-49.99	22.12	1.81
7 End	8	-49.99	-43.20	-50.89
8 Start	8	-68.47	24.83	-42.64
8 End	3	-86.87	24.83	31.86
9 Start	8	-25.16	25.27	-8.25
9 End	9	-25.16	-40.06	-45.22
10 Start	9	-40.06	25.16	-45.22
10 End	5	-58.45	25.16	30.25

Cell color: Normal, Hinge

Sign convention:  Diagrams,  Local axes

Decimal places: 4 Displacement, 4 Rotation, 2 Force, 2 Moment

OK

## 3. Support Reactions

EngiLab Beam.2D 2015 Lite - Analysis results

Node Displacements | Element End Forces | **Support Reactions** | Full Report (RTF) | Analysis Validation

Node ID	X-Force	Y-Force	Z-Moment
▶ 1	-20.56	78.31	52.33
4	-30.38	171.91	61.71
6	-29.07	121.46	60.22

Cell color: Free, Constrained, Spring

Positive: Global axes

Decimal places: 4 Displacement, 4 Rotation, 2 Force, 2 Moment

OK

### 4. Full Analysis Report (RTF)

The screenshot shows the 'Full Report (RTF)' tab in the EngiLab Beam.2D 2015 Lite software. The report is titled 'EngiLab Beam.2D 2015 Analysis Report' for model 'Model1.bea', created on Sunday, 10 May 2015, at 21:33:17. The report is divided into three main sections: Node Displacements, Element End Forces, and Support Reactions. A right-hand sidebar contains settings for display options (Displacement, Forces, Reactions, Color, B&W) and decimal places for Displacement, Rotation, Force, and Moment.

**Node Displacements**

Node ID	X-Displacement	Y-Displacement	Z-Rotation
1	0.0000	0.0000	0.0000
2	0.0009	0.0000	-0.0004
3	0.0009	-0.0001	-0.0003
4	0.0000	0.0000	0.0000
5	0.0009	0.0000	-0.0003
6	0.0000	0.0000	0.0000
7	0.0022	0.0000	-0.0004
8	0.0021	-0.0001	-0.0002
9	0.0021	-0.0001	-0.0002

Sign convention: Positive according to Global Axes  
 Displacements reported with 4 decimal places  
 Rotations (in Radians) reported with 4 decimal places  
 Background color: Free DOF Constrained DOF Spring DOF

**Element End Forces**

Element ID	Axial Force	Shear Force	Bending Moment
1 Start	-78.31	20.56	-52.33
1 End	-59.91	20.56	9.34
2 Start	-9.45	19.40	7.55
2 End	-9.45	-45.93	-58.79
3 Start	-153.52	30.38	-29.43
3 End	-171.91	30.38	61.71
4 Start	-3.91	20.72	2.50
4 End	-3.91	-44.61	-57.23
5 Start	-103.06	29.07	-26.98
5 End	-121.46	29.07	60.22
6 Start	-40.52	0.01	1.79
6 End	-22.12	0.01	1.81
7 Start	-49.99	22.12	1.81
7 End	-49.99	-43.20	-50.89
8 Start	-68.47	24.83	-42.64
8 End	-86.87	24.83	31.86
9 Start	-25.16	25.27	-8.25
9 End	-25.16	-40.06	-45.22
10 Start	-40.06	25.16	-45.22
10 End	-58.45	25.16	30.25

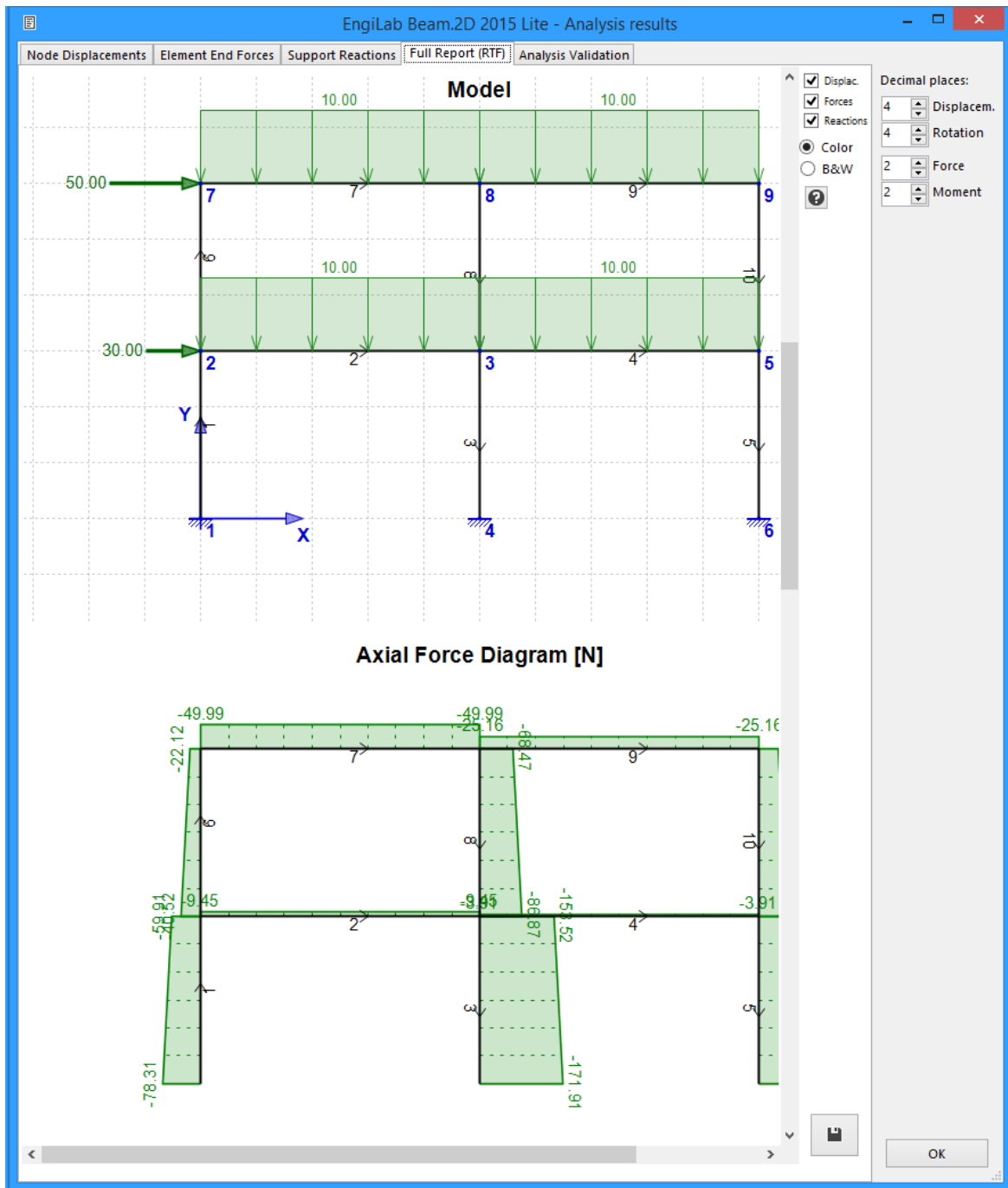
Sign convention: According to the diagrams' sign convention  
 Axial force: Positive when member is in tension  
 Shear force: Positive when it rotates the member clockwise  
 Bending moment: Positive when the "bottom" fiber is in tension ("top" fiber in compression)  
 Forces reported with 2 decimal places  
 Bending moments reported with 2 decimal places  
 Background color: Normal DOF Hinge DOF

**Support Reactions**

Node ID	X-Force	Y-Force	Z-Moment
1	-20.56	78.31	52.33
4	-30.38	171.91	61.71
6	-29.07	121.46	60.22

Sign convention: Positive according to Global Axes  
 Forces reported with 2 decimal places  
 Bending moments reported with 2 decimal places

The Full Analysis Reports includes also the pictures of the Model, N, V, M Diagrams, Deformation and Free Body Diagram, as shown below.



## 5. Analysis Validation

### 5a. Global (Model) Equilibrium

The screenshot shows the 'Analysis Validation' window in EngiLab Beam.2D 2015 Lite. The 'Global (Model) Equilibrium' tab is selected. The table below displays the equilibrium results for the global model.

Load/Reaction type	$\Sigma(FX)$	$\Sigma(FY)$	$\Sigma(MZ \text{ at } 0,0)$
▶ Nodal Loads	80.00	0.00	-390.00
Elemental Loads	0.00	-200.00	-1000.00
Acceleration (Body) Loads	0.00	-171.68	-858.38
Support Reactions	-80.00	371.68	2248.38
<b>SUM</b>	<b>0.00</b>	<b>0.00</b>	<b>0.00</b>

On the right side of the window, there are controls for 'Decimal places' and a 'Positive: Global axes' indicator. The 'Decimal places' are set to 4 for Displacement and Rotation, and 2 for Force and Moment. The 'Positive: Global axes' indicator shows a coordinate system with X and Y axes and a positive rotation arrow.

### 5b. Individual Element Equilibrium

The screenshot shows the 'Analysis Validation' window in EngiLab Beam.2D 2015 Lite. The 'Individual Element Equilibrium' tab is selected. The table below displays the equilibrium results for individual elements.

Element ID	FX Elemental Force	FY Body Force	FX End Force i	FX End Force j	SUM $\Sigma FX$	FY Elemental Force	FY Body Force	FY End Force i	FY End Force j	SUM $\Sigma FY$	SUM $\Sigma MZ \text{ at } i$	SUM $\Sigma MZ \text{ at } j$
▶ 1	0.00	0.00	-20.56	20.56	0.00	0.00	-18.39	78.31	-59.91	0.00	0.00	0.00
2	0.00	0.00	9.45	-9.45	0.00	-50.00	-15.33	19.40	45.93	0.00	0.00	0.00
3	0.00	0.00	30.38	-30.38	0.00	0.00	-18.39	-153.52	171.91	0.00	0.00	0.00
4	0.00	0.00	3.91	-3.91	0.00	-50.00	-15.33	20.72	44.61	0.00	0.00	0.00
5	0.00	0.00	29.07	-29.07	0.00	0.00	-18.39	-103.06	121.46	0.00	0.00	0.00
6	0.00	0.00	-0.01	0.01	0.00	0.00	-18.39	40.52	-22.12	0.00	0.00	0.00
7	0.00	0.00	49.99	-49.99	0.00	-50.00	-15.33	22.12	43.20	0.00	0.00	0.00
8	0.00	0.00	24.83	-24.83	0.00	0.00	-18.39	-68.47	86.87	0.00	0.00	0.00
9	0.00	0.00	25.16	-25.16	0.00	-50.00	-15.33	25.27	40.06	0.00	0.00	0.00
10	0.00	0.00	25.16	-25.16	0.00	0.00	-18.39	-40.06	58.45	0.00	0.00	0.00

On the right side of the window, there are controls for 'Decimal places' and a 'Positive: Global axes' indicator. The 'Decimal places' are set to 4 for Displacement and Rotation, and 2 for Force and Moment. The 'Positive: Global axes' indicator shows a coordinate system with X and Y axes and a positive rotation arrow.

*This page intentionally left blank.*



# Chapter

---



## License Agreement

## 8 License Agreement

- [License Agreement \(Lite Edition\)](#)
- [License Agreement \(Pro Edition\)](#)

### 8.1 EULA (Lite Edition)

#### EngiLab Beam.2D LITE Edition End User License Agreement (EULA)

**IMPORTANT:** Read the following terms carefully before installing, using and copying **EngiLab Beam.2D LITE Edition** (the "Software Product"). By installing, using and copying the Software Product you agree to accept all of the following terms. If you do not agree with the terms of this EULA, you must not use the Software Product.

This End-User License Agreement ("EULA"), effective as of the date you accept the terms hereof, is a legal agreement entered into between EngiLab Co. ("EngiLab") and you (either an individual or a single entity), the end user of the Software Product identified above, which includes computer software and may include associated media, printed materials, and online or electronic documentation. The parties agree as follows:

#### 1. SCOPE

In accordance with the terms herein, EngiLab grants to you, and you accept from EngiLab, a non-exclusive, non-transferable and non-assignable, limited license to use the current version of EngiLab's Software Product. The Software Product is protected by copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. The Software Product is licensed, not sold. All rights not expressly granted are reserved by EngiLab.

#### 2. DEFINITIONS

Software Product means the EngiLab free (Lite Edition) software accompanying this EULA, which includes executable modules and electronic documentation and may include associated media, printed materials and information available at the product web site protected by copyright laws. The Software Product also includes any updates and supplements to the original Software Product provided by EngiLab. Any software provided along with the Software Product that is associated with a separate end-user license agreement is licensed to you under the terms of that license agreement.

#### 3. GRANT OF LICENSE

EngiLab grants you a limited, non-exclusive, non-transferable, non-renewable license to install, use, access, display, run, or otherwise interact with the Software Product. This Lite Edition of the Software Product is provided to you free of charge for your own **personal, academic or educational use only**, and **it may not be used for any commercial purposes**. For commercial use that includes all fields that do not belong to pure personal use: e.g. all kinds of use in the range of any profession or for the utilization of additional professional features, you must license the Professional (PRO) Edition of the Software Product which is subject to charging.

Besides the limitation for non-commercial use, the Lite Edition of the Software Product has also other limitations compared to the Pro Edition, on the total Number of Nodes, Materials and Sections of a Model. The Lite Edition of the Software product can create and modify any structural Model, but it can only analyze Models with up to 10 Nodes, 3 Materials and 3 Sections.

The EULA permits use of the Software Product during unlimited period free of charge. The Software

Product may be used as many times as you like, for as long as you like. You may copy the Software Product only for your own use or for your own backup purposes, provided that you keep this copyright notice and disclaimer of warranty intact. You may not copy or distribute copies of the Software Product to any third party for any use. You must not charge money or fees for the Software Product to anyone.

#### **4. DESCRIPTION OF OTHER RIGHTS AND LIMITATIONS**

4.1 You may not resell, or otherwise transfer for value, the Software Product under any circumstances. You may not charge any fees for the copy or use of the Software Product itself. You must not represent in any way that you are selling the Software Product itself.

4.2 You may not make modifications to the Software Product, or decompile, disassemble, reverse engineer or modify the Software Product or any portion of it. You may not combine other commercial applications with, or otherwise prepare derivative works of the Software.

4.3 The Software Product is licensed as a single product. Its component parts may not be separated not within the Software Product.

4.4 This EULA does not grant you any rights in connection with any trademarks or service marks of EngiLab.

4.5 You may not rent, lease, or lend the Software Product to other users.

4.6 You may not make any transfers of this EULA and Software Product to a third party.

#### **5. COPYRIGHT**

The title and all copyrights in and to the Software Product (including but not limited to any images, photographs, animations, video, audio, music, text, and "applets" incorporated into the Software Product), the accompanying printed materials, and all copies of the Software Product are owned at all the time by EngiLab. All title and intellectual property rights in and to the content which may be accessed through use of the Software Product is the property of the respective content owner and may be protected by applicable copyright or other intellectual property laws and treaties. This EULA grants you no rights to use such content. Software Product documentation is provided in electronic form. You may print copies of such electronic documentation.

#### **6. LIMITED WARRANTY**

EngiLab has made every effort to make the use of the Software Product as reliable and safe as possible. To the best of our knowledge this software is accurate and complies with the standards of good engineering practice. 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. EngiLab expressly disclaims any warranty that the Software Product will meet your requirements or operate under your specific conditions of use. EngiLab makes no warranty that operation of the Software Product will be secure, error free, or free from interruption. The Software Product and any related documentation is provided "as is" without warranty of any kind, either express or implied, including, without limitation, the implied warranties or merchantability, fitness for a particular purpose, or noninfringement. The entire risk arising out of use or performance of the Software Product remains with you.

YOU MUST DETERMINE WHETHER THE SOFTWARE PRODUCT SUFFICIENTLY MEETS YOUR REQUIREMENTS FOR SECURITY AND UNINTERRUPTABILITY. YOU BEAR SOLE RESPONSIBILITY AND ALL LIABILITY FOR ANY LOSS INCURRED DUE TO FAILURE OF THE SOFTWARE PRODUCT TO MEET YOUR REQUIREMENTS. ENGI LAB WILL NOT, UNDER ANY CIRCUMSTANCES, BE RESPONSIBLE OR LIABLE FOR THE LOSS OF DATA ON ANY COMPUTER OR INFORMATION STORAGE DEVICE.

## **7. DISCLAIMER OF DAMAGES**

Under no circumstances shall EngiLab, its directors, officers, employees or agents be liable to you or any other party for indirect, consequential, special, incidental, punitive, or exemplary damages of any kind (including, without limitation, damages for loss of business profits, business interruption, loss of business information, or any other pecuniary loss) resulting from this EULA, or from the furnishing, performance, installation, use, or inability to use the Software Product, whether due to a breach of contract, breach of warranty, or the negligence of EngiLab or any other party, even if EngiLab is advised beforehand of the possibility of such damages and known defects. To the extent that the applicable jurisdiction limits EngiLab's ability to disclaim any implied warranties, this disclaimer shall be effective to the maximum extent permitted.

## **8. TERM AND TERMINATION**

8.1 This EULA comes into effect when you install the Software Product on your computer, and is effective for the entire period of use of the Software Product.

8.2 Any use in violation of this EULA shall constitute not only breach of this EULA, but a violation of national and international copyright laws. Any use of the Software Product that infringes upon EngiLab's intellectual property rights or that is for commercial purposes will be investigated and EngiLab shall have the right to take appropriate civil and criminal legal action.

8.3 Without prejudice to any other rights, EngiLab may terminate this EULA if you fail to comply with the terms and conditions of this EULA. In such event, you must destroy all copies of the Software Product and all of its component parts.

## 8.2 EULA (Pro Edition)

### EngiLab Beam.2D PRO Edition End User License Agreement (EULA)

**IMPORTANT:** Read the following terms carefully before installing, using and copying **EngiLab Beam.2D PRO Edition** (the "Software Product"). By installing, using and copying the Software Product you agree to accept all of the following terms. If you do not agree with the terms of this EULA, you must not use the Software Product.

This End-User License Agreement ("EULA"), effective as of the date you accept the terms hereof, is a legal agreement entered into between EngiLab Co. ("EngiLab") and you (either an individual or a single entity), the end user of the Software Product identified above, which includes computer software and may include associated media, printed materials, and online or electronic documentation. The parties agree as follows:

#### 1. SCOPE

In accordance with the terms herein, EngiLab grants to you, and you accept from EngiLab, a non-exclusive, non-transferable and non-assignable, limited license to use the current version of EngiLab's Software Product. The Software Product is protected by copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. The Software Product is licensed, not sold. All rights not expressly granted are reserved by EngiLab.

#### 2. DEFINITIONS

Software Product means the EngiLab paid (Pro Edition) software accompanying this EULA, which includes executable modules and electronic documentation and may include associated media, printed materials and information available at the product web site protected by copyright laws. The Software Product also includes any updates and supplements to the original Software Product provided by EngiLab. Any software provided along with the Software Product that is associated with a separate end-user license agreement is licensed to you under the terms of that license agreement.

#### 3. GRANT OF LICENSE

EngiLab grants you a limited, non-exclusive, non-transferable, non-renewable license to install, use, access, display, run, or otherwise interact with the Software Product. This Pro Edition of the Software Product is provided to you for any legal use (**personal, educational, academic, non-profit or commercial use**).

The Pro Edition of the Software Product requires a **License Key** in order to operate. Upon purchase of the Software Product, the user automatically receives a License Key via email which is to be used with the specific Software Product. The License Key is bound to the specific user via the "Registration Name" that was given by the user and it certifies that the copy of the program is original.

The Software Product needs also an **Activation Key** in order to be activated and run. Normally, after the user enters the License Key, the program connects with EngiLab Servers in order to obtain an Activation Key online. The Activation Key is bound to the specific License Key and to the specific Computer that asked for it. Unlike the License Key that can be used in more than one Computers, an Activation Key obtained for one Computer cannot be used to activate the Software Product on another Computer.

**One User - Two Machines per License Policy:** The maximum number of allowed Activations per License Key is two (2). This means that a user having purchased the Software Product and being given a License Key is eligible for 2 Activation Keys, for use with two different Computers, provided only one computer is in use at any given time. The two different Computers will have the same

License Key, but different Activation Keys. As a result, a user can use the same License Key for his primary (e.g. Desktop) and secondary (e.g. Laptop) Computer and each Computer can obtain a unique Activation Key bound to it.

The EULA permits use of the Software Product during unlimited period of time. The Software Product may be used as many times as you like, for as long as you like. You may copy the Software Product only for your own use or for your own backup purposes, provided that you keep this copyright notice and disclaimer of warranty intact. You may not copy or distribute copies of the Software Product to any third party for any use. You must not charge money or fees for the Software Product to anyone.

#### **4. DESCRIPTION OF OTHER RIGHTS AND LIMITATIONS**

4.1 You may not resell, or otherwise transfer for value, the Software Product under any circumstances. You may not charge any fees for the copy or use of the Software Product itself. You must not represent in any way that you are selling the Software Product itself.

4.2 You may not make modifications to the Software Product, or decompile, disassemble, reverse engineer or modify the Software Product or any portion of it. You may not combine other commercial applications with, or otherwise prepare derivative works of the Software.

4.3 The Software Product is licensed as a single product. Its component parts may not be separated not within the Software Product.

4.4 This EULA does not grant you any rights in connection with any trademarks or service marks of EngiLab.

4.5 You may not rent, lease, or lend the Software Product to other users.

4.6 You may not make any transfers of this EULA and Software Product to a third party.

#### **5. COPYRIGHT**

The title and all copyrights in and to the Software Product (including but not limited to any images, photographs, animations, video, audio, music, text, and "applets" incorporated into the Software Product), the accompanying printed materials, and all copies of the Software Product are owned at all the time by EngiLab. All title and intellectual property rights in and to the content which may be accessed through use of the Software Product is the property of the respective content owner and may be protected by applicable copyright or other intellectual property laws and treaties. This EULA grants you no rights to use such content. Software Product documentation is provided in electronic form. You may print copies of such electronic documentation.

#### **6. LIMITED WARRANTY**

EngiLab has made every effort to make the use of the Software Product as reliable and safe as possible. To the best of our knowledge this software is accurate and complies with the standards of good engineering practice. 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. EngiLab expressly disclaims any warranty that the Software Product will meet your requirements or operate under your specific conditions of use. EngiLab makes no warranty that operation of the Software Product will be secure, error free, or free from interruption. The Software Product and any related documentation is provided "as is" without warranty of any kind, either express or implied, including, without limitation, the implied warranties or merchantability, fitness for a particular purpose, or noninfringement. The entire risk arising out of use or performance of the Software Product remains with you.

YOU MUST DETERMINE WHETHER THE SOFTWARE PRODUCT SUFFICIENTLY MEETS YOUR

REQUIREMENTS FOR SECURITY AND UNINTERRUPTABILITY. YOU BEAR SOLE RESPONSIBILITY AND ALL LIABILITY FOR ANY LOSS INCURRED DUE TO FAILURE OF THE SOFTWARE PRODUCT TO MEET YOUR REQUIREMENTS. ENGI LAB WILL NOT, UNDER ANY CIRCUMSTANCES, BE RESPONSIBLE OR LIABLE FOR THE LOSS OF DATA ON ANY COMPUTER OR INFORMATION STORAGE DEVICE.

## **7. DISCLAIMER OF DAMAGES**

Under no circumstances shall EngiLab, its directors, officers, employees or agents be liable to you or any other party for indirect, consequential, special, incidental, punitive, or exemplary damages of any kind (including, without limitation, damages for loss of business profits, business interruption, loss of business information, or any other pecuniary loss) resulting from this EULA, or from the furnishing, performance, installation, use, or inability to use the Software Product, whether due to a breach of contract, breach of warranty, or the negligence of EngiLab or any other party, even if EngiLab is advised beforehand of the possibility of such damages and known defects. To the extent that the applicable jurisdiction limits EngiLab's ability to disclaim any implied warranties, this disclaimer shall be effective to the maximum extent permitted.

## **8. TERM AND TERMINATION**

8.1 This EULA comes into effect when you install the Software Product on your computer, and is effective for the entire period of use of the Software Product.

8.2 Any use in violation of this EULA shall constitute not only breach of this EULA, but a violation of national and international copyright laws. Any use of the Software Product that infringes upon EngiLab's intellectual property rights will be investigated and EngiLab shall have the right to take appropriate civil and criminal legal action.

8.3 Without prejudice to any other rights, EngiLab may terminate this EULA if you fail to comply with the terms and conditions of this EULA. In such event, you must destroy all copies of the Software Product and all of its component parts.



[www.engilab.com](http://www.engilab.com)