

EngiLab Frame.2D 2021

v3.4

(v3.4.7920.0)

User Manual



www.engilab.com

This page intentionally left blank.

EngiLab Frame.2D 2021 v3.4 User Manual

(c) 2021 EngiLab PC

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 PC

Document type

User Manual

Program name

EngiLab Frame.2D

Program version

v3.4.7920.0

Document version

v1.0

Document release date

Sep. 7, 2021

"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

Chapter 1 Introduction to EngiLab Frame.2D	1
1 Overview	2
2 Lite and Pro Editions	3
Chapter 2 Structural Modeling	5
1 System of Units	6
2 Setting up a model	12
3 Model Properties	14
Materials	14
Sections	18
Nodes	23
Elements	36
Nodal Loads	47
Elemental Loads	49
Body (Acceleration) Loads	52
Chapter 3 Tools	55
1 Import Materials/Sections	56
2 Import DXF	57
3 Convert Model to Truss	58
Chapter 4 Analysis and Analysis results	61
1 Analysis	62
2 N, V, M Diagrams	65
3 Deformation	71
4 Free Body Diagram (F)	73
5 Analysis results	75
Node Displacements	76
Element End Forces	77
Support Reactions	78
Full Report (RTF)	79
Analysis Validation	80
Chapter 5 Settings	83
1 General (Settings)	84
2 Model (Settings)	85
3 NVM Diagrams (Settings)	86
4 Colors (Settings)	88
5 Decimals (Settings)	90
6 Fonts (Settings)	91
7 Results (Settings)	92

Chapter 6 Useful information	95
1 Ready to-analyze Examples	96
2 Tips on Modeling hinges	97
3 Tips on Modeling symmetric structures	99
Chapter 7 Example Problems	103
1 Example Problem 1	104
Overview - Example 1	104
Step 1. Preparation of the input data	106
Step 2. Define Materials	107
Step 3. Define Sections	109
Step 4. Draw the Model on screen	117
Step 5. Edit Nodes	121
Step 6. Edit Elements	124
Step 7. Define Nodal Loads	126
Step 8. Define Elemental Loads	131
Step 9. Define Body (Acceleration) Loads	136
Step 10. Run the Analysis	138
Step 11. View N, V, M Diagrams, Model Deformation and Free Body Diagram	139
Step 12. View the analytical results	148
Chapter 8 License Agreement	153
1 EULA (Lite Edition)	154
2 EULA (Pro Edition)	157

Chapter

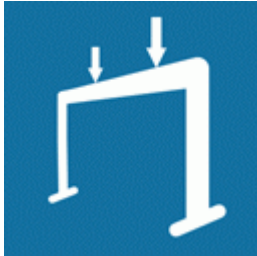


Introduction to EngiLab Frame.2D

1 Introduction to EngiLab Frame.2D

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

1.1 Overview



EngiLab Frame.2D 2021

v3.4 (v3.4.7920.0)

Copyright (c) 2021 EngiLab PC.

All rights reserved



www.engilab.com

EngiLab Frame.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

Technical support

EngiLab offers technical support via email at 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 Vista, 7, 8, 10, etc.)
- Name and exact version of the product being used.
- Brief description of the problem.
- Detailed instructions on how to reproduce 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. We take customer satisfaction very seriously and always welcome your feedback.

Compatibility

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

- Windows Vista, Windows 7, 8, 10, 11
- 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 Frame.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
- 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

Pro Edition

- The Pro Edition is available for purchase at 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, 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

Introduction

EngiLab Frame.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. This way there is no limitation in the system of units that can be used. In fact, **any consistent system of units can be used**. So, how does it work?

Imagine Newton's Second Law of Motion. The equation is $F = m * a$,

where F is the force, m is the mass of the object and a is the acceleration. The second law states that the rate of change of momentum of a body over time is directly proportional to the force applied, and occurs in the same direction as the applied force. This law is global and it applies everywhere, no matter what units we use for the quantities Force, Mass and Acceleration. But these units need to be consistent, i.e. the units we use for mass and acceleration will define the unit we take for force when we apply the formula (without any conversions). For example if we use $m=20$ kg and $a=10$ m/s², then we take $F=200$ and the unit of Force is then kg*m/s², which means N (Newton). The result is therefore $F=200$ N. In a similar way, we can use the same formula with different units, as long as we keep things consistent.

As long as we use **consistent units**, there is no need for any unnecessary conversions. What you give (inputs) will define what you take (outputs). The same applies to EngiLab Frame.2D. There are no units in the program itself but the user needs to know what units he/she uses for the inputs. The outputs will simply comply. Below you can find a thorough explanation of how exactly this works and also instructions on using the most common units.

Implementation

Before starting to define any model in the program, 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.

EngiLab strongly recommends the use of "EngiLab Units" for all unit conversions. EngiLab Units is a free unit conversion program that is available for download at www.engilab.com. EngiLab Units 2021 (v3.1) supports 584 units in 20 unit categories, including distance, acceleration, pressure (stress) and others. **All units needed in EngiLab Frame.2D are supported by EngiLab Units.**



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)
Acceleration	L/T^2	(1 Length unit) / (1 time unit) ²	m/s^2
Mass *	$F \cdot T^2 / L$	(1 force unit) / (1 acceleration unit)	$kN/(m/s^2) = Mg = t *$
Density	$F \cdot T^2 / L^4$	(1 mass unit) / (1 length unit) ³	t/m^3
Stress	F/L^2	(1 force unit) / (1 length unit) ²	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)
Acceleration	L/T^2	(1 Length unit) / (1 time unit) ²	m/s^2
Force *	$M \cdot L/T^2$	(1 mass unit) · (1 acceleration unit)	$kg \cdot m/s^2 = N *$
Density	M/L^3	(1 mass unit) / (1 length unit) ³	kg/m^3
Stress	$M/L/T^2$	(1 force unit) / (1 length unit) ²	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. Any column can be used for the inputs (and outputs) of the program.

Quantity	SI (MKS)	MTS	mmNS	US Unit (ft)	US Unit (in)
Length	m	m	mm	ft	in
Force	N	kN	N	lbf	lbf
Mass	$\text{kg} = \text{N} \cdot \text{s}^2 / \text{m}$	$\text{t (tonne)} = \text{kN} \cdot \text{s}^2 / \text{m}$	$\text{t} = \text{N} \cdot \text{s}^2 / \text{mm}$	slug $\text{lbf} \cdot \text{s}^2 / \text{ft}$	blob $\text{lbf} \cdot \text{s}^2 / \text{in}$
Time	s	s	s	s	s
Stress	Pa (N/m^2)	kPa (kN/m^2)	MPa (N/mm^2) MN/ m^2)	lbf/ft^2	psi (lbf/in^2)
Density	kg/m^3	t/m^3	t/mm^3	slug/ft^3	$\text{lbf} \cdot \text{s}^2 / \text{in}^4$
Acceleration	m/s^2	m/s^2	mm/s^2	ft/s^2	in/s^2

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^3	7.85 t/m^3	7.85×10^{-9} t/mm^3	15.23151461 slug/ft^3 ($\text{lbf} \cdot \text{s}^2 / \text{ft}^4$)	$7.34544493 \times 10^{-4}$ blob/ in^3 ($\text{lbf} \cdot \text{s}^2 / \text{in}^4$)
Steel Modulus of Elasticity (E)	210×10^9 N/m^2	210×10^6 kN/m^2	210×10^3 MPa	4,385,941,188 .96153 lbf/ft^2 (psf)	30,457,924.92 33 lbf/ft^2 (psi)
Earth Gravity acceleration	9.80665 m/s^2	9.80665 m/s^2	9806.65 mm/s^2	32.17404856 ft/s^2	386.088583 in/s^2

Notes:

- 1 t (tonne) = 10^3 kg = 1 Mg. It is a mass that accelerates by 1 m/s^2 when a force of 1 kN is exerted on it.
- 1 slug = 1 $\text{lbf} \cdot \text{s}^2 / \text{ft}$. It is a mass that accelerates by 1 ft/s^2 when a force of 1 pound-force (lbf) is exerted on it.
- 1 blob = 1 $\text{lbf} \cdot \text{s}^2 / \text{in}$. It is a mass that accelerates by 1 in/s^2 when a force of 1 pound-force (lbf) is exerted on it.
- 1 MPa = 1 MN/m^2 = 1 N/mm^2

Practical example

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

- Length: **m**

- Force: **kN**
- Time: **s**

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

Quantity	Unit used
Node Coordinates X, Y	m
Material Elastic Modulus E	kPa = kN/m ²
Material Density d	t/m ³
Section Area A	m ²
Section Moment of Inertia I	m ⁴
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 ²

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


Quantity	Unit used
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**

- 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^3 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



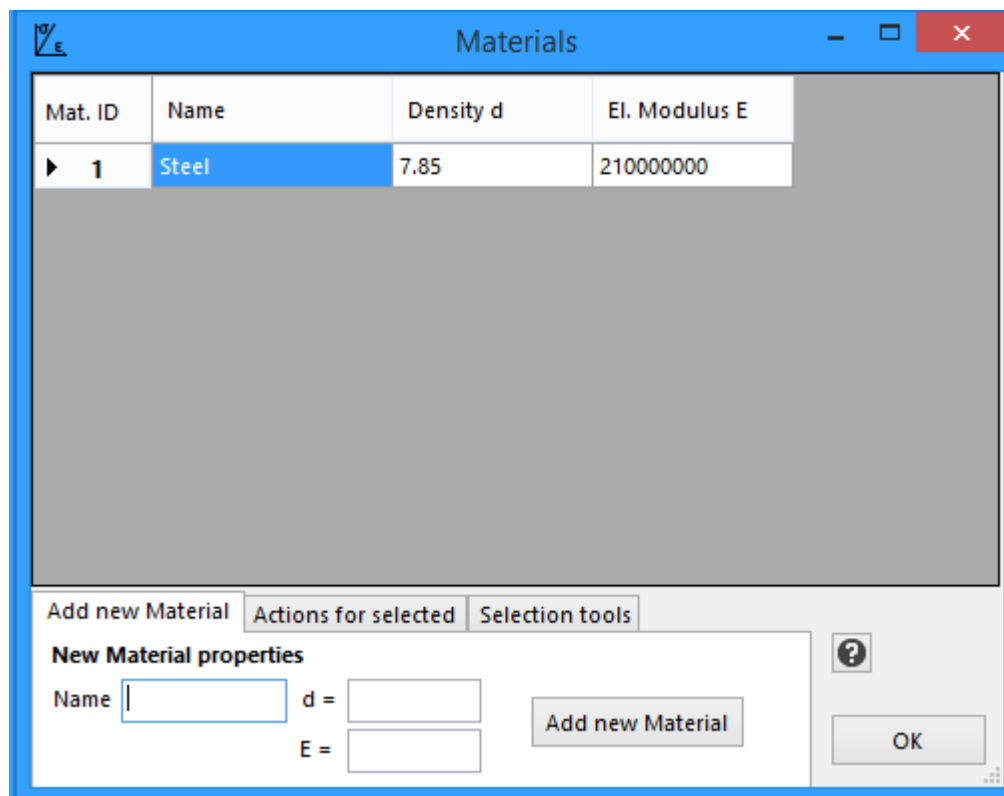
Each element is assigned a **Material**. The Material properties are the following:

- **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 the Density equal to zero for a Material, you can model Weightless Elements.

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



Add new Material

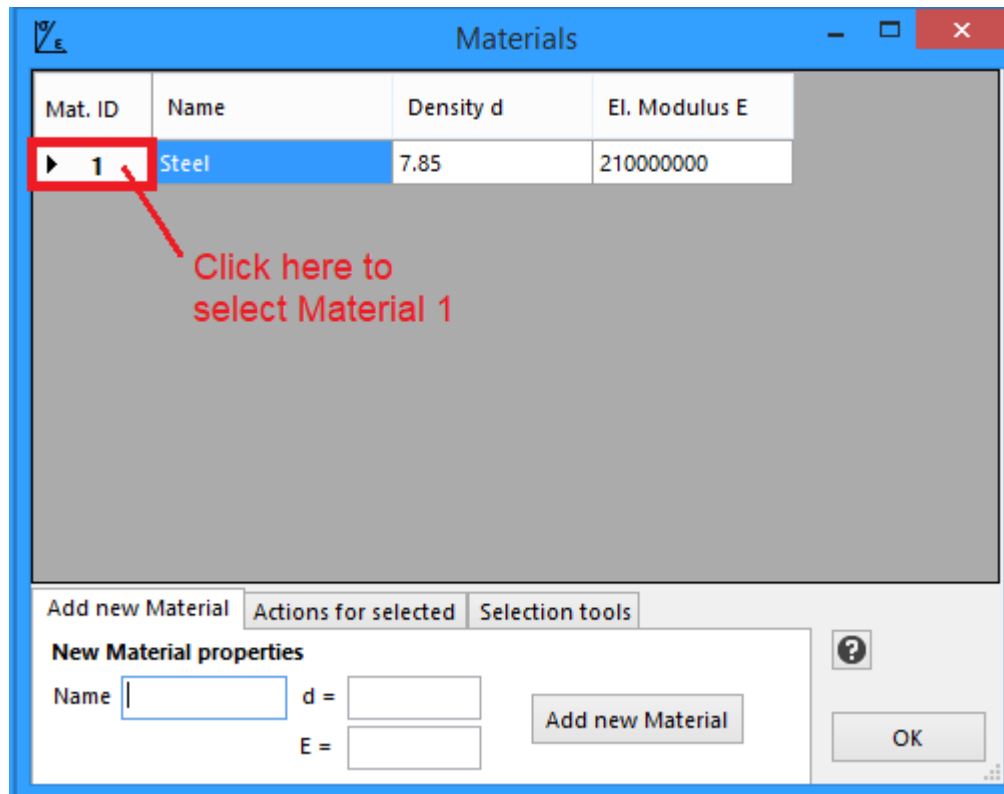
To add a new Material, type the Name (optional), Density (optional) and Elastic Modulus E of the Material and click the "Add new Material" button.

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.

Select existing Material(s)

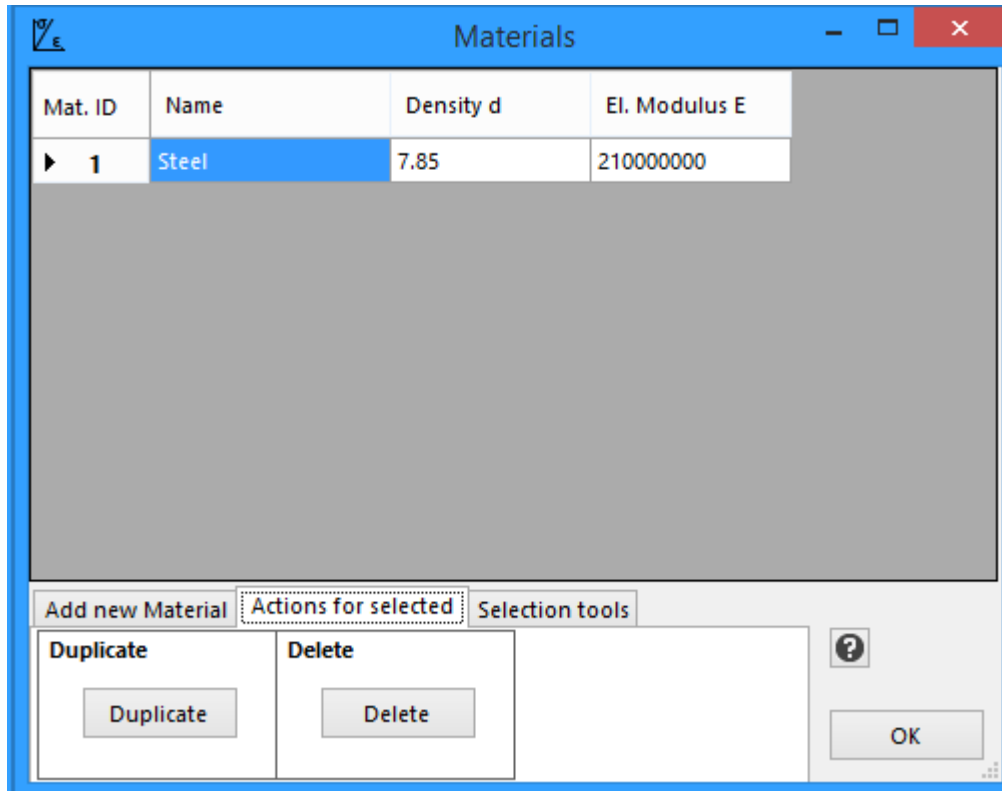
You can click on a row number to select one or more Materials. Then you can apply actions to the selected Materials, as shown next.



Actions for selected Materials

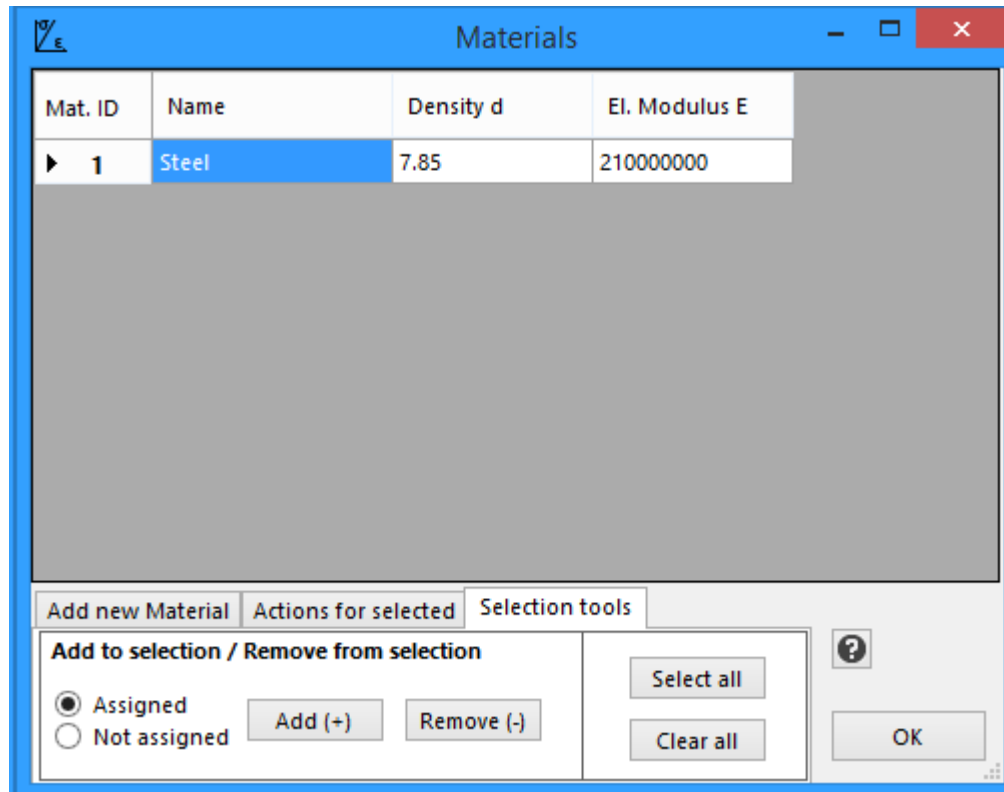
In order to perform these actions, **you have to first select the entire row(s) of the corresponding Material(s)** and then apply the action.

- **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.

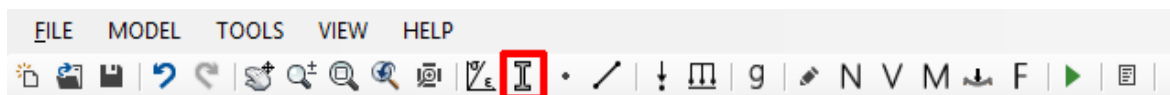


Materials selection tools

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.



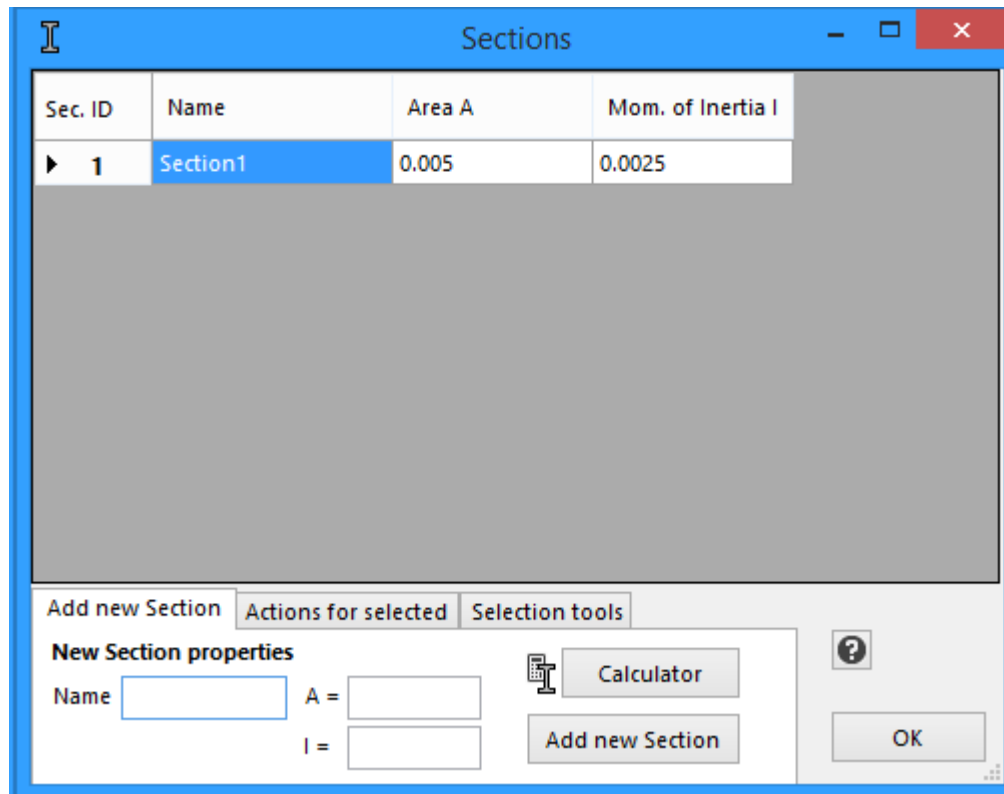
2.3.2 Sections



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

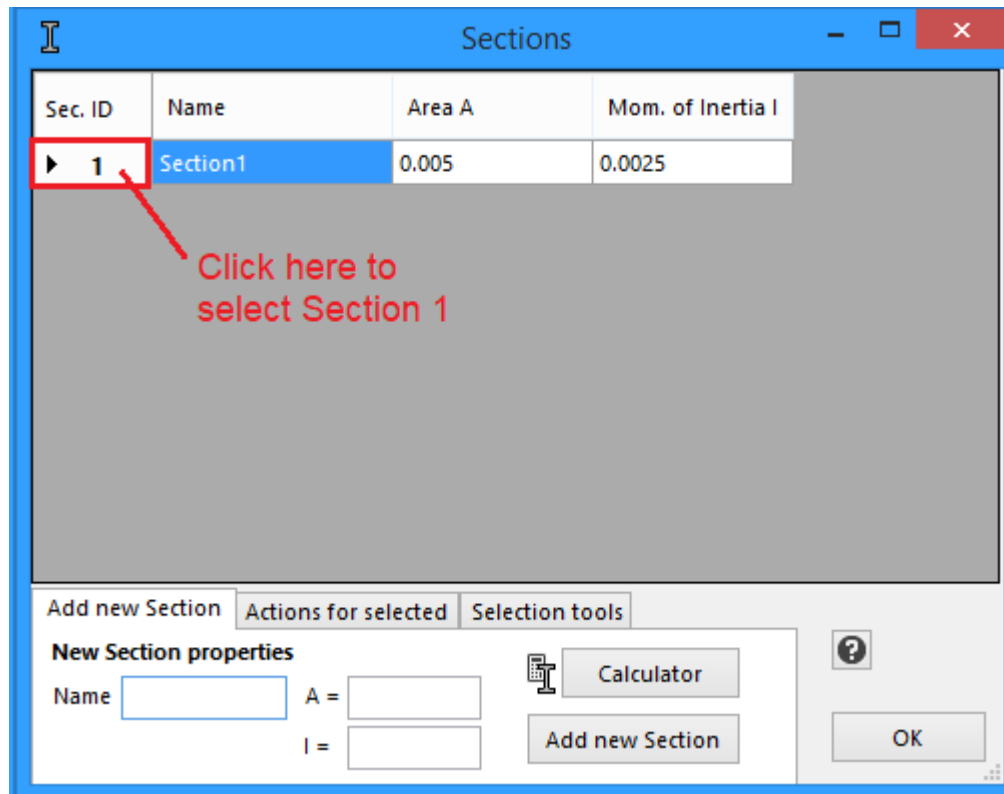
To add a new Section, type the Name (optional), Area A and Moment of Inertia I of the Section and click the "Add new Section" button.

Edit existing Section(s)

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.

Select an existing Section

You can click on a row number to select one or more Sections. Then you can apply actions to the selected Sections, as shown in "Actions for selected Sections".



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.

Prop.	Details	Value
yC	Centroid y	0.15
zC	Centroid z	0.3
A	Area	0.00831
I_y	Mom. of Inertia y	0.0005537064
I_z	Mom. of Inertia z	4.05061E-05
yPNA	Plastic Neutral axi...	0.15
zPNA	Plastic Neutral axi...	0.3
Wply	Plastic Modulus y	0.002019105
Wplz	Plastic Modulus z	0.0004086375

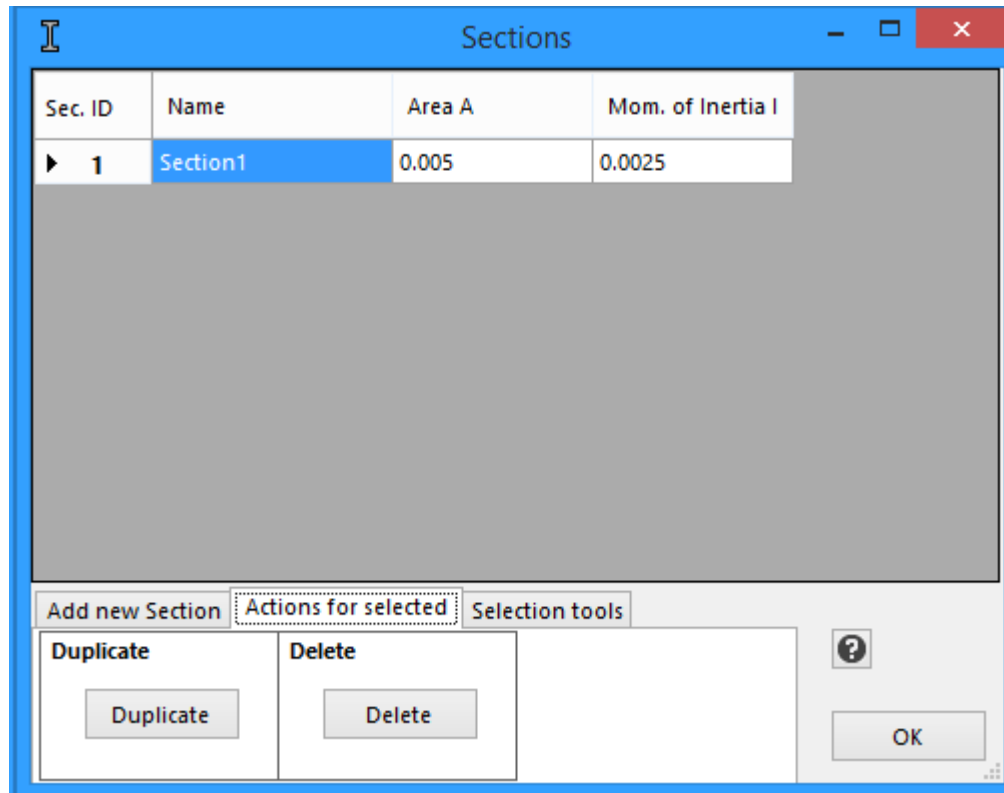
The available sections are the following:

- Square bar
- Square tube
- Rectangular bar
- Rectangular tube
- Circular bar
- Circular tube
- ✚ Cross (Normal)
- ✚ Cross (General)
- ⌚ Wide flange (Normal)
- ⌚ Wide flange (General)
- ⌚ T-Shape
- ⌚ C-Shape (Normal)
- ⌚ C-Shape (General)
- ⌚ L-Shape (Normal)
- ⌚ L-Shape (General)

Actions for selected Sections

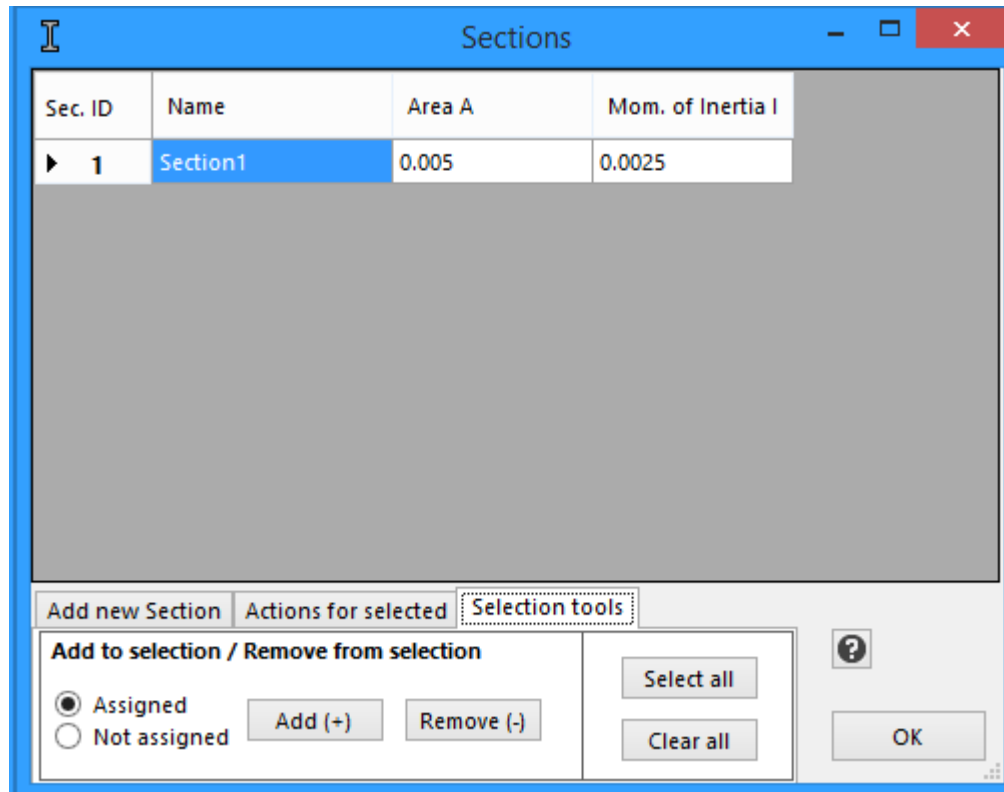
In order to perform these actions, **you have to first select the entire row(s) of the corresponding Section(s)** and then apply the action.

- **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 spring)
- Fixed (Fully constrained, not able to move at all)
- With a spring on (Given the spring stiffness, the spring provides a reaction force that is proportional to the displacement of the corresponding DOF).

Note that a DOF with a spring on is at an intermediate state between Free and Fixed. If the spring stiffness is zero, then it will behave exactly as a free DOF, while if the spring stiffness reaches infinity, it will behave as a fixed DOF. Any value in between zero and infinity will provide some stiffness.

Node properties

The Node properties are the following:

- **Coordinates: X, Y**
- **Constraints: DX-Con, DY-Con, RZ-Con.** Each one can be Checked or Not checked.
- **Springs stiffness: DX-Stiff, DY-Stiff, RZ-Stiff.** Each stiffness component has a real positive value, or a zero value. The stiffness cannot have a negative value.

• Nodes

Node ID	X	Y	DX-Con	DY-Con	RZ-Con	DX-Stiff	DY-Stiff	RZ-Stiff
1	2	0	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
2	2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
3	4	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
4	4	1.1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
5	5.8	0.4	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
6	3.5	-0.8	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
7	5.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
8	7.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
9	10.4	-0.8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
10	8.2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0

Add new Node Selection tools Renumber Move/Copy Set Constraints Delete

New Node coordinates

X=

Y=

Add new

Add new (Relative)

?

↶ ↷

OK

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 (DX, DY or RZ) of the Node is constrained or not. DX and DY are the X and Y displacements of the node, respectively. RZ is the rotation around the Z axis.

Each DOF can be:

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

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

Picture	DX-Constraint	DY-Constraint	RZ-Constraint	Description
	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 displacement of the corresponding DOF. Each stiffness component has a real positive value, or a zero value. The stiffness cannot have a negative value.

Note: A DOF that has a spring should not be constrained, as there is no point in that - Any constrained DOF will not have a displacement and as a result the spring reaction will be zero. 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 fully constrained.

Add new Node

To add a new Node, you can type the X and Y Coordinates of the Node and click the "Add new" button. This uses the **absolute coordinates X, Y** for the definition of the new Node.

• Nodes

Node ID	X	Y	DX-Con	DY-Con	RZ-Con	DX-Stiff	DY-Stiff	RZ-Stiff
▶ 1	2	0	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
2	2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
3	4	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
4	4	1.1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
5	5.8	0.4	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
6	3.5	-0.8	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
7	5.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
8	7.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
9	10.4	-0.8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
10	8.2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0

Add new Node Selection tools Renumber Move/Copy Set Constraints Delete

New Node coordinates
 X=
 Y=

Add new
 Add new (Relative)

?
 ↶ ↷
 OK

Another convenient way to define a new Node is by using relative coordinates, based on the coordinates of a previously selected Node. For example, you select Node (row) 10, you set the X and Y values, and then you click "**Add new (Relative)**". The new Node is defined in relation to the previously selected Node. This is very convenient when defining Nodes with a fixed distance between them. In this case, the previous Node acts as the selected Node, so you can keep adding Nodes using relative coordinates based on the previously defined Node.

Note: If multiple Nodes are selected and the "Add new (Relative)" button is clicked, then the Node with the highest ID (of the selected ones) is taken into account as the reference Node for the definition of the new Node.

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 on the screen.

Select existing Node(s)

You can click on a row number to select one or more Nodes. Then you can apply actions to the selected Nodes (renumber, move/copy, set constraints, delete, etc).

Click to select ALL

Click to select Nodes

Node ID	X	Y	DX-Con	DY-Con	RZ-Con	DX-Stiff	DY-Stiff	RZ-Stiff
1	2	0	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
2	2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
3	4	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
4	4	1.1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
5	5.8	0.4	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
6	3.5	-0.8	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
7	5.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
8	7.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
9	10.4	-0.8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
10	8.2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0

Add new Node Selection tools Renumber Move/Copy Set Constraints Delete

New Node coordinates

X=

Y=

Add new

Add new (Relative)

?

↶ ↷

OK

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. You can also Select all Nodes, or Clear the selection.

• Nodes

Node ID	X	Y	DX-Con	DY-Con	RZ-Con	DX-Stiff	DY-Stiff	RZ-Stiff
▶ 1	2	0	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
2	2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
3	4	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
4	4	1.1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
5	5.8	0.4	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
6	3.5	-0.8	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
7	5.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
8	7.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
9	10.4	-0.8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
10	8.2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0

Add new Node Selection tools Renumber Move/Copy Set Constraints Delete

Add to / Remove from selection

☒ Connected Add (+) Select all
☐ Not connected Remove (-) Clear selection

? ↶ ↷ OK

Renumber Nodes

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

A tool is available for the renumbering of (the selected) Nodes, based on various criteria:

- X Coordinate
- Y Coordinate
- (X + Y)
- (X - Y)
- (X² + Y²)

First, the Nodes have to be selected. Then we select the "Sort by" field and then the sort order (ascending or descending). Finally, we click the "Renumber" button.

Important: Renumbering is done for the selected Nodes. The non-selected Nodes remain unchanged. To renumber all Nodes, all Nodes have to be selected first.

• Nodes

Node ID	X	Y	DX-Con	DY-Con	RZ-Con	DX-Stiff	DY-Stiff	RZ-Stiff
▶ 1	2	0	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
2	2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
3	4	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
4	4	1.1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
5	5.8	0.4	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
6	3.5	-0.8	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
7	5.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
8	7.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
9	10.4	-0.8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
10	8.2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0

Add new Node
Selection tools
Renumber
Move/Copy
Set Constraints
Delete

Renumber selected Nodes

Sort by
X Coordinate

Sort order
☒ Ascending
☐ Descending

Renumber

?

↶

↷

OK

Move/Copy Nodes

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

This tool is used for moving or copying Nodes. The user specifies the Move/Copy Vector $\{V\}=\{dX, dY\}$.

• Nodes

Node ID	X	Y	DX-Con	DY-Con	RZ-Con	DX-Stiff	DY-Stiff	RZ-Stiff
▶ 1	2	0	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
2	2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
3	4	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
4	4	1.1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
5	5.8	0.4	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
6	3.5	-0.8	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
7	5.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
8	7.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
9	10.4	-0.8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
10	8.2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0

Add new Node Selection tools Renumber Move/Copy Set Constraints Delete

Move / Copy selected Nodes

dX= ☐ Move ☒ Copy
 dY= ☒ Copy Constraints ☐ Copy Springs ☐ Copy Nodal Loads

No of times

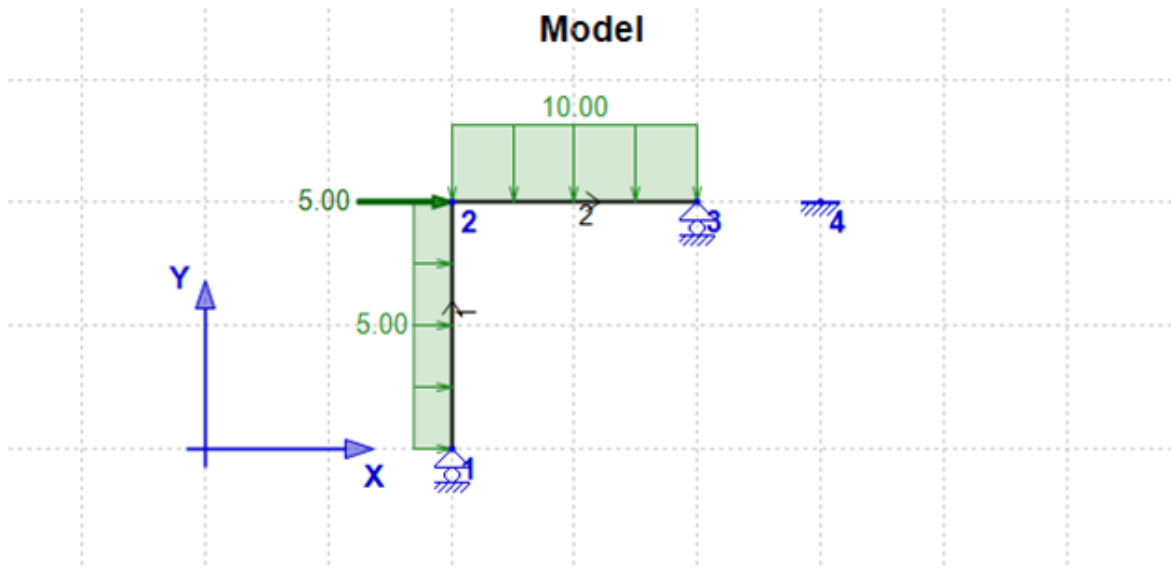
Apply

OK

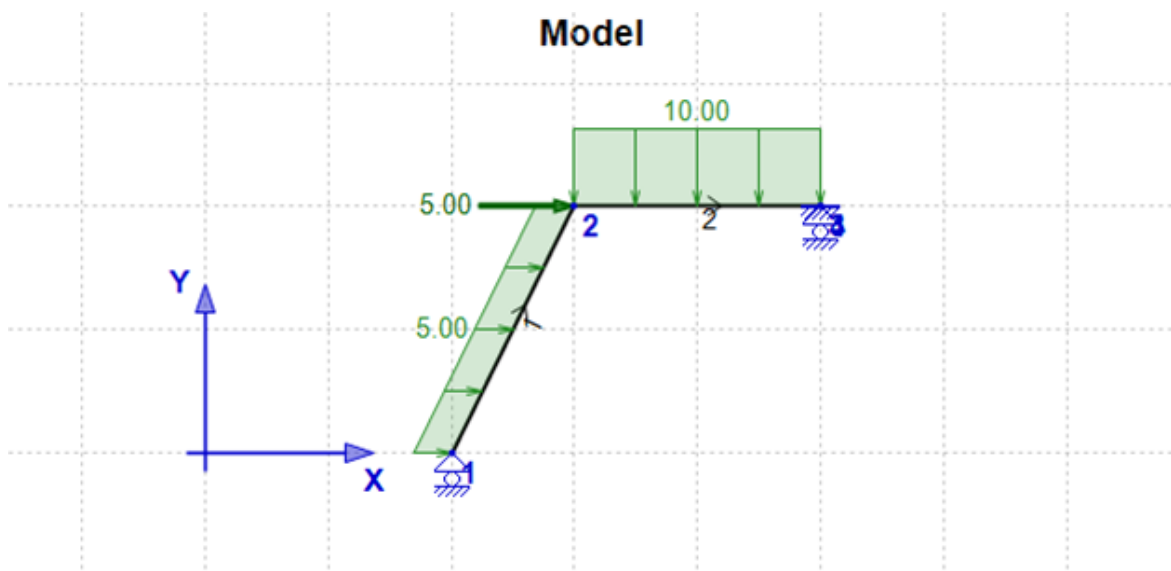
Move Nodes

- Multiple Nodes (one or more) can be selected to be moved at once.
- Move Nodes can only be applied once (only one time), not a number of times (only copy works with the option to apply it over a number of times).
- With the Move command, several nodes can be moved to a new location. The user defines the move vector as {dX, dY}.
- Nodes are moved together with their supports, springs and any nodal loads on them.
- If another Node is already at the new position, it is not affected. In the end there will be two Nodes with different IDs, at the same position.
- If elements are connected to the Nodes that move, they will be moved also together with the Nodes, as Elements depend on their Nodes for their start and end points.
- If both Nodes (start Node i and end Node j) of an Element move, then the Element itself is moved without any change in its length or its other properties. The Element simply has a new position in the 2D plane, as both its Nodes move the same.
- If only one Node (either the start Node i or the end Node j) of an Element moves, then the Element's length will change. An Elemental load on the Element will not change its value (as it is defined per unit length of the element), but the total load (total force) on the element will do change, due to the change in the length of the Element.

Example 1 – Moving Nodes



In the above example, Nodes 2 and 3 are moved with the vector $\{DX=1, DY=0\}$. The result of the operation is the following:



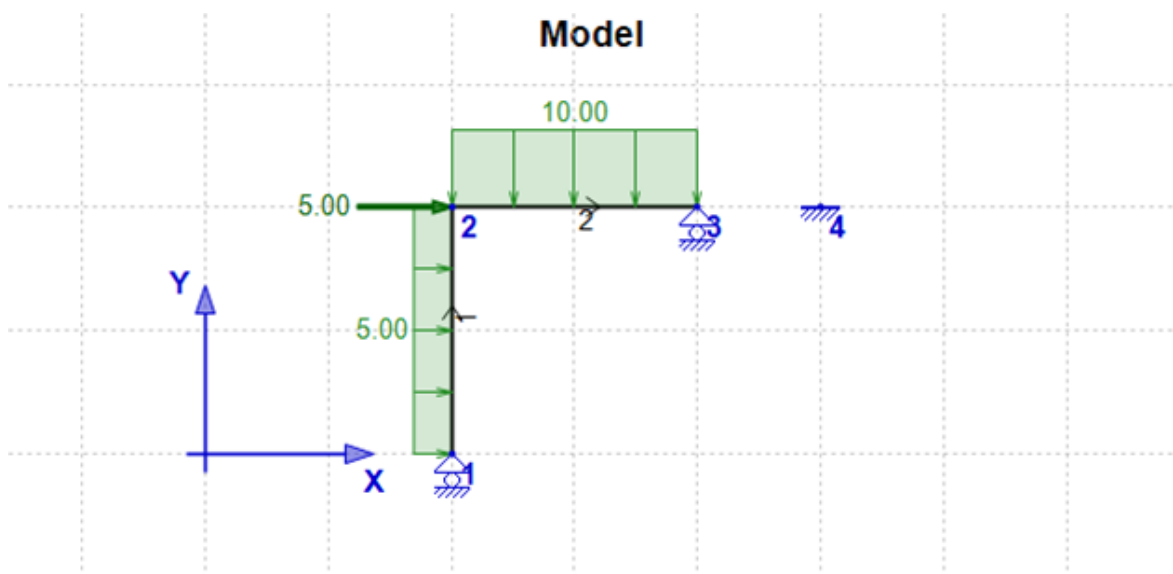
- Note the change in the length of Element 1 because of Node 2 moving to point (3, 2) from point (2, 2).
- The length of Element 2 is not affected as both its Nodes (start and end) have moved the same.
- The uniform load on Element 1 remains 5 units of force per unit of length, but the total load on Element 1 has now changed to $5 \times 2.24 = 11.2$ from $5 \times 2 = 10$ as it refers to the new length of the element, which is now increased.
- Node 3 has moved to (5, 2) where Node 4 was already previously. Node 4 is not affected and in the end, there will be two Nodes at the same location (5, 2).

Copy Nodes

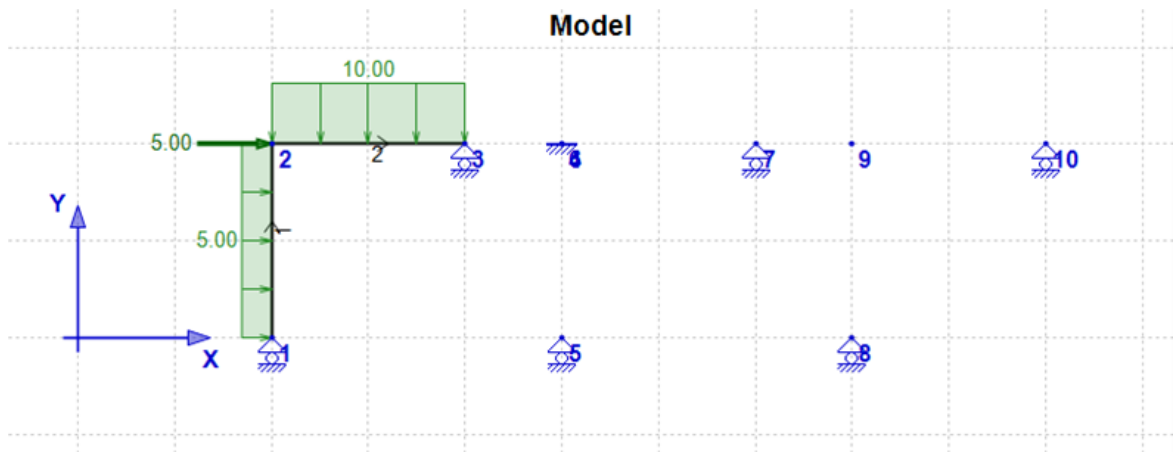
- Multiple Nodes (one or more) can be selected to be copied at once.
- Copy Nodes can be applied multiple times (up to 50) with a single click.
- With the Copy command, several Nodes can be copied to a new location. The user defines the copy vector as $\{dX, dY\}$ and the number of times the operation will be applied.
- If the corresponding check boxes are checked, Nodes are copied together with their Constraints (supports), Springs and Nodal Loads (forces or moments)
- By default, the "Copy Constraints" and the "Copy Springs" check boxes are checked, while the "Copy Nodal Loads" check box is NOT checked. All these can be changed by the user.
- If another Node is already at the new position, it is not affected. In the end there will be two Nodes with different IDs, at the same position.
- Copy Nodes does not affect any Elements, as elements depend on existing Nodes only.

Example 2 – Copying Nodes

Copy Nodes works similarly with Move Nodes, but this time new Nodes are created instead of moving already existing ones. In the previous model of Example 1:



We now copy Nodes 1, 2 and 3 with the vector $\{DX=3, DY=0\}$, 2 times. The result of the operation is the following:



- Node 4 was already at (5, 2) and is not affected. New Node 6 is also created at the same location (5, 2).
- Constraints (and springs) are copied together with the Nodes.
- Nodal forces (or moments) are NOT copied together with the Nodes (see Node 2 and new Nodes 6 and 9).
- Copy Nodes does not affect any Elements as elements depend on existing Nodes only.

Set Constraints

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

This tool is used for setting constraints on selected Nodes. You can set constraints for more than one Nodes at a time. First select the Nodes (rows) that you want to set the constraints on. Set the constraints by checking the DX, DY or RZ constraint checkbox. Finally, click the "Set" button to perform the operation.


• Nodes

Node ID	X	Y	DX-Con	DY-Con	RZ-Con	DX-Stiff	DY-Stiff	RZ-Stiff
▶ 1	2	0	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
2	2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
3	4	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
4	4	1.1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
5	5.8	0.4	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
6	3.5	-0.8	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
7	5.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
8	7.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
9	10.4	-0.8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
10	8.2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0

Add new Node
Selection tools
Renumber
Move/Copy
Set Constraints
Delete

Set Constraints to selected Nodes

☐ DX-Constraint
☐ DY-Constraint
☐ RZ-Constraint



Set

?

↶ ↷

OK

Delete Nodes

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

This tool is used for deleting Nodes. You can Delete more than one Nodes at a time. First select the Nodes (rows) that you want to delete. Then, click the "Delete" button to delete the Nodes.

• Nodes

Node ID	X	Y	DX-Con	DY-Con	RZ-Con	DX-Stiff	DY-Stiff	RZ-Stiff
▶ 1	2	0	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
2	2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
3	4	2	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
4	4	1.1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
5	5.8	0.4	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
6	3.5	-0.8	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0
7	5.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
8	7.1	-0.8	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
9	10.4	-0.8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	0	0	0
10	8.2	2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0

Add new Node Selection tools Renumber Move/Copy Set Constraints Delete

Delete selected Nodes

Delete

? ↶ ↷ OK

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.

Element properties

The **Element** user-defined properties are the following:

- **Material**
- **Section**
- **Nodes: Node i** (Start Node), **Node j** (End Node)
- **Hinges: Hinge i** (Hinge at Start Node i), **Hinge j** (Hinge at End Node j), each one either Checked or Not Checked

The derived Element properties (**Read-only properties**, calculated automatically by the program) are the following:

- **Length:** Length of the Element (in distance units, for example m). It is calculated based on the locations of the Start and End Nodes, as $L = \sqrt{(X_j - X_i)^2 + (Y_j - Y_i)^2}$
- **Volume:** Volume of the Element (in distance³ units, for example m³). It is calculated as the Length of the Element times the Area of the Cross section, $V = L \cdot A$.
- **Mass:** Mass of the Element (in mass units, for example ton or kg, based on the system used). It is calculated as the Volume of the Element times the Density of the Material, $M = V \cdot d$.

Elem. ID	Material	Section	Node i	Node j	Hinge i	Hinge j	Length	Volume	Mass
1	1. Steel	1. Beam	1	2	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
2	1. Steel	1. Beam	2	3	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
3	2. Concrete	1. Beam	4	5	<input type="checkbox"/>	<input type="checkbox"/>	1.93	0.00483	0.01207
4	2. Concrete	2. Column	6	7	<input type="checkbox"/>	<input type="checkbox"/>	1.60	0.00320	0.00800
5	1. Steel	1. Beam	7	8	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
6	1. Steel	1. Beam	8	9	<input type="checkbox"/>	<input type="checkbox"/>	3.30	0.00825	0.06476
7	2. Concrete	1. Beam	3	4	<input type="checkbox"/>	<input type="checkbox"/>	0.90	0.00225	0.00562
8	2. Concrete	1. Beam	4	6	<input type="checkbox"/>	<input type="checkbox"/>	1.96	0.00491	0.01228
9	1. Steel	1. Beam	5	8	<input type="checkbox"/>	<input type="checkbox"/>	1.77	0.00442	0.03472
10	1. Steel	1. Beam	9	10	<input type="checkbox"/>	<input type="checkbox"/>	3.56	0.00890	0.06988
11	1. Steel	1. Beam	10	5	<input type="checkbox"/>	<input type="checkbox"/>	2.88	0.00721	0.05661
12	1. Steel	1. Beam	4	10	<input type="checkbox"/>	<input type="checkbox"/>	4.30	0.01074	0.08430

Add new Element Selection tools Renummer Move/Copy Divide Delete

New Element properties

Material 1. Steel
Section 1. Beam

Node i
Node j

☐ i Hinges ☐ j

Add new

? ↶ ↷ 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" 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 on the screen.

Select existing Element(s)

You can click on a row number to select one or more Elements. Then you can apply actions to the selected Elements (renumber, move/copy, divide, delete, etc).

Click to select ALL Click to select Elements

Elem. ID	Material	Section	Node i	Node j	Hinge i	Hinge j	Length	Volume	Mass
1	1. Steel	1. Beam	1	2	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
2	1. Steel	1. Beam	2	3	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
3	2. Concrete	1. Beam	4	5	<input type="checkbox"/>	<input type="checkbox"/>	1.93	0.00483	0.01207
4	2. Concrete	2. Column	6	7	<input type="checkbox"/>	<input type="checkbox"/>	1.60	0.00320	0.00800
5	1. Steel	1. Beam	7	8	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
6	1. Steel	1. Beam	8	9	<input type="checkbox"/>	<input type="checkbox"/>	3.30	0.00825	0.06476
7	2. Concrete	1. Beam	3	4	<input type="checkbox"/>	<input type="checkbox"/>	0.90	0.00225	0.00562
8	2. Concrete	1. Beam	4	6	<input type="checkbox"/>	<input type="checkbox"/>	1.96	0.00491	0.01228
9	1. Steel	1. Beam	5	8	<input type="checkbox"/>	<input type="checkbox"/>	1.77	0.00442	0.03472
10	1. Steel	1. Beam	9	10	<input type="checkbox"/>	<input type="checkbox"/>	3.56	0.00890	0.06988
11	1. Steel	1. Beam	10	5	<input type="checkbox"/>	<input type="checkbox"/>	2.88	0.00721	0.05661
12	1. Steel	1. Beam	4	10	<input type="checkbox"/>	<input type="checkbox"/>	4.30	0.01074	0.08430

Add new Element Selection tools Renumber Move/Copy Divide Delete

New Element properties

Material: 1. Steel Section: 1. Beam Node i: Node j:

☐ i Hinges j ☐

Add new

OK

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 a specific Section. This is useful when for example you need to Delete all the Elements with a specific Section. You can also Select all Elements, or Clear the selection.

Elements

Elem. ID	Material	Section	Node i	Node j	Hinge i	Hinge j	Length	Volume	Mass
1	1. Steel	1. Beam	1	2	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
2	1. Steel	1. Beam	2	3	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
3	2. Concrete	1. Beam	4	5	<input type="checkbox"/>	<input type="checkbox"/>	1.93	0.00483	0.01207
4	2. Concrete	2. Column	6	7	<input type="checkbox"/>	<input type="checkbox"/>	1.60	0.00320	0.00800
5	1. Steel	1. Beam	7	8	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
6	1. Steel	1. Beam	8	9	<input type="checkbox"/>	<input type="checkbox"/>	3.30	0.00825	0.06476
7	2. Concrete	1. Beam	3	4	<input type="checkbox"/>	<input type="checkbox"/>	0.90	0.00225	0.00562
8	2. Concrete	1. Beam	4	6	<input type="checkbox"/>	<input type="checkbox"/>	1.96	0.00491	0.01228
9	1. Steel	1. Beam	5	8	<input type="checkbox"/>	<input type="checkbox"/>	1.77	0.00442	0.03472
10	1. Steel	1. Beam	9	10	<input type="checkbox"/>	<input type="checkbox"/>	3.56	0.00890	0.06988
11	1. Steel	1. Beam	10	5	<input type="checkbox"/>	<input type="checkbox"/>	2.88	0.00721	0.05661
12	1. Steel	1. Beam	4	10	<input type="checkbox"/>	<input type="checkbox"/>	4.30	0.01074	0.08430

Add new Element Selection tools Renumber Move/Copy Divide Delete

Add to / Remove from selection

Mat ☐ 1. Steel

Sec. ☐ 1. Beam

Renumber Elements

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

Elem. ID	Material	Section	Node i	Node j	Hinge i	Hinge j	Length	Volume	Mass
1	1. Steel	1. Beam	1	2	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
2	1. Steel	1. Beam	2	3	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
3	2. Concrete	1. Beam	4	5	<input type="checkbox"/>	<input type="checkbox"/>	1.93	0.00483	0.01207
4	2. Concrete	2. Column	6	7	<input type="checkbox"/>	<input type="checkbox"/>	1.60	0.00320	0.00800
5	1. Steel	1. Beam	7	8	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
6	1. Steel	1. Beam	8	9	<input type="checkbox"/>	<input type="checkbox"/>	3.30	0.00825	0.06476
7	2. Concrete	1. Beam	3	4	<input type="checkbox"/>	<input type="checkbox"/>	0.90	0.00225	0.00562
8	2. Concrete	1. Beam	4	6	<input type="checkbox"/>	<input type="checkbox"/>	1.96	0.00491	0.01228
9	1. Steel	1. Beam	5	8	<input type="checkbox"/>	<input type="checkbox"/>	1.77	0.00442	0.03472
10	1. Steel	1. Beam	9	10	<input type="checkbox"/>	<input type="checkbox"/>	3.56	0.00890	0.06988
11	1. Steel	1. Beam	10	5	<input type="checkbox"/>	<input type="checkbox"/>	2.88	0.00721	0.05661
12	1. Steel	1. Beam	4	10	<input type="checkbox"/>	<input type="checkbox"/>	4.30	0.01074	0.08430

Add new Element Selection tools **Renumber** Move/Copy Divide Delete

Renumber selected Elements

Sort by: Min/Max X-Coord. Sort order: ☒ Ascending ☐ Descending

[Renumber]

[?] [↶] [↷] [OK]

This tool is for the renumbering of (the selected) Elements, based on various criteria:

- **"Min/Max X-Coord."**: If the sort order is "Ascending", then the Min X-Coordinate of the Element is used. If the sort order is "Descending", then the Max X-Coordinate is used.
- **"Min/Max Y-Coord."**: If the sort order is "Ascending", then the Min Y-Coordinate of the Element is used. If the sort order is "Descending", then the Max Y-Coordinate is used.
- **"Min/Max Node ID"**: If the sort order is "Ascending", then the Min Node ID of the Element is used. If the sort order is "Descending", then the Max Node ID is used.
- **"Start Node (i) ID"**
- **"Start Node (i) X-Coord."**
- **"Start Node (i) Y-Coord."**
- **"End Node (j) ID"**
- **"End Node (j) X-Coord."**
- **"End Node (j) Y-Coord."**

First, the Elements have to be selected. Then we select the "Sort by" field and then the sort order (ascending or descending). Finally, we click the "Renumber" button.

Important: Renumbering is done for the selected Elements. The non-selected Elements remain unchanged. To renumber all Elements, all Elements have to be selected first.

Move/Copy Elements

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

This tool is used for moving or copying Elements. The user specifies the Move/Copy Vector $\{V\}=\{dX, dY\}$.

The screenshot shows a software window titled 'Elements' with a table of 12 elements and a control panel below it.

Elem. ID	Material	Section	Node i	Node j	Hinge i	Hinge j	Length	Volume	Mass
1	1. Steel	1. Beam	1	2	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
2	1. Steel	1. Beam	2	3	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
3	2. Concrete	1. Beam	4	5	<input type="checkbox"/>	<input type="checkbox"/>	1.93	0.00483	0.01207
4	2. Concrete	2. Column	6	7	<input type="checkbox"/>	<input type="checkbox"/>	1.60	0.00320	0.00800
5	1. Steel	1. Beam	7	8	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
6	1. Steel	1. Beam	8	9	<input type="checkbox"/>	<input type="checkbox"/>	3.30	0.00825	0.06476
7	2. Concrete	1. Beam	3	4	<input type="checkbox"/>	<input type="checkbox"/>	0.90	0.00225	0.00562
8	2. Concrete	1. Beam	4	6	<input type="checkbox"/>	<input type="checkbox"/>	1.96	0.00491	0.01228
9	1. Steel	1. Beam	5	8	<input type="checkbox"/>	<input type="checkbox"/>	1.77	0.00442	0.03472
10	1. Steel	1. Beam	9	10	<input type="checkbox"/>	<input type="checkbox"/>	3.56	0.00890	0.06988
11	1. Steel	1. Beam	10	5	<input type="checkbox"/>	<input type="checkbox"/>	2.88	0.00721	0.05661
12	1. Steel	1. Beam	4	10	<input type="checkbox"/>	<input type="checkbox"/>	4.30	0.01074	0.08430

Below the table is a control panel with tabs: 'Add new Element', 'Selection tools', 'Renumber', 'Move/Copy' (selected), 'Divide', and 'Delete'.

The 'Move / Copy selected Elements' section contains:

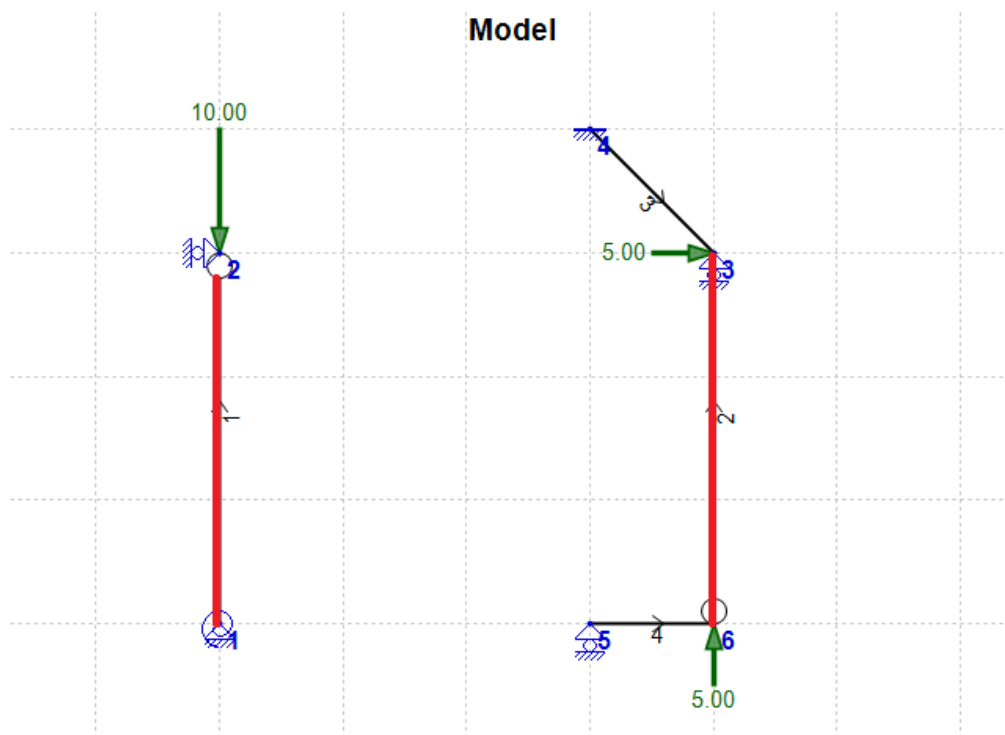
- Input fields for $dX=$ and $dY=$.
- Radio buttons for 'Move' (selected) and 'Copy'.
- A 'No of times' input field set to 1.
- A checkbox labeled 'Copy Elemental Loads also' which is checked.
- An 'Apply' button.
- Help, Undo, and Redo icons.
- An 'OK' button.

Move Elements

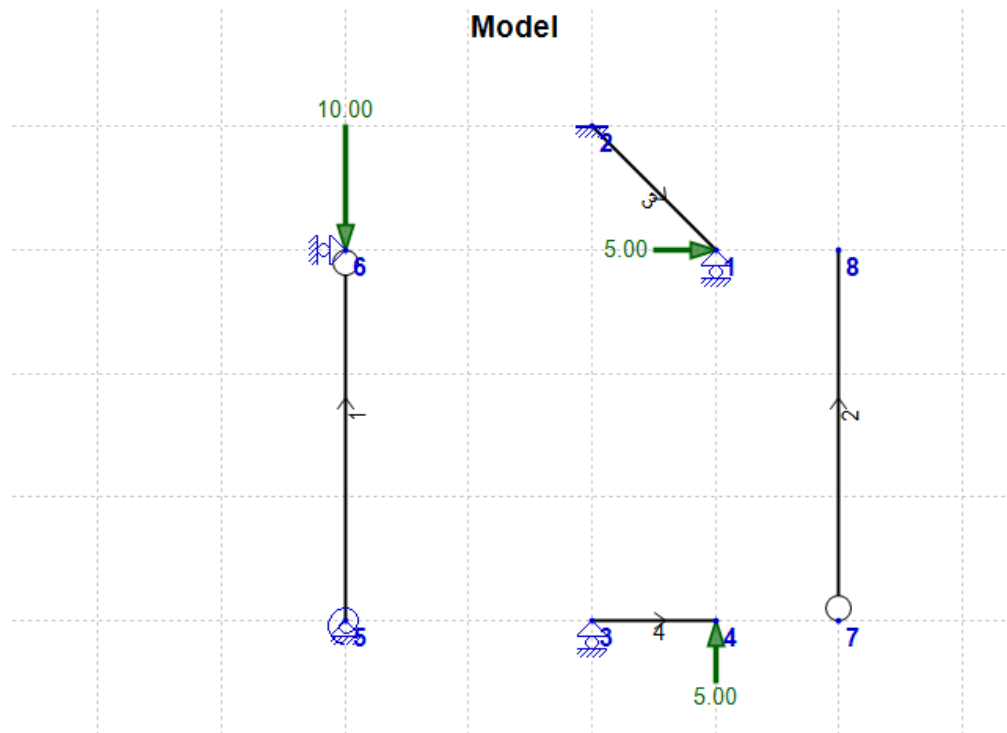
- Multiple Elements (one or more) can be selected to be moved at once.
- Move Elements can only be applied once (only one time), not a number of times (only copy works with the option to apply it over a number of times).
- With the Move command, several Elements can be moved together to new locations. The user defines the move vector as $\{dX, dY\}$.
- Elements are moved together with their hinges and any Elemental Loads on them.
- If an Element is moved that is not connected to other Elements, then its start and end nodes are actually moved (**see Example 1** below), together with any Constraints, Springs and Nodal Loads on them. As a result, the element moves.

- If an Element is moved that was previously connected to other Elements via common Nodes (Elements that are not selected to move also), then the Element is detached from the other Elements and new Nodes are generated to define its new position. The previous Nodes stay connected with the other Elements, together with any Constraints, Nodal Loads or Springs they had (**see Example 1** below).
- If an Element is moved and there is already an existing Node in the new position (either at the start i or at the end j of the new position of the Element, or at both ends i and j), then the existing Node is used for the definition of the Element, i.e. the Element is connected to the existing Node (**see Example 2** below).

Example 1 - Moving Elements, detaching Elements from Nodes

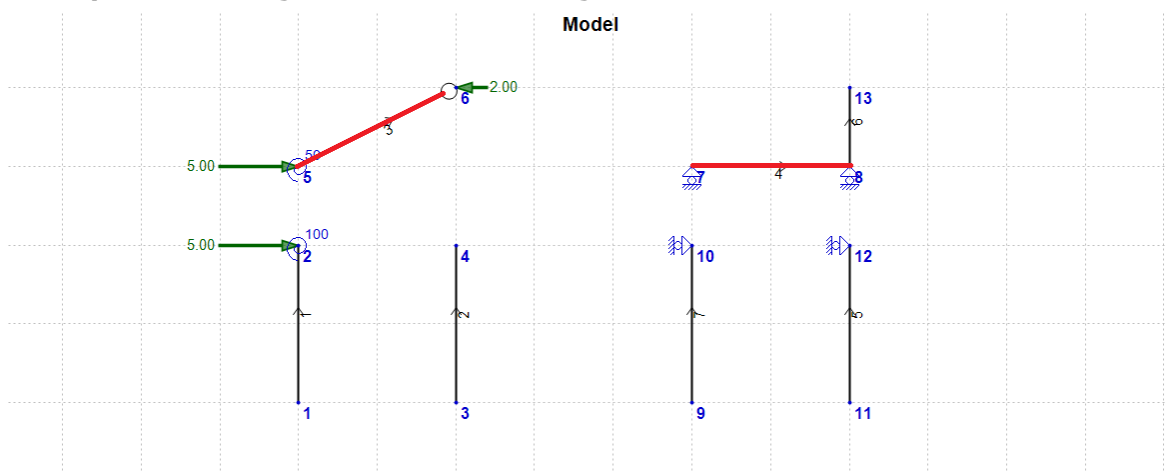


In the above example, Elements 1 and 2 are moved with the vector $\{DX=1, DY=0\}$. The result of the operation is the following:

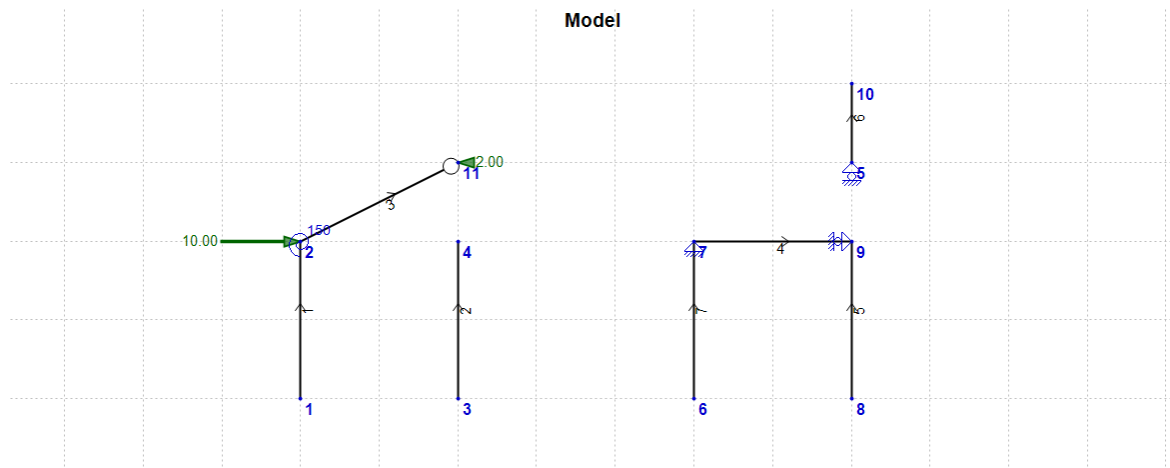


- Element 1 is moved together with its Nodes.
- Element 2 is detached from Elements 3 and 4 and it is now on its own, not connected to other Elements.
- 2 new Nodes have been automatically generated for this (Nodes 7 and 8 in the second figure).
- Some Node renumbering has occurred as a result of the operation. You can use the Nodes renumbering tool to renumber Nodes, if needed.
- The Nodal Load on Node 2 is moved together with the Node (numbered as 6 after the operation). The Nodal Loads on Nodes 3 and 6 stay with the old Nodes (numbered as 1 and 4 after the operation).

Example 2 - Moving Elements, attaching Elements to Nodes



In the above example, Elements 3 and 4 are moved with the vector $\{DX=0, DY=-1\}$. The result of the operation is the following:



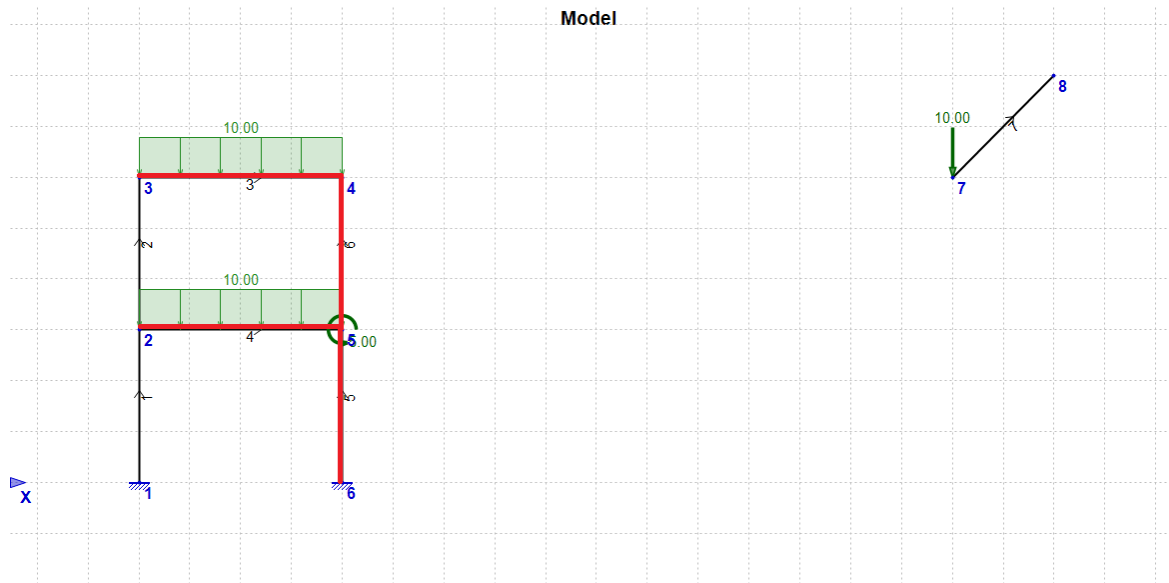
- The start Node i of Element 3 is attached (connected) to the existing Node 2. As a result, Element 3 is now connected to Element 1. Nodal loads and springs are combined in this case. The new Nodal Load on Node 2 is $5+5=10$, while the rotational spring has a stiffness of $50+100=150$.
- Element 4 is detached from Node 8 (new numbering: Node 5). And now Element 4 is connected to Nodes 10 and 12 (new numbering: Nodes 7 and 9). The two supports are combined on the left part of the element. New Node 7 now has a pinned support, while the previous supports were two rollers that are now combined. Supports are not combined at the right part of the element, as the roller support of Node 8 (new numbering: Node 5) stays behind, together with the Node which is needed for the definition of Element 6.

Copy Elements

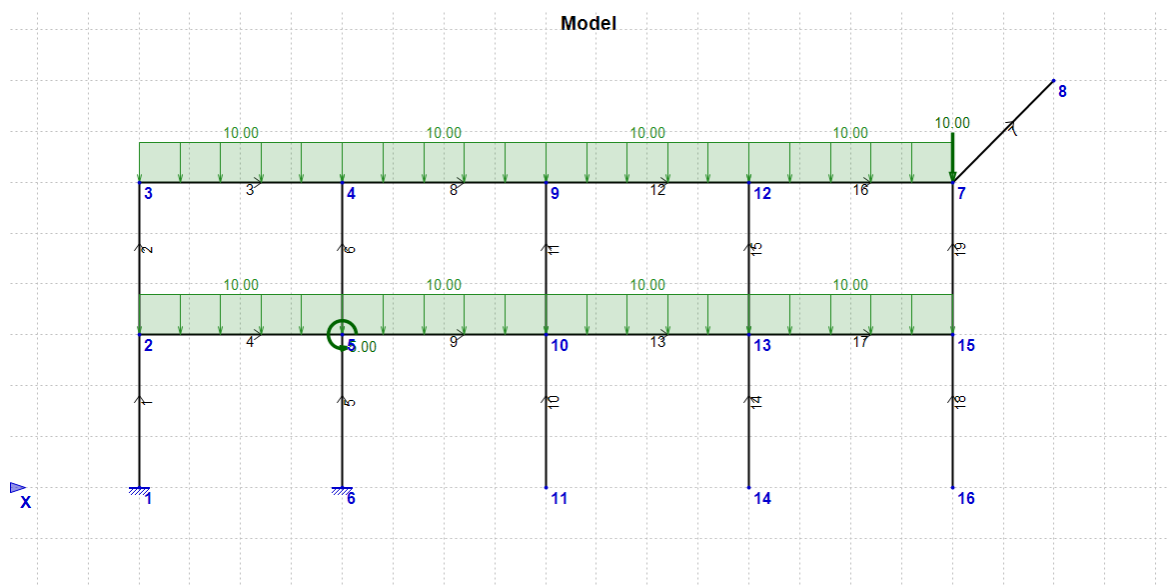
- Multiple Elements (one or more) can be selected to be copied at once.
- Copy Elements can be applied multiple times (up to 50) with a single click.
- With the Copy command, several Elements can be copied to new locations. The user defines the copy vector as $\{dX, dY\}$ and the number of times the operation will be applied.
- If the check box "Copy Elemental Loads" is checked, Elements are copied together with their Elemental Loads.
- Any hinges (start i and/or end j) are copied together with the Elements.
- By default, the "Copy Elemental Loads" check box is checked. This can be changed by the user.

Example 3 – Copying Elements

Copy Elements works similarly with Move Elements, but this time new Elements are created instead of moving already existing ones. See the example below:



We now copy Elements 3, 4, 5 and 6 with the vector $\{DX=4, DY=0\}$, 3 times. The "Copy Elemental Loads" check box is checked. The result of the operation is the following:



- New Elements and Nodes are created and we now have a fully connected frame structure.
- Node 7 was already at (20, 6) and it is used for the connections of Elements 16 and 19. No new Node is created there, the old Node is instead used.
- Elemental Loads are copied together with the Elements, because the check box "Copy Elemental Loads" was checked.
- This technique offers a very fast and efficient solution to create structures such as frames, with repeated patterns.

Divide Elements

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

Select the Elements (rows). Type the number of the division segments (up to 40) and then click the "Divide" button. You can Divide more than one Elements at a time.

Elem. ID	Material	Section	Node i	Node j	Hinge i	Hinge j	Length	Volume	Mass
1	1. Steel	1. Beam	1	2	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
2	1. Steel	1. Beam	2	3	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
3	2. Concrete	1. Beam	4	5	<input type="checkbox"/>	<input type="checkbox"/>	1.93	0.00483	0.01207
4	2. Concrete	2. Column	6	7	<input type="checkbox"/>	<input type="checkbox"/>	1.60	0.00320	0.00800
5	1. Steel	1. Beam	7	8	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
6	1. Steel	1. Beam	8	9	<input type="checkbox"/>	<input type="checkbox"/>	3.30	0.00825	0.06476
7	2. Concrete	1. Beam	3	4	<input type="checkbox"/>	<input type="checkbox"/>	0.90	0.00225	0.00562
8	2. Concrete	1. Beam	4	6	<input type="checkbox"/>	<input type="checkbox"/>	1.96	0.00491	0.01228
9	1. Steel	1. Beam	5	8	<input type="checkbox"/>	<input type="checkbox"/>	1.77	0.00442	0.03472
10	1. Steel	1. Beam	9	10	<input type="checkbox"/>	<input type="checkbox"/>	3.56	0.00890	0.06988
11	1. Steel	1. Beam	10	5	<input type="checkbox"/>	<input type="checkbox"/>	2.88	0.00721	0.05661
12	1. Steel	1. Beam	4	10	<input type="checkbox"/>	<input type="checkbox"/>	4.30	0.01074	0.08430

Add new Element Selection tools Renumber Move/Copy **Divide** Delete

Divide selected Elements

No of Segments:

Delete Elements

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

This tool is used for deleting Elements. First select the Elements (rows) that you want to delete. Then, click the "Delete" button to delete the Elements. You can Delete more than one Elements at a time.

Elements

Elem. ID	Material	Section	Node i	Node j	Hinge i	Hinge j	Length	Volume	Mass
1	1. Steel	1. Beam	1	2	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
2	1. Steel	1. Beam	2	3	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
3	2. Concrete	1. Beam	4	5	<input type="checkbox"/>	<input type="checkbox"/>	1.93	0.00483	0.01207
4	2. Concrete	2. Column	6	7	<input type="checkbox"/>	<input type="checkbox"/>	1.60	0.00320	0.00800
5	1. Steel	1. Beam	7	8	<input type="checkbox"/>	<input type="checkbox"/>	2.00	0.00500	0.03925
6	1. Steel	1. Beam	8	9	<input type="checkbox"/>	<input type="checkbox"/>	3.30	0.00825	0.06476
7	2. Concrete	1. Beam	3	4	<input type="checkbox"/>	<input type="checkbox"/>	0.90	0.00225	0.00562
8	2. Concrete	1. Beam	4	6	<input type="checkbox"/>	<input type="checkbox"/>	1.96	0.00491	0.01228
9	1. Steel	1. Beam	5	8	<input type="checkbox"/>	<input type="checkbox"/>	1.77	0.00442	0.03472
10	1. Steel	1. Beam	9	10	<input type="checkbox"/>	<input type="checkbox"/>	3.56	0.00890	0.06988
11	1. Steel	1. Beam	10	5	<input type="checkbox"/>	<input type="checkbox"/>	2.88	0.00721	0.05661
12	1. Steel	1. Beam	4	10	<input type="checkbox"/>	<input type="checkbox"/>	4.30	0.01074	0.08430

Add new Element Selection tools Renumber Move/Copy Divide **Delete**

Delete selected Elements

Delete

? ↶ ↷ OK

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. If you have already defined an Element and you would like to define a point load on it, then you can use the Divide Elements command (in the Elements form) to divide the Element and form the Node to apply the load on.

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

- **Force FX**, positive when pointing to the right
- **Force FY**, positive when pointing up
- **Bending Moment MZ**, positive when counter-clockwise

	Node ID	Force FX	Force FY	Moment MZ
1	2	30		
2	7	50		

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

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.

	Node ID	Force FX	Force FY	Moment MZ
1	2	30		
2	7	50		

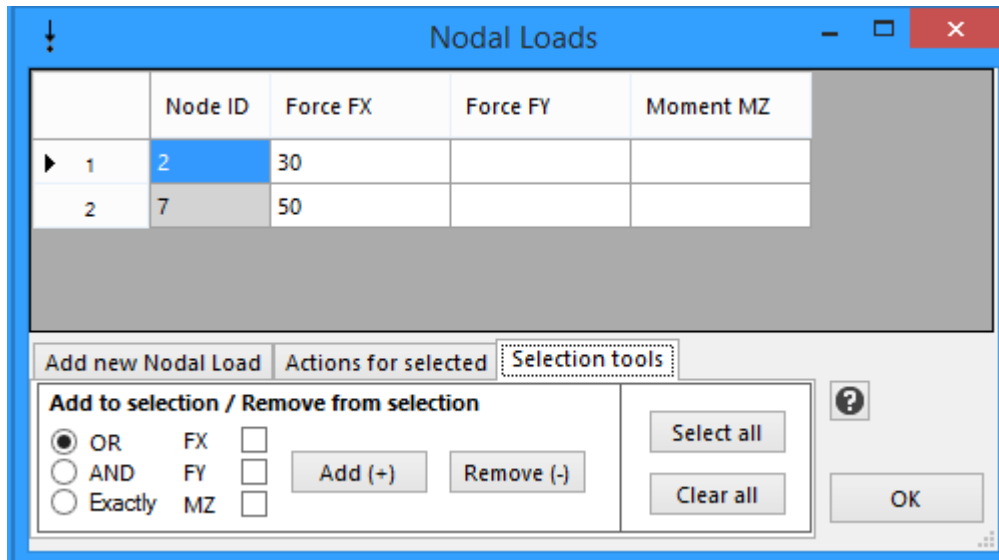
Add new Nodal Load Actions for selected Selection tools

Delete

? OK

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 Frame.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.

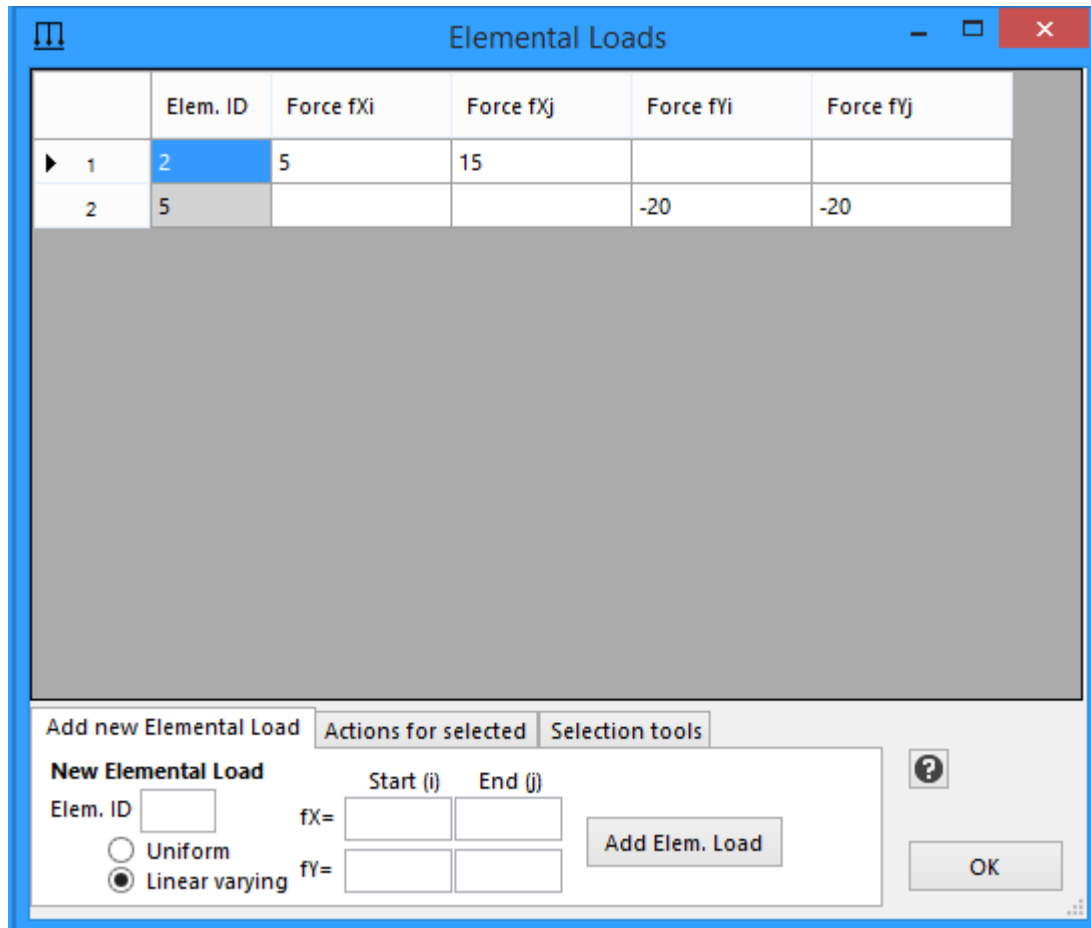
The Elemental Load properties are the following:

Elemental Load in X-Direction

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

Elemental Load in Y-Direction

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



	Elem. ID	Force f_{Xi}	Force f_{Xj}	Force f_{Yi}	Force f_{Yj}
▶ 1	2	5	15		
2	5			-20	-20

Add new Elemental Load Actions for selected Selection tools

New Elemental Load

Elem. ID fX= fY=

☐ Uniform ☒ Linear varying

Add Elem. Load ? OK

Add new Elemental Load

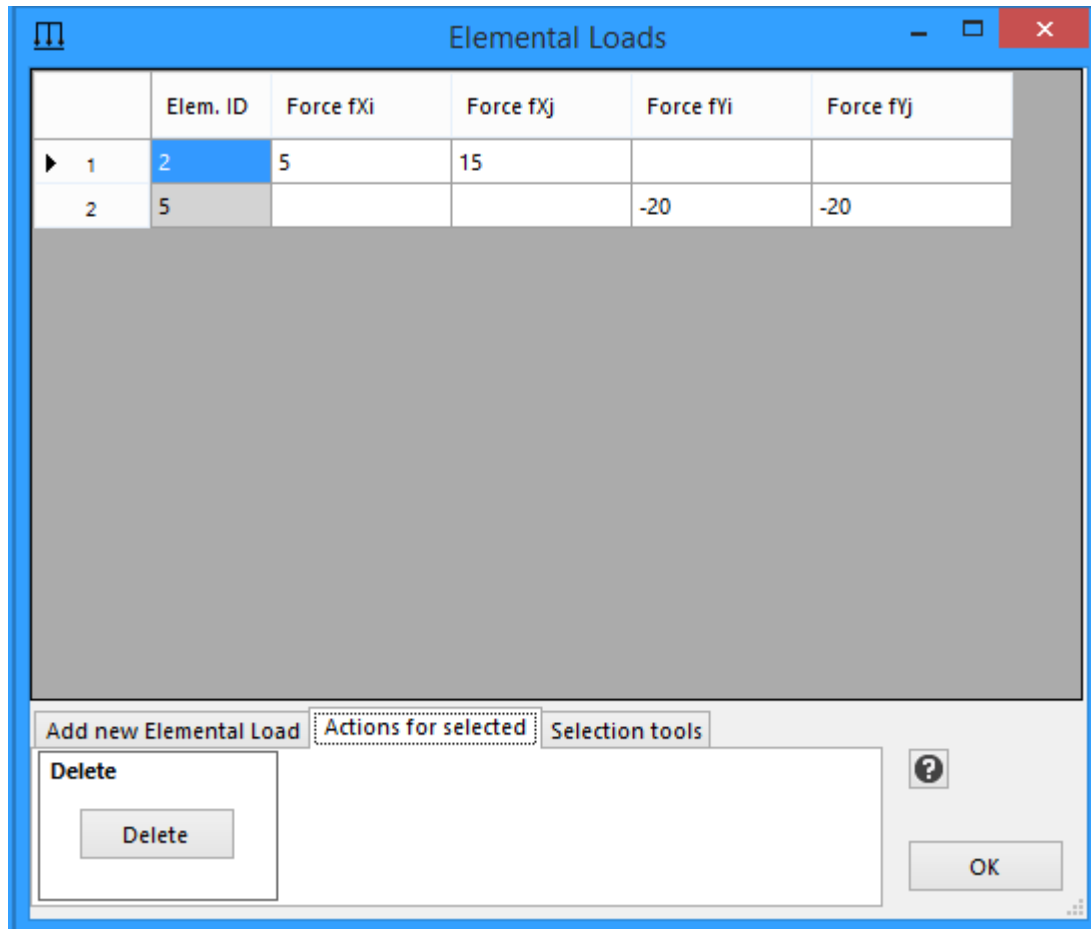
To add a new Elemental Load, first specify the Element ID. Then select Uniform or Linear varying Load. In general, all Elemental Loads are considered as linear varying loads by the program, but if you select a uniform load then the program will ask only for two values $f_X = f_{Xi} = f_{Xj}$ and $f_Y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

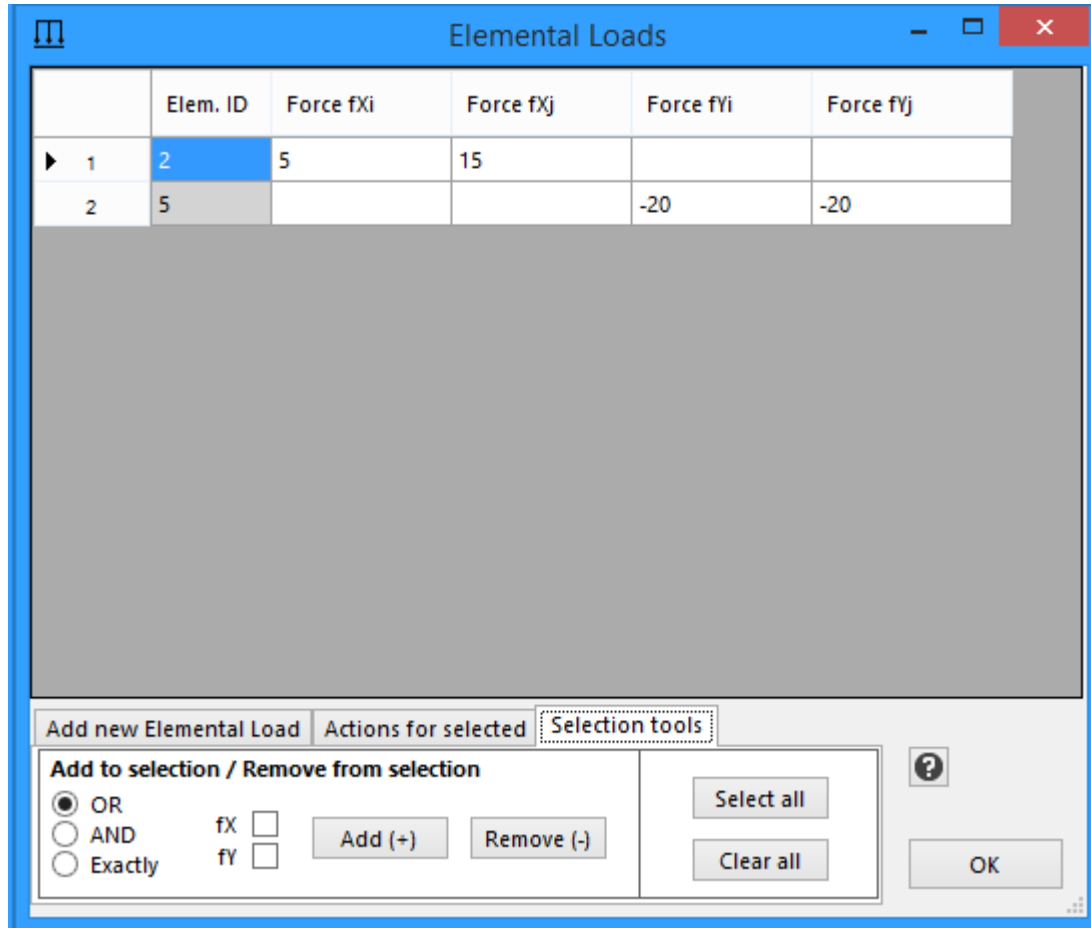
In order to perform these actions, **you have to first select the entire row(s) of the corresponding Elemental Load(s)** and then apply the action.

- **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. The values of the body loads can be seen on the form (read-only properties).

Elem. ID	Material Density	Section Area	X-Body Load	Y-Body Load	Length
1	2.5	0.25	0	-6.13125	3.00
2	2.5	0.125	0	-3.065625	5.00
3	2.5	0.25	0	-6.13125	3.00
4	2.5	0.125	0	-3.065625	5.00
5	2.5	0.25	0	-6.13125	3.00
6	2.5	0.25	0	-6.13125	3.00
7	2.5	0.125	0	-3.065625	5.00
8	2.5	0.25	0	-6.13125	3.00
9	2.5	0.125	0	-3.065625	5.00
10	2.5	0.25	0	-6.13125	3.00

Linear Acceleration Vector

aX =

aY =

Useful applications of body (Acceleration) loads

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 a Density value). Note that Density is an optional property of a Material. If there is no Density for the Material of an Element, then the self-weight of the Element will be considered as zero and there will be no body load.

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.

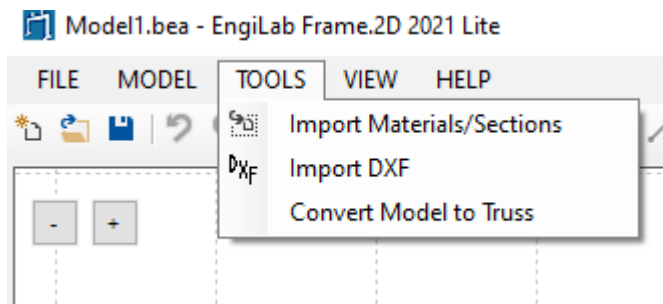
This page intentionally left blank.

Chapter



Tools

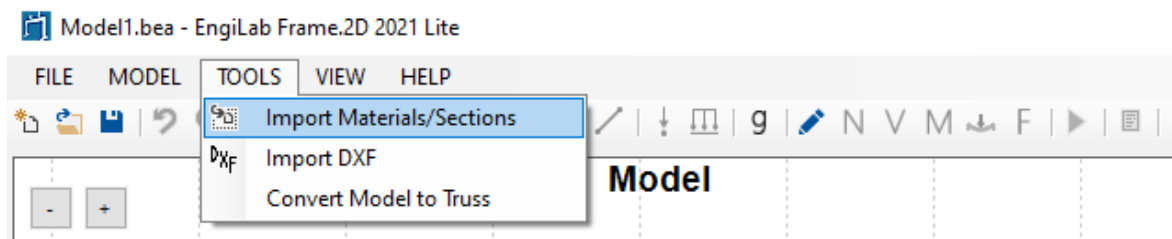
3 Tools



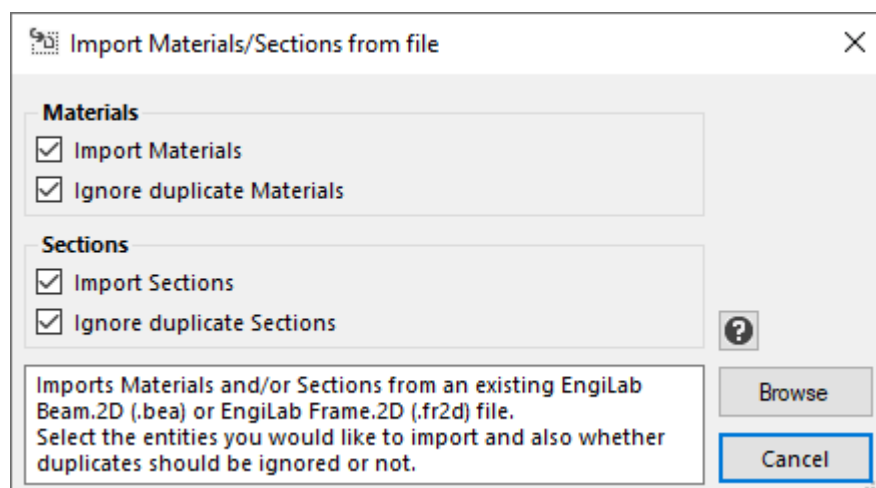
The Tools Menu of the program offers the following options:

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

3.1 Import Materials/Sections

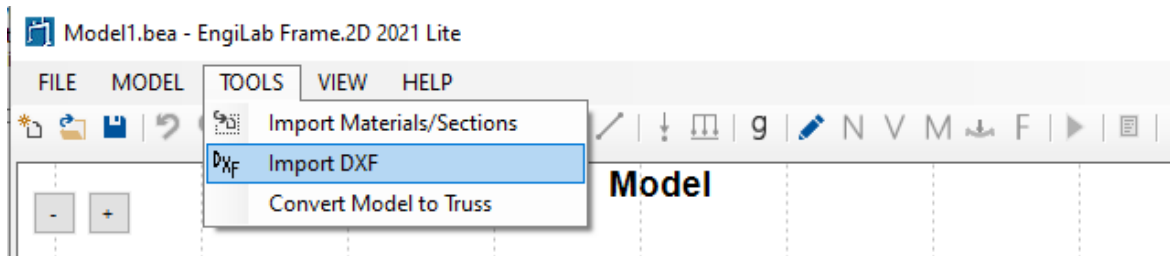


The user can import **Materials** and/or **Sections** from an existing EngiLab Frame.2D file. You can import Materials/Sections from an older EngiLab Beam.2D file (**.bea** file) or from a newer EngiLab Frame.2D file (**.fr2d** file).



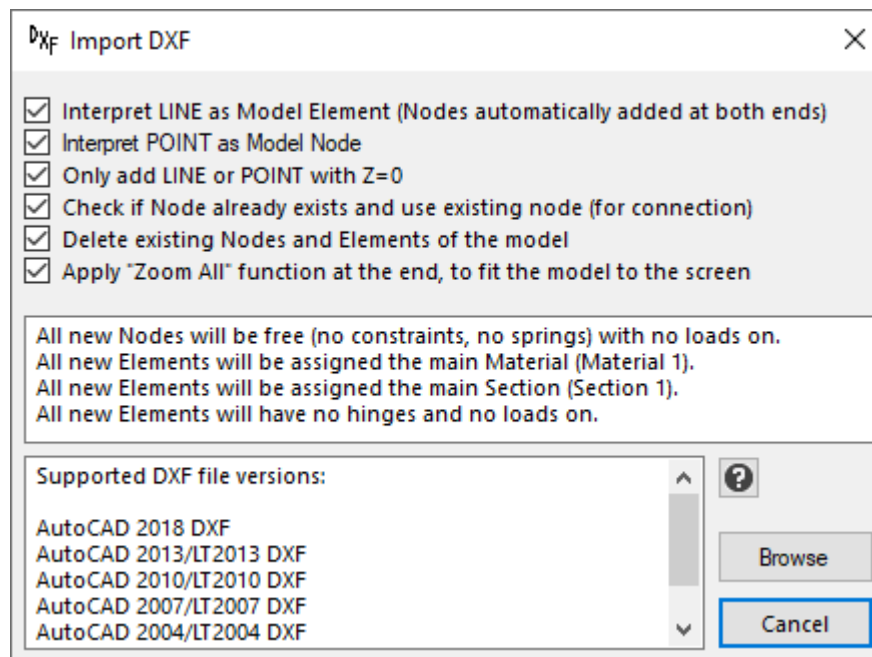
The user can choose to import Materials and/or Sections and also whether duplicate Materials and/or Sections should be ignored.

3.2 Import DXF



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 Frame.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:

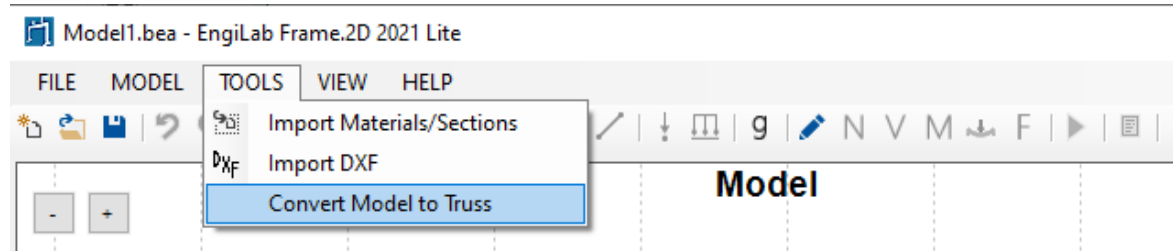
- All new Nodes will be free (no constraints, no springs) with no loads on them.
- All new Elements will be assigned the main Material (Material 1).
- All new Elements will be assigned the main Section (Section 1).
- All new Elements will have no hinges, and no loads on them.

Supported DXF file versions:

- AutoCAD 2018 DXF
- AutoCAD 2013/LT2013 DXF
- AutoCAD 2010/LT2010 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 any newer versions of AUTOCAD DXF files.

3.3 Convert Model to Truss

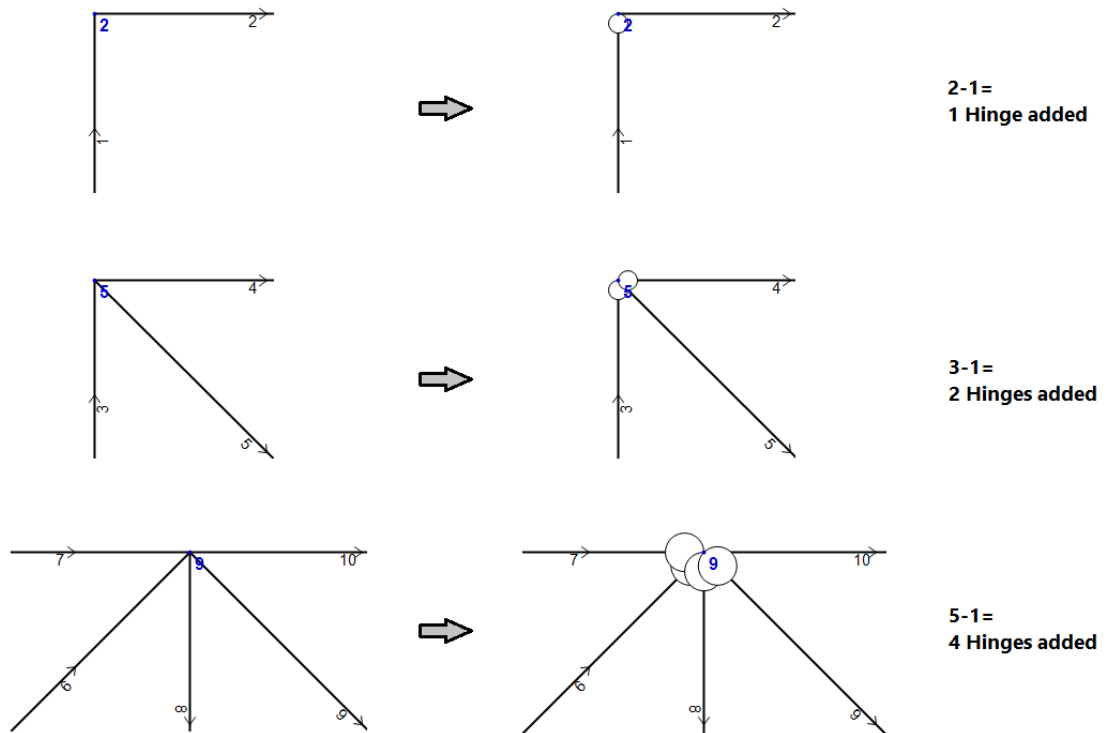


This command converts a **Frame** model to a **Truss**. A Truss model has only axial tension (no shear, no bending moment).

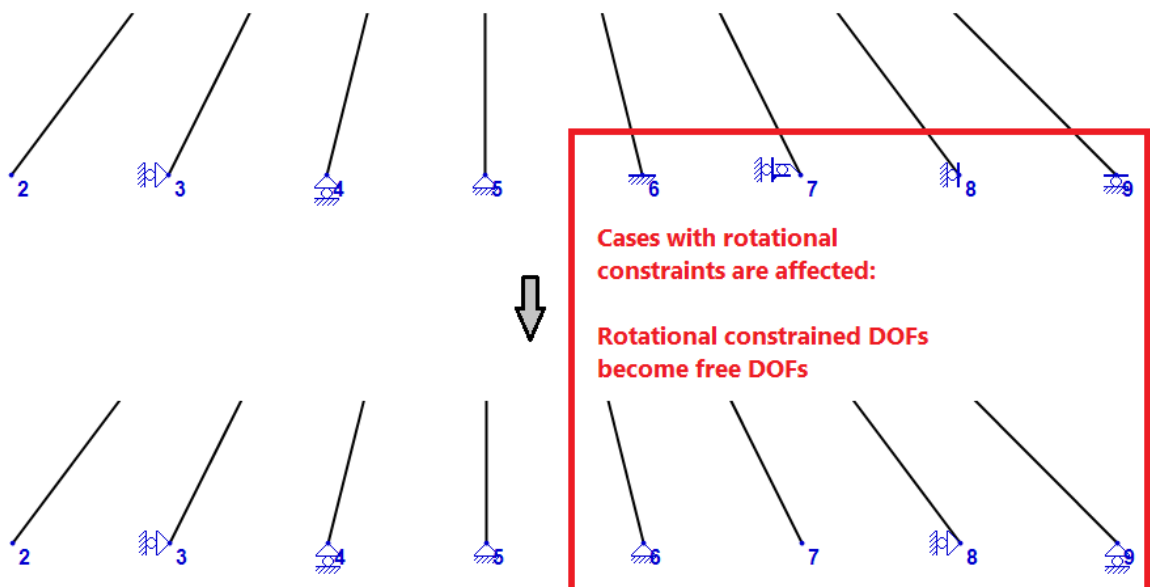
Important: Please note that if a Body (Acceleration) Load is present, then a Truss model can have also shear and bending, for example bending of a truss element due to its self weight. In this case, each element acts as a simply supported beam with a uniform load on.

In order to convert a **Frame model** to a **Truss model**, 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.

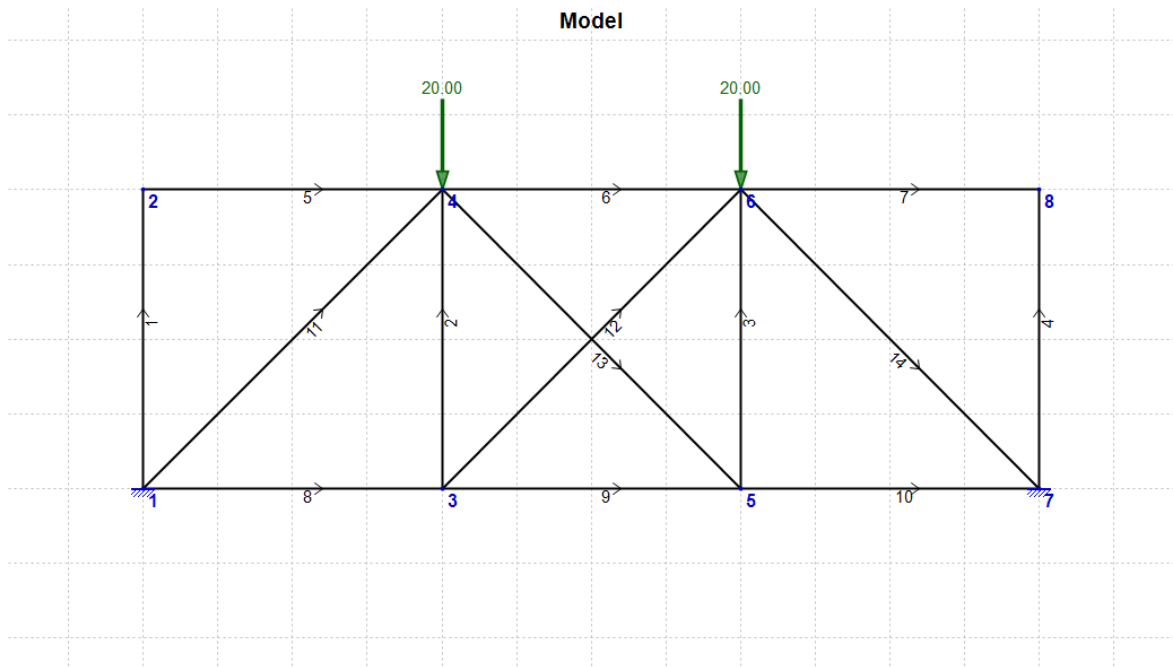


2. **Supports are converted to Truss-Model supports**, by releasing any constraints on the rotational Degree of Freedom (**RZ-Con**). The other two DOFs of each node are not affected. The picture below explains this conversion for every support case in EngiLab Frame.2D.

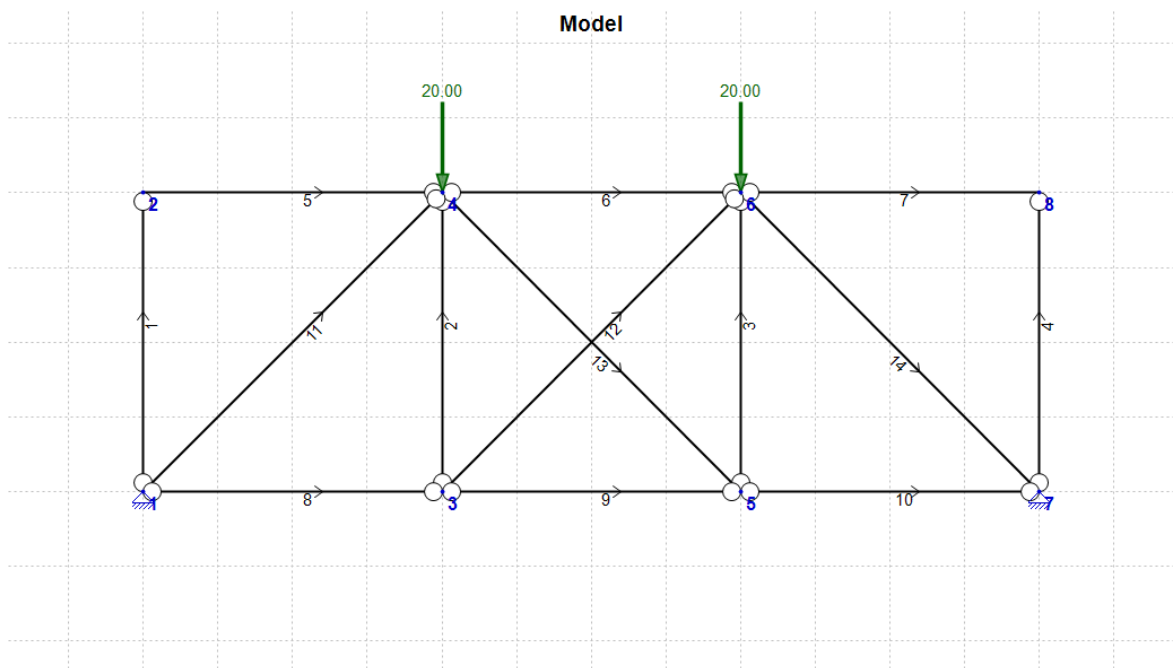


See the example below.

The picture below shows the initial Frame Model. If you analyze this Frame model, you will get axial forces, shear forces and bending moments in all elements.



The picture below shows the Model after it has been converted to a Truss. If you analyze this Truss model, you will get ONLY axial forces. No shear forces and bending moments will be present in the elements.



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 effective Stiffness Matrix $[K]$ as well as the vector of the effective External Forces $\{F\}$. The supports are also taken into account to form $[K]$ and $\{F\}$. 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 effective Stiffness Matrix $[K]$ is reversible (Matrix Determinant greater than zero) and no problems will occur during the analysis process.

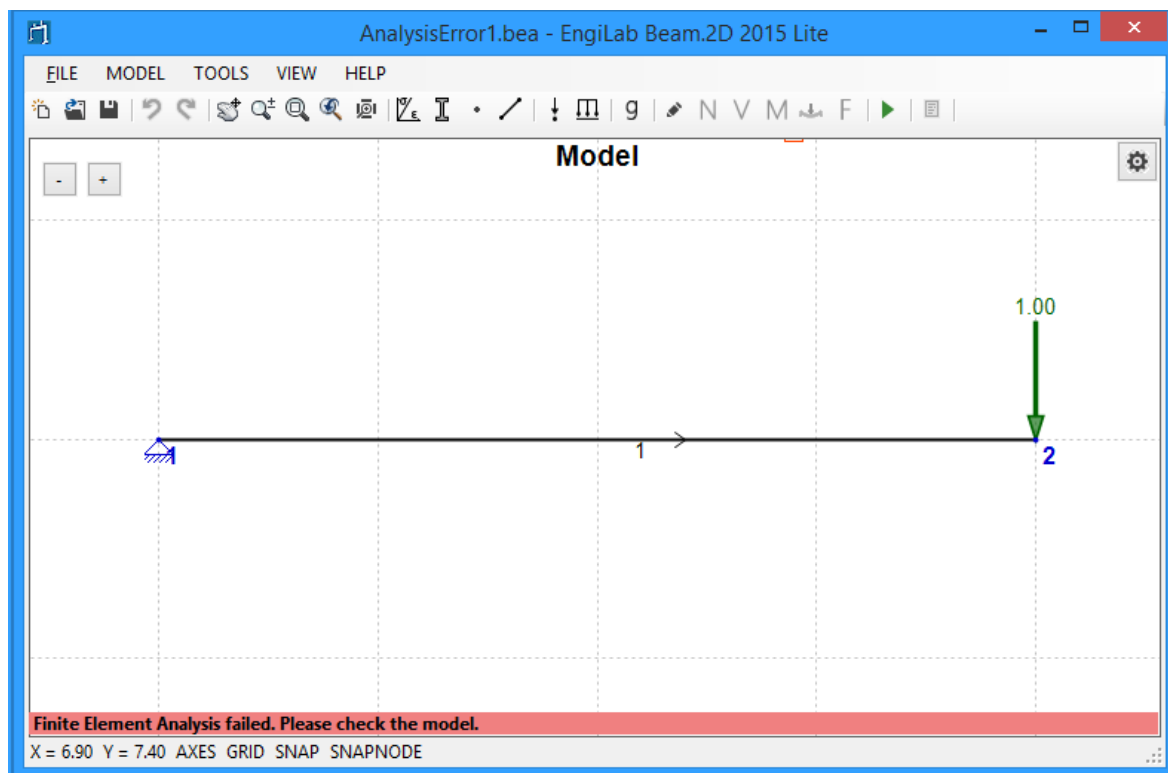
Errors during the analysis process

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

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

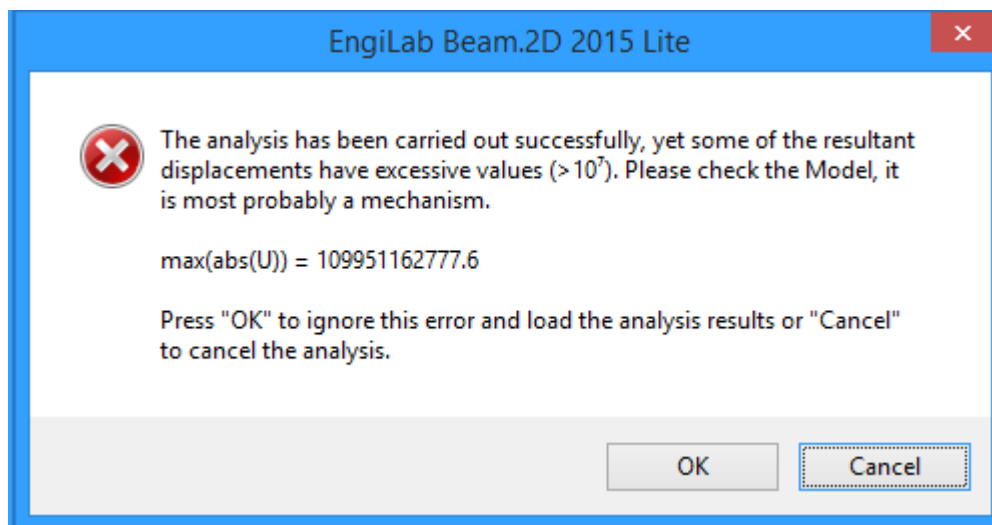
This error occurs due to the model stiffness matrix being non-reversible. There may be a zero-element appearing at the matrix diagonal. In this case the determinant of the model effective stiffness matrix equals to zero, the matrix is non-reversible and the linear system of equations cannot be solved for the displacements. The structure is essentially a mechanism that cannot be analyzed.

Example: An 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 any means. The model cannot be analyzed and the programs gives us the error message, as shown below.

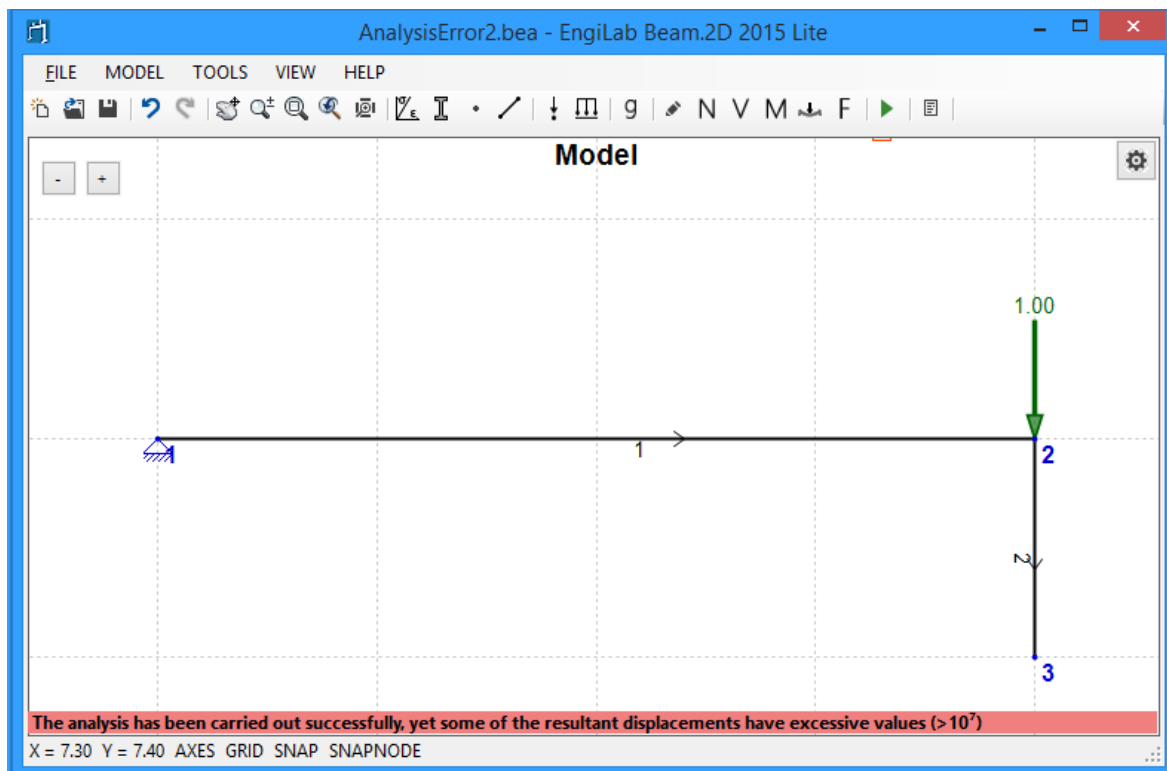


2. 'The analysis has been carried out successfully, yet some of the resultant displacements have excessive values ($>10^7$). Please check the Model, it is most probably a mechanism.'

This case does not have to do with a computational error occurring during the analysis process. The analysis has been carried out successfully by the computer, 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. The model is obviously a mechanism, as was the one above, but this time the computer gives us the second error message, not the first.




Possible reasons for such error messages:

- There are Nodes that are not connected to the model via Elements.
- The Constraints (supports) are inadequate (under-constrained Model), for example the two models 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

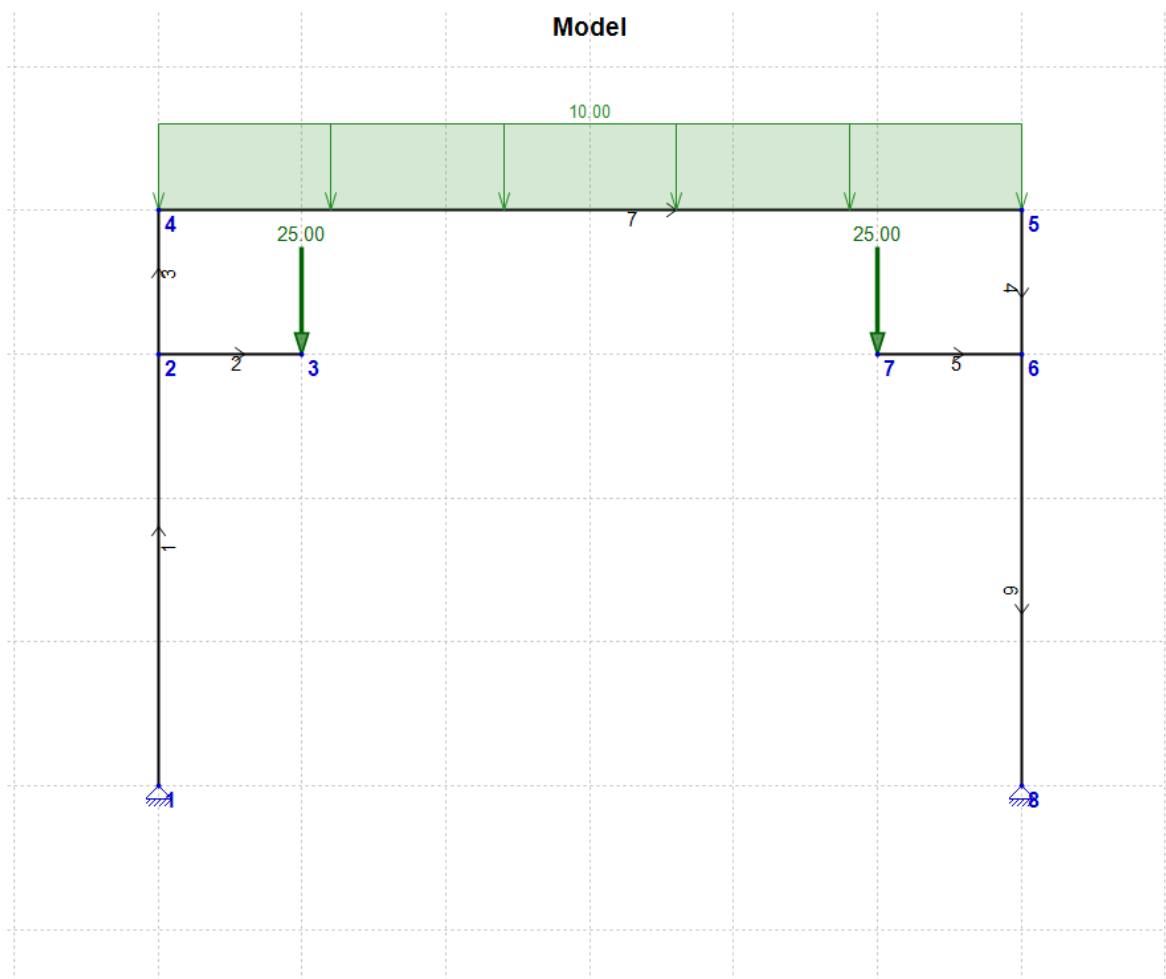


After setting up the model and analyzing it (Clicking the 'Analyze' button ) you can click **N**, **V** or **M** to see the **Axial Force** Diagram, **Shear Force** Diagram or **Bending Moment** Diagram.

- **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.

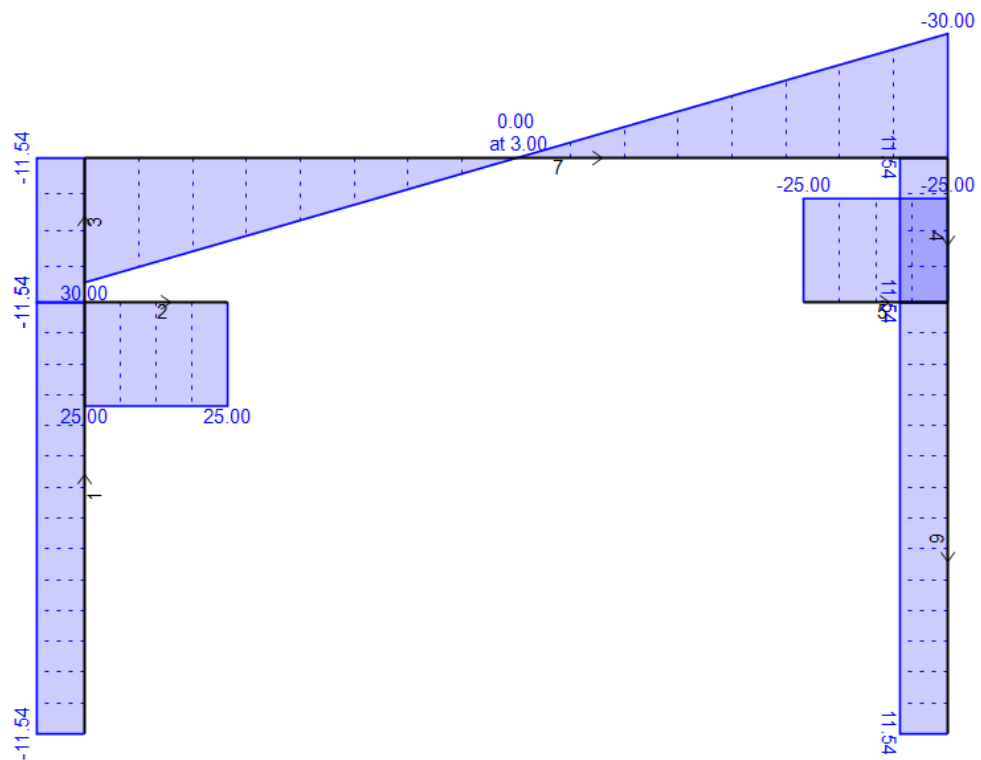
See also: [NVM Diagrams \(Settings\)](#).

Example

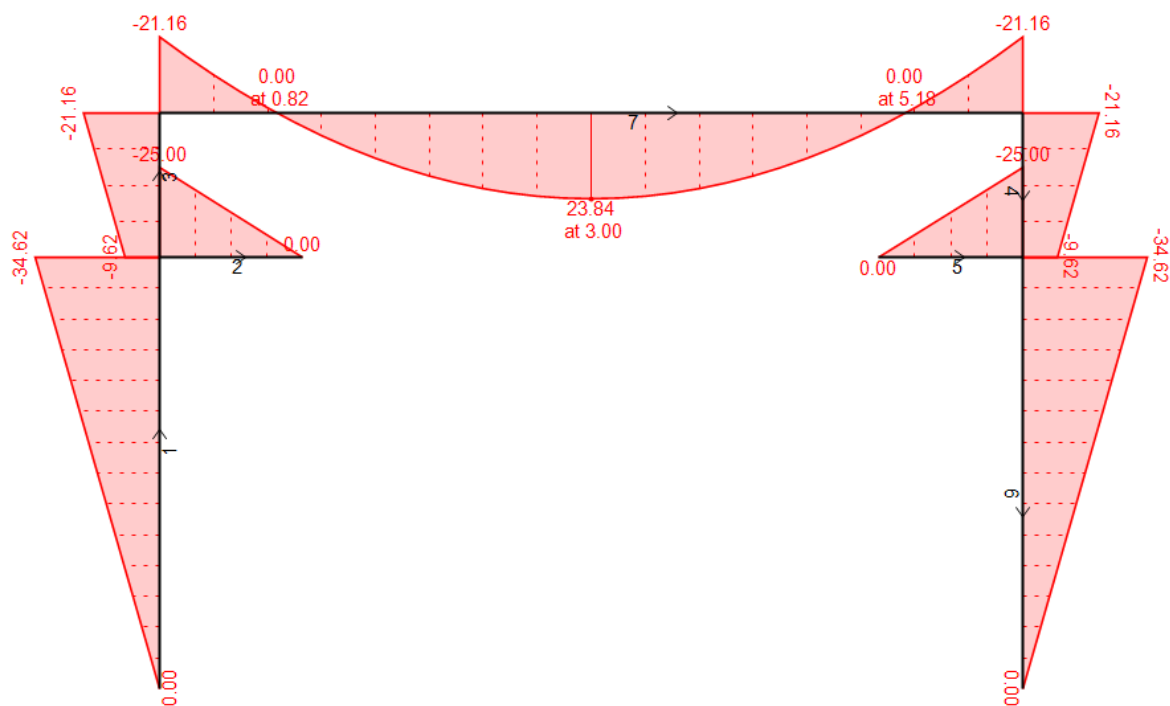
Axial Force Diagram [N]



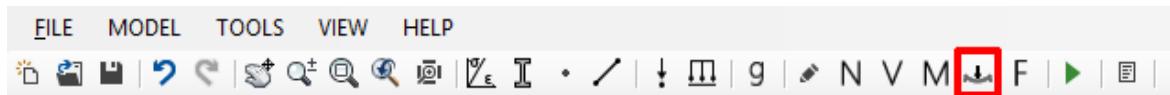
Shear Force Diagram [V]





Bending Moment Diagram [M]



4.3 Deformation



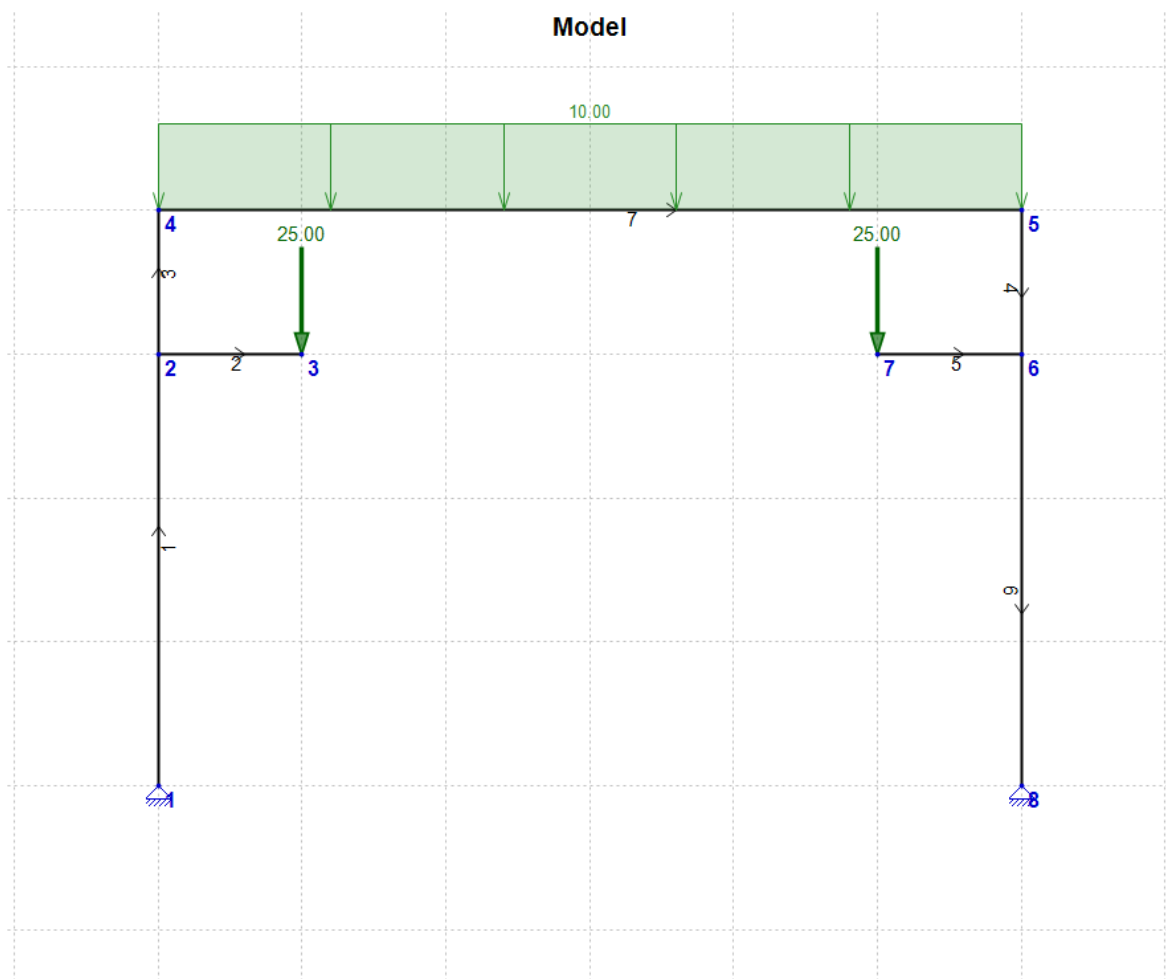
EngiLab Frame.2D uses the Finite Element Method (FEM) to analyze the Model. According to FEM, displacements are computed only for Nodes. Loads within elements (elemental loads) are distributed to the connecting Nodes and the analysis results give the Node Displacements vector (displacements at the positions 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 Frame.2D uses special computational techniques that give 100% accurate results without any approximations, for the displacements (and forces) 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.

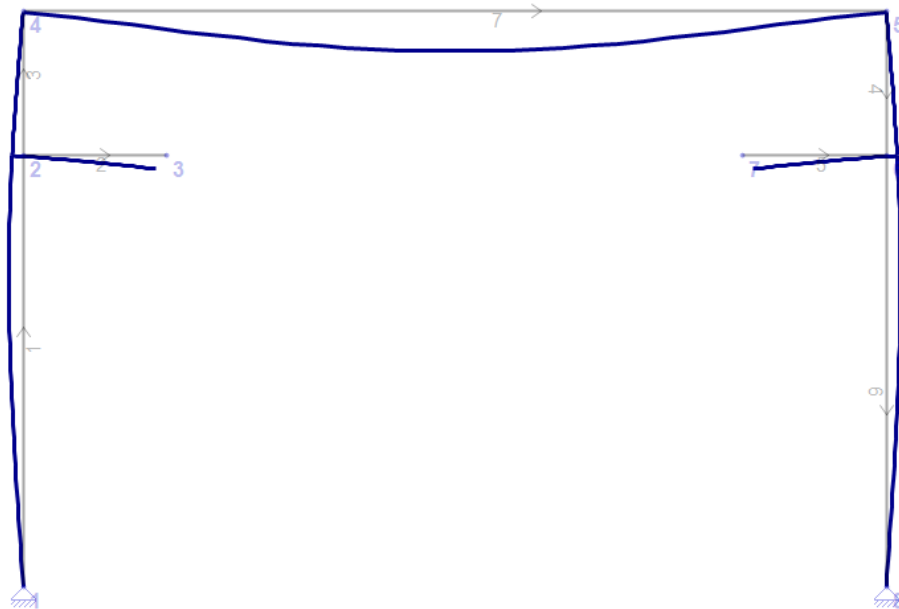
- : Deformation

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)



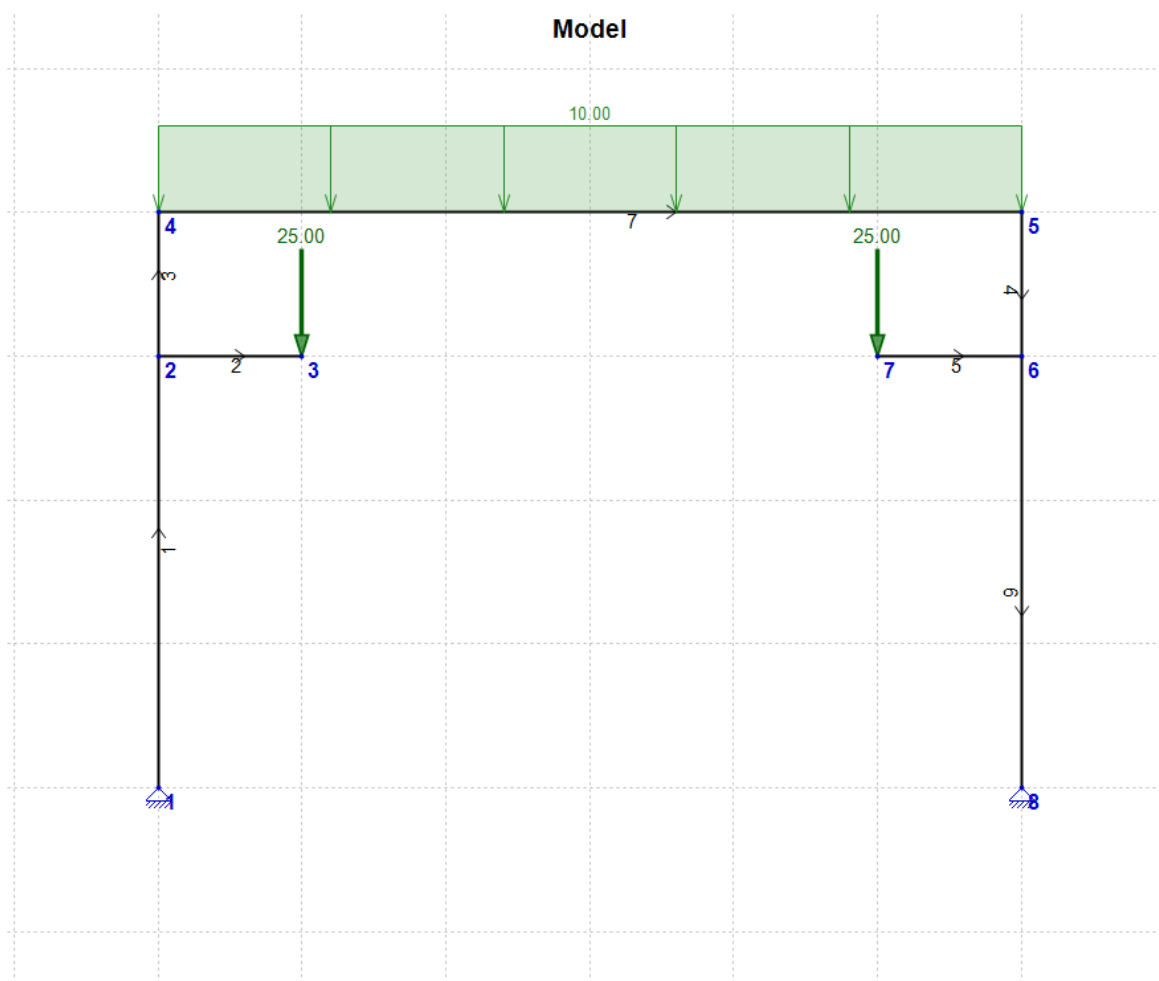
After setting up the model and analyzing it (Clicking the 'Analyze' button ) you can click **F** to see the Free Body Diagram of the Model.

- **F** : Free Body Diagram

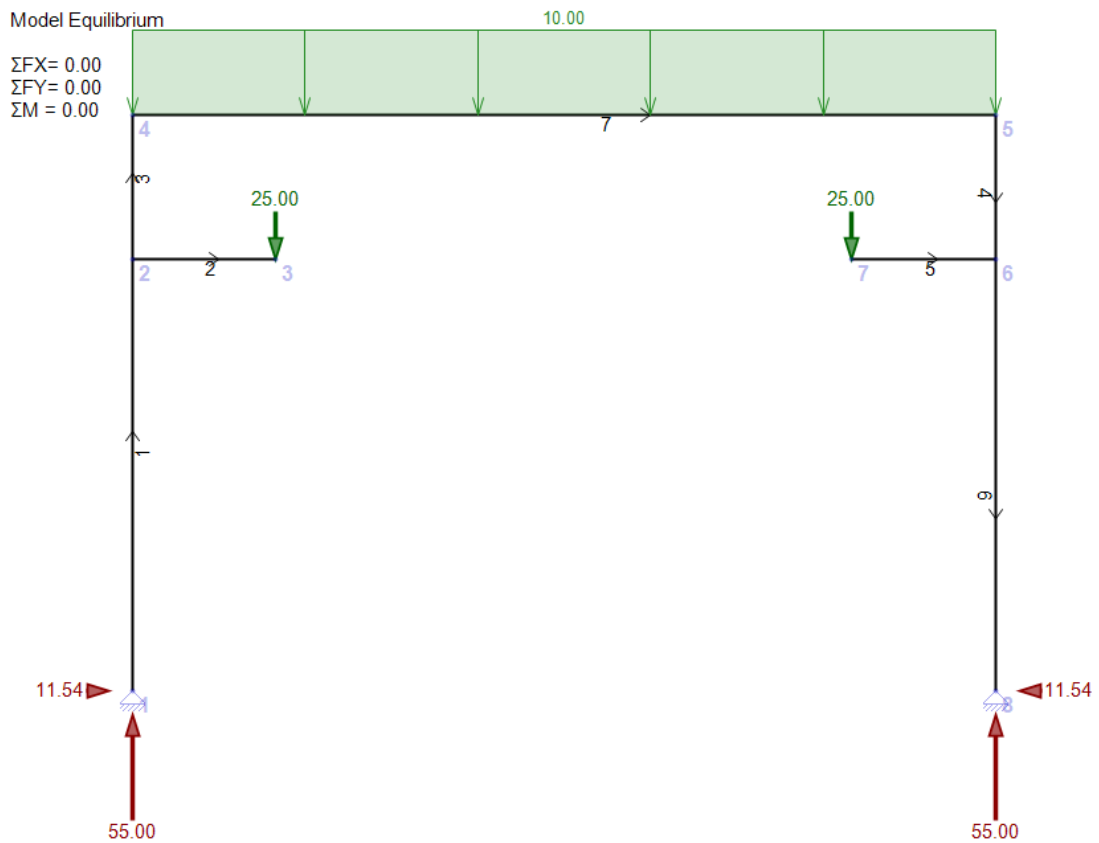
The Free Body Diagram shows the support reactions on screen and also the calculations of the equilibrium of the Model.

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

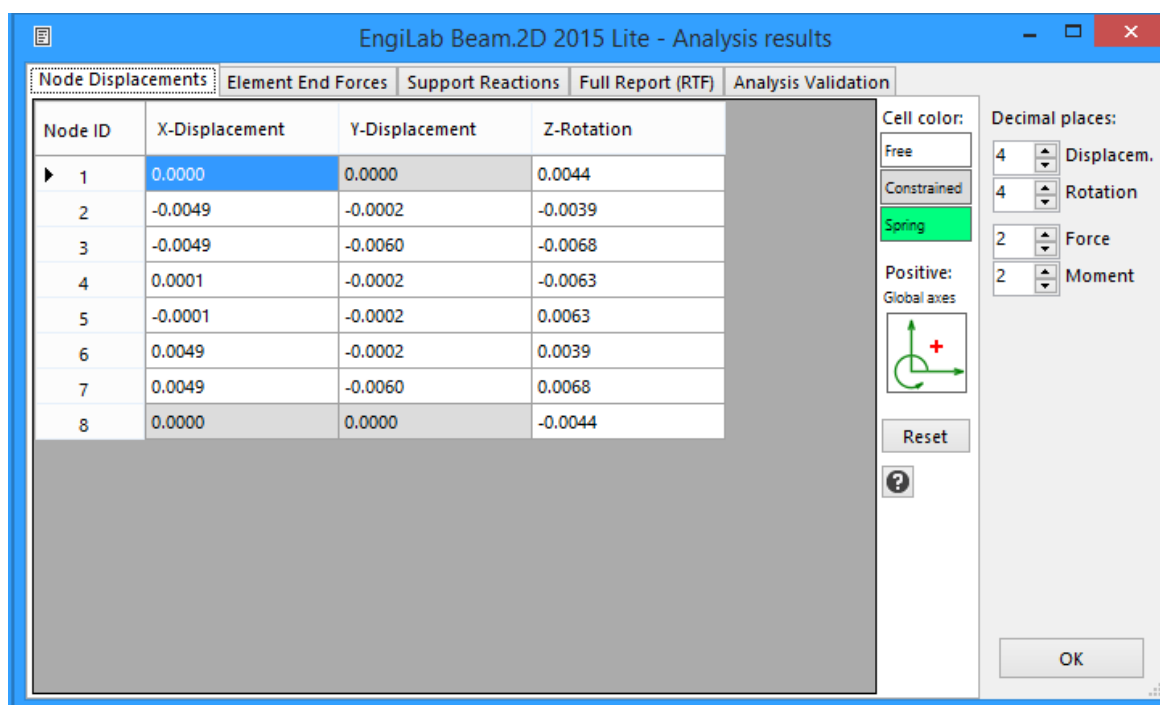


By clicking the Analysis results button, you get a new form where you can see the following:

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

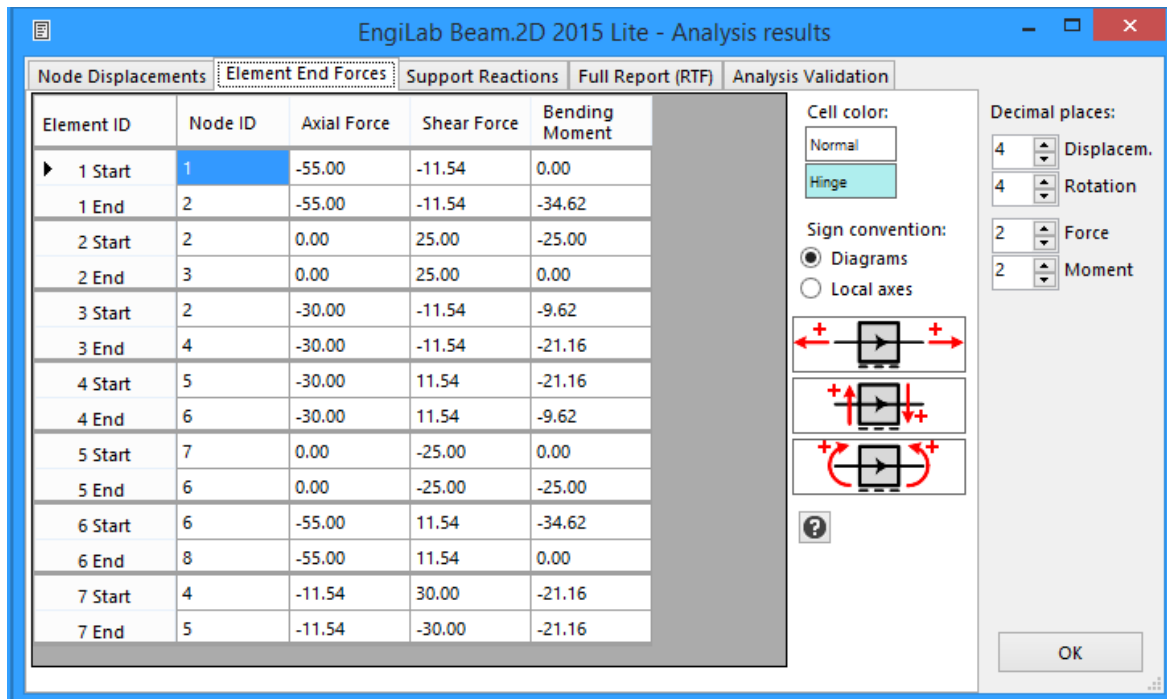
4.5.1 Node Displacements

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



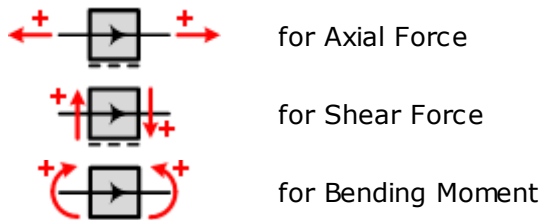
4.5.2 Element End Forces

Shows the End Forces of the Elements.



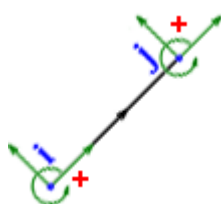
There are two options for the sign convention:

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 \(Settings\)](#).

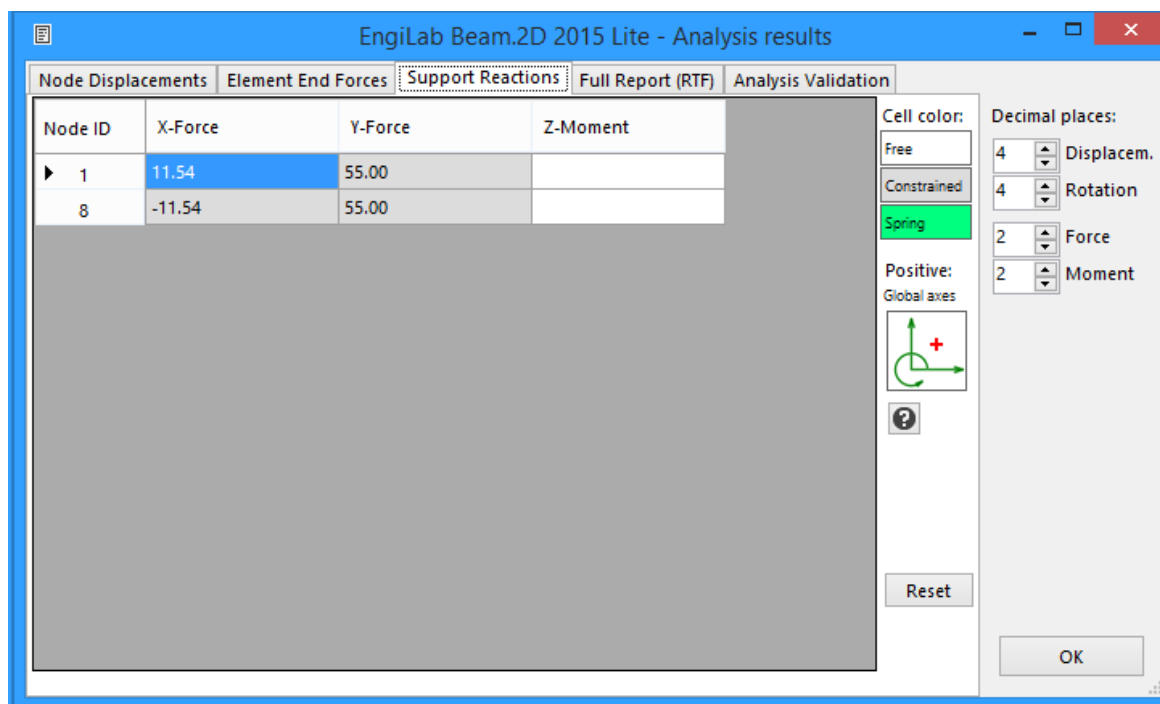
2. Use the sign convention of the Finite Element Analysis (Element local axes), as shown below. This sign convention cannot change.



4.5.3 Support Reactions

This tab 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.



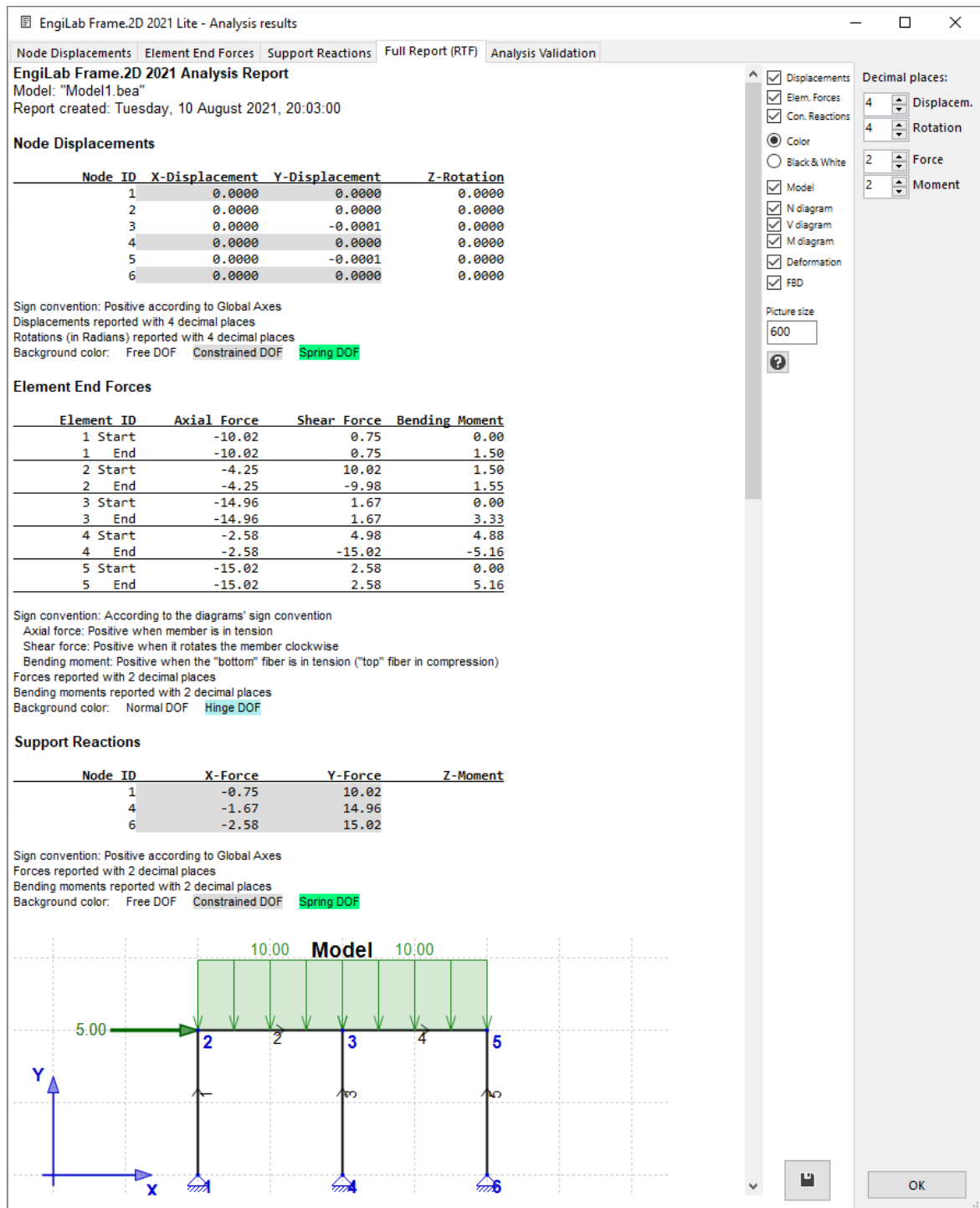
4.5.4 Full Report (RTF)

This tab shows all the Analysis results in Rich Text Format (RTF). You can easily copy the document and paste it anywhere (for example in MS Word), or you can click the "Save" button and save the document in RTF format which can be opened with a text editor, such as MS Word.

All the diagrams are also included in the Full Report by default. The following images are added:

1. Model
2. Axial Force Diagram (N)
3. Shear Force Diagram (V)
4. Bending Moment Diagram (M)
5. Deformation (D)
6. Free Body Diagram (F)

The user can choose which elements to include in the Full Report. In EngiLab Frame.2D 2021 the user can choose the (max) picture size (in Pixels) for these images.



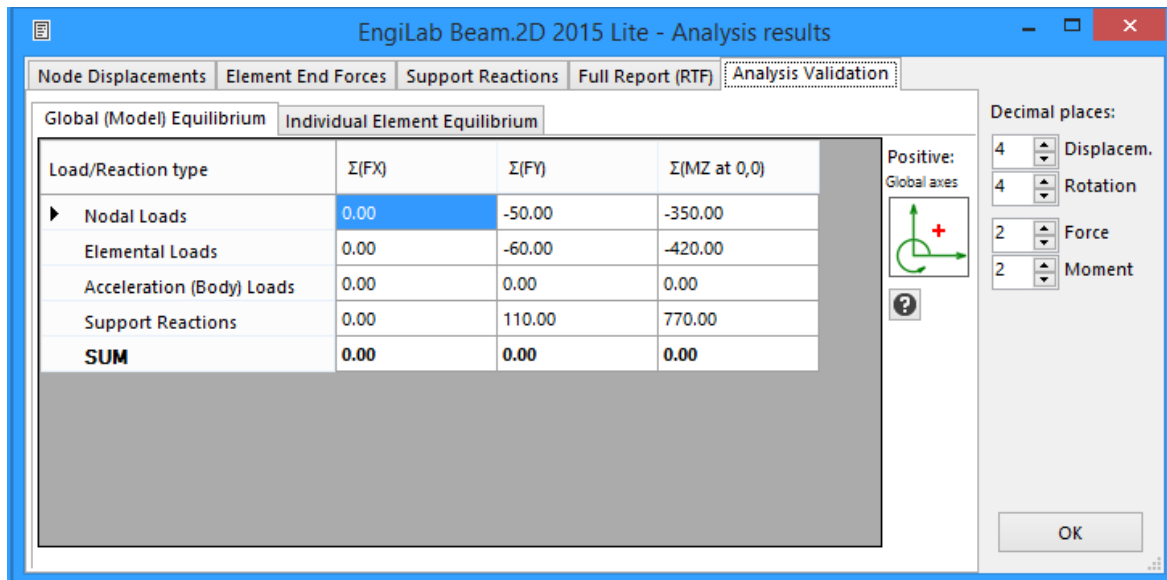
4.5.5 Analysis Validation

This tab 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 (in 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 sub-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.

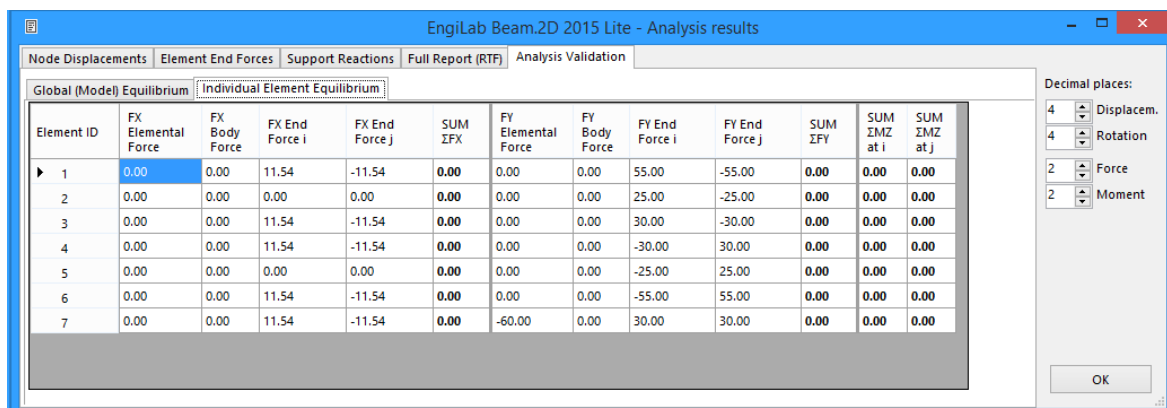
Important: If you see values other than zero in the last row (SUM, in bold), then there is a problem with the Model and the Model needs to be checked. The Model is probably a mechanism and any results should not be taken into account.



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.

Important: If you see values other than zero in these four columns (in bold), then there is a problem with the Model and the Model needs to be checked. The Model is probably a mechanism and any results should not be taken into account.



This page intentionally left blank.

Chapter

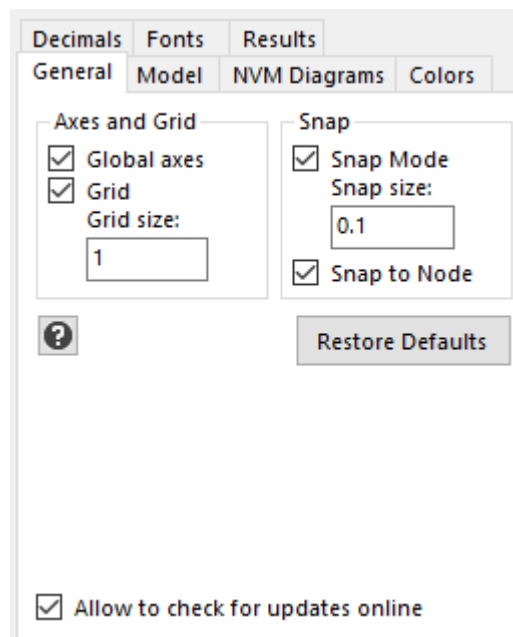


Settings

5 Settings

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

5.1 General (Settings)



Available settings:

- **Global Axes:** Shows/Hides the Global Axes
- **Grid:** Shows/Hides the Grid
- **Grid size:** Sets the size of the Grid shown on screen
- **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)

Decimals Fonts Results
General Model NVM Diagrams Colors

Nodes group

- ☒ Nodes
- ☒ Numbering
- ☒ Constraints
- ☐ Coordinates
- ☒ Springs
- ☐ Spring values

Elements group

- ☒ Elements
- ☒ Numbering
- ☒ Hinges
- ☒ Orientation
- ☐ Material
- ☐ Section
- ☐ Length

Loads group

- ☒ Nodal Loads
- ☒ Nodal Loads values
- ☒ Elemental Loads
- ☒ Elemental Loads values

Nodal Loads representation

- ☒ Show the Resultant force
- ☐ Show the X, Y Components

Acceleration

- ☒ Acceleration vector
- ☒ Acceleration values

? Restore Defaults

Each setting Shows/Hides the corresponding object on screen.

5.3 NVM Diagrams (Settings)

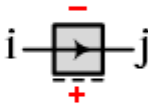
Decimals Fonts Results

General Model **NVM Diagrams** Colors

Show in NVM Diagrams


- ☒ Nodal values
- ☒ Min, Max values and positions
- ☒ Zero values and positions
- ☐ Intermediate values
- ☒ Intermediate lines
- ☒ Diagram Fill

Positive direction

☒ "Bottom" fiber 


☐ "Top" fiber

Axial Force positive

☒ Tension 


☐ Compression

Shear Force positive


☒ Clockwise 

☐ Counter-Clockwise

Bending Moment positive

☒ "Bot." fib. in tension 

☐ "Top" fiber in tension

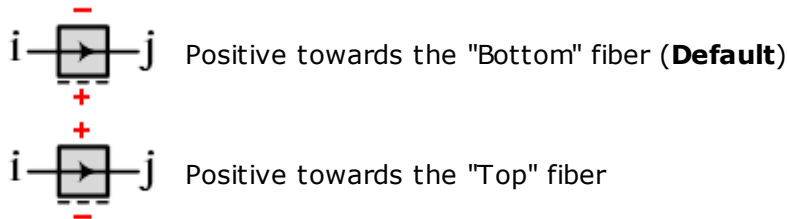
 Restore Defaults

Available settings:

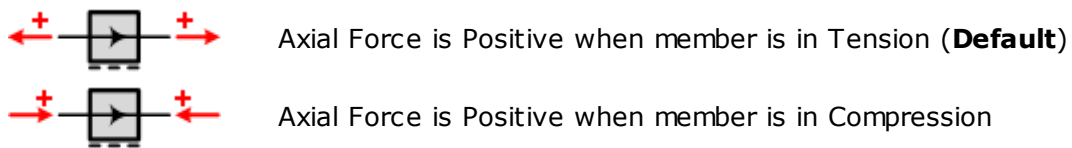
- **Nodal values:** Shows/Hides the Nodal values of the diagrams (Axial Force, Shear Force or Bending Moment).
- **Min, Max values and positions:** Shows/Hides the (local) Minimum and Maximum values of the diagrams and the corresponding positions along an Element.
- **Zero values and positions:** Shows/Hides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

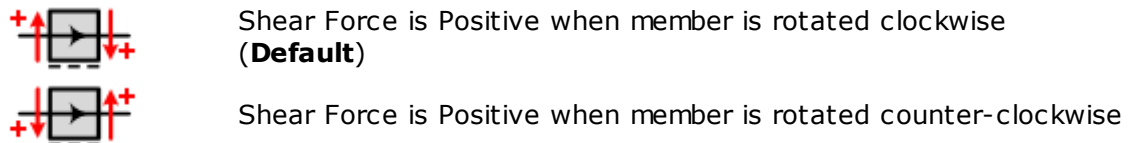
Determines the positive direction of the diagrams. Applies to all three diagrams (Axial Force, Shear Force or Bending Moment). There are two options:

**Axial Force positive**

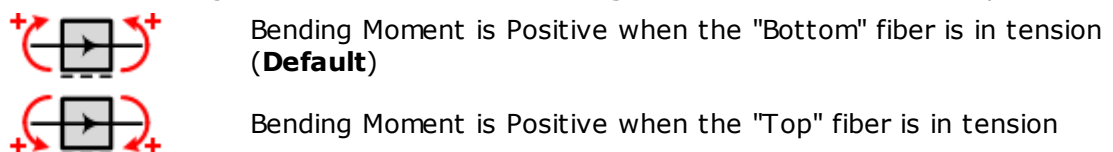
Determines the sign convention for the **Axial Force**. There are two options:

**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)

The screenshot shows the 'Colors' settings dialog box. It has a tabbed interface with the following tabs: Decimals, Fonts, Results, General, Model, NVM Diagrams, and Colors. The 'Colors' tab is selected. The settings are organized into several sections:

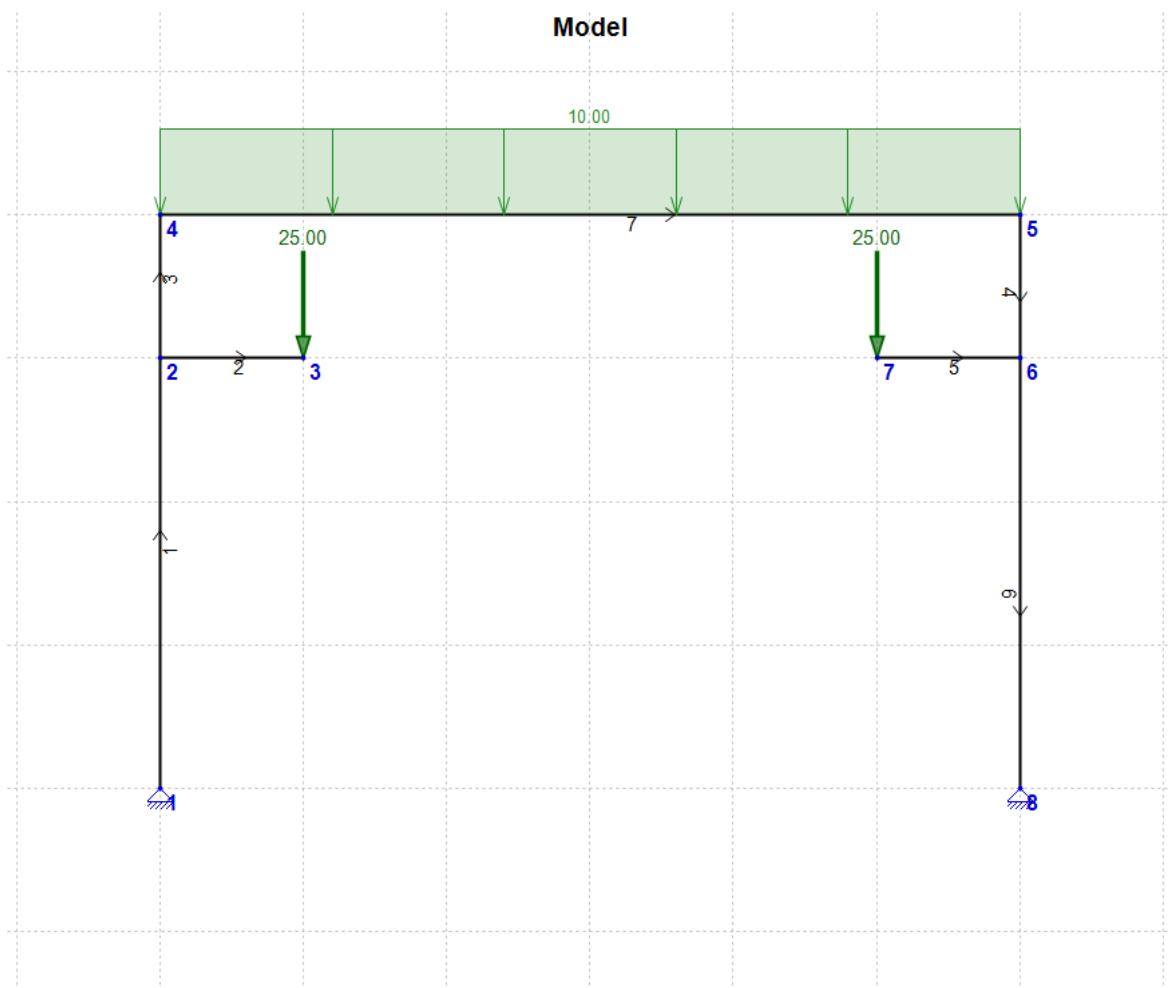
- General:**
 - Background (White)
 - Axes (Blue)
 - Grid (Gray)
- Model:**
 - Nodes (Blue)
 - Elements (Black)
 - Nodal Loads (Green)
 - Elem. Loads (Green)
 - Acceleration (Red)
- Graphical User Interface:**
 - Active Node (Red)
 - Active Element (Red)
 - Snap Rectangle (Orange)
 - Zoom Window (Gray)
 - Cursor values in NVMDF Drawings (Black)
- NVM Diagrams:**
 - Axial Force Transparency % (80)
 - Shear Force Transparency % (80)
 - Bend. Mom. Transparency % (80)
- D (Deformation) Drawing:**
 - Deformed Model (Blue)
- F (Free Body Diagram) Drawing:**
 - Support Reactions (Red)
 - Element End Forces (Red)
 - Equilibrium Calculations (Black)

At the bottom of the dialog, there is a help icon (question mark) and two buttons: 'Light Theme' and 'Dark Theme'.

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)

These settings influence only the appearance of the Model inputs and the Analysis outputs. For example, if the Coordinate is set to 2 decimals, then you will see $X=2.12$ on screen, even if the real X coordinate of a Node is 2.116. Similarly in the results, a displacement may be 0.003123442 but if 4 decimals are selected, then you will see it as 0.0031.

Note: All available digits are taken into consideration for the analysis, at all times. There are no approximations other than the rounding errors made by the computer at all times. These settings are only for the presentation of the inputs and outputs and do not affect the analysis procedure or the analysis results.

5.6 Fonts (Settings)

Category	Setting	Value
General	Picture Title	14
	Axes	13
Model	Nodes	12
	Elements	11
	Nodal Loads	11
	Elemental Loads	11
	Acceleration	11
NVM Diagrams, D and F Drawings	NVM Nodal, Min, Max, Zero 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

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

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.



for Axial Force



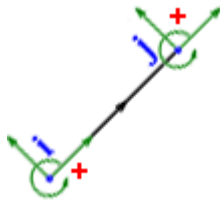
for Shear Force



for Bending Moment

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 highlight with colors / No colors (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.
- **Elements to include:** You can choose what exactly elements to include in the Full Report
- **Picture size (pixels):** You can select the (max) size of the images, in pixels.

This page intentionally left blank.

Chapter

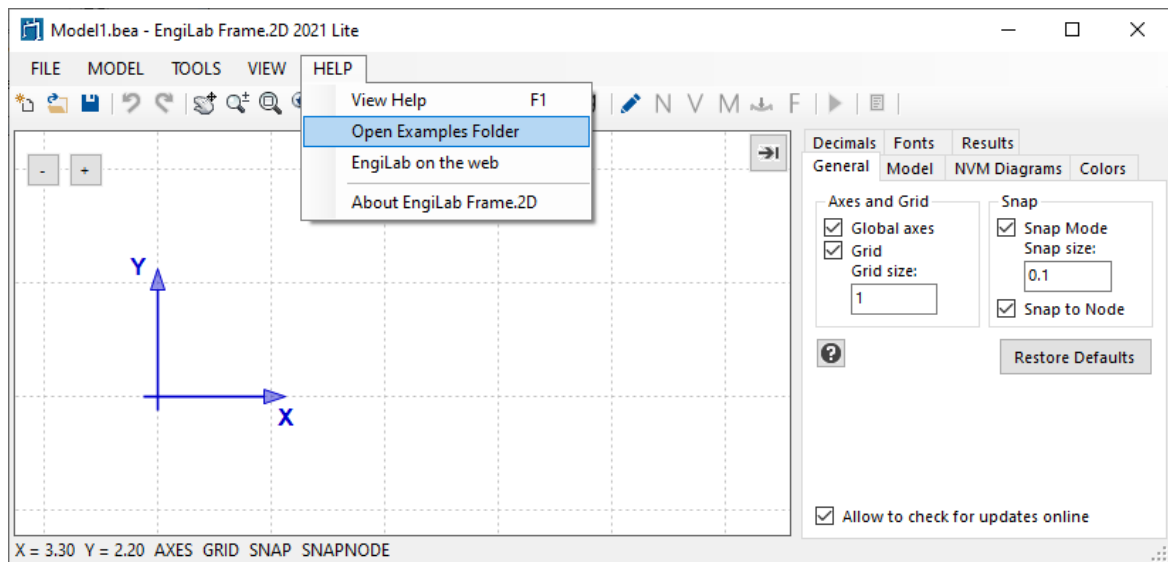


Useful information

6 Useful information

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


6.1 Ready to-analyze Examples



In the folder **\Examples** (which is located where the program was installed), you can find ready-to-analyze EngiLab Frame.2D input files (.fr2d) that you can open and analyze within EngiLab Frame.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 Frame.2D 2021 Lite\Examples** for the Lite Edition and **C:\Program Files (x86)\EngiLab\EngiLab Frame.2D 2021 Pro\Examples** for the Pro Edition).
- Select a .fr2d 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

6.2 Tips on Modeling hinges

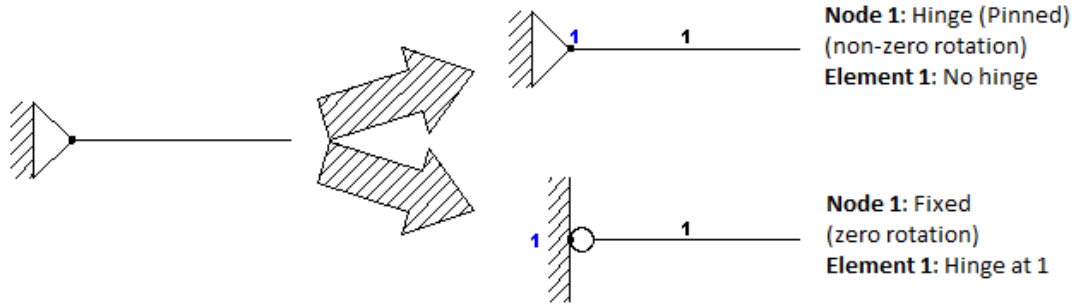
The hinges (rotational releases) of a model is an issue which requires our attention. There are two kinds of hinges: **External hinges (Pinned constraints)** and **Internal hinges**.

External hinges (Pinned constraints)

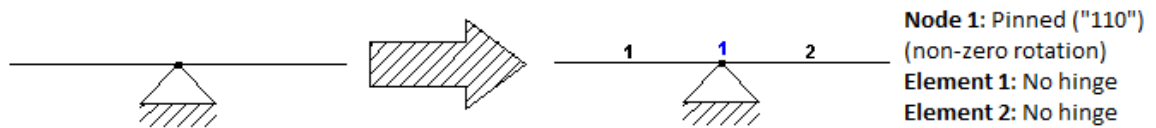
1. An external **hinge to which only ONE element is connected** can be given in a model using two possible ways (the result of the analysis should be the same):

- **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.



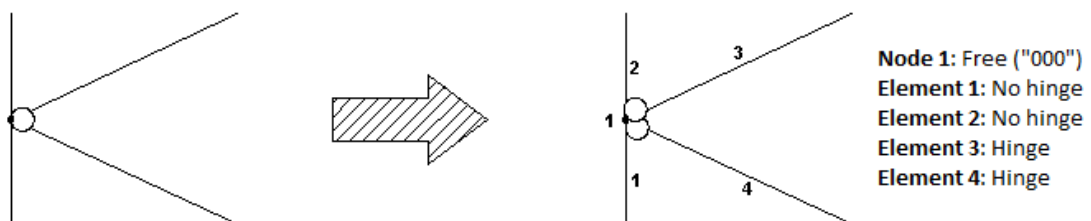
2. An **external hinge to which more than one elements are connected** must be given in the Model as follows:



Internal hinges

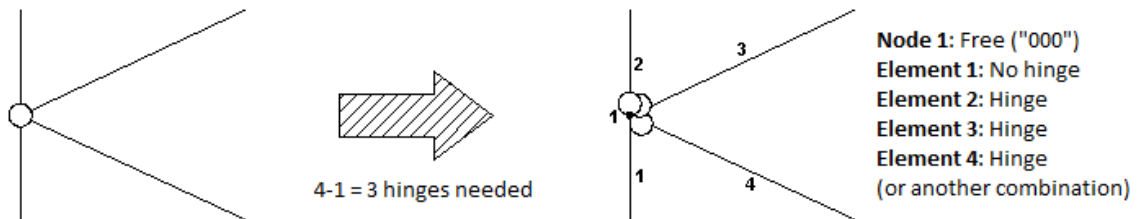
An **internal hinge** must be given always as a hinge of one or more elements as follows:

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.

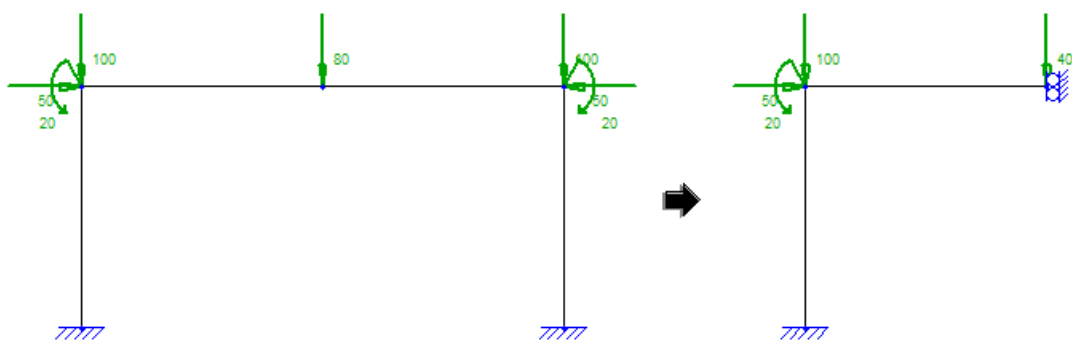


Note: In the case of a 'Full' internal hinge, each connecting element has its own rotation at the hinge end. The program only calculates the rotation of the element with no hinge. In the example below, the rotation of element 1 at end 1 will be calculated, while the rotations of the elements 2, 3 and 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.

6.3 Tips on Modeling symmetric structures

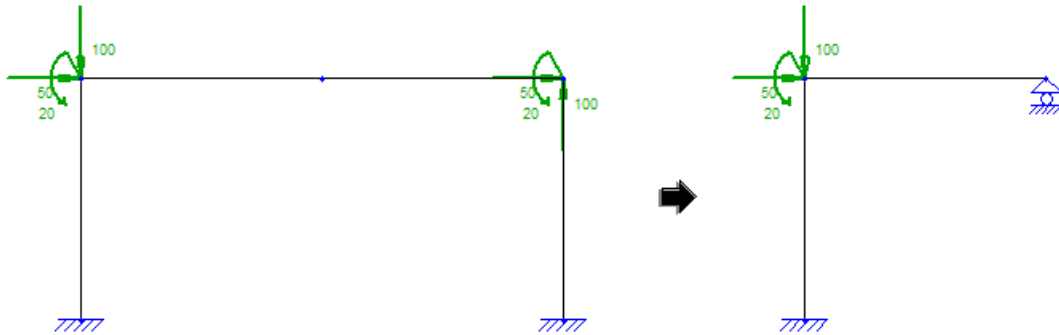
If a structure is symmetric (Symmetric structure with symmetric / anti-symmetric loads), then the size of the finite element model can be reduced, which, in turn, reduces the time and cost of the analysis. For each plane of symmetry in the model, the model size can be reduced by a factor of approximately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

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



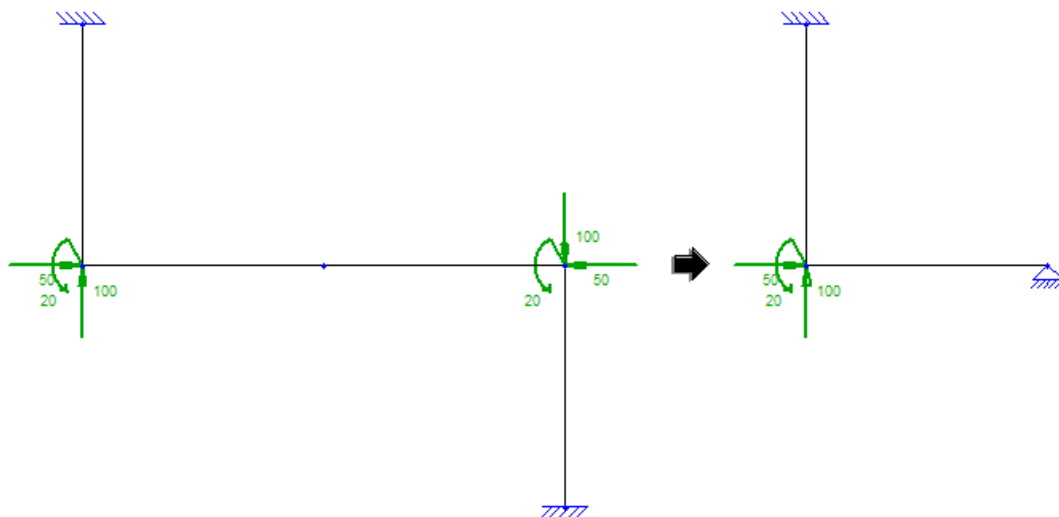
For the above example, half the structure can be analyzed using a "101" constraint for the node on the symmetric axis. Note that if there is a load applied on that node, **half the load has to be applied on the constrained node of the second model.**

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

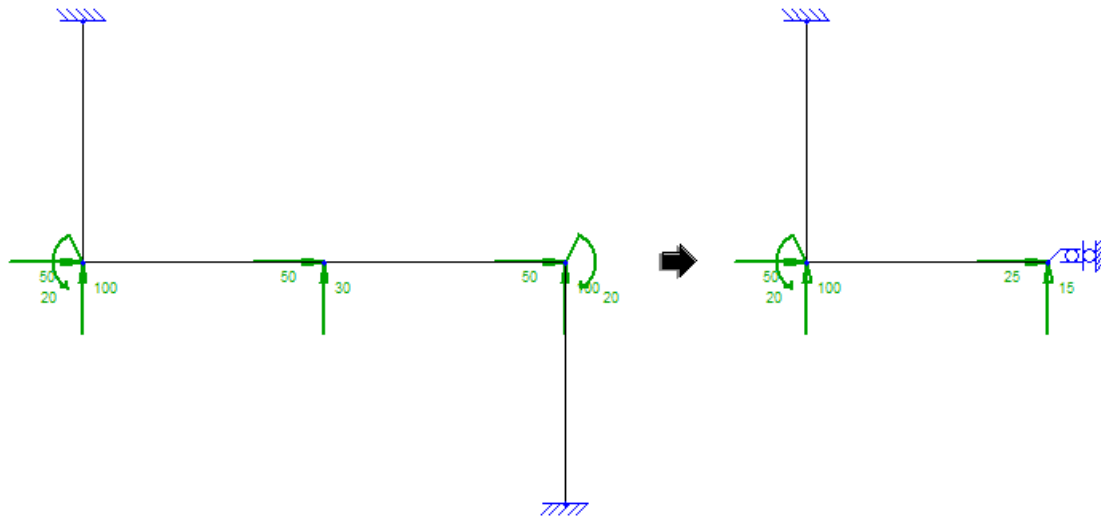


For the above example, half the structure can be analyzed using an "010" (x-Roller) constraint for the node on the symmetric axis.

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.**

This page intentionally left blank.

Chapter



Example Problems

7 Example Problems

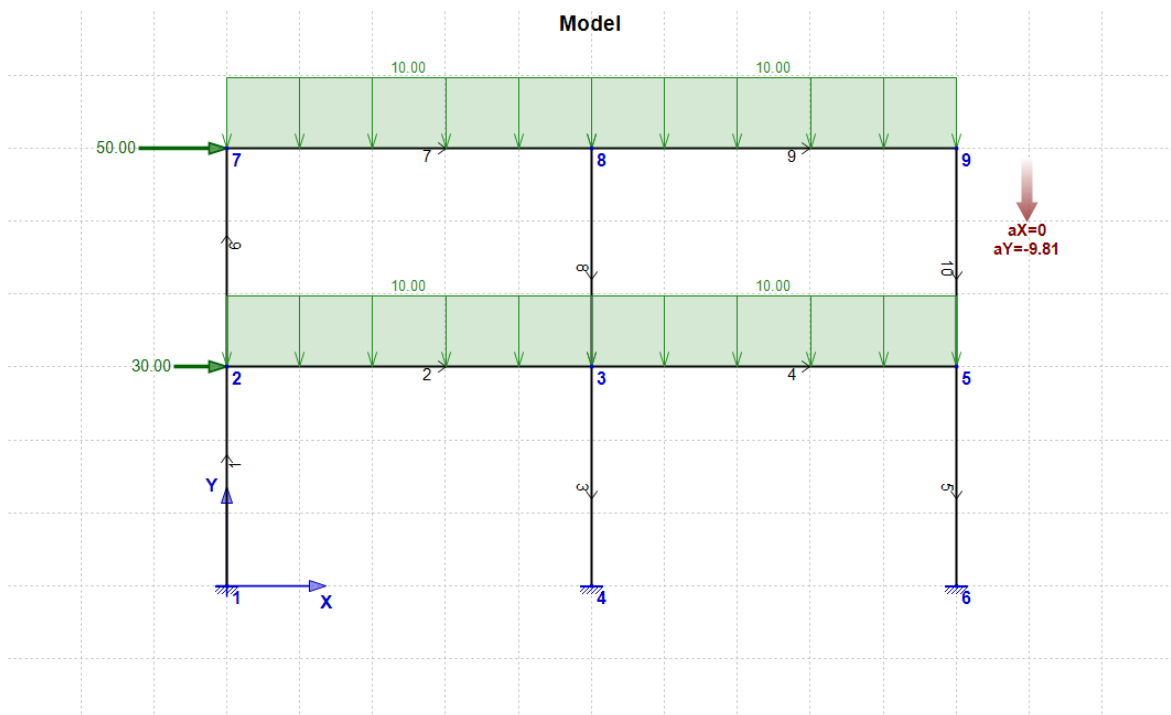
- [Example Problem 1](#)

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/m³**

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, i.e. 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)

The geometry of the structure is as shown in the figure above, where **the Grid is 1 m x 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. Even if the orientation of the Elements is different, the analysis results will not change.

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/s²** (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](#).

EngiLab strongly recommends the use of "EngiLab Units" for all unit conversions. EngiLab Units is a free unit conversion program that is available for download at www.engilab.com. EngiLab Units 2021 (v3.1) supports 584 units in 20 unit categories, including distance, acceleration, pressure (stress) and others. **All units needed in EngiLab Frame.2D are supported by EngiLab 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)². In our example: **m/s²**
- The **Mass unit** is given by: $F \cdot T^2/L$ (1 force unit) / (1 acceleration unit). In our example: $kN/(m/s^2) = Mg = t$ (metric ton) (**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)³. In our example: **t/m³** (**IMPORTANT**: t/m³ has to be used instead of kg/m³!)
- The **Stress unit** is given by: F/L^2 (1 force unit) / (1 length unit)². In our example: **kN/m²**

Note: The mass unit (in our example 1 t) is the mass that accelerates by the acceleration unit rate (in our example 1 m/s²) 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/m²**
- **d** = 2500 kg/m³ = **2.5 t/m³**

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/s²**

7.1.3 Step 2. Define Materials



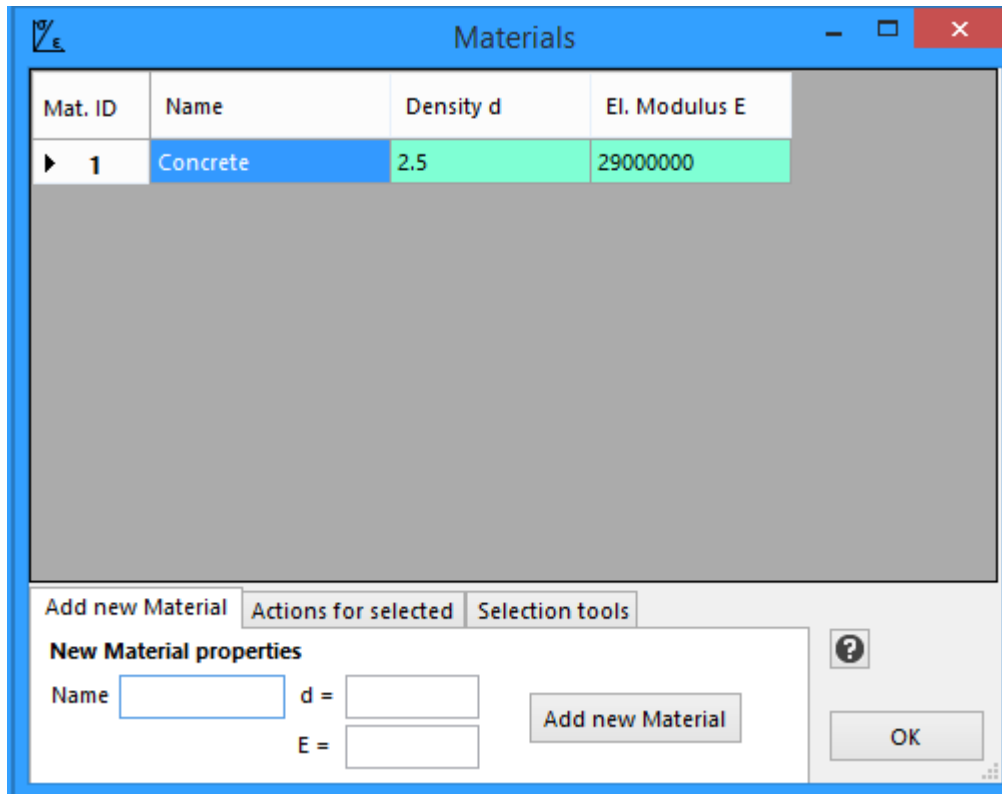
After the preparation of the input data, it is time to start working with the program.

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

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

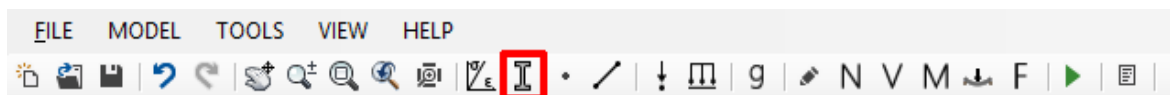
The screenshot shows the 'Materials' dialog box. It features a table with the following columns: 'Mat. ID', 'Name', 'Density d', and 'El. Modulus E'. Below the table, there are three tabs: 'Add new Material', 'Actions for selected', and 'Selection tools'. The 'Add new Material' tab is active, showing the 'New Material properties' section. This section includes input fields for 'Name' (filled with 'Concrete'), 'd' (filled with '2.5'), and 'E' (filled with '29000000'). There are also buttons for 'Add new Material' and 'OK'.

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.

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

Add new Section | **Actions for selected** | **Selection tools**

New Section properties

Name: A = I =

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 Frame.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.

Section Properties Calculator

Section Type: **Square bar**

a =

Decimals:

Prop.	Details	Value
yC	Centroid y	0.25
zC	Centroid z	0.25
A	Area	0.25
Iy	Mom. of Inertia y	0.00520833
Iz	Mom. of Inertia z	0.00520833
yPNA	Plastic Neutral axis...	0.25
zPNA	Plastic Neutral axis...	0.25
Wply	Plastic Modulus y	0.03125
Wplz	Plastic Modulus z	0.03125

Apply y-y Apply z-z Cancel

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_y are transferred to the main Section form, as shown below.

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

Add new Section | Actions for selected | Selection tools

New Section properties

Name: Columns A = 0.25 I = 0.00520833

Calculator Add new Section OK

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

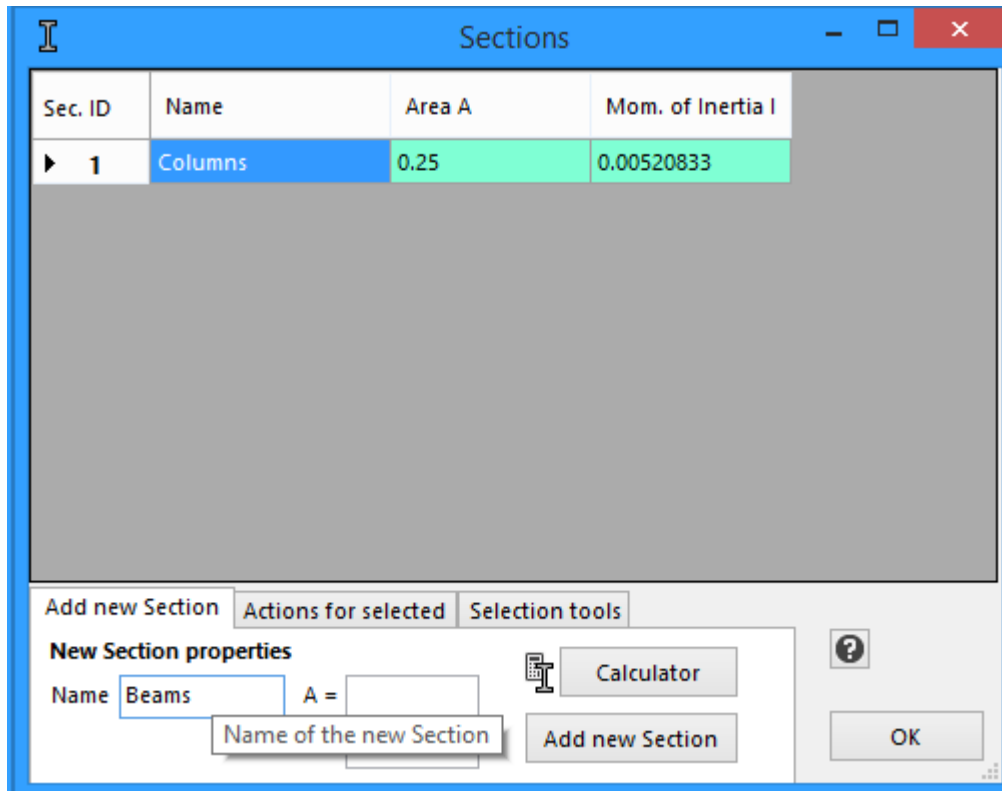
Sec. ID	Name	Area A	Mom. of Inertia I
1	Columns	0.25	0.00520833

New Section properties

Name A = I =

Calculator Add new Section OK

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.

Section Properties Calculator

Section Type: **Rectangular bar**

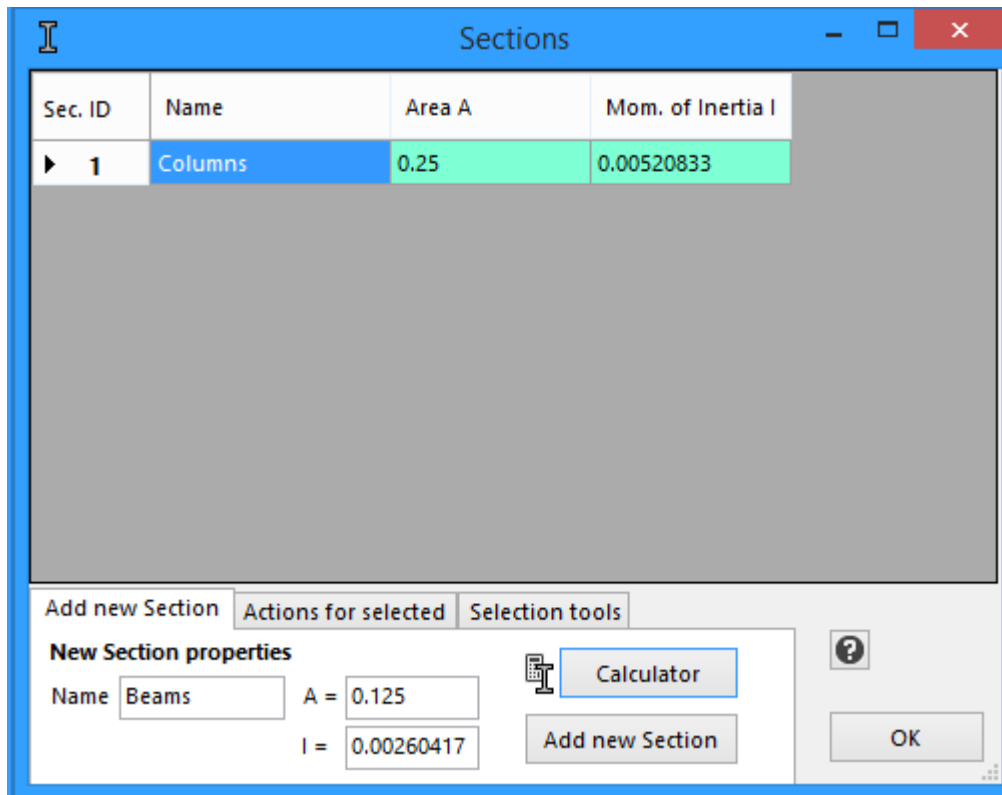
a = 0.25
b = 0.50

Decimals 8

Prop.	Details	Value
yC	Centroid y	0.125
zC	Centroid z	0.25
A	Area	0.125
Iy	Mom. of Inertia y	0.00260417
Iz	Mom. of Inertia z	0.00065104
yPNA	Plastic Neutral axis...	0.125
zPNA	Plastic Neutral axis...	0.25
Wply	Plastic Modulus y	0.015625
Wplz	Plastic Modulus z	0.0078125

Apply y-y Apply z-z Cancel

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.



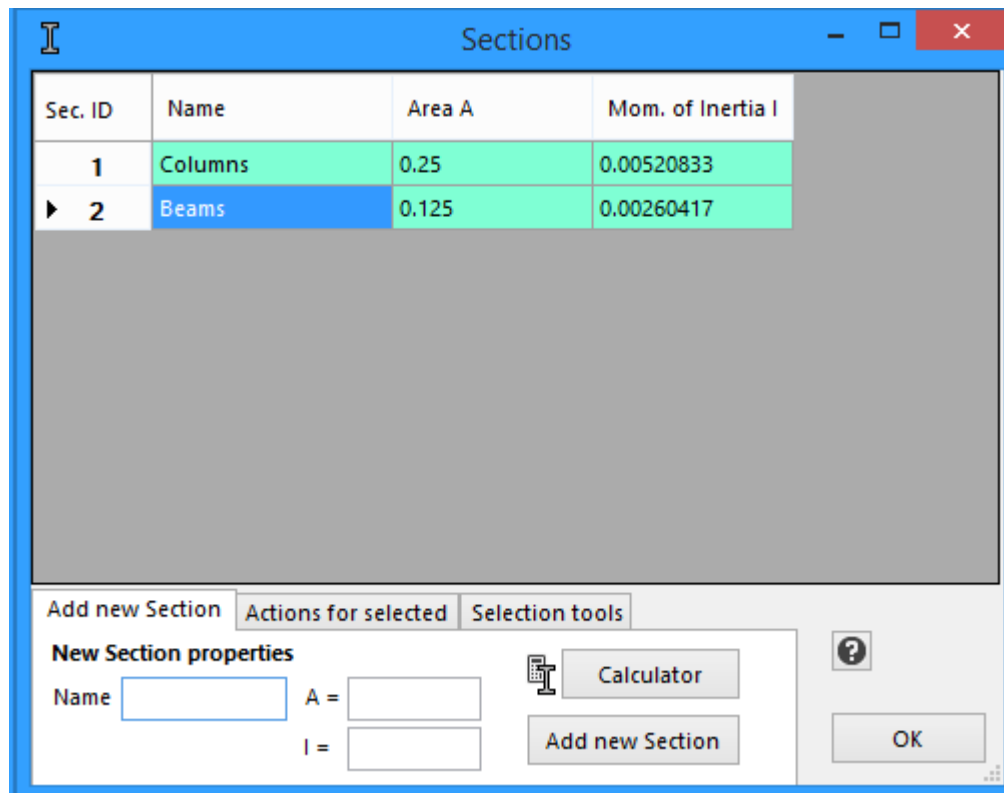
Sec. ID	Name	Area A	Mom. of Inertia I
▶ 1	Columns	0.25	0.00520833

Add new Section Actions for selected Selection tools

New Section properties

Name: A = I =

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



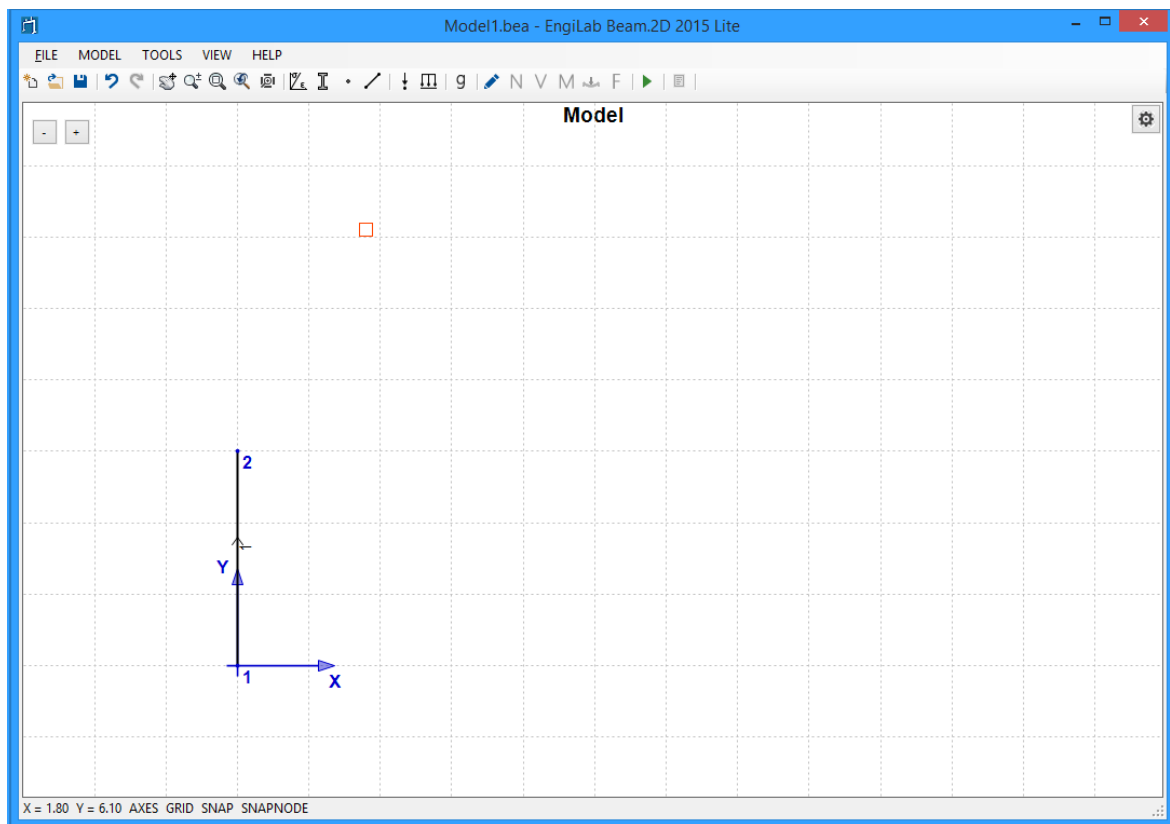
Then **Click OK** to exit the Sections form.

7.1.5 Step 4. Draw the Model on screen

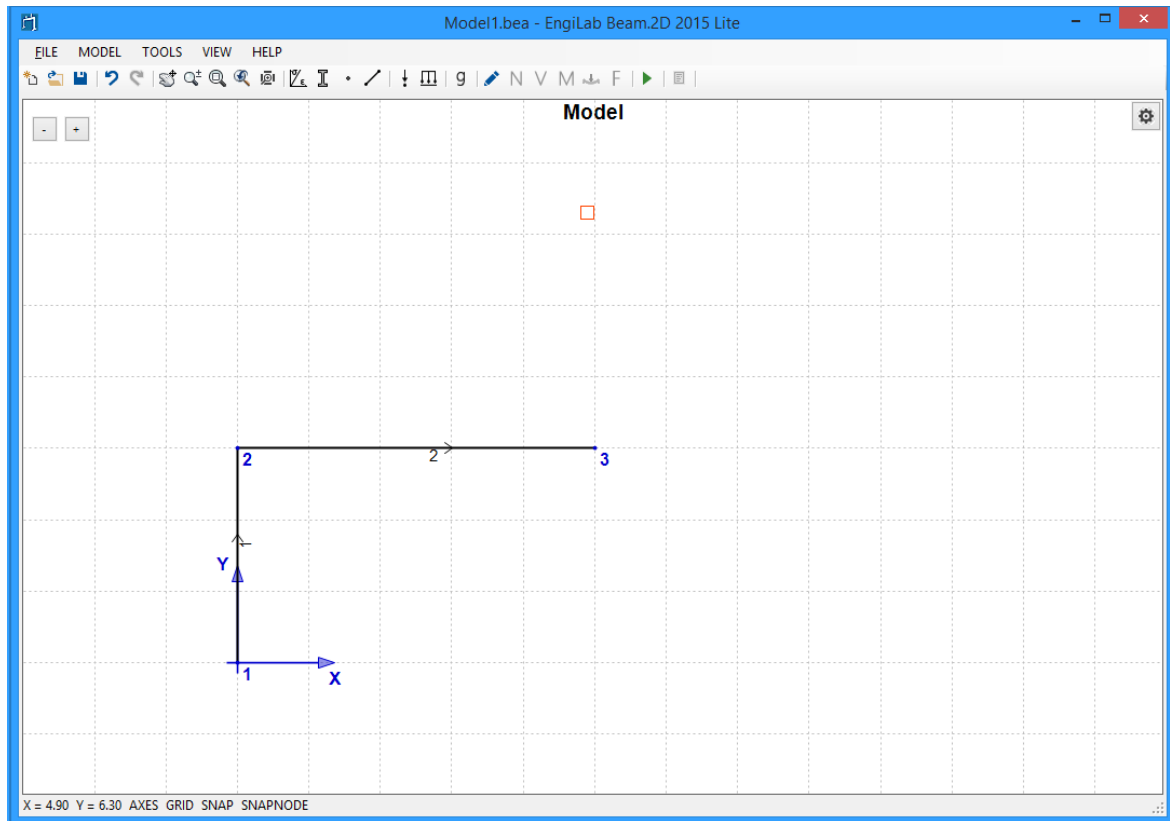
After you have defined the Material and the Sections, you can **start drawing your Model on screen**, as follows:

- 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.

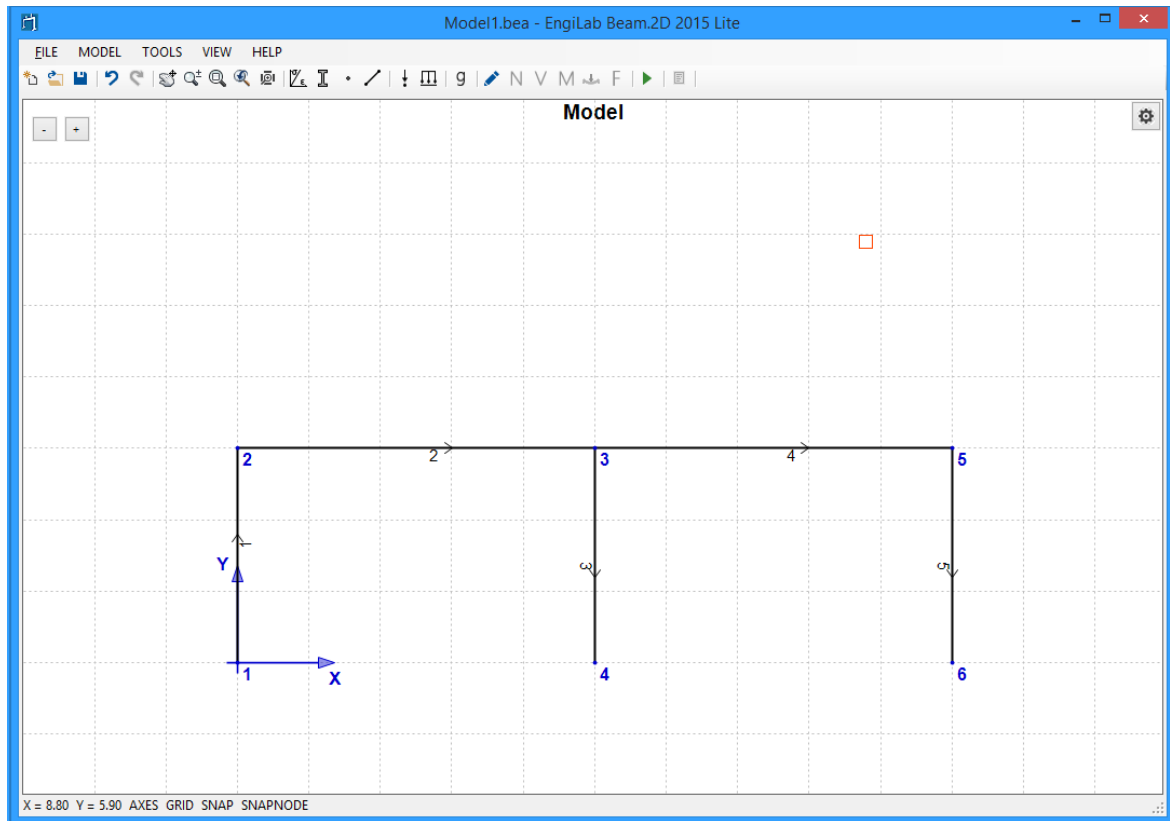
You can start at any point. Let's start from point 0,0 (Origin of axes). Draw the first column as shown below. Release the left button after you have done a distance of 3 in the Y Direction.



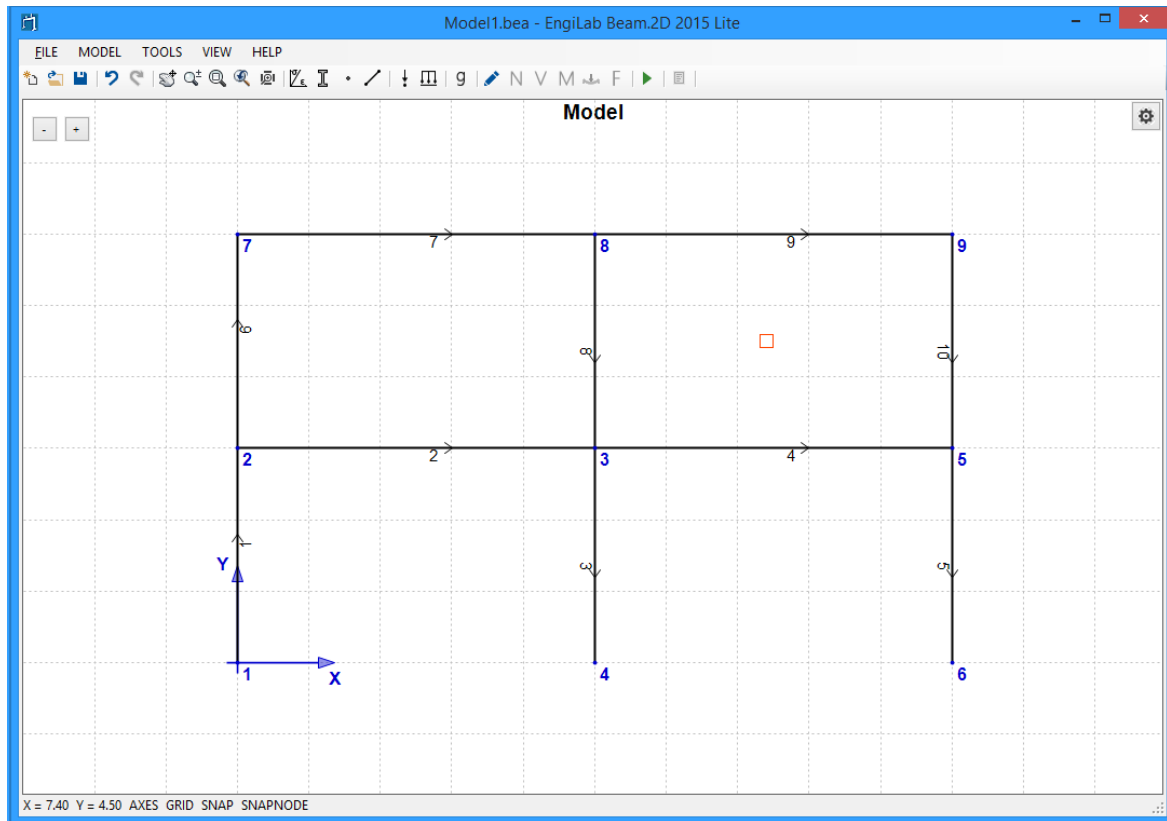
Continue with the Beam. Hold down the left button starting at Node 2 and release it at distance 5 in the X Direction. The picture should be as follows.



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](#).

Nodes

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

Add new Node
Actions for selected
Selection tools

New Node coordinates

X=
Y=

Add new Node

?

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.

Nodes


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=

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
☐ i Hinges

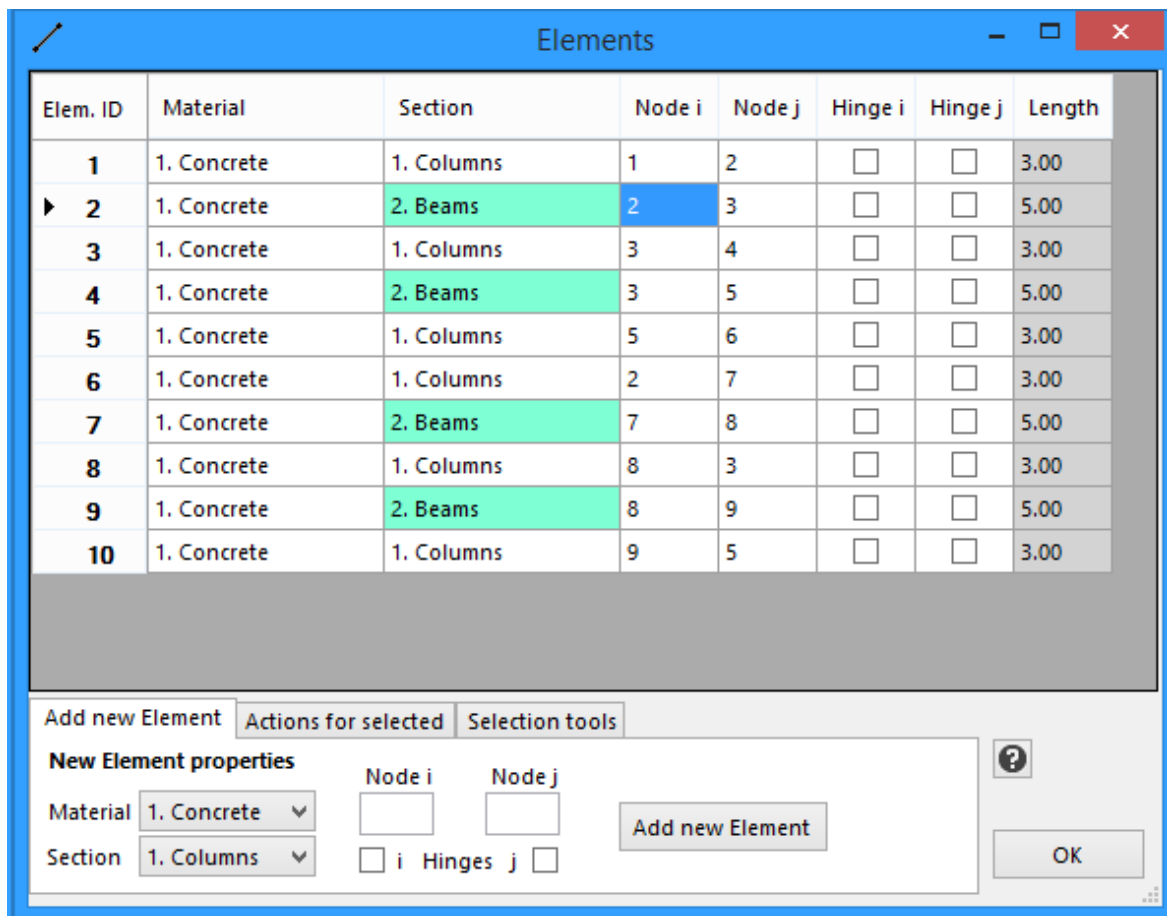
Node j
j ☐

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 image shows a software window titled "Nodal Loads". It features a table with four columns: "Node ID", "Force FX", "Force FY", and "Moment MZ". The table is currently empty. Below the table, there are three tabs: "Add new Nodal Load", "Actions for selected", and "Selection tools". The "Add new Nodal Load" tab is active. Within this tab, there is a section titled "New Nodal Load" which includes a "Node ID" input field, two radio buttons for "Cartesian (FX, FY)" (selected) and "Polar (F, θ)", and three input fields for "FX=", "FY=", and "MZ=". To the right of these fields is an "Add New" button. Further right, there is a help icon (a question mark in a square) and an "OK" button.

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) ☐ Polar (F, θ)

FX= FY= MZ=

?

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

The screenshot shows the 'Nodal Loads' window with a table and a 'New Nodal Load' section.

	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) ☐ Polar (F, θ)

FX= FY= MZ=

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

Nodal Loads

	Node ID	Force FX	Force FY	Moment MZ
▶ 1	2	30		

Add new Nodal Load Actions for selected Selection tools

New Nodal Load

Node ID:

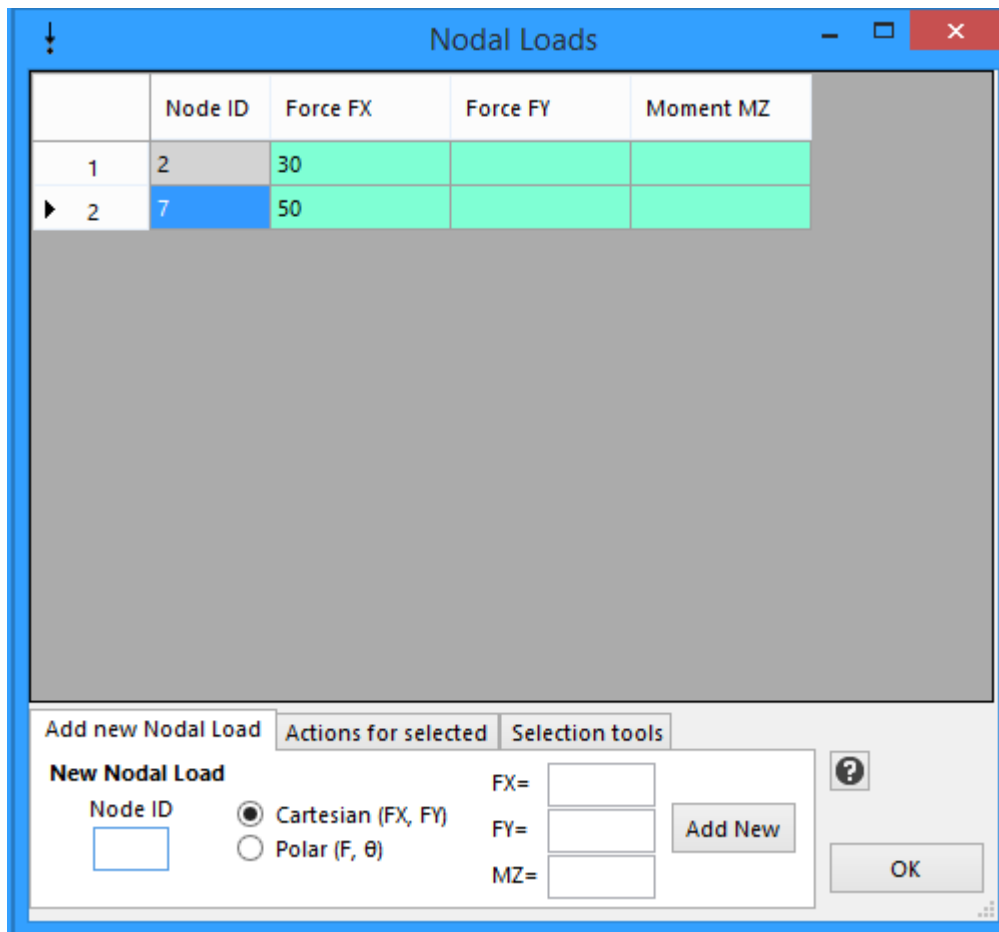
☒ Cartesian (FX, FY) ☐ Polar (F, θ)

FX= FY=

MZ=

Add New ? OK

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



	Node ID	Force FX	Force FY	Moment MZ
1	2	30		
2	7	50		

Add new Nodal Load | **Actions for selected** | **Selection tools**

New Nodal Load

Node ID

☒ Cartesian (FX, FY) ☐ Polar (F, θ)

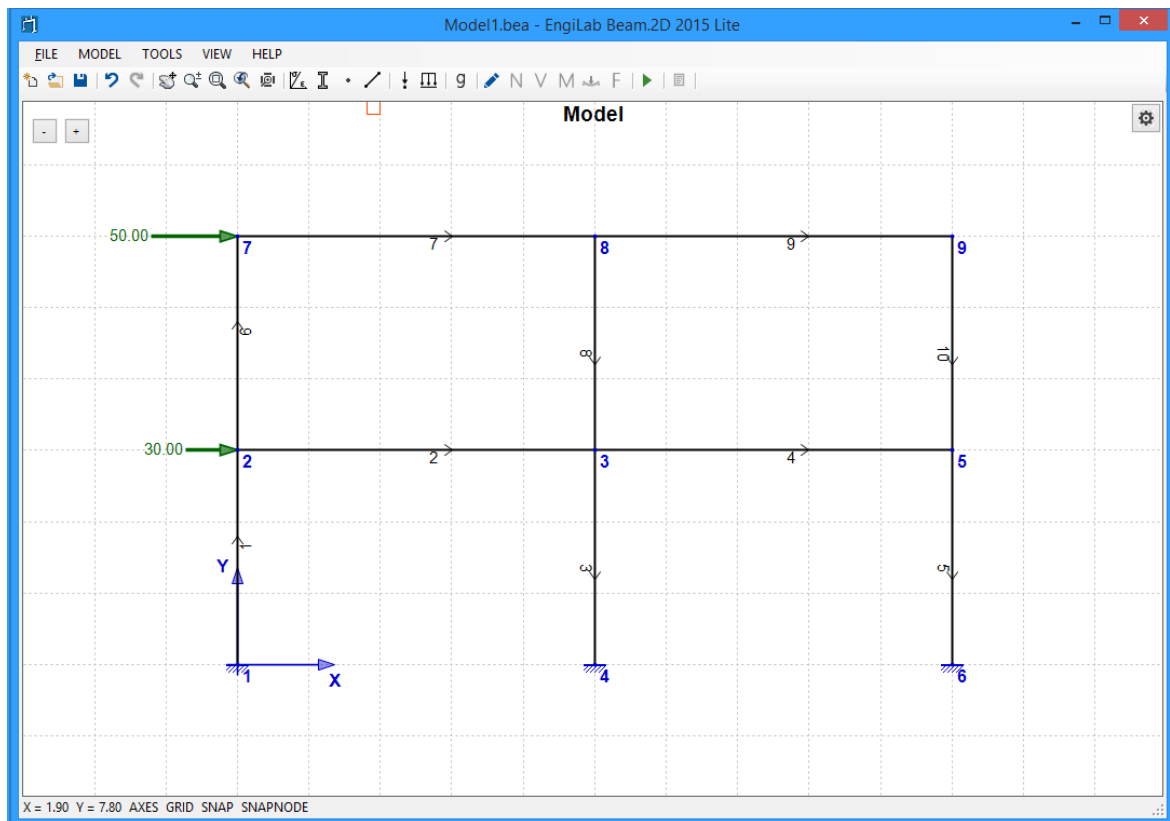
FX=

FY=

MZ=

Add New **OK**

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.

The 'Elemental Loads' dialog box is shown. It features a blue title bar with the text 'Elemental Loads' and standard window controls. The main area contains a table with the following headers: 'Elem. ID', 'Force fXi', 'Force fXj', 'Force fYi', and 'Force fYj'. Below the table, there are three tabs: 'Add new Elemental Load', 'Actions for selected', and 'Selection tools'. The 'Add new Elemental Load' tab is active, showing a 'New Elemental Load' section. This section includes an 'Elem. ID' input field, 'fX=' and 'fY=' input fields, and two radio buttons: 'Uniform' (selected) and 'Linear varying'. To the right of these fields is an 'Add Elem. Load' button. At the bottom right of the dialog is an 'OK' button. A help icon (?) is also present in the bottom right area.

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

Elem. ID	Force f_{Xi}	Force f_{Xj}	Force f_{Yi}	Force f_{Yj}
----------	----------------	----------------	----------------	----------------

Add new Elemental Load Actions for selected Selection tools

New Elemental Load

Elem. ID $f_X =$

☒ Uniform $f_Y =$

☐ Linear varying

Add Elem. Load OK

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:

	Elem. ID	Force fXi	Force fXj	Force fYi	Force fYj
1	2			-10	-10
2	4			-10	-10
3	7			-10	-10
4	9			-10	-10

Add new Elemental Load Actions for selected Selection tools

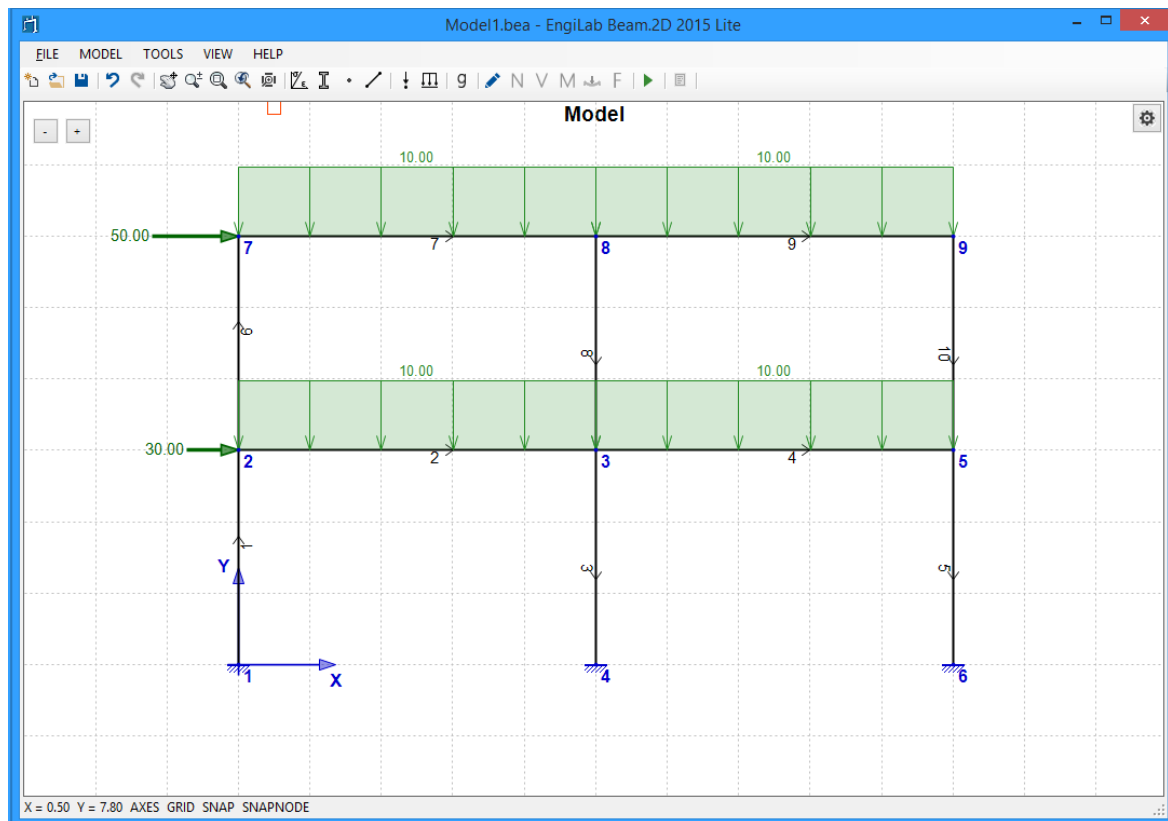
New Elemental Load

Elem. ID fX= fY=

☒ Uniform ☐ Linear varying

Add Elem. Load OK

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](#).

The dialog box titled "Body (Acceleration) Loads" contains a table with the following data:

Elem. ID	Material Density	Section Area	X-Body Load	Y-Body Load	Length
1	2.5	0.25	0	0	3.00
2	2.5	0.125	0	0	5.00
3	2.5	0.25	0	0	3.00
4	2.5	0.125	0	0	5.00
5	2.5	0.25	0	0	3.00
6	2.5	0.25	0	0	3.00
7	2.5	0.125	0	0	5.00
8	2.5	0.25	0	0	3.00
9	2.5	0.125	0	0	5.00
10	2.5	0.25	0	0	3.00

Below the table is a section titled "Linear Acceleration Vector" with input fields for $a_X =$ and $a_Y =$. A "Delete" button is next to the a_Y field. A tooltip points to the a_Y field with the text "Y-Component of the acceleration vector". There is also a "?" icon and an "OK" button.

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.

Elem. ID	Material Density	Section Area	X-Body Load	Y-Body Load	Length
1	2.5	0.25	0	-6.13125	3.00
2	2.5	0.125	0	-3.065625	5.00
3	2.5	0.25	0	-6.13125	3.00
4	2.5	0.125	0	-3.065625	5.00
5	2.5	0.25	0	-6.13125	3.00
6	2.5	0.25	0	-6.13125	3.00
7	2.5	0.125	0	-3.065625	5.00
8	2.5	0.25	0	-6.13125	3.00
9	2.5	0.125	0	-3.065625	5.00
10	2.5	0.25	0	-6.13125	3.00

Y-Acceleration set to -9.81

Linear Acceleration Vector

aX =

aY =

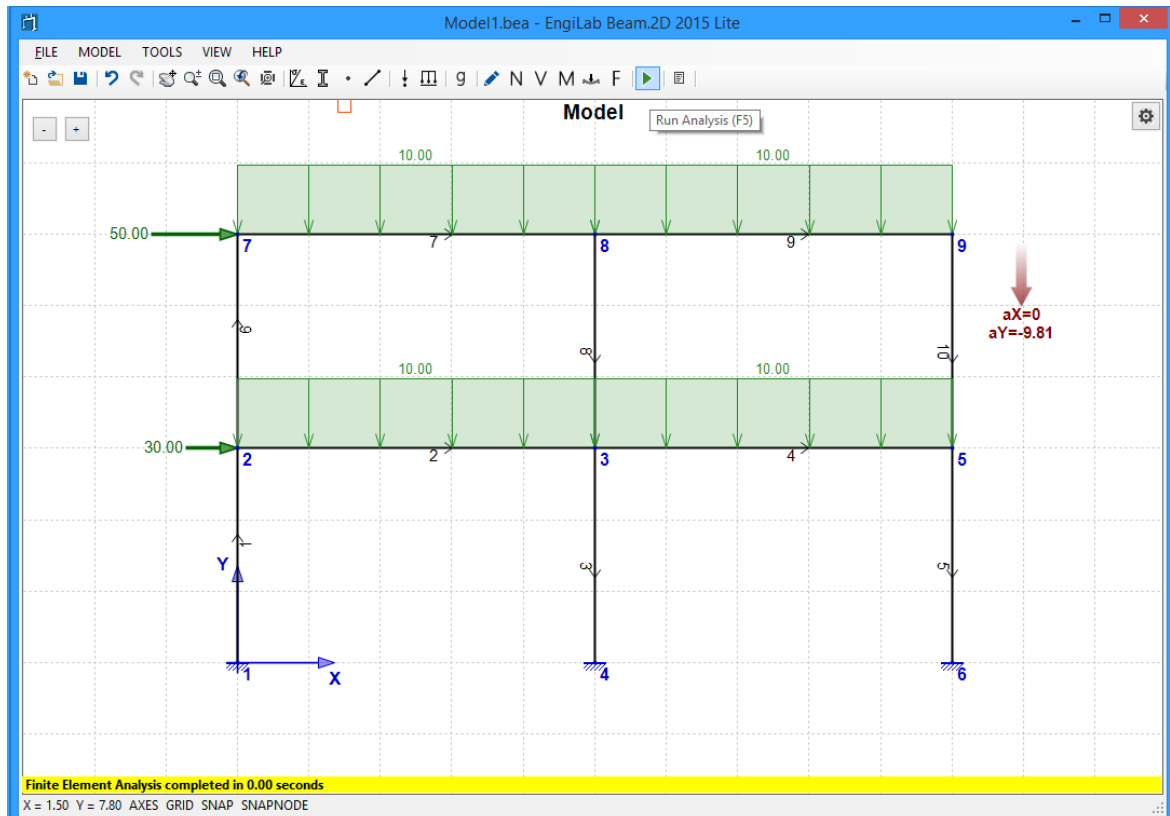
You see that the program has automatically calculated the self weight (in kN/m) for each Element. This is a read-only property. **Click OK to exit the Body (Acceleration) Loads.**

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.



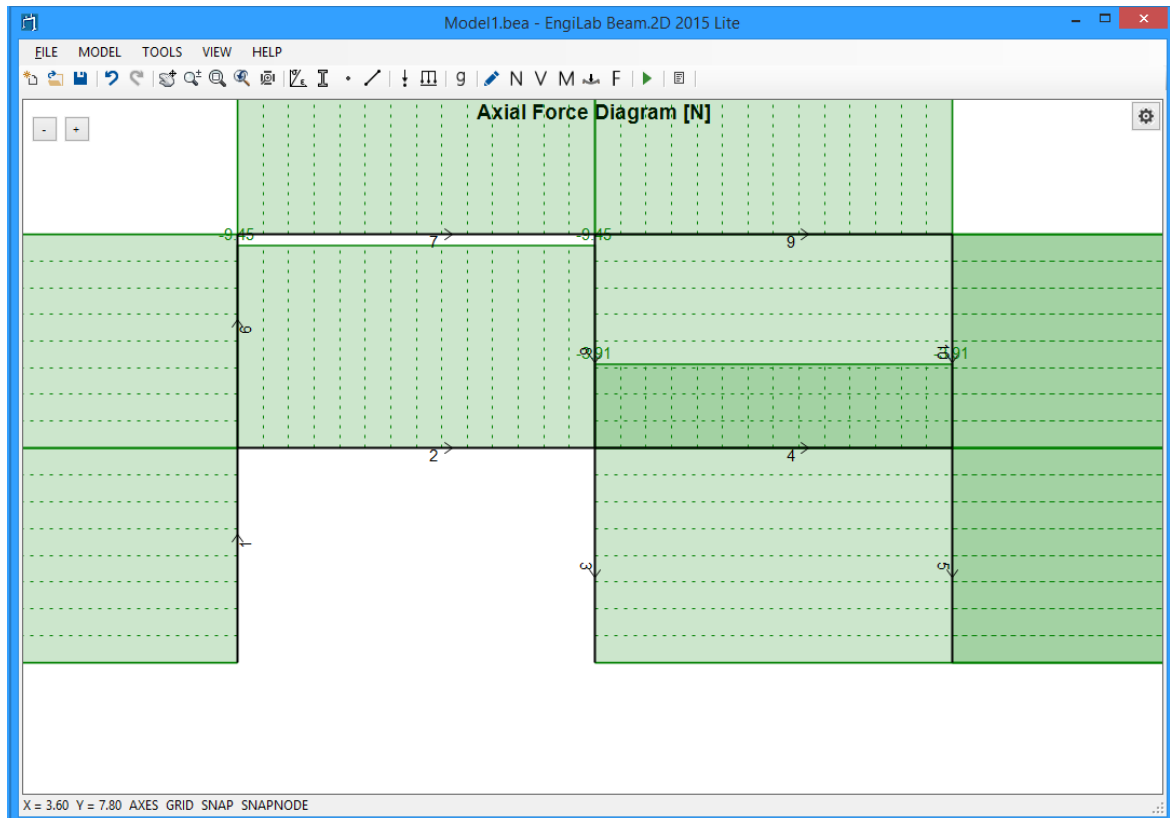
The Finite Element Analysis has been completed successfully.


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



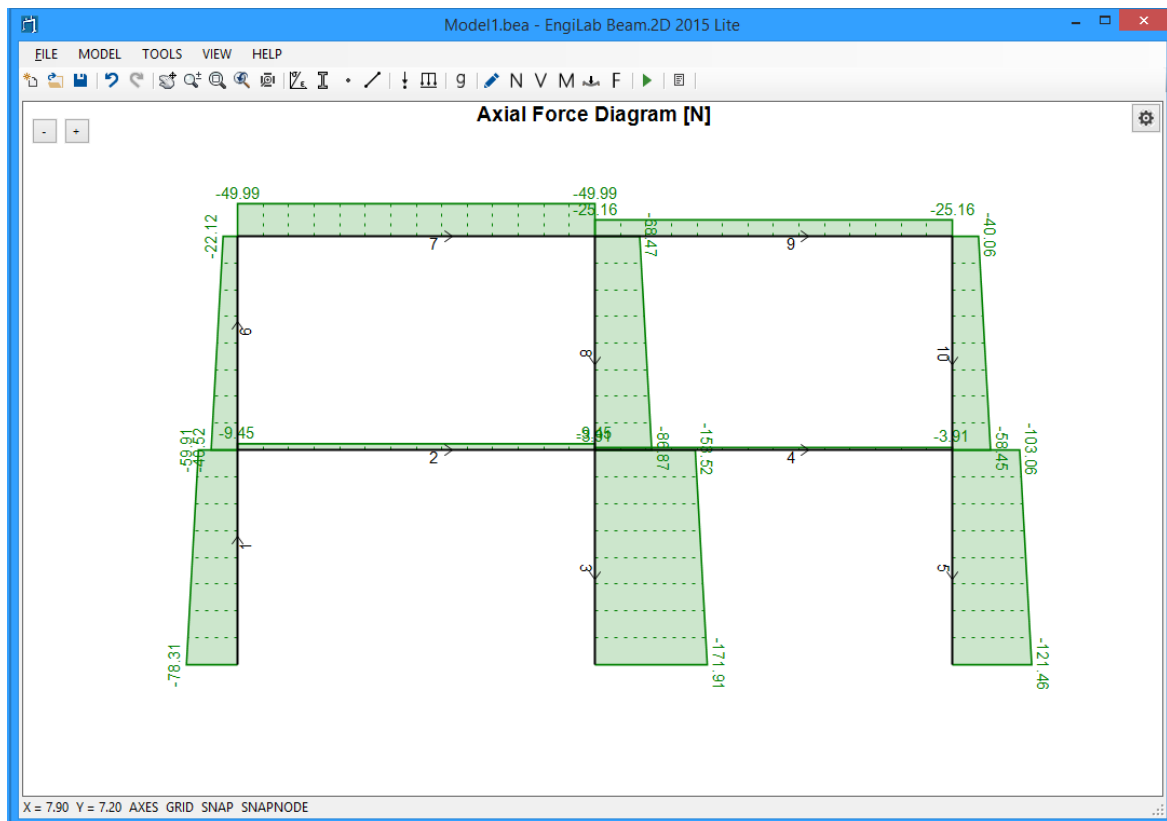
1. Axial Force Diagram

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





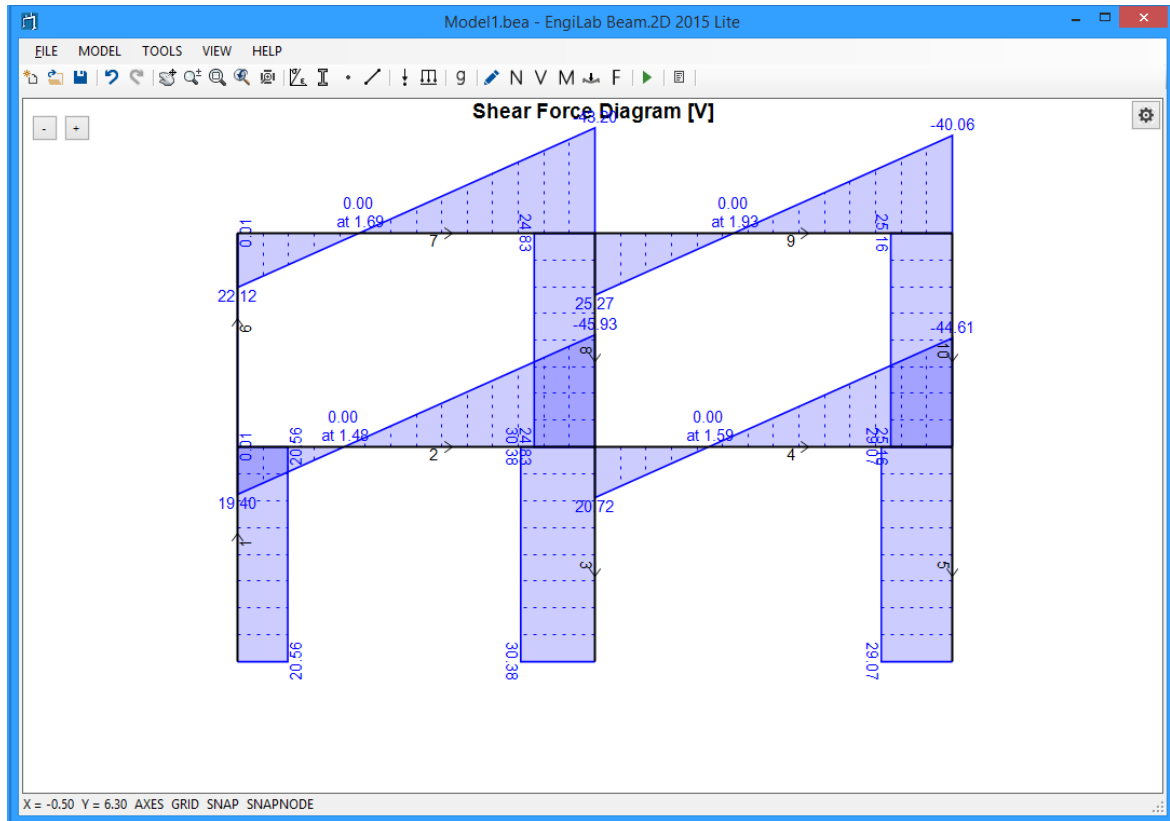
The Axial Force Diagram is out of Scale. Click the **"Zoom All"** button  to automatically scale the Diagram.

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




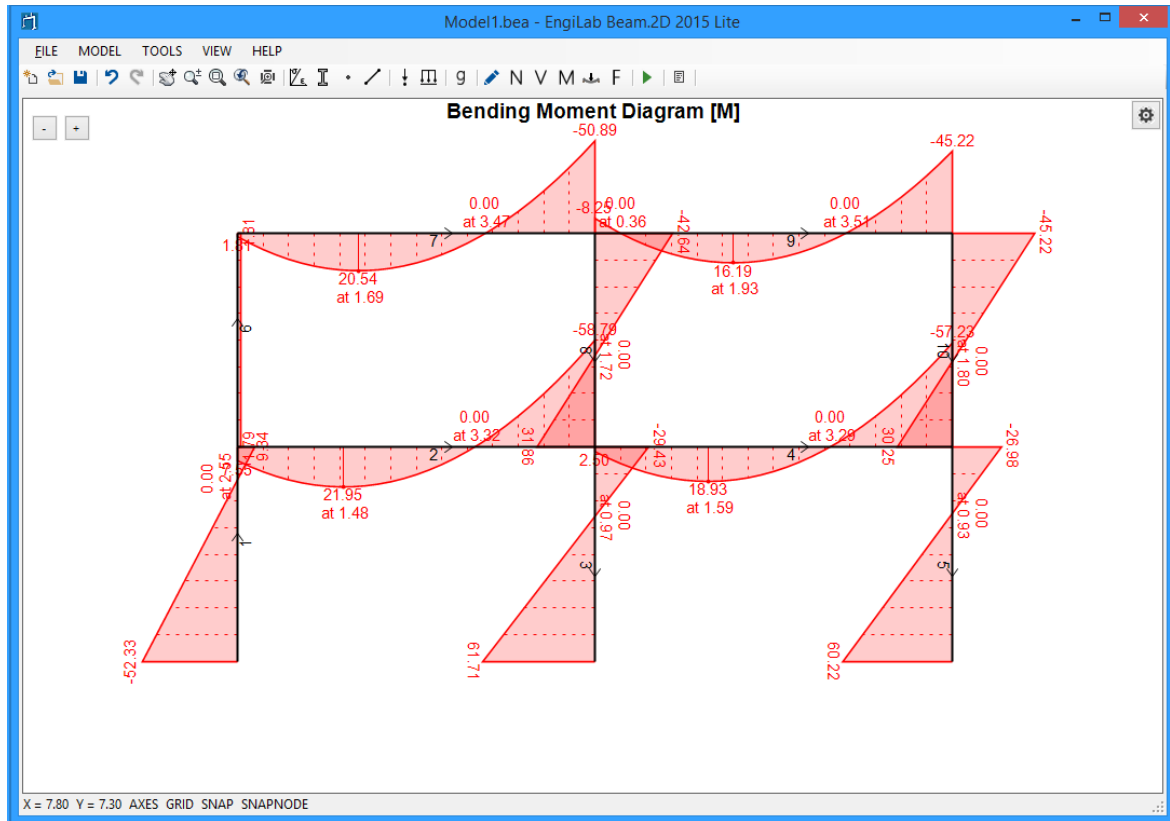
2. Shear Force Diagram

Click , 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.

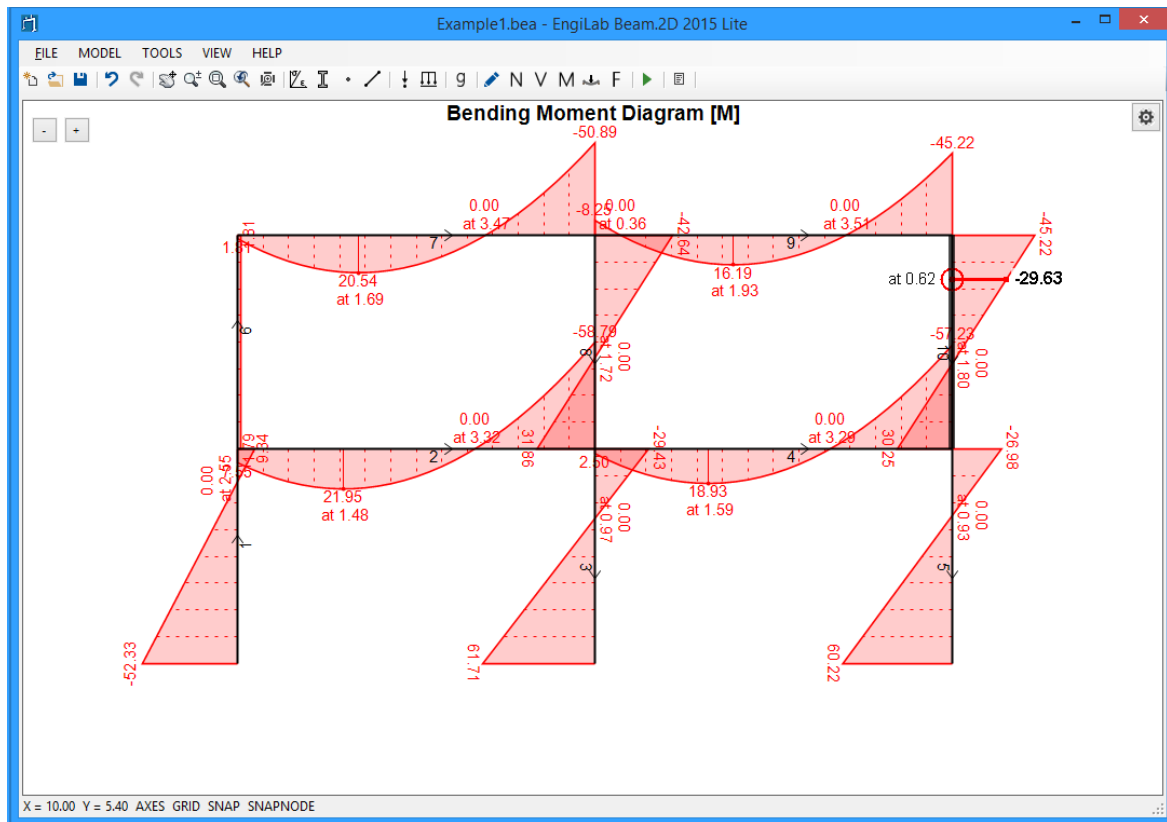


3. Bending Moment Diagram


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.

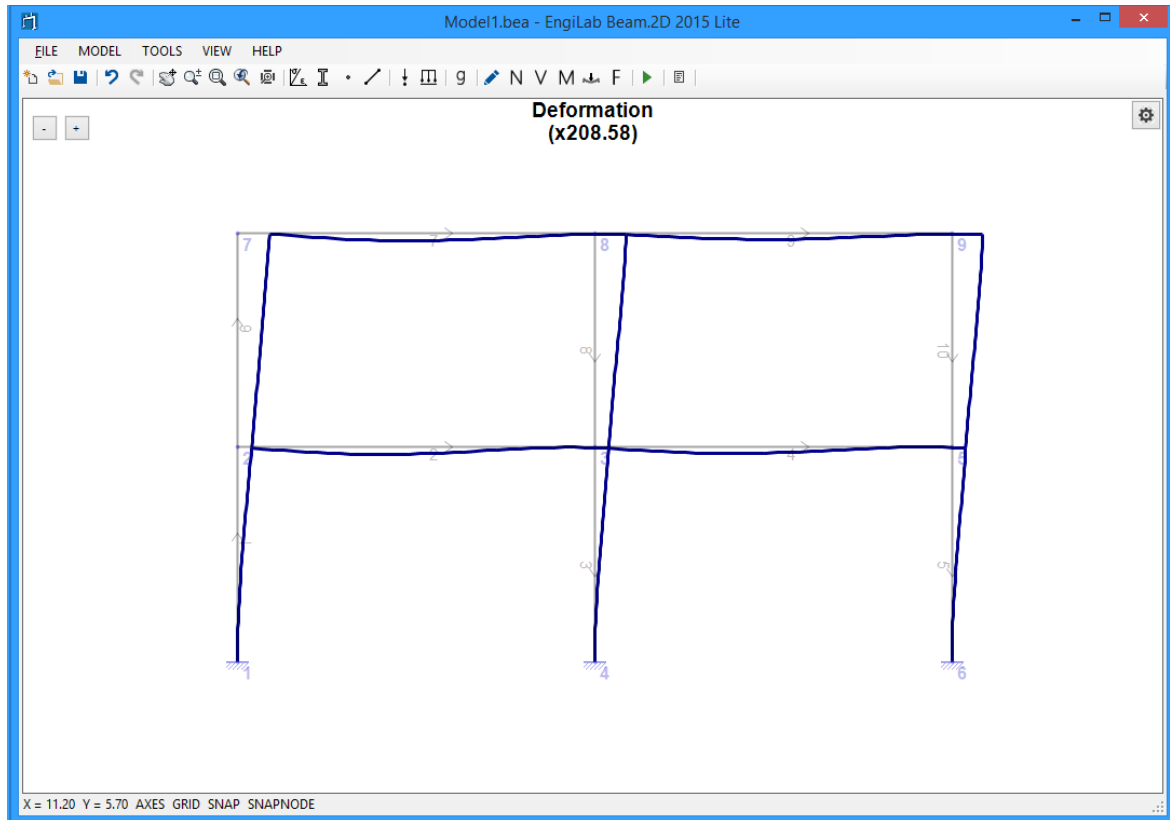


Note that if you move the pointer over an Element, you can read the corresponding value of the Diagram, as shown below for the Bending Moment Diagram case. This happens for all Diagrams and also for the Deformation and the Free Body Diagram.

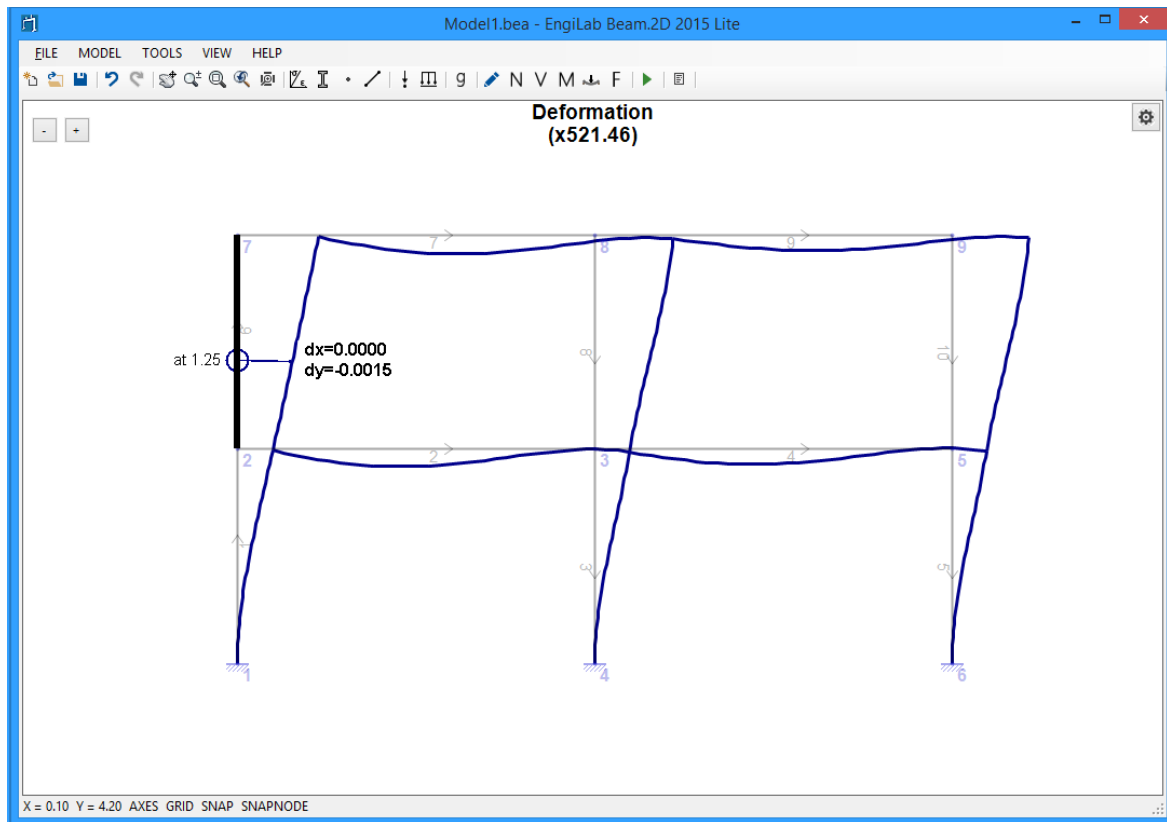


4. Deformation

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.

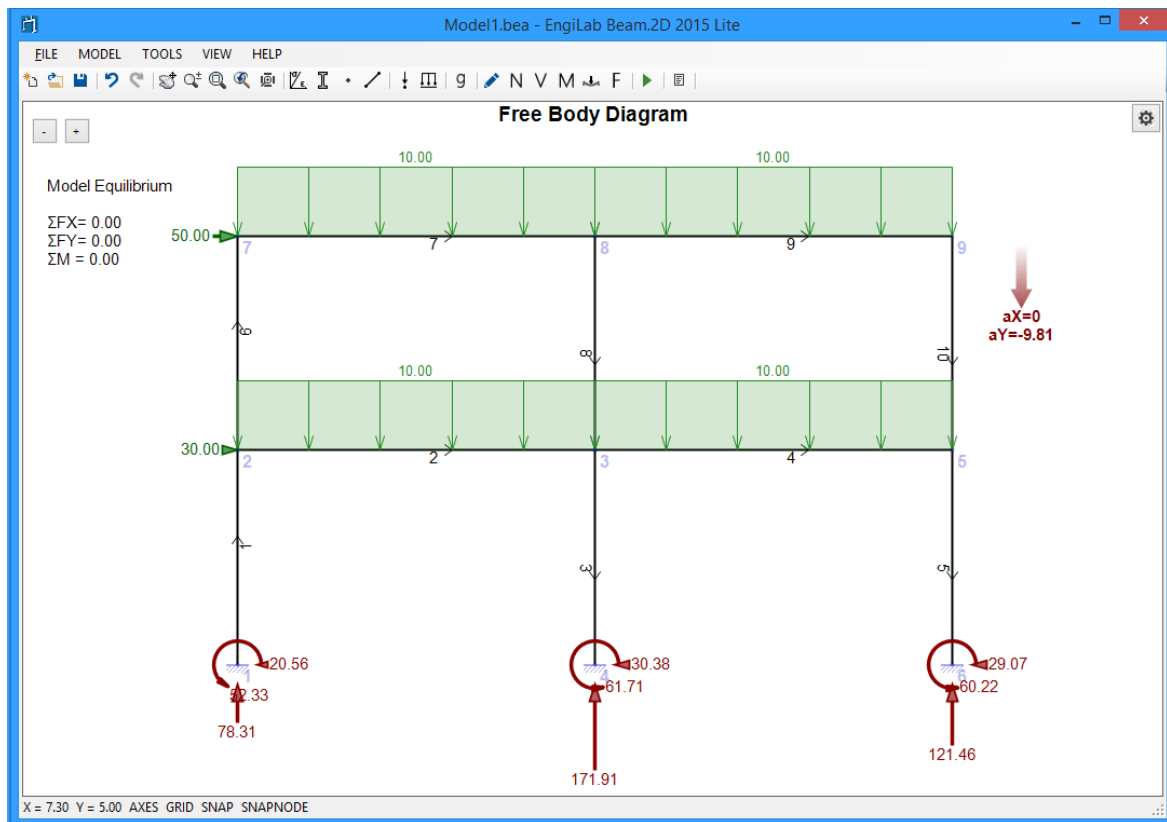


See above that you can get the deformation values on screen, if the mouse pointer moves over an element.

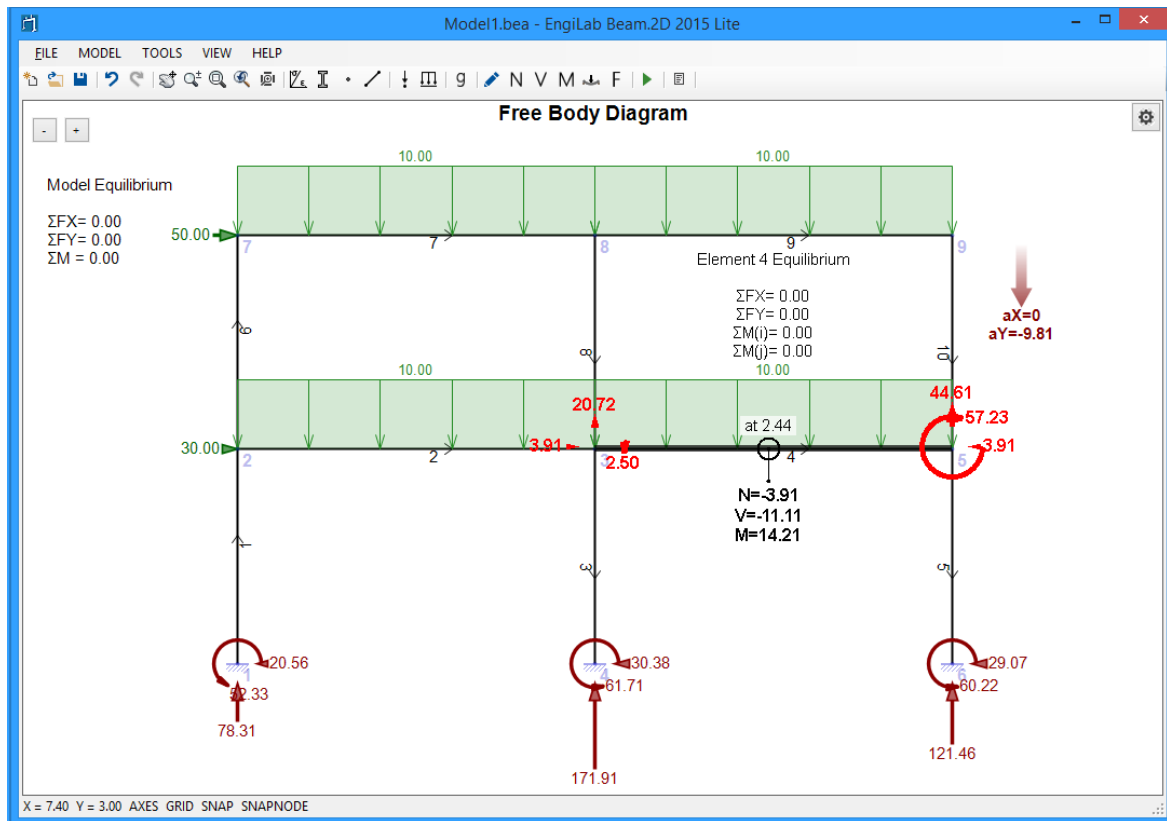
IMPORTANT: The values that are given on screen are the x and y displacements of the corresponding point of each element **in Local Element Axes**. For example, in the picture below Element 6 goes upward, which means that the Element x-Axis is pointing upwards and the Element y-Axis is pointing to the 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 (ΣF_X , ΣF_Y , $\Sigma M(i)$, $\Sigma M(j)$) 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 ΣF_Y . There is an external uniform load with value 10, acting along Element 4 (Length = 5 m), which gives an external load of $10 \times 5 = 50$ 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 = $0.125 \times 5 \times 2.5 \times 9.81 = 15.33$ kN.

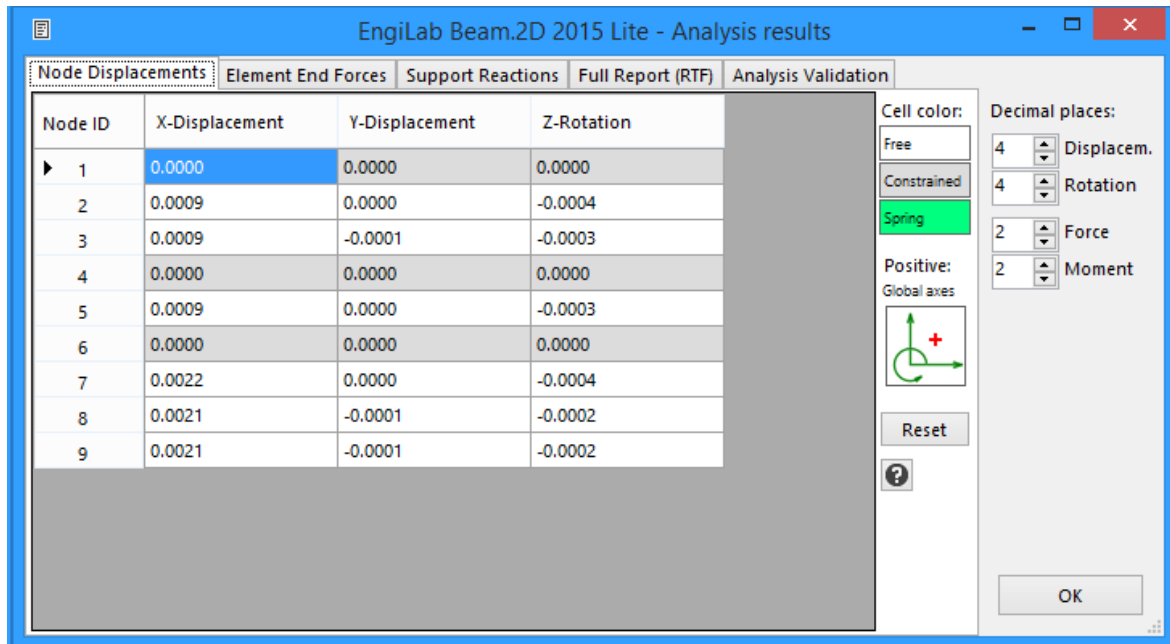
So, the correct equilibrium calculation should be: $\Sigma F_Y = 50 + 15.33 - 65.33 = 0.00$ kN which is reported by the program for Element 4.

7.1.13 Step 12. View the analytical results



Click  to see the analytical results.

1. Node Displacements



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)

EngiLab Beam.2D 2015 Lite - Analysis results

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

EngiLab Beam.2D 2015 Analysis Report
 Model: "Model1.bea"
 Report created: Sunday, 10 May 2015, 21:33:17

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

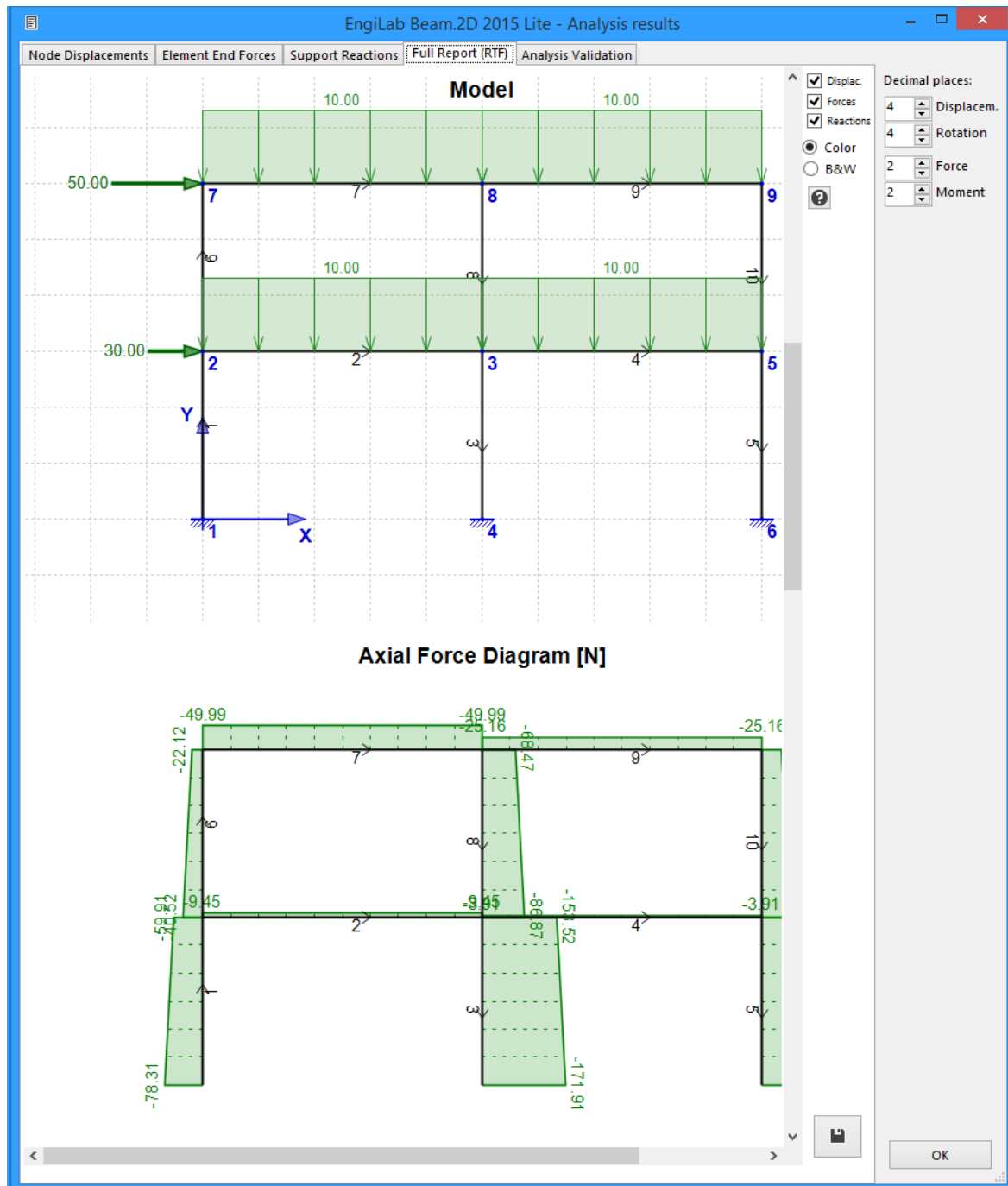
Sign convention: Positive according to Global Axes
 Forces reported with 2 decimal places
 Bending moments reported with 2 decimal places

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

Color: ☒ Color ☐ B&W

OK

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

EngiLab Beam.2D 2015 Lite - Analysis results

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

Global (Model) Equilibrium | Individual Element Equilibrium

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
SUM	0.00	0.00	0.00

Positive: Global axes

Decimal places:
 4 Displacement
 4 Rotation
 2 Force
 2 Moment

OK

5b. Individual Element Equilibrium

EngiLab Beam.2D 2015 Lite - Analysis results

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

Global (Model) Equilibrium | Individual Element Equilibrium

Element ID	FX Elemental Force	FY Body Force	FX End Force i	FX End Force j	SUM ΣFX	FY Elemental Force	FY Body Force	FY End Force i	FY End Force j	SUM ΣFY	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

Decimal places:
 4 Displacement
 4 Rotation
 2 Force
 2 Moment

OK

Chapter



License Agreement

8 License Agreement

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

8.1 EULA (Lite Edition)

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

IMPORTANT: Read the following terms carefully before installing, using and copying **EngiLab Frame.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 PC ("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 Frame.2D PRO Edition End User License Agreement (EULA)

IMPORTANT: Read the following terms carefully before installing, using and copying **EngiLab Frame.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 PC ("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. ENGILAB 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