

# EngiLab Frame.2D 2022

## v4.2

(v4.2.8183.0)

# User Manual

EngiLab

www.engilab.com

*This page intentionally left blank.*

# EngiLab Frame.2D 2022 v4.2 User Manual

**(c) 2022 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**

v4.2.8183.0

**Document version**

v1.0

**Document release date**

May. 28, 2022

*"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

EngiLab

*This page intentionally left blank.*

# Table of Contents


## Chapter 1 Introduction to EngiLab Frame.2D 1

1 Overview ... 2
2 What is a 2D Frame? ... 3
3 Ready to-analyze Examples ... 5
4 Lite and Pro Editions ... 6

## Chapter 2 Structural Modeling 9

1 Setting up a model ... 10
2 Model Properties ... 13
- Units ... 13
  - Consistent System ... 16
  - Predefined Systems ... 22
  - Custom System ... 23
- Materials ... 24
- Sections ... 28
  - Import Sections from Database ... 35
- Nodes ... 36
- Elements ... 49
- Nodal Loads ... 61
- Elemental Loads ... 64
- Body (Acceleration) Loads ... 67

## Chapter 3 Tools 71

1 Import Materials/Sections ... 72
2 Import DXF ... 73
3 Convert Model to Truss ... 74

## Chapter 4 Analysis, Results and Validation 79

1 Analysis ... 80
2 N, V, M Diagrams ... 84
3 Deformation ... 87
4 Free Body Diagram (F) ... 89
5 Analysis Results Form ... 91
- Node Displacements ... 92
- Support Reactions ... 93
- Element End Forces ... 94
- Full Report ... 96

6 Analysis Validation Form ... 97
- Global (Model) Equilibrium ... 98
- Elements Equilibrium ... 99

## Chapter 5 Settings 101

1 General (Settings) ... 102
2 Model (Settings) ... 103


(c) 2022 EngiLab PC

3 NVM Diagrams (Settings) ... 104
4 Colors (Settings) ... 106
5 Fonts (Settings) ... 109
6 Report (Settings) ... 110

Chapter 6 Useful information 113

1 Tips on Modeling hinges ... 114
2 Tips on Modeling symmetric structures ... 116

Chapter 7 Example Problems 119

1 Example Problem 1 (based on the 2015 version) ... 120
Overview - Example 1 ... 120
Step 1. Preparation of the input data ... 123
Step 2. Define Materials ... 126
Step 3. Define Sections ... 128
Step 4. Draw the Model on screen ... 136
Step 5. Edit Nodes ... 140
Step 6. Edit Elements ... 143
Step 7. Define Nodal Loads ... 145
Step 8. Define Elemental Loads ... 150
Step 9. Define Body (Acceleration) Loads ... 154
Step 10. Run the Analysis ... 156
Step 11. View N, V, M Diagrams, Model Deformation and Free Body Diagram 157
Step 12. View the analytical results ... 166
2 Example Problem 2 (based on the 2022 version) ... 172
Example 2 - Hand Calculations ... 174
Step 1. Define the System of Units ... 175
Step 2. Define Materials ... 177
Step 3. Define Sections ... 180
Step 4. Draw the Model on screen ... 183
Step 5. Edit Nodes ... 189
Step 6. Edit Elements ... 193
Step 7. Define Nodal Loads ... 198
Step 8. Define Elemental Loads ... 201
Step 9. Define Body (Acceleration) Loads ... 202
Step 10. Run the Analysis ... 204
Step 11. View the Analysis Results on screen ... 205
Step 12. View the analytical results ... 216
Analysis Validation (optional) ... 219

Chapter 8 License Agreement 221

1 EULA (Lite Edition) ... 222
2 EULA (Pro Edition) ... 225

(c) 2022 EngiLab PC

# Chapter



## Introduction to EngiLab Frame.2D

# 1 Introduction to EngiLab Frame.2D

## 1.1 Overview



**EngiLab Frame.2D 2022**

**v4.2** (v4.2.8183.0)

Copyright (c) 2022 EngiLab PC.
All rights reserved



www.engilab.com

EngiLab Frame.2D is a powerful structural engineering software for the **linear static analysis of plane (2D) frames** for Windows, featuring a fast, rich and intuitive user interface for pre-processing and post-processing of the results. It uses the **Finite Element Method (FEM)** for the analysis, with 3 DOFs per Node. EngiLab Frame.2D can draw the Model Deformation, Free Body Diagram, Axial Force, Shear Force and Bending Moment Diagrams, among others.

### 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:

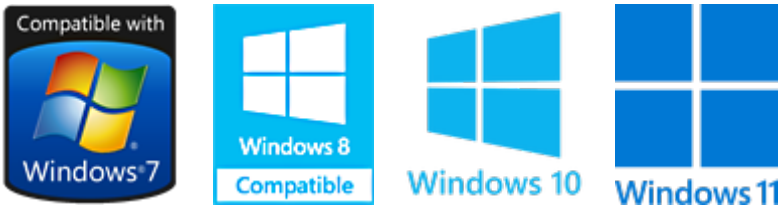
- Operating system (Windows 7, 8, 10, 11, etc.)
- Name and exact version of the product being used.
- Brief description of the problem.
- Detailed instructions on how to reproduce the problem, and/or an input file as an attachment.

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, Windows 8, Windows 8.1, Windows 10, Windows 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 What is a 2D Frame?

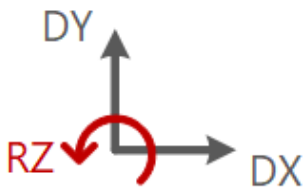
### About 2D Frames

EngiLab Frame.2D specializes in the analysis of 2D Frames. In EngiLab Frame.2D, each Node has 3 Degrees Of Freedom (DOFs), the DX-Displacement, the DY-Displacement and the RZ-Rotation. The figure below explains this and highlights the differences between **EngiLab Frame.2D** and **EngiLab Truss.2D** programs.

(c) 2022 EngiLab PC




2D Truss
2 DOFs per Node




2D Frame
3 DOFs per Node



**EngiLab Truss.2D**



**EngiLab Frame.2D**




Axial Force only



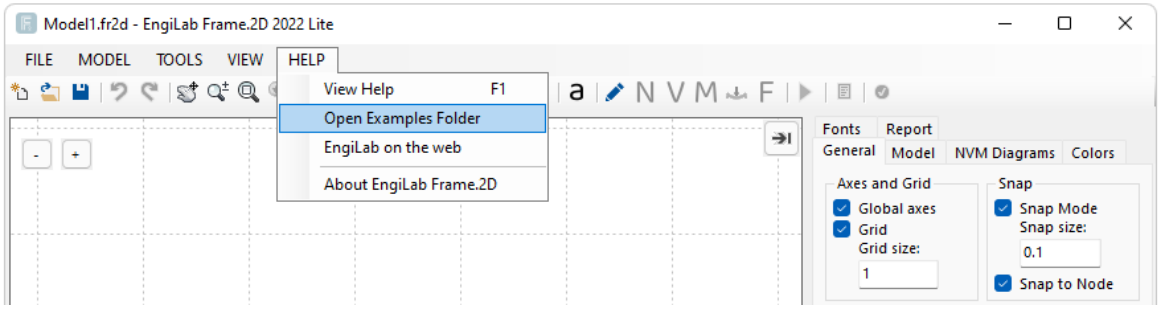

Axial Force
Shear Force
Bending Moment

Frames are very common in engineering. A Frame in EngiLab Frame.2D has the following properties:

- All members (Elements) are straight, connecting two Nodes.
- Members are connected only at the ends of the members. If two Elements share a common Node, they are connected through this Node.
- Members can have internal hinges at either of their ends (Start i and End j). A hinge is a pin joint which means that moment will be zero at that point for the particular member.
- A member has Axial Force, Shear Force and Bending Moment, at any of its points along its length.
- Point Loads are always Nodal Loads. Nodal Loads can include FX-Force, FY-Force and MZ-Moment.
- Along an Element, there can be distributed load in the form of a uniform load or a linear varying load.

(c) 2022 EngiLab PC

## 1.3 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 2022 Lite\Examples** for the Lite Edition and **C:\Program Files (x86) \EngiLab\EngiLab Frame.2D 2022 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.

(c) 2022 EngiLab PC

**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
- Support Reactions
- Element End Forces
- Full Report

## 1.4 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

- **Completely 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, no surprises.
- 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.

(c) 2022 EngiLab PC

- Technical Support for the Lite Edition is only provided via email at support@engilab.com

(c) 2022 EngiLab PC

### 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.

(c) 2022 EngiLab PC

# Chapter



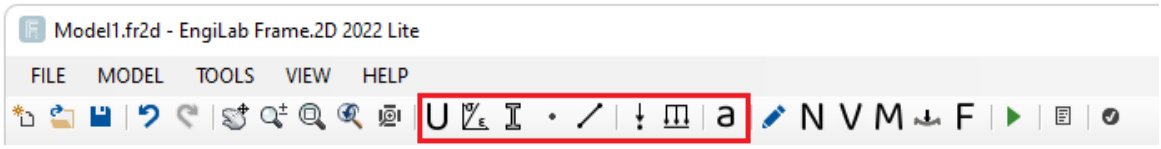
## Structural Modeling

# 2 Structural Modeling

## 2.1 Setting up a model

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

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




**1**. Click U to define the **System of Units**. This is a very important step for the model. For details, see Units

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

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

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

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

(c) 2022 EngiLab PC

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

**5**. Click to:

- Move existing **Nodes** to their exact positions, if needed. For example, a Node with X-Coordinate 5.8 is defined on screen with Snap Size = 0.1, but it should be moved to the exact position 5.85.
- Define or change Nodal Constraints (supports).
- Define Springs.
- Use tools related to Nodes, such as the Move Nodes tool, Copy Nodes tool, and others.

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 the Default Material and the Default Section, but the user can change this later on, if needed.
- Define Hinges at Element ends. Each Element can have a hinge at either end (Start-i or End-j or both).
- See properties of Elements such as Length, Angle, Volume, Mass, and others.

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 example to take into account the self weight of the structure. For details, see Body (Acceleration) Loads.

- If you want to take into account the self-weight of Elements as an additional uniform 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 set the earth gravitational

(c) 2022 EngiLab PC

acceleration with a minus (-) sign at the Y direction - this means gravity acting towards -Y global axis.

- **Example:**

Using Consistent Units, 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.

Using another System of Units, you simply have to enter the value of the standard earth gravitational acceleration in this system (check the Acceleration unit used in Units).

### B. Analyze the Model




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

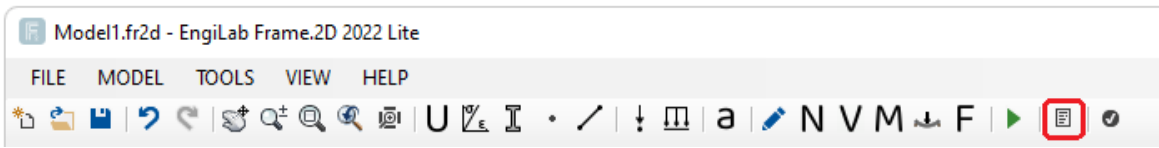
### C. Post-Processing: See the Analysis Results




**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:

(c) 2022 EngiLab PC

- Node Displacements
- Support Reactions
- Element End Forces
- Full Analysis Report

### D. Analysis Validation (optional)




Optionally, you can click to see the Analysis Validation Reports and make sure that everything is OK with the Analysis and its results.

- Global (Model) Equilibrium
- Elements Equilibrium

## 2.2 Model Properties

### 2.2.1 Units




Before starting to define any model in the program, you need to select a **System of Units** to use. EngiLab Frame.2D supports various Systems of Units:

- **Consistent System**
- **Predefined Systems**
  - Default Metric
  - Default US
  - International SI
- **Custom System (user-defined)**

**EngiLab strongly recommends the use of "EngiLab Units" software for any unit conversions needed. EngiLab Units is a free but powerful unit conversion program that is available for download at** www.engilab.com. EngiLab Units 2022 (v3.5) supports

(c) 2022 EngiLab PC

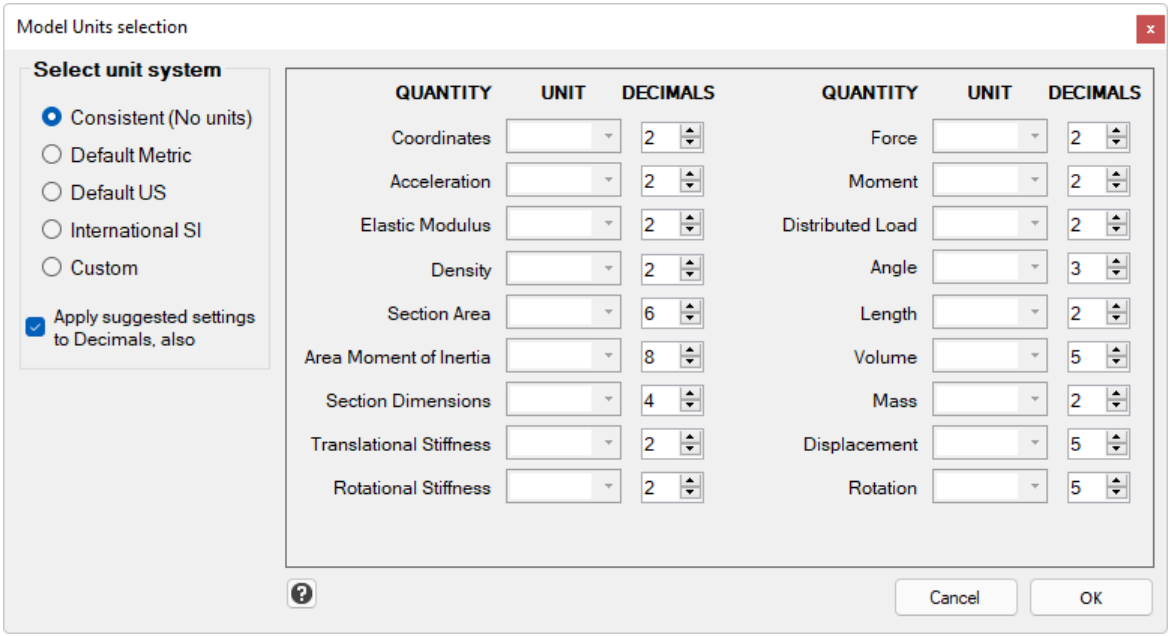
715 units in 22 unit categories, including distance, acceleration, pressure (stress) and others. **All units needed in EngiLab Frame.2D are supported by EngiLab Units**.




The Pre-defined Systems of Units are standard and cannot be changed. They are based on commonly used unit systems around the world.

The Custom System is fully customizable by the user.

If you click the "Units" button in the Toolbar, you get the Units form. The screenshot below depicts the Consistent System of Units. This system is special and is explained in detail in the Section Consistent Units




**Consistent System of Units**

The pre-defined Systems of Units are standard and cannot be changed. The Custom system is fully customizable by the user.

There are **18 types of units** used in the program. The various Units and their use, are explained in detail in the following figure:

| QUANTITY | USER INPUT | PROGRAM / ANALYSIS OUTPUT |
|---|---|---|
| Coordinates | X, Y Node Coordinates | |
| Acceleration | X, Y Model Acceleration | |
| Elastic Modulus | Material Elastic Modulus | |
| Density | Material Density | |
| Section Area | Section Area | |
| Area Moment of Inertia | Section Area Moment of Inertia | |
| Section Dimensions | Section Dimensions (in the Section Properties Calculator, used to define a Section). **This property is NOT saved to the input file**. | |
| Translational Stiffness | KX, KY Spring Stiffness | |
| Rotational Stiffness | KZ Spring Stiffness | |
| Force | FX, FY Nodal Load | Element End Force (Axial and Shear Force), Support Reaction Force, Sum of Forces |
| Moment | MZ Nodal Load | Sum of Moments |
| Distributed Load | fXi, fXj, fYi, fYj Elemental Load | Body (Acceleration) Load on Elements |
| Angle | Polar Coordinates in Nodal Loads (used to define a Nodal Load). **This property is NOT saved to the input file**. | Element Orientation Angle |
| Length | | Element Length |
| Volume | | Element Volume |
| Mass | | Element Mass |
| Displacement | | Node Displacement |
| Rotation | | Node Rotation |

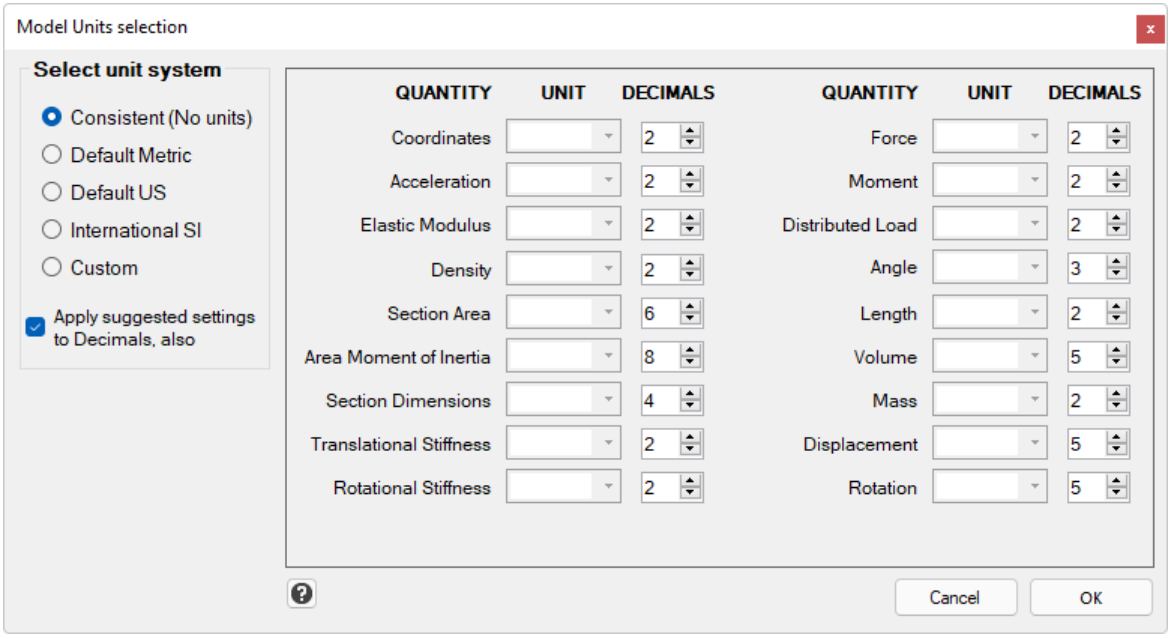
(c) 2022 EngiLab PC

Note that some units are used only for Model input data (for example, the Elastic Modulus unit), while other units are used for both inputs and outputs (for example, the Force unit). Others are used only for outputs (for example, the Displacement unit).

Note that Angle and Rotation are separate units in EngiLab Frame.2D. The Angle unit is used to show the Element Orientation Angle, while the Rotation unit is used for the rotational degree of freedom of the Nodes (Node Rotation in Nodal Displacements).

#### 2.2.1.1 Consistent System

The Consistent System of Units is the one shown in the form below. In fact, in this system, no specific units are used for any of the quantities, this is why all unit fields are empty. The Units can be anything (any compatible unit for the quantity in question), provided that some rules are followed, as explained next.




**Consistent System of Units**

**How does a Consistent System of Units 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 three 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/s$^2$, then we take F=200 and the unit of Force is then kg*m/s$^2$, 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 always 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). When using Consistent Units, the program will not show any units, 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 this way.

## Implementation

All input data must be specified in **consistent units**. As a result, the analysis results will also comply to that system. The important point about using Consistent Units is the necessity to stick with units that work correctly together - not to mix units that do not have a correct relationship with each other.

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

(c) 2022 EngiLab PC

### 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 $m/s^2$).

(c) 2022 EngiLab PC

## 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 | kg = N·s$^2$/m | t (tonne) = kN·s$^2$/m | t = N·s$^2$/mm | slug = lbf·s$^2$/ft | blob = lbf·s$^2$/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$^3$ | lbf·s$^2$/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$^3$ (lbf·s$^2$/ft$^4$) | 7.34544493×10$^{-4}$ blob/in$^3$ (lbf·s$^2$/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) | 30,457,924.9233 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$^2$ |

(c) 2022 EngiLab PC

**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 lbf·$s^2$/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 lbf·$s^2$/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**

(c) 2022 EngiLab PC

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 |
| Mass | t=Mg |
| Volume | $m^3$ |

* Rotations (where applicable) are ALWAYS given in RADIANS in a Consistent System of Units.

**Note**: In the above example, if one wants to apply self-weight to the structure, he/she can add the standard earth gravitational acceleration at the -Y direction as: **aY = -9.80665** (i.e. $m/s^2$)

(c) 2022 EngiLab PC

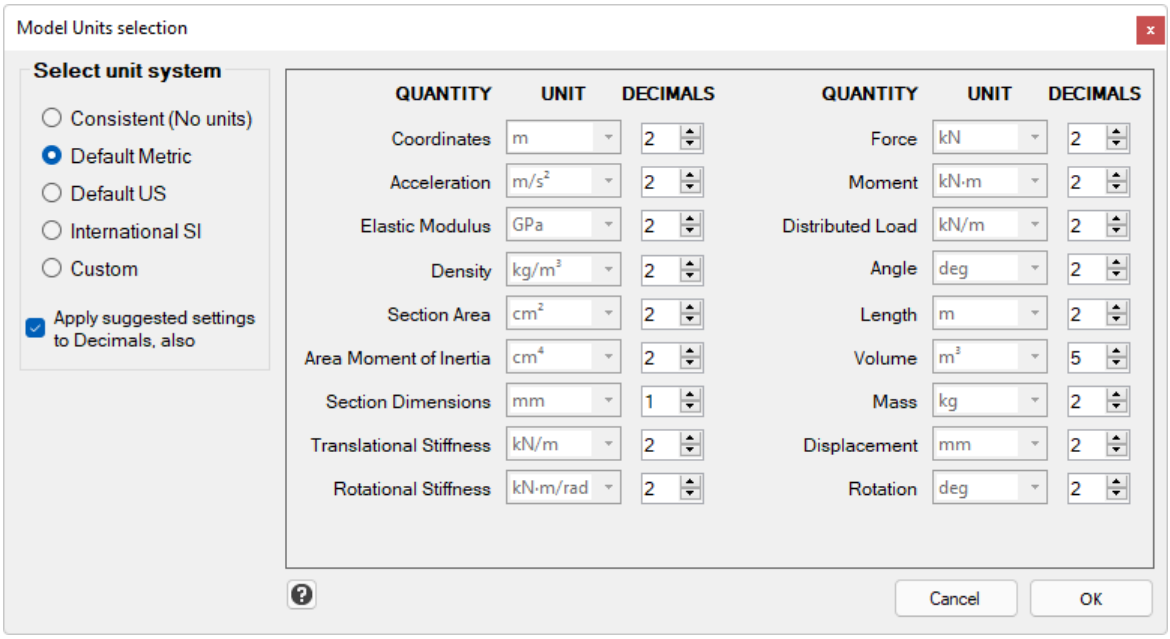
#### 2.2.1.2 Predefined Systems

EngiLab Frame.2D supports the following predefined Systems of Units:

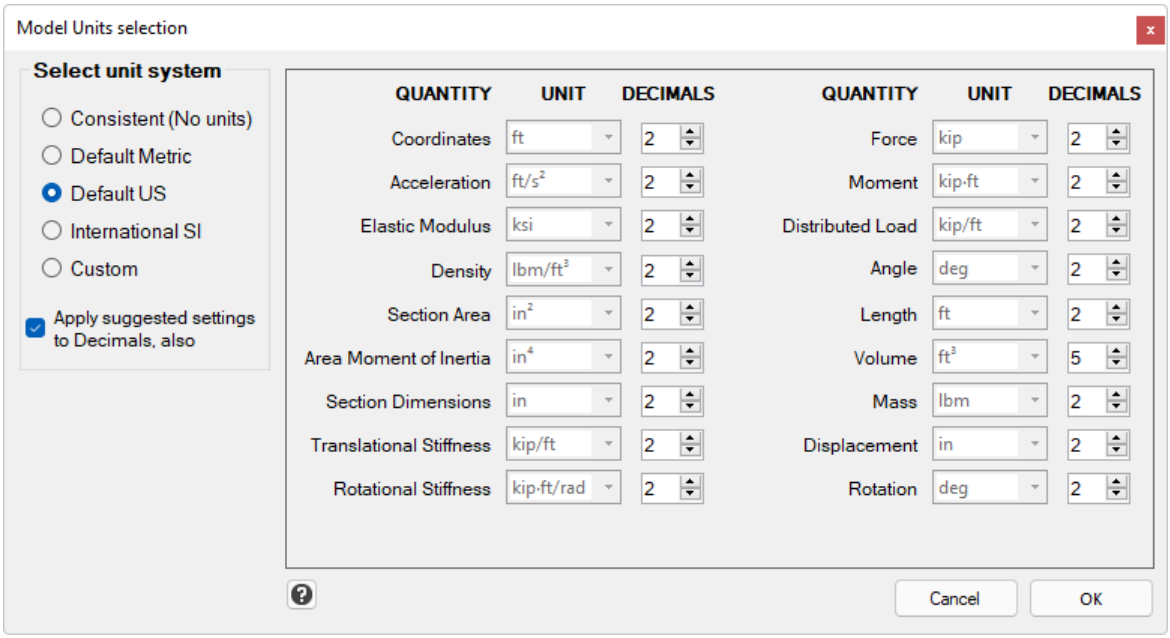
- **Default Metric**
- **Default US**
- **International SI**

The three systems are shown in the screenshots below. In these systems Units are "locked" and cannot be changed. The Decimal places for each quantity can be changed. These Decimal places are for illustration purposes, for showing the results or the Model data on screen and do not affect any calculations or the accuracy of the Analysis.

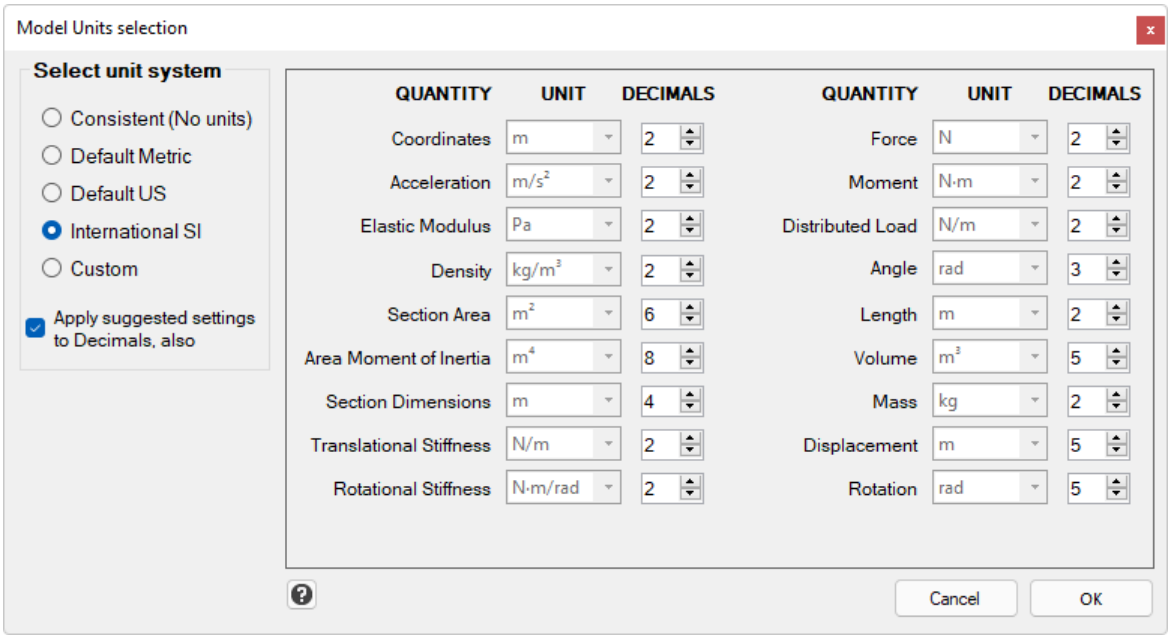
**Tip**: If you want to change a predefined System of Units, all you need to do is to select it and then click "**Custom**". This will unlock all fields and you can change any of them. The system will change to "Custom" but it will be based on the previously selected system.




**Default Metric System of Units**




**Default US System of Units**




**International SI System of Units**

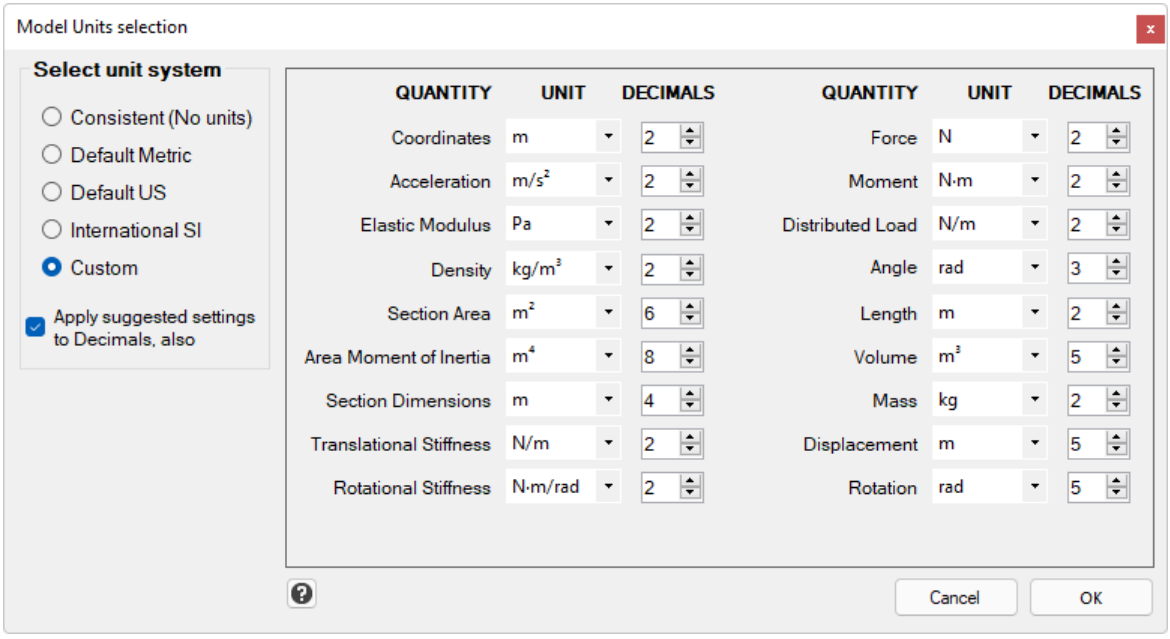
### 2.2.1.3 Custom System

The Custom System of Units is fully customizable by the user. A variety of units is offered for each unit category, for the user to select from.

**Tip**: You can first select a predefined system and then you can click "**Custom**". This will "unlock" all the unit fields and then you can choose whatever you want.

**Note**: Units cannot be left empty. On the other hand, if ALL units are empty, this is acceptable as this is the definition of the Consistent System of Units. In other words, either ALL units can be empty (i.e. Consistent System) or no units can be empty.

(c) 2022 EngiLab PC




**Custom System of Units**

## 2.2.2 Materials




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

- **Name** (optional, max. number of characters in Name: 20)
- **Elastic Modulus E** (or Modulus of Elasticity, or Young's Modulus) - Essential property
- **Density d** (optional, can be omitted, see below)

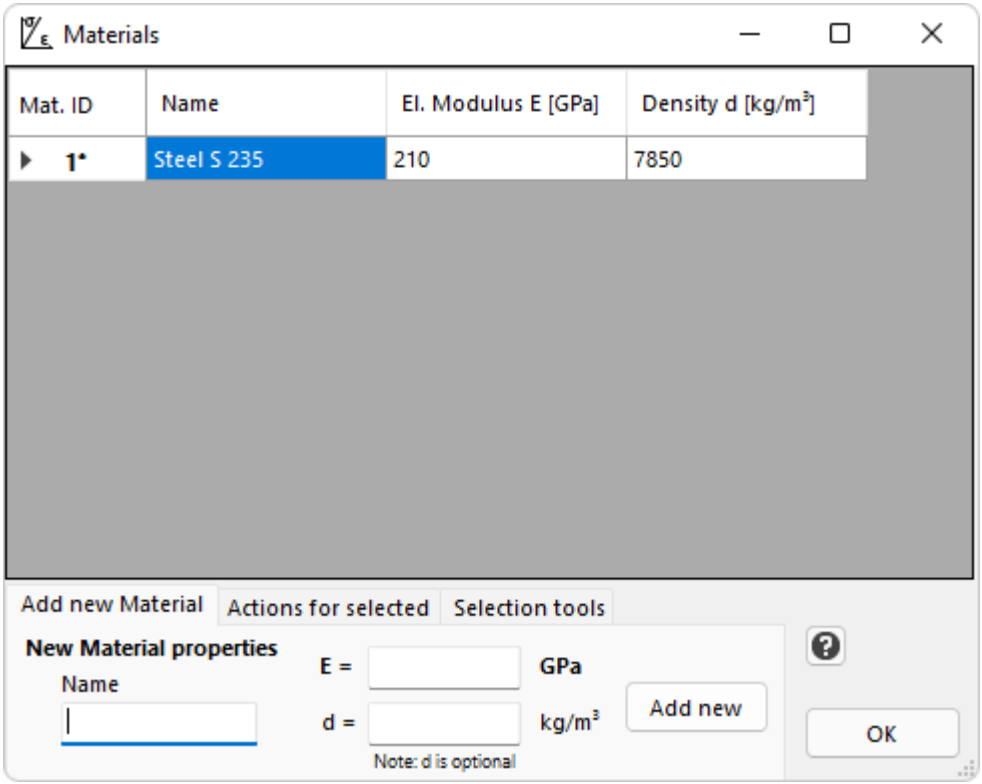
The **Elastic Modulus E** (essential property) of the Material is an important parameter affecting the Model Stiffness Matrix. It cannot be omitted and it cannot have a zero value.

The **Density d** (optional) 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

(c) 2022 EngiLab PC

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), Elastic Modulus E and Density d (optional) of the Material and click the "Add new Material" button.

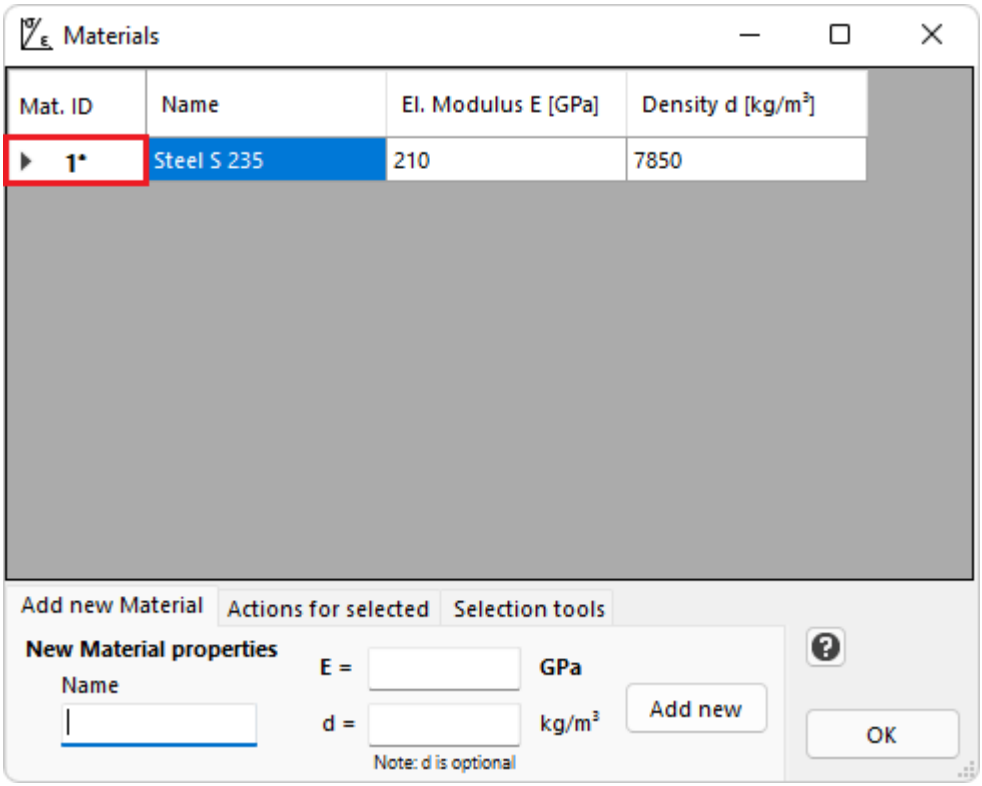
### Edit an existing Material

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.

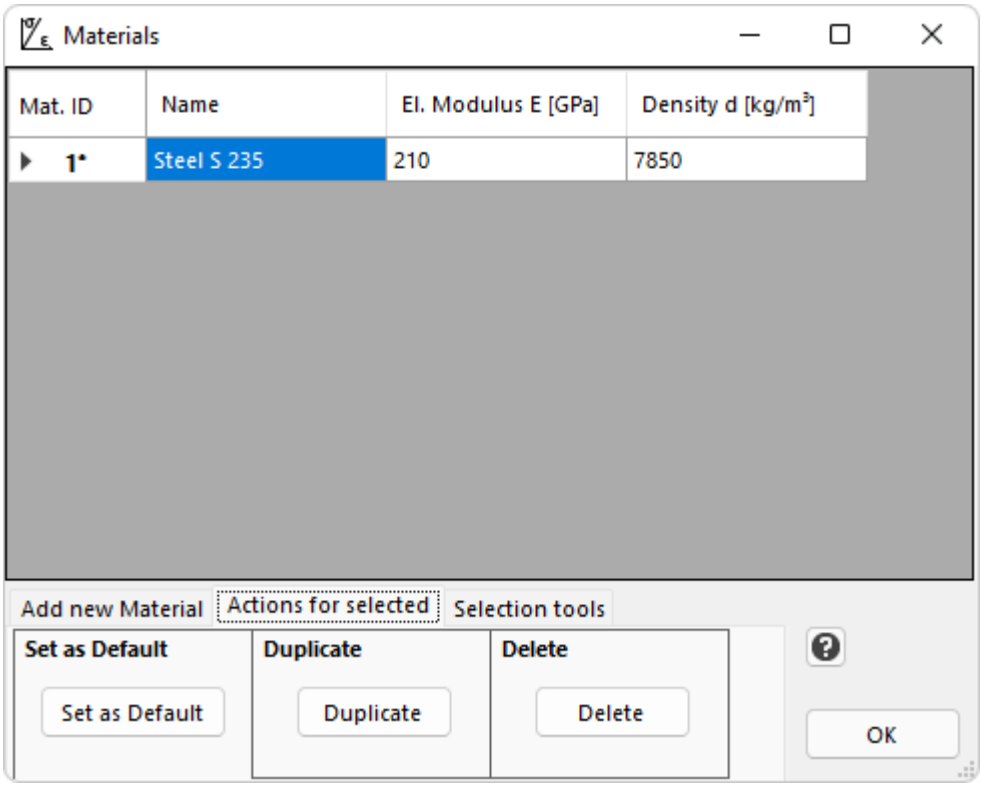
(c) 2022 EngiLab PC


Materials
Mat. ID
Name
El. Modulus E [GPa]
Density d [kg/m³]
1*
Steel S 235
210
7850
Add new Material
Actions for selected
Selection tools
New Material properties
Name
E =
GPa
d =
kg/m³
Note: d is optional
Add new
OK

### 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.

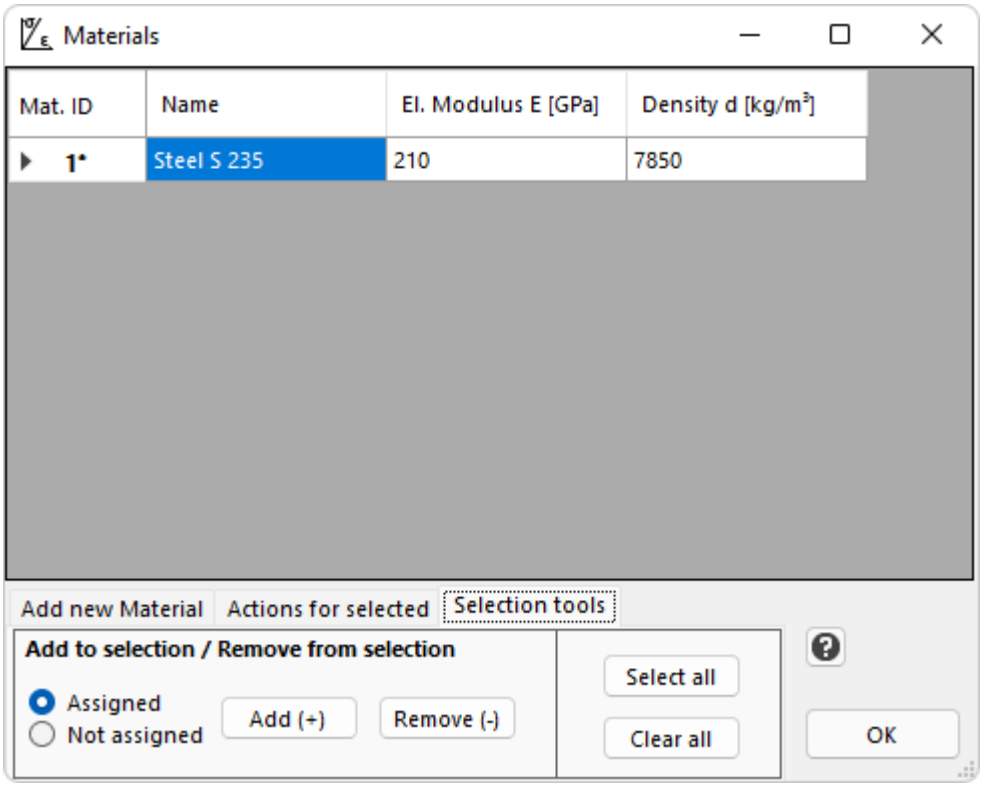
- **Set as Default**. Sets the selected Material as the Default Material. Any Elements drawn on the screen are assigned the Default Material.
- **Duplicate Material**(s). Click the "Duplicate" button. You can Duplicate more than one Materials at a time.
- **Delete Material**(s). Click the "Delete" button. You can Delete more than one Materials at a time.




(c) 2022 EngiLab PC

### 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.




## 2.2.3 Sections

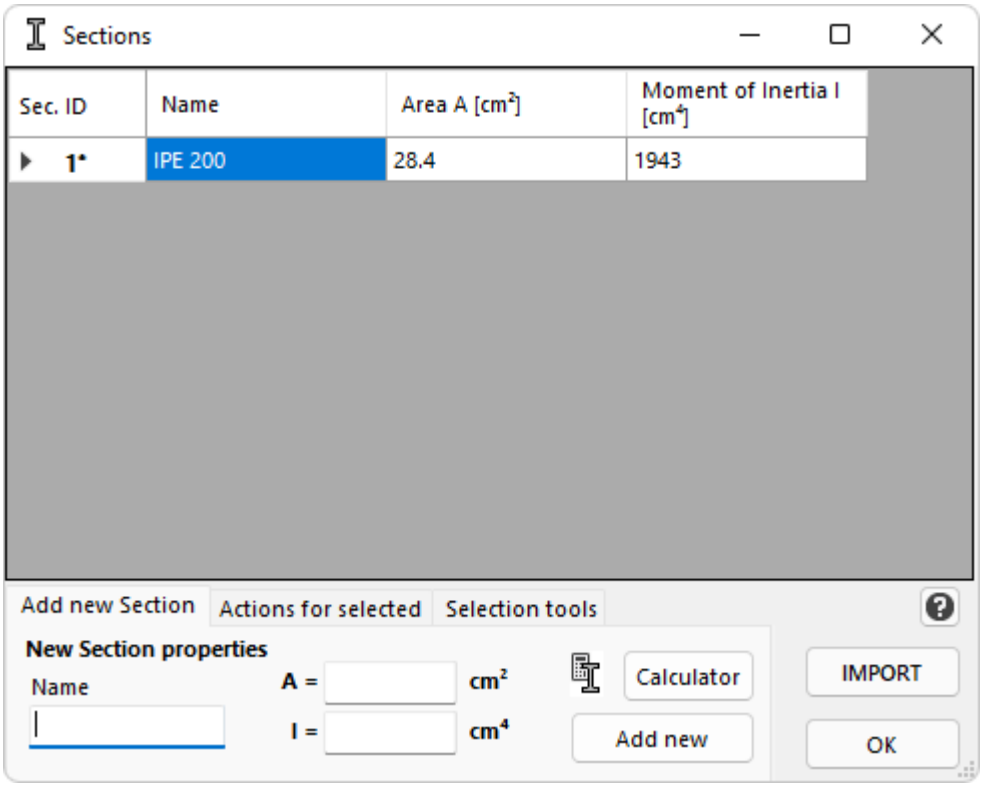



Each element is assigned a **Section**. The Section properties are the following:

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

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

(c) 2022 EngiLab PC




### 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.

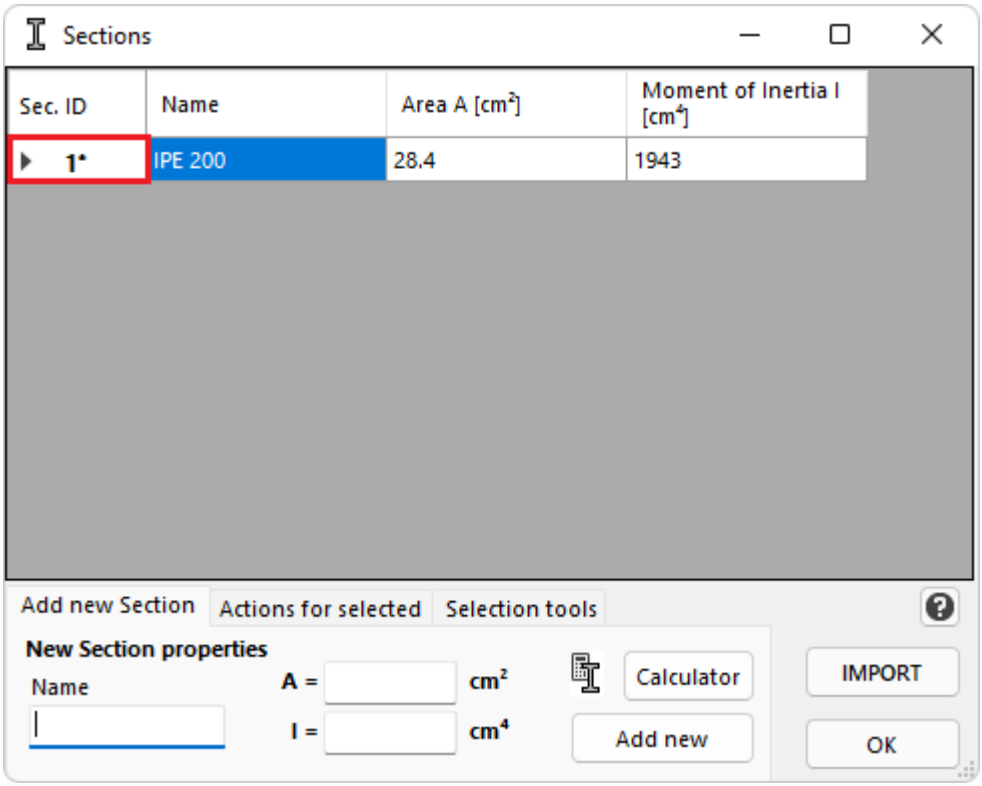
### Edit existing Section(s)

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".

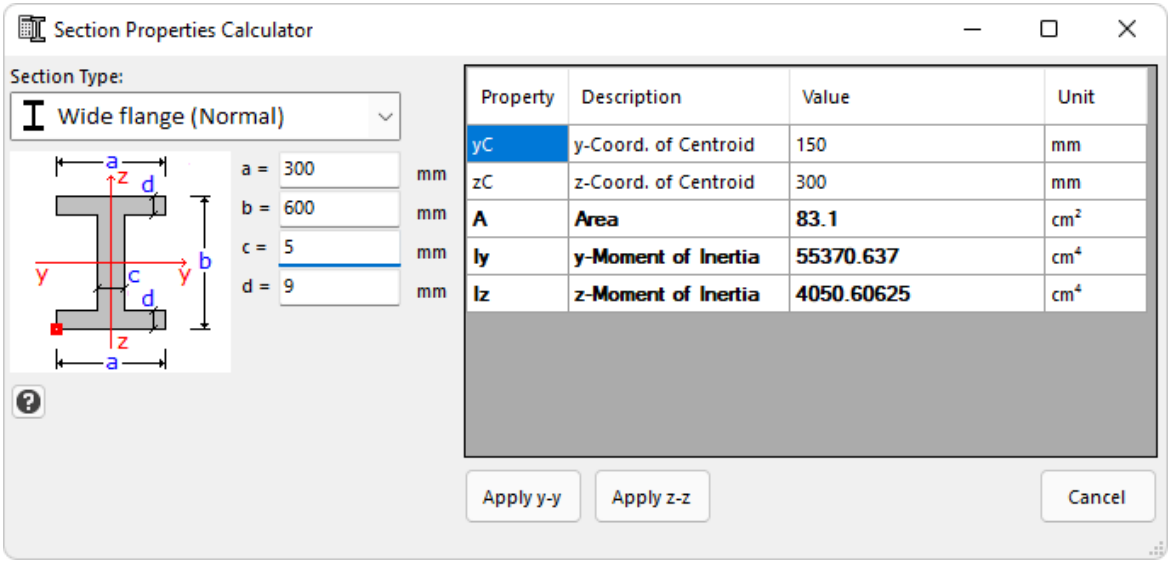
(c) 2022 EngiLab PC


Sections
Sec. ID
Name
Area A [cm²]
Moment of Inertia I [cm⁴]
1*
IPE 200
28.4
1943
Add new Section
Actions for selected
Selection tools
New Section properties
Name
A =
cm²
I =
cm⁴
Calculator
Add new
IMPORT
OK

## Section Properties Calculator

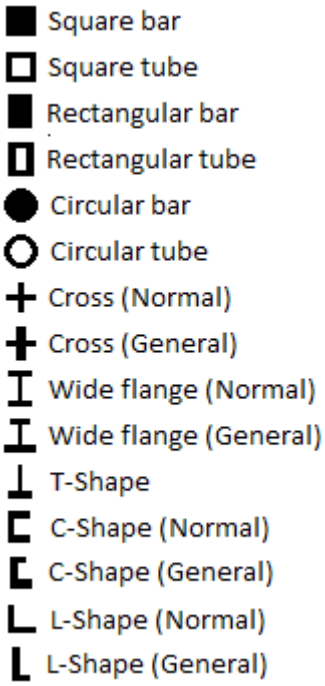
By clicking the "**Calculator**" button, a new form appears, as shown below. You can use the Section Properties Calculator to calculate the properties of various section shapes. The various section shapes are shown in the second figure below. The Calculator calculates various properties. The program needs only to take the **Area A** and **Moment of Inertia I** property.

- Click "**Apply y-y**" to apply the Area A and the $I_y$ Moment of Inertia
- Click "**Apply z-z**" to apply the Area A and the $I_z$ Moment of Inertia




**The available sections are the following:**

(c) 2022 EngiLab PC

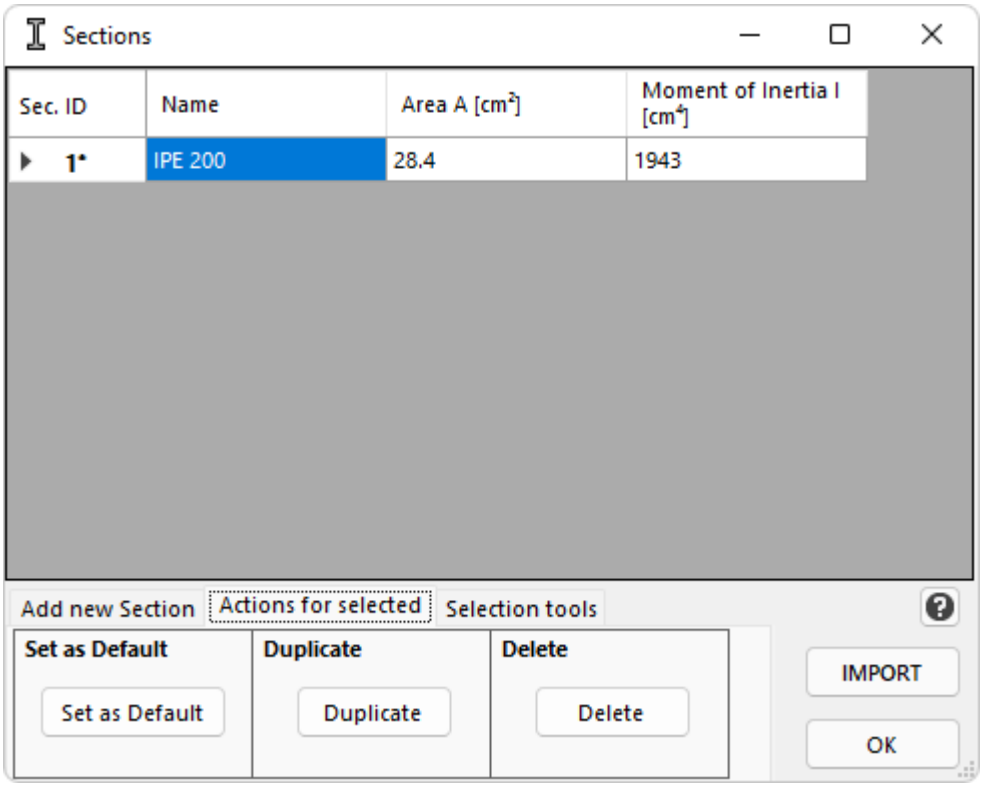



(c) 2022 EngiLab PC

### 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.

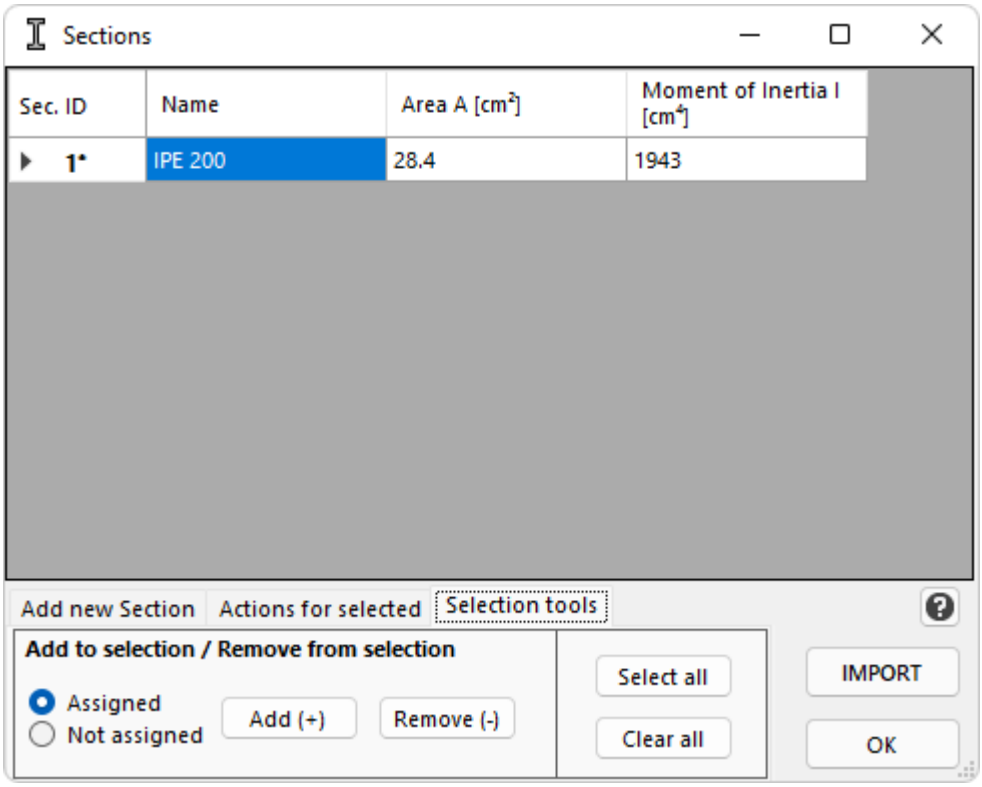
- **Set as Default**. Sets the selected Section as the Default Section. Any Elements drawn on the screen are assigned the Default Section.
- **Duplicate Section**(s). Click the "Duplicate" button. You can Duplicate more than one Sections at a time.
- **Delete Section**(s). Click the "Delete" button. You can Delete more than one Sections at a time.



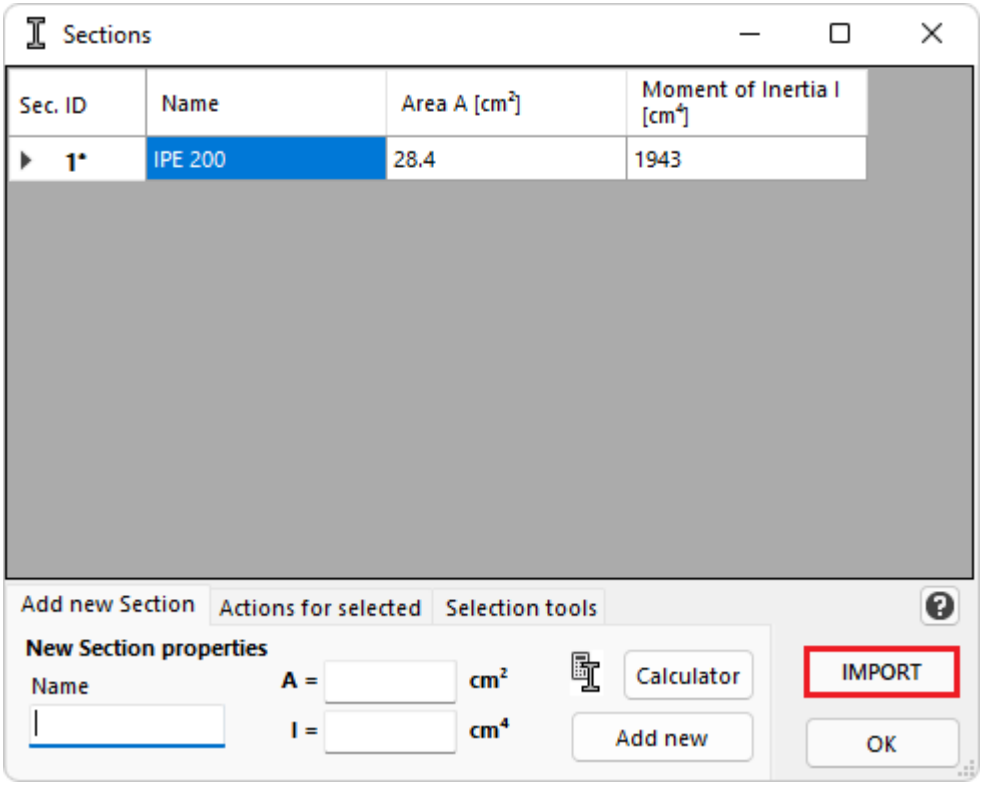

(c) 2022 EngiLab PC

### 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.



#### 2.2.3.1 Import Sections from Database

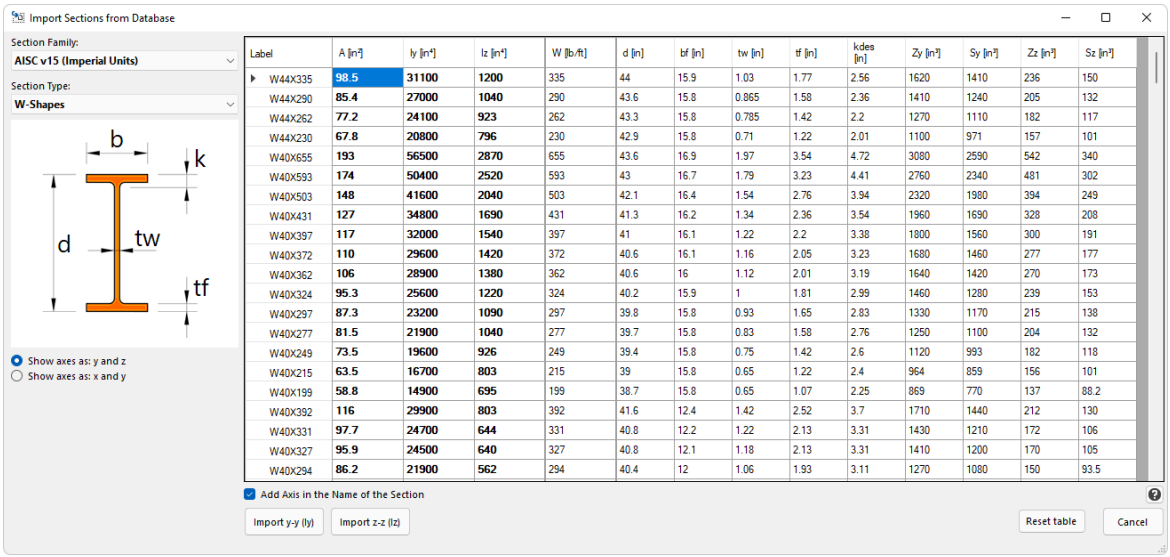



Click "**IMPORT**" to import sections from a Database. Currently, the following section Databases are supported by the program:

- **AISC v15 (Imperial Units)**, with 2091 sections (W-Shapes, M-Shapes, HP-Shapes, S-Shapes, C-Shapes, MC-Shapes, MT-Shapes, WT-Shapes, ST-Shapes, HSS_Rectangular, Angles_L, HSS_Round, PIPE, Double_Angles_2L)
- **AISC v15 (Metric Units)**, with 2091 sections (W-Shapes, M-Shapes, HP-Shapes, S-Shapes, C-Shapes, MC-Shapes, MT-Shapes, WT-Shapes, ST-Shapes, HSS_Rectangular, Angles_L, HSS_Round, PIPE, Double_Angles_2L)
- **European**, with 143 sections (IPE, IPE-A, IPN, HE-A, HE-B, HE-M, UPE)

By clicking **IMPORT** we get a new form.

(c) 2022 EngiLab PC




| Label | A [in²] | Iy [in⁴] | Iz [in⁴] | W [lb/ft] | d [in] | bf [in] | tw [in] | tf [in] | kdes [in] | Zy [in³] | Sy [in³] | Zz [in³] | Sz [in³] |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| W44X335 | 98.5 | 31100 | 1200 | 335 | 44 | 15.9 | 1.03 | 1.77 | 2.56 | 1620 | 1410 | 236 | 150 |
| W44X290 | 85.4 | 27000 | 1040 | 290 | 43.6 | 15.8 | 0.865 | 1.58 | 2.36 | 1410 | 1240 | 205 | 132 |
| W44X262 | 77.2 | 24100 | 923 | 262 | 43.3 | 15.8 | 0.785 | 1.42 | 2.2 | 1270 | 1110 | 182 | 117 |
| W44X230 | 67.8 | 20800 | 796 | 230 | 42.9 | 15.8 | 0.71 | 1.22 | 2.01 | 1100 | 971 | 157 | 101 |
| W40X655 | 193 | 56500 | 2870 | 655 | 43.6 | 16.9 | 1.97 | 3.54 | 4.72 | 3080 | 2590 | 542 | 340 |
| W40X593 | 174 | 50400 | 2520 | 593 | 43 | 16.7 | 1.79 | 3.23 | 4.41 | 2760 | 2340 | 481 | 302 |
| W40X503 | 148 | 41600 | 2040 | 503 | 42.1 | 16.4 | 1.54 | 2.76 | 3.94 | 2320 | 1980 | 394 | 249 |
| W40X431 | 127 | 34800 | 1690 | 431 | 41.3 | 16.2 | 1.34 | 2.36 | 3.54 | 1960 | 1690 | 328 | 208 |
| W40X397 | 117 | 32000 | 1540 | 397 | 41 | 16.1 | 1.22 | 2.2 | 3.38 | 1800 | 1560 | 300 | 191 |
| W40X372 | 110 | 29600 | 1420 | 372 | 40.6 | 16.1 | 1.16 | 2.05 | 3.23 | 1680 | 1460 | 277 | 177 |
| W40X362 | 106 | 28900 | 1380 | 362 | 40.6 | 16 | 1.12 | 2.01 | 3.19 | 1640 | 1420 | 270 | 173 |
| W40X324 | 95.3 | 25600 | 1220 | 324 | 40.2 | 15.9 | 1 | 1.81 | 2.99 | 1460 | 1280 | 239 | 153 |
| W40X297 | 87.3 | 23200 | 1090 | 297 | 39.8 | 15.8 | 0.93 | 1.65 | 2.83 | 1330 | 1170 | 215 | 138 |
| W40X277 | 81.5 | 21900 | 1040 | 277 | 39.7 | 15.8 | 0.83 | 1.58 | 2.76 | 1250 | 1100 | 204 | 132 |
| W40X249 | 73.5 | 19600 | 926 | 249 | 39.4 | 15.8 | 0.75 | 1.42 | 2.6 | 1120 | 993 | 182 | 118 |
| W40X215 | 63.5 | 16700 | 803 | 215 | 39 | 15.8 | 0.65 | 1.22 | 2.4 | 964 | 859 | 156 | 101 |
| W40X199 | 58.8 | 14900 | 695 | 199 | 38.7 | 15.8 | 0.65 | 1.07 | 2.25 | 869 | 770 | 137 | 88.2 |
| W40X392 | 116 | 29900 | 803 | 392 | 41.6 | 12.4 | 1.42 | 2.52 | 3.7 | 1710 | 1440 | 212 | 130 |
| W40X331 | 97.7 | 24700 | 644 | 331 | 40.8 | 12.2 | 1.22 | 2.13 | 3.31 | 1430 | 1210 | 172 | 106 |
| W40X327 | 95.9 | 24500 | 640 | 327 | 40.8 | 12.1 | 1.18 | 2.13 | 3.31 | 1410 | 1200 | 170 | 105 |
| W40X294 | 86.2 | 21900 | 562 | 294 | 40.4 | 12 | 1.06 | 1.93 | 3.11 | 1270 | 1080 | 150 | 93.5 |

Select entire rows from the table and then click "**Import y-y (Iy)**" or "**Import z-z (Iz)**" to import the section to the Model.

- "**Import y-y (Iy)**": Imports the Section Name, Area A and **Moment of Inertia Iy** (usually the strong axis of the section)
- "**Import z-z (Iz)**": Imports the Section Name, Area A and **Moment of Inertia Iz** (usually the weak axis of the section)

## 2.2.4 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:**

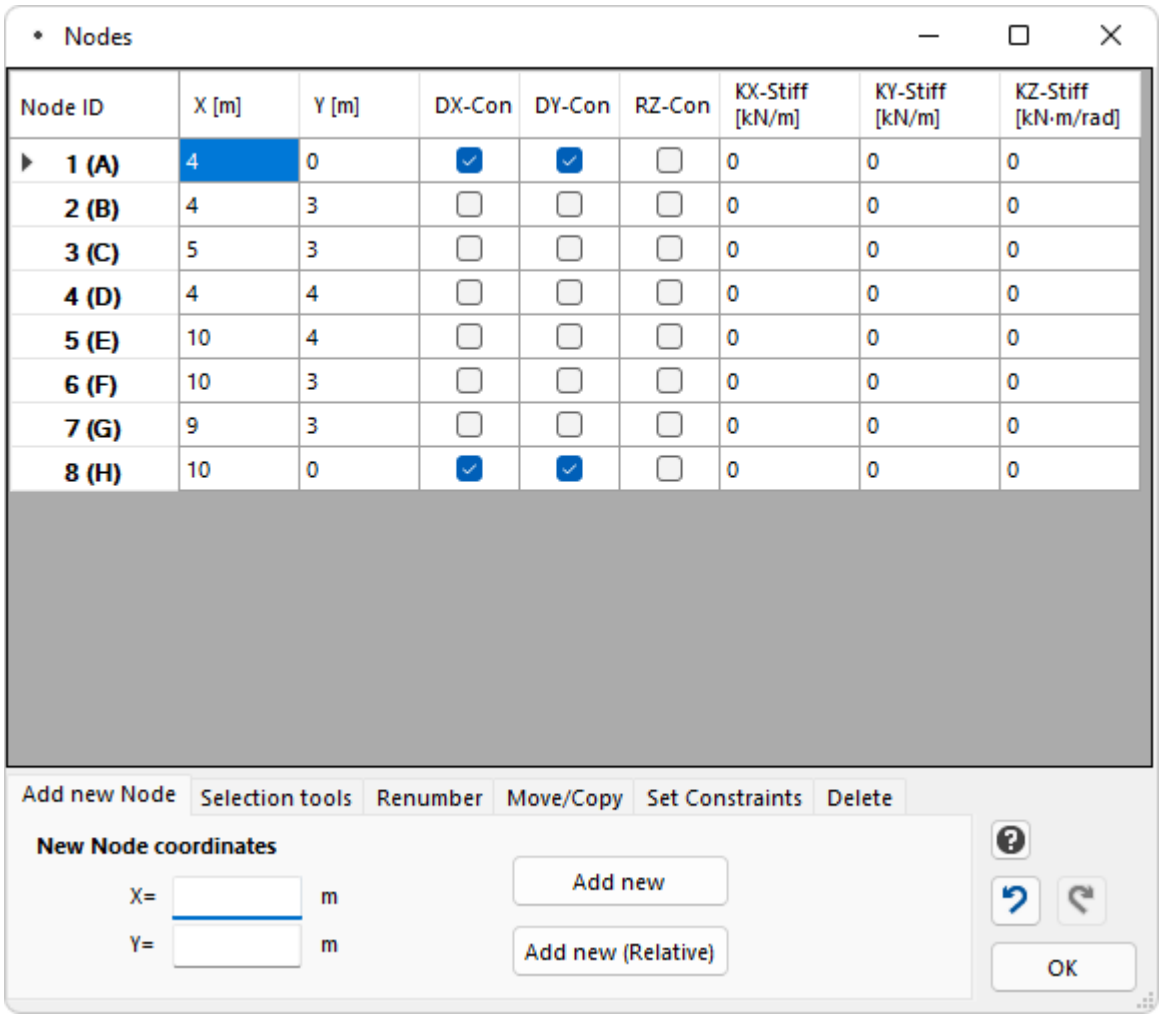
- Free to move (No support, No spring)
- Fixed (Fully constrained, not able to move at all)
- With a spring on (Given the spring stiffness, the spring provides a reaction force that is proportional to the displacement of the corresponding DOF).

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.




(c) 2022 EngiLab PC

### Coordinates

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

### Constraints

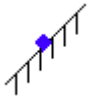
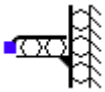
The properties **DX-Con**, **DY-Con**, **RZ-Con** determine whether the corresponding DOF (DX, DY or RZ) of the Node is constrained or not. DX and DY are the X and Y displacements of the node, respectively. RZ is the rotation around the Z axis.

Each DOF can be:

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

(c) 2022 EngiLab PC

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

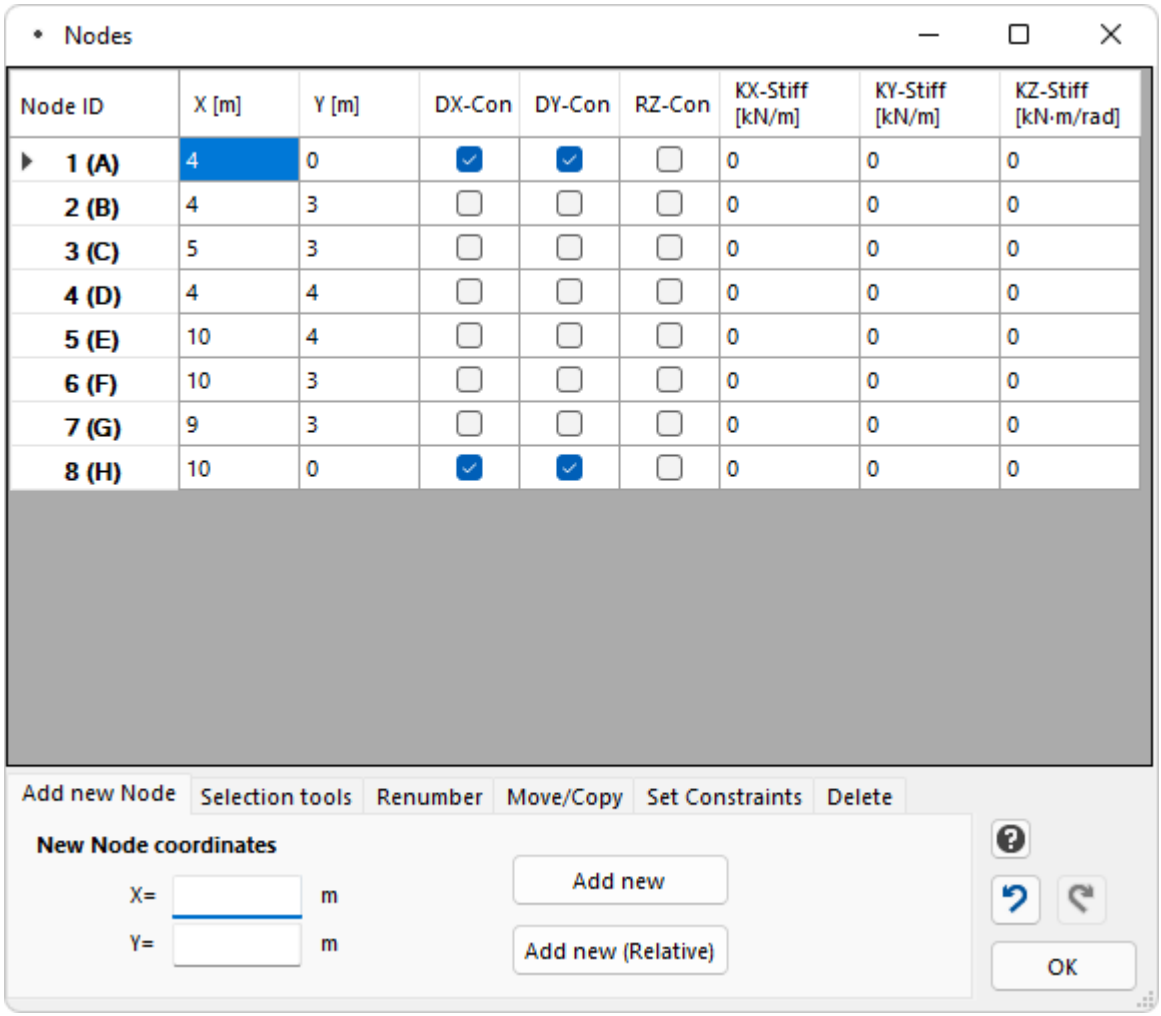
The properties **DX-Stiff**, **DY-Stiff**, **RZ-Stiff** determine the Stiffness of the Spring (Elastic constant) of the corresponding DOF. A spring provides a spring reaction force that is proportional to the displacement of the corresponding DOF. Each stiffness component has a real positive value, or 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

(c) 2022 EngiLab PC

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.




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.

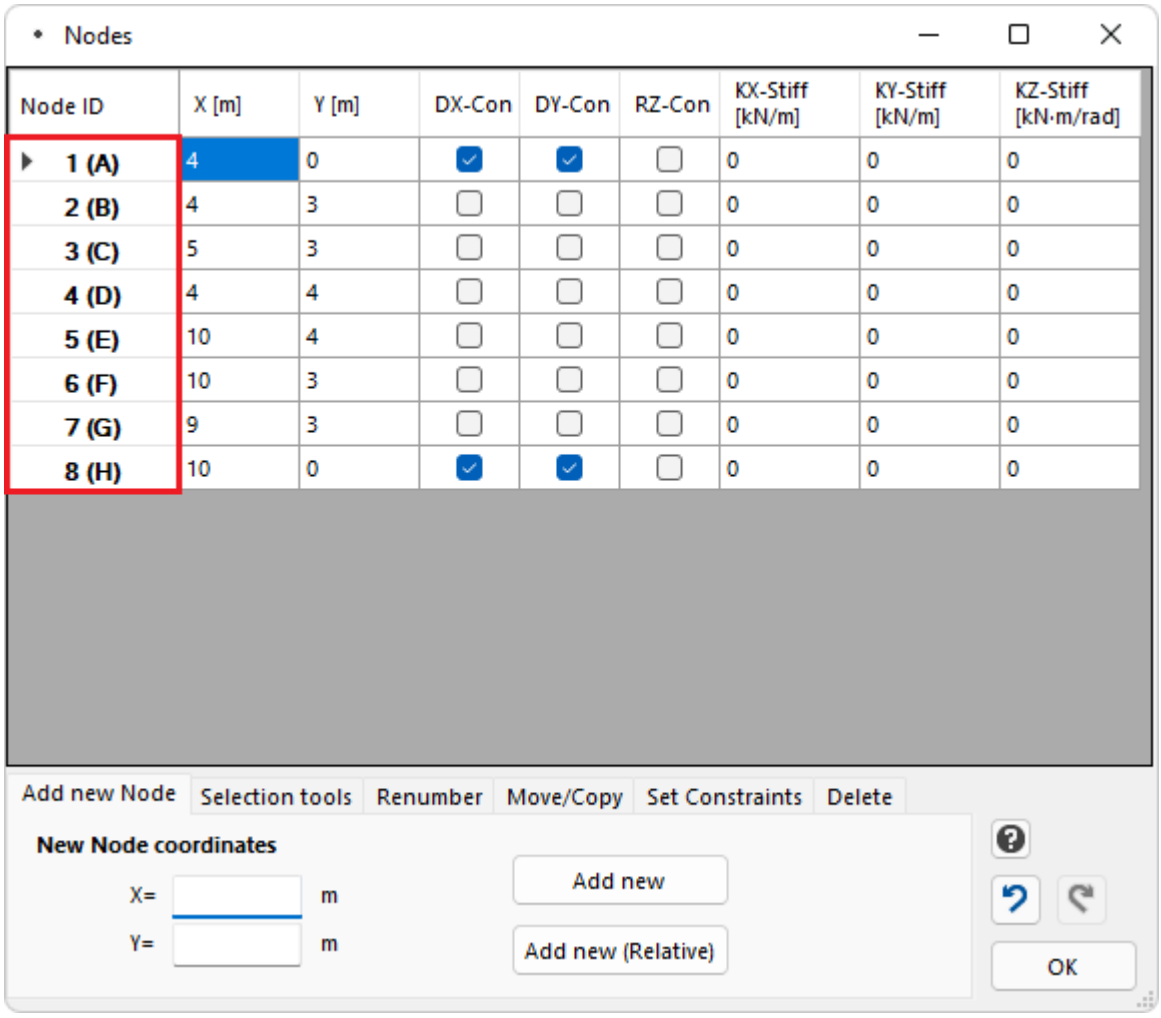
**Note**: If multiple Nodes are selected and the "Add new (Relative)" button is clicked, then the Node with the highest ID (of the selected ones) is taken into account as the reference Node for the definition of the new Node.

### 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)

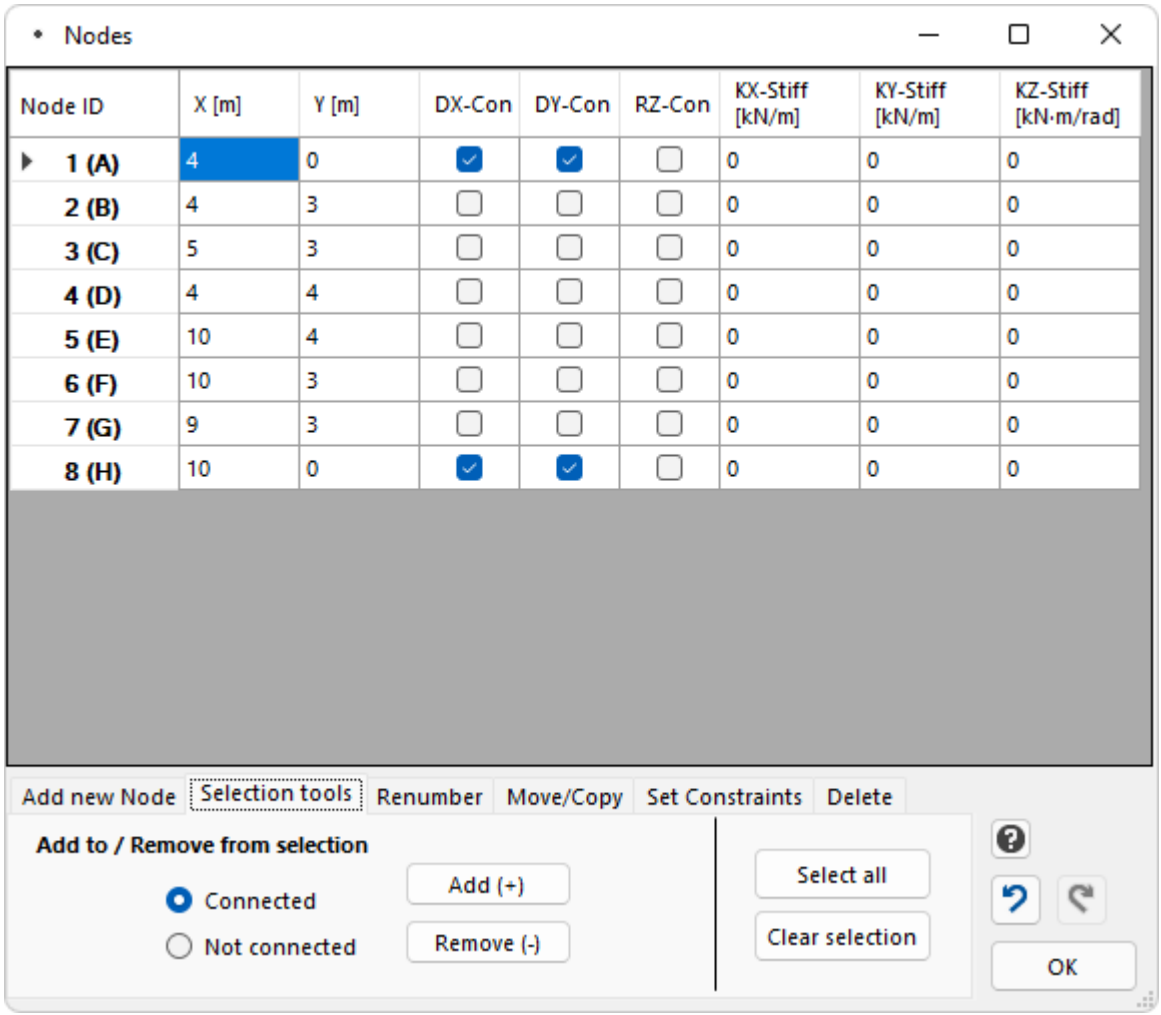
You can click on a row number to select one or more Nodes. Then you can apply actions to the selected Nodes (renumber, move/copy, set constraints, delete, etc).




### 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.

(c) 2022 EngiLab PC




## 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.

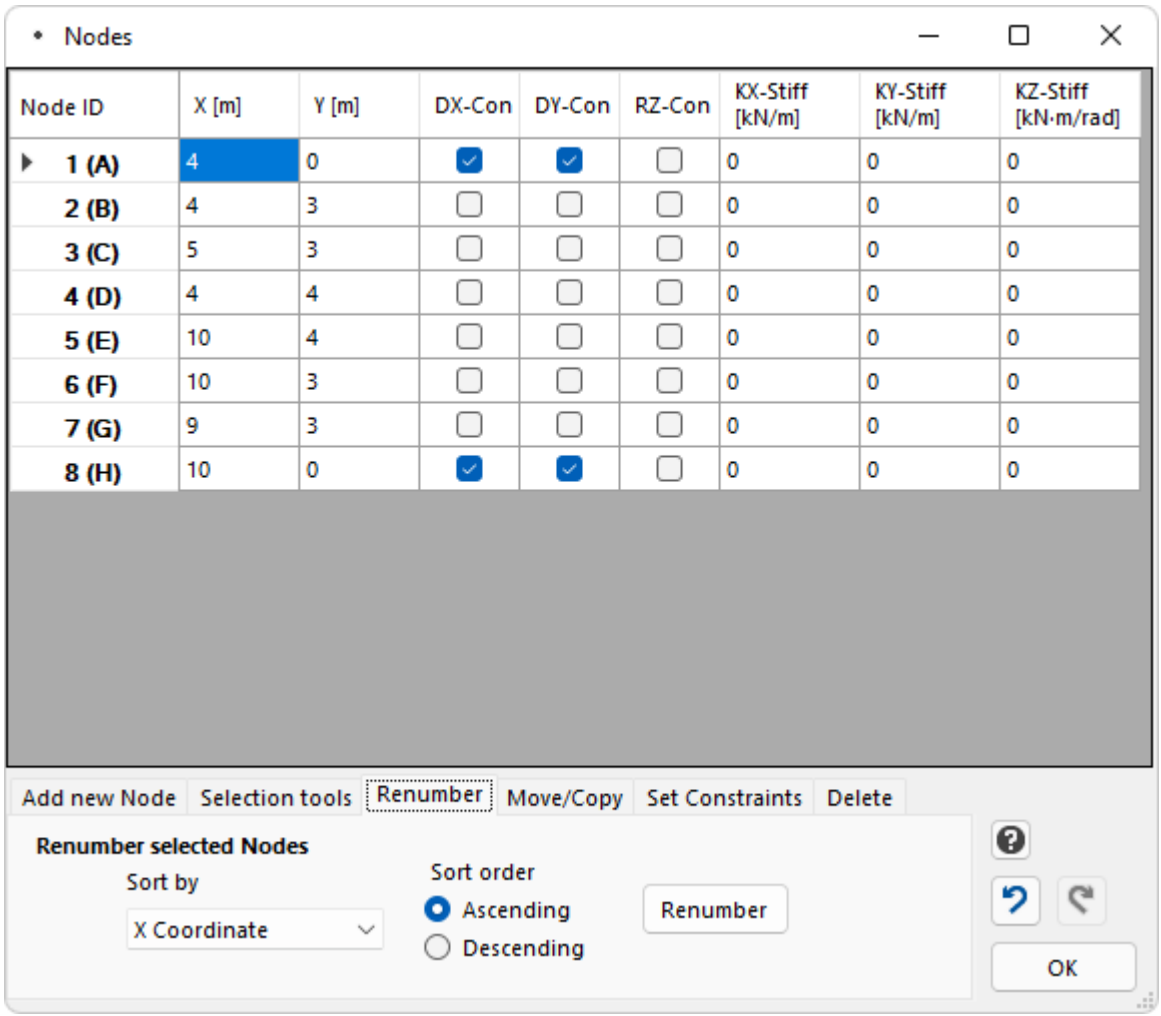
A tool is available for the renumbering of (the selected) Nodes, based on various criteria:

- X Coordinate
- Y Coordinate
- (X + Y)
- (X - Y)
- ($X^2$ + $Y^2$)

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.

(c) 2022 EngiLab PC

**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.

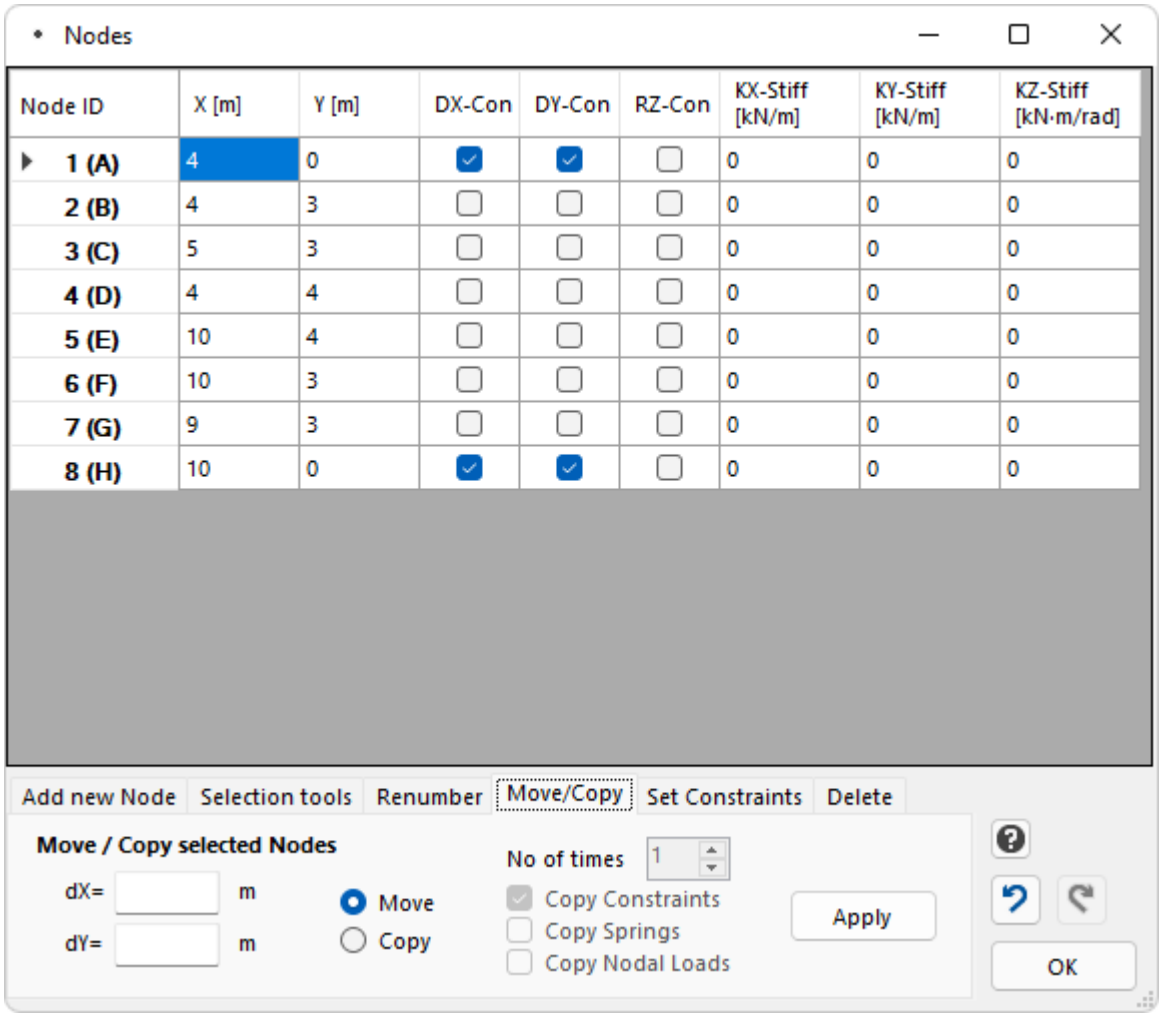



## Move/Copy Nodes

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}.

(c) 2022 EngiLab PC



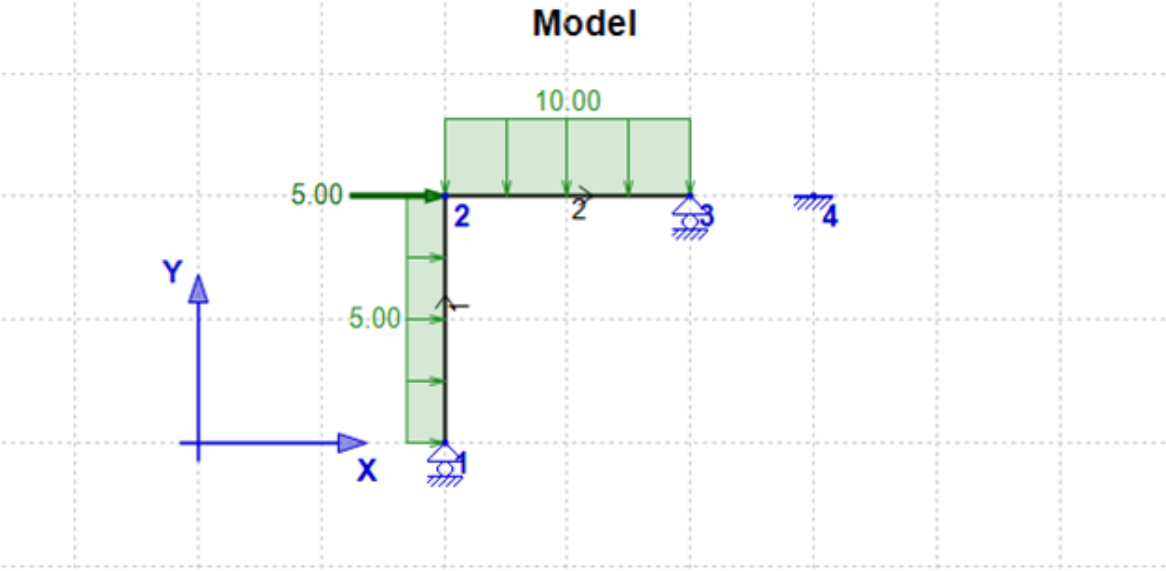

### Move Nodes

- 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.

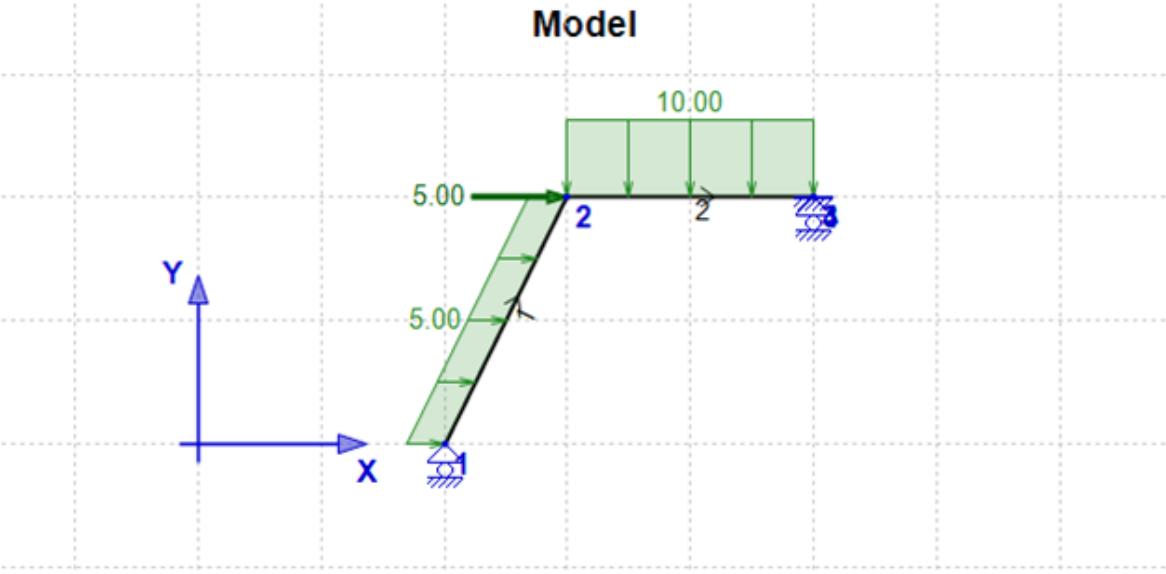
(c) 2022 EngiLab PC

- 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:




(c) 2022 EngiLab PC

- 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*2.24= 11.2 from 5*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).

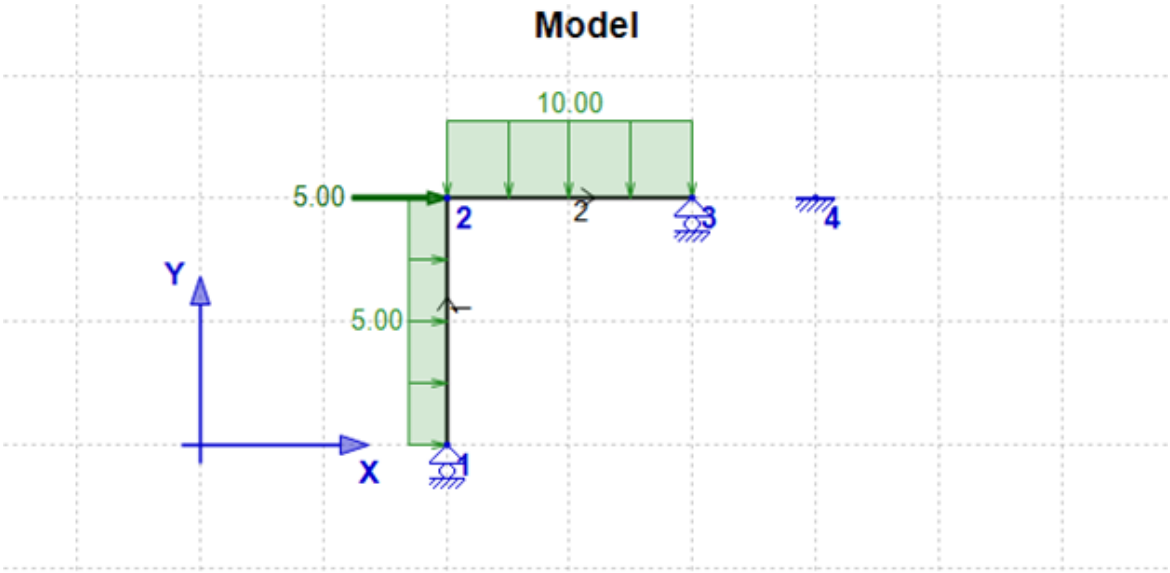
**Copy Nodes**

- 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.

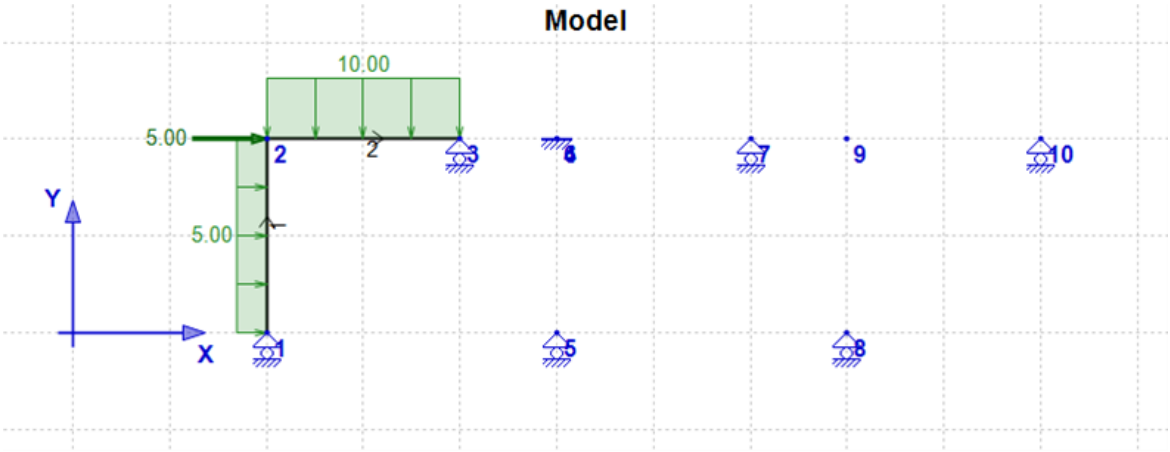
**Example 2 – Copying Nodes**

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:

(c) 2022 EngiLab PC




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:



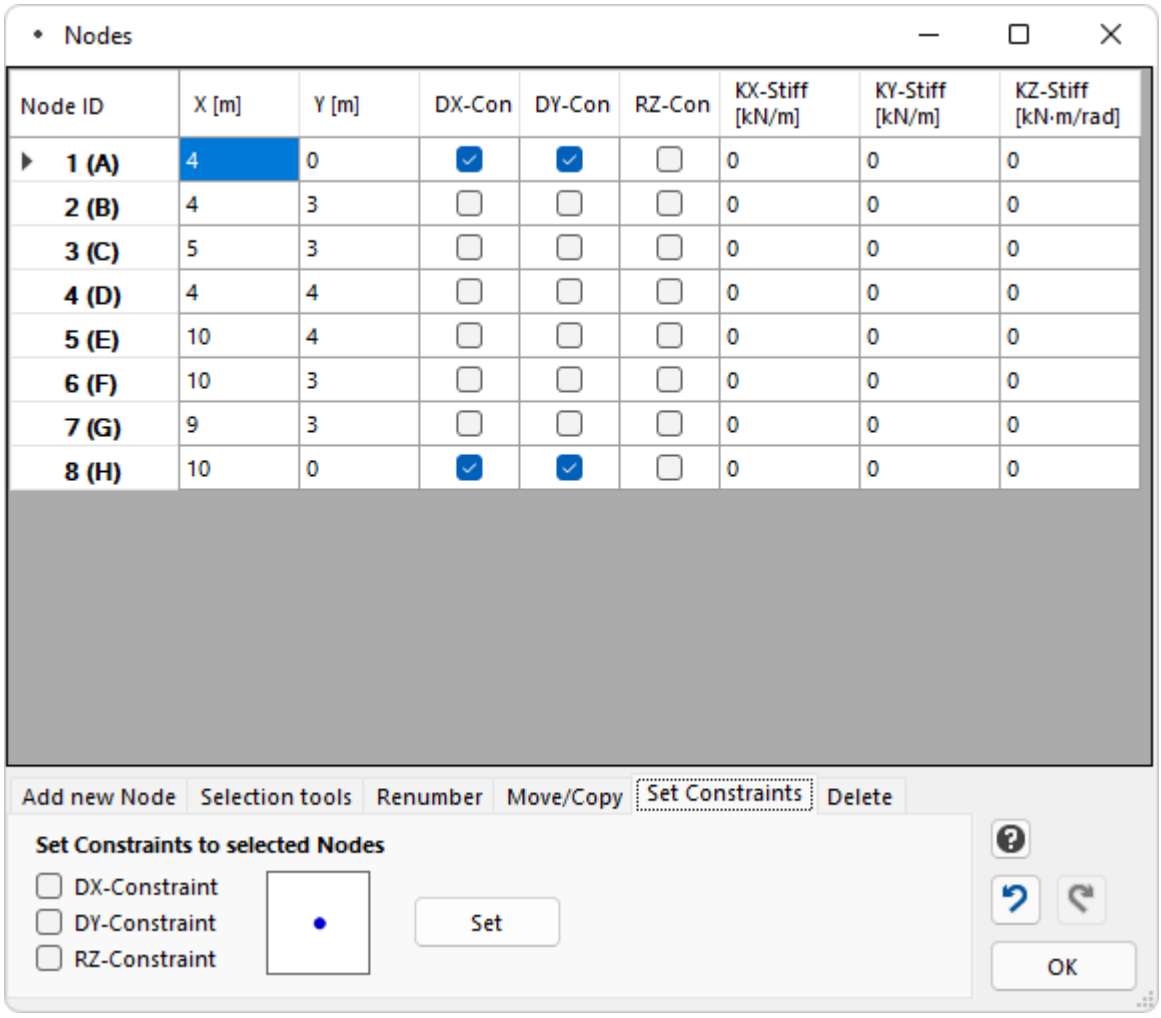

- Node 4 was already at (5, 2) and is not affected. New Node 6 is also created at the same location (5, 2).
- Constraints (and springs) are copied together with the Nodes.
- Nodal forces (or moments) are NOT copied together with the Nodes (see Node 2 and new Nodes 6 and 9).
- Copy Nodes does not affect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

## Set Constraints

(c) 2022 EngiLab PC

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.

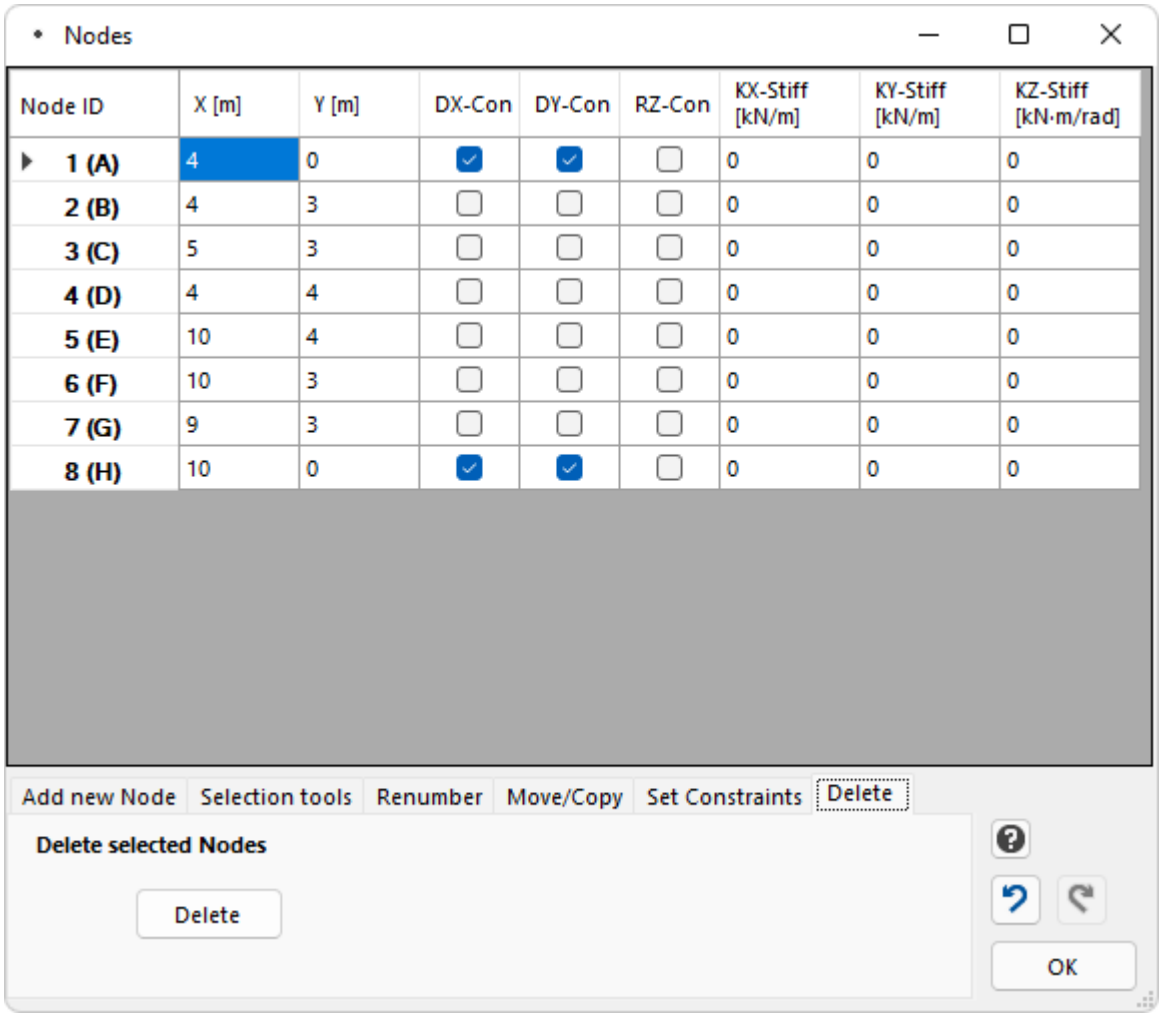
This tool is used for setting constraints on selected Nodes. You can set constraints for more than one Nodes at a time. First select the Nodes (rows) that you want to set the constraints on. Set the constraints by checking the DX, DY or RZ constraint checkbox. Finally, click the "Set" button to perform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




## 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.




## 2.2.5 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.

(c) 2022 EngiLab PC

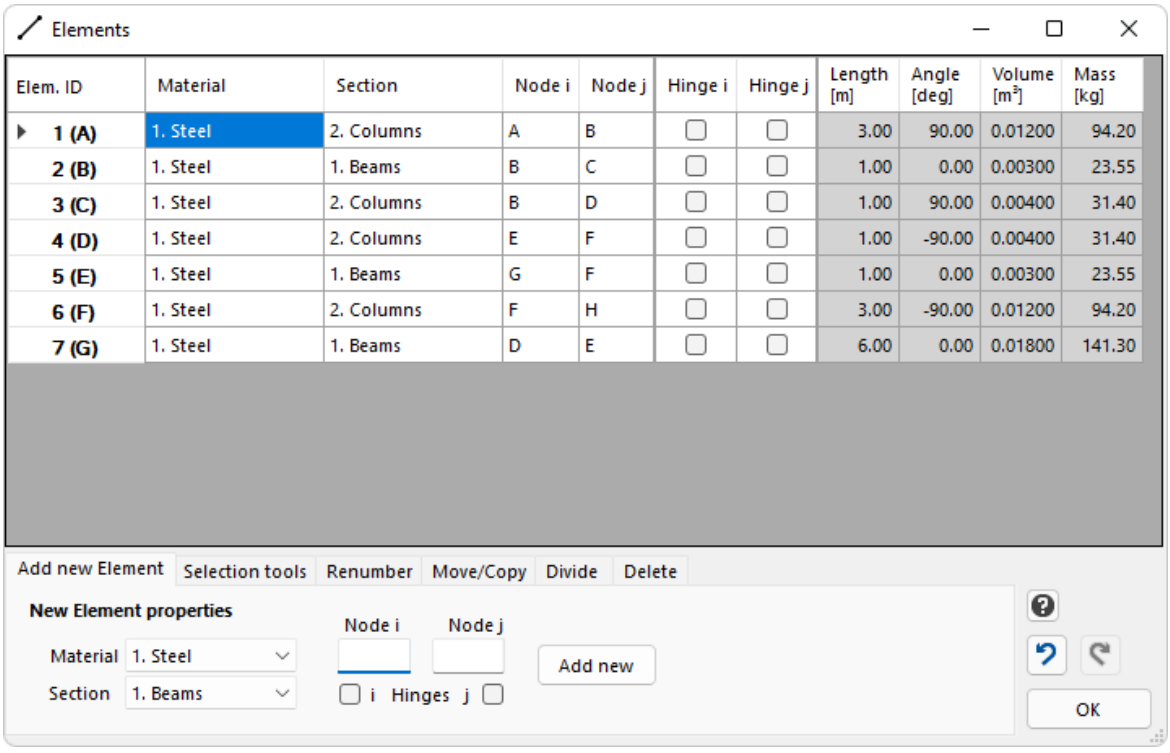
## Element properties

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

- **Material ID**
- **Section ID**
- **Node i ID** (Start Node)
- **Node j ID** (End Node)
- **Hinge i** (Hinge at Start Node i), Checked or Not Checked
- **Hinge j** (Hinge at End Node j), 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 Length units, for example m). It is calculated based on the locations of the Start (i) and End (j) Nodes, as $L=sqrt[(Xj-Xi)^2+(Yj-Yi)^2]$.
- **Angle:** Orientation Angle of the Element (in Angle units, for example deg), measured counter-clockwise from the horizontal. When the Angle is zero, the Element points to the right (+X Global axis).
- **Volume**: Volume of the Element (in Volume units, for example $m^3$). It is calculated as the Length of the Element times the Area of the Cross section, V=L*A.
- **Mass**: Mass of the Element (in Mass units, for example kg). It is calculated as the Volume of the Element times the Density of the Material, M=V*d.



### Add new Element

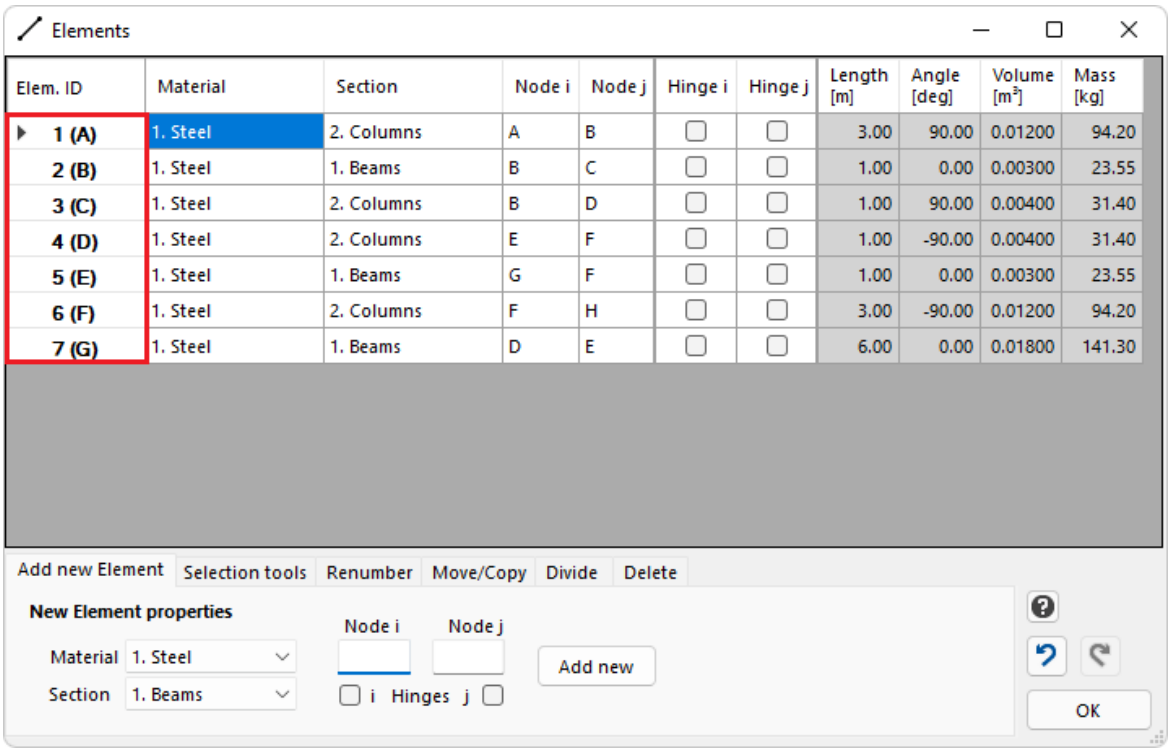
To add a new Element, select the Material and the Section and type the Start and End Nodes of the new Element. You can optionally add hinges to the new Element. Then click the "Add new" button.

### 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)

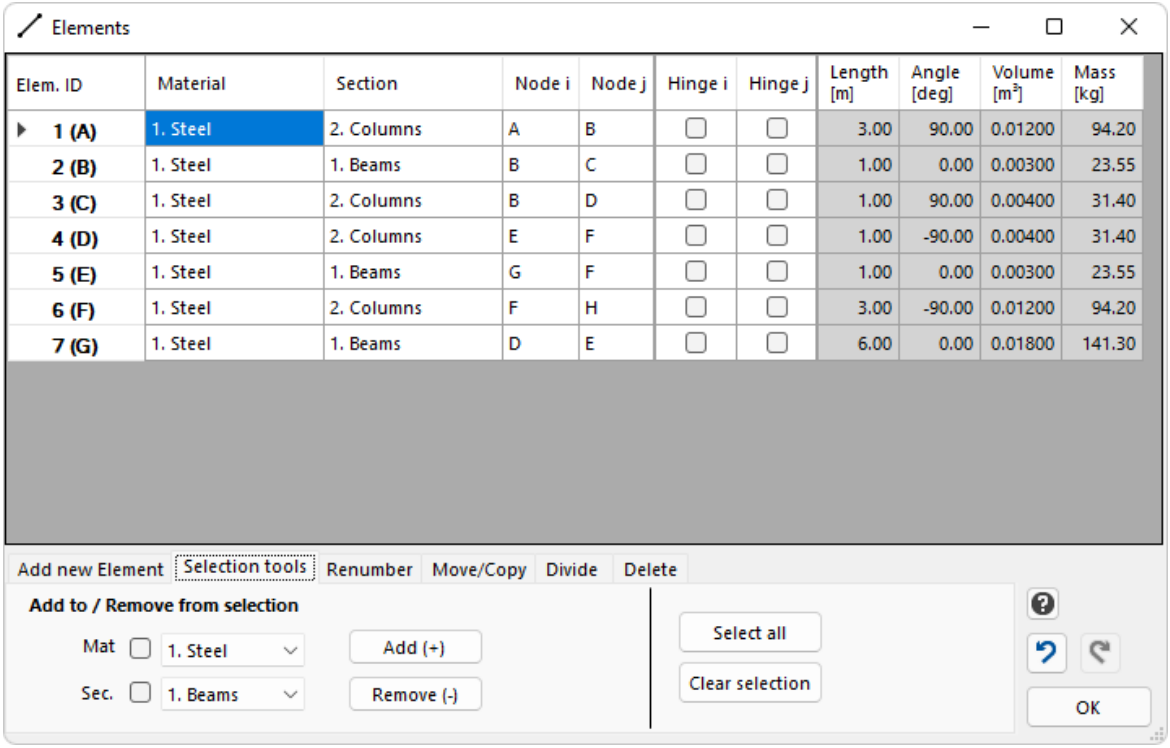
You can click on a row number to select one or more Elements. Then you can apply actions to the selected Elements (renumber, move/copy, divide, delete, etc).




### Selection tools

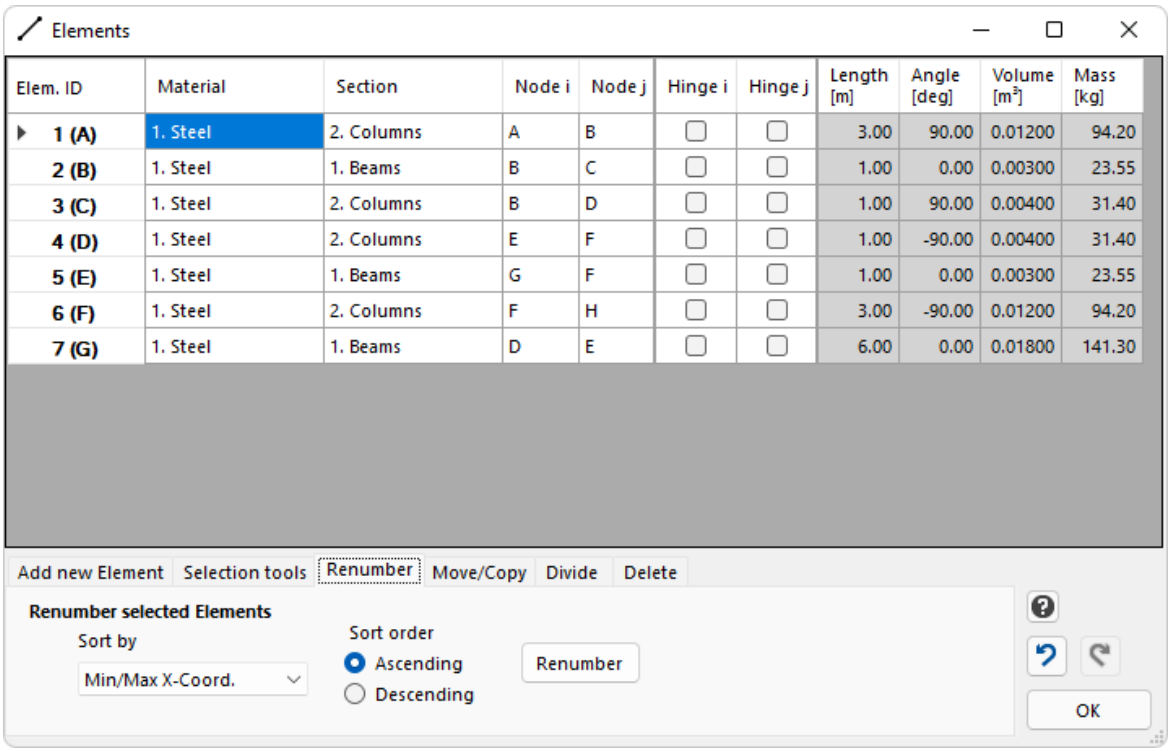
This tab provides tools for the selection of Elements. For example, you can select (add to selection) or deselect (remove from selection) all the Elements that are assigned a specific Material and/or a specific Section. This is useful when for example you need to Delete all the Elements with a specific Section. You can also Select all Elements, or Clear the selection.

(c) 2022 EngiLab PC




## 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.




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.

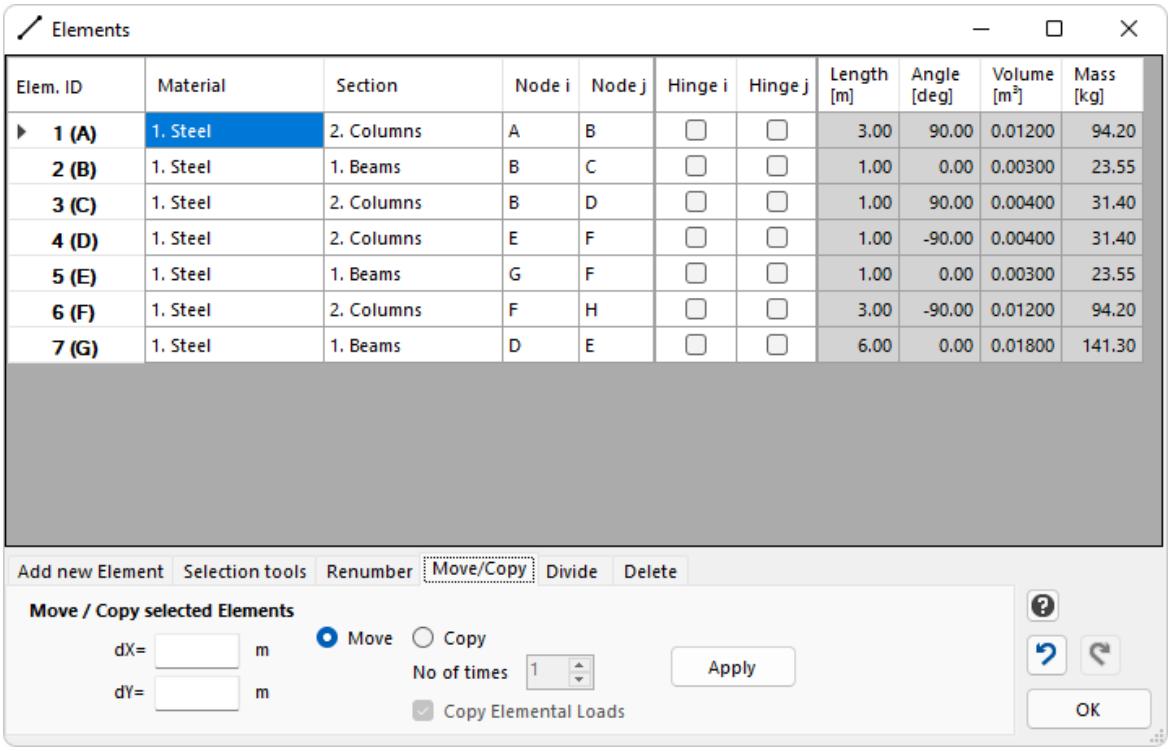
(c) 2022 EngiLab PC

**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}.



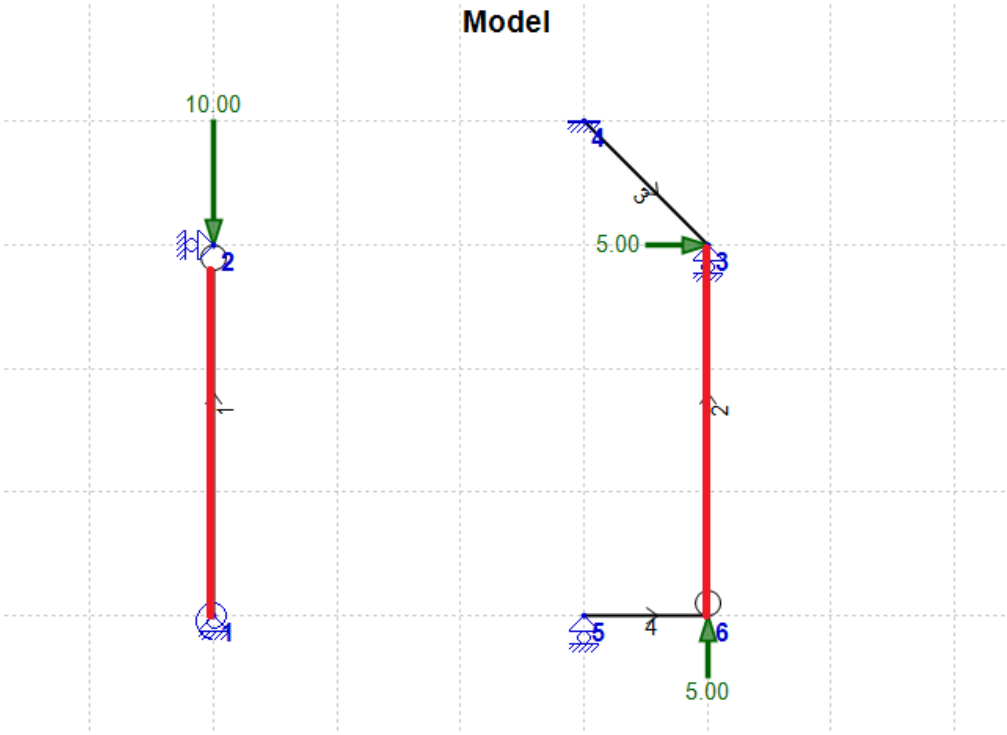

### Move Elements

- 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.

(c) 2022 EngiLab PC

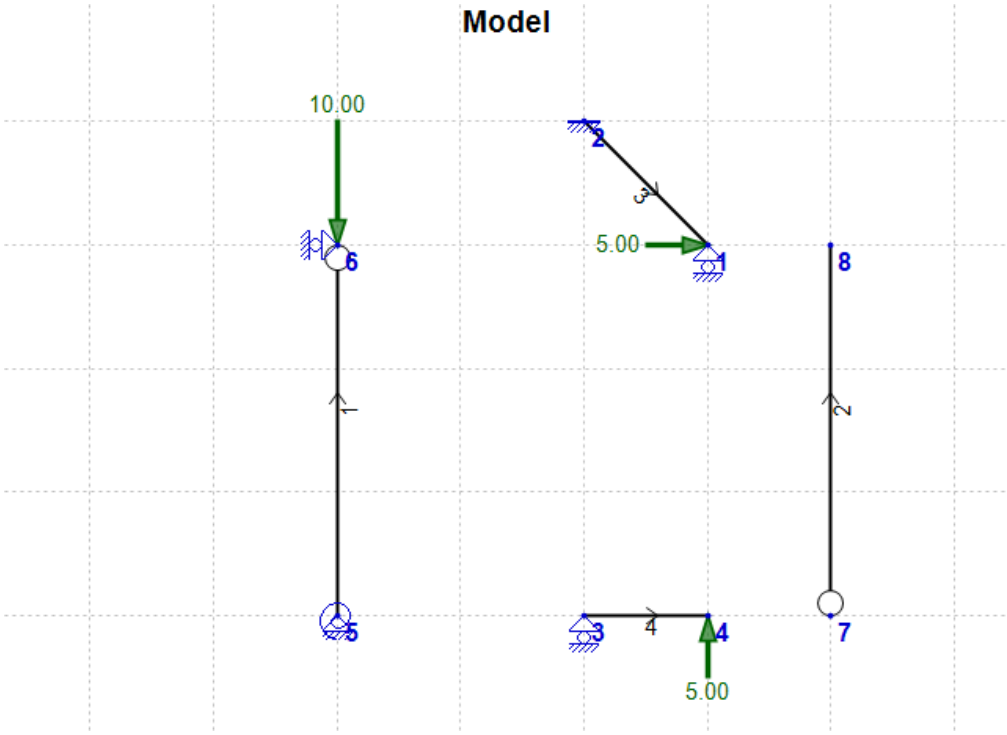
- If an Element is moved that was previously connected to other Elements via common Nodes (Elements that are not selected to move also), then the Element is detached from the other Elements and new Nodes are generated to define its new position. The previous Nodes stay connected with the other Elements, 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:

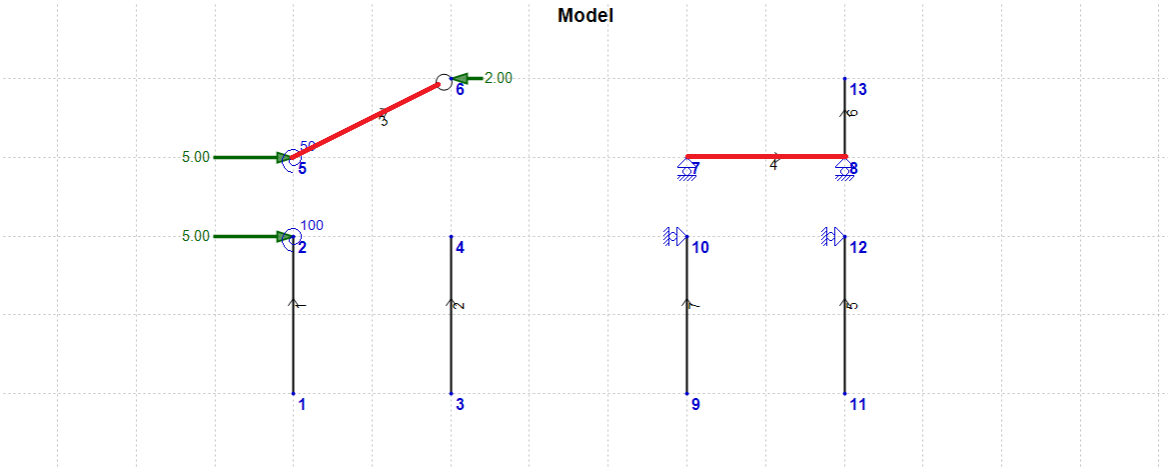
(c) 2022 EngiLab PC



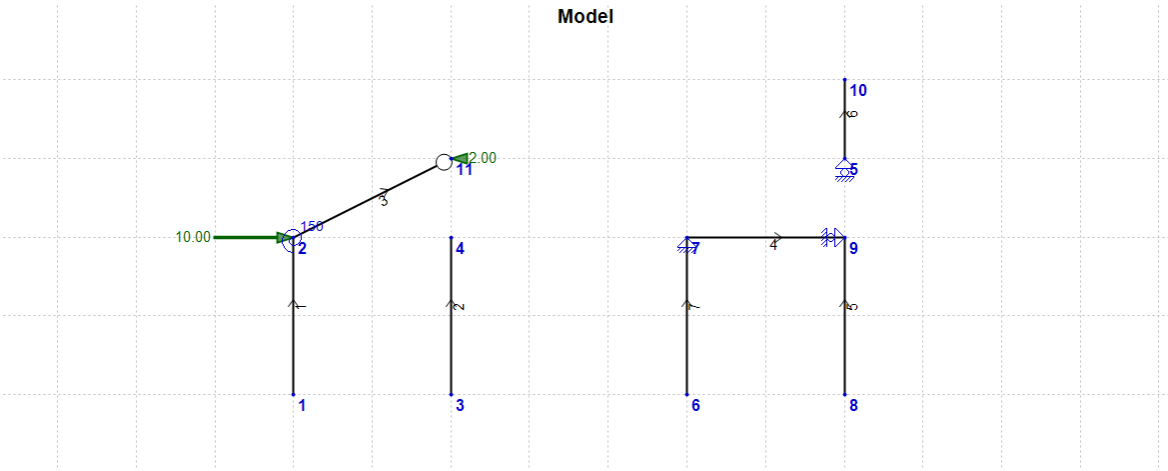

- Element 1 is moved together with its Nodes.
- Element 2 is detached from Elements 3 and 4 and it is now on its own, not connected to other Elements.
- 2 new Nodes have been automatically generated for this (Nodes 7 and 8 in the second figure).
- Some Node renumbering has occurred as a result of the 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**

(c) 2022 EngiLab PC




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

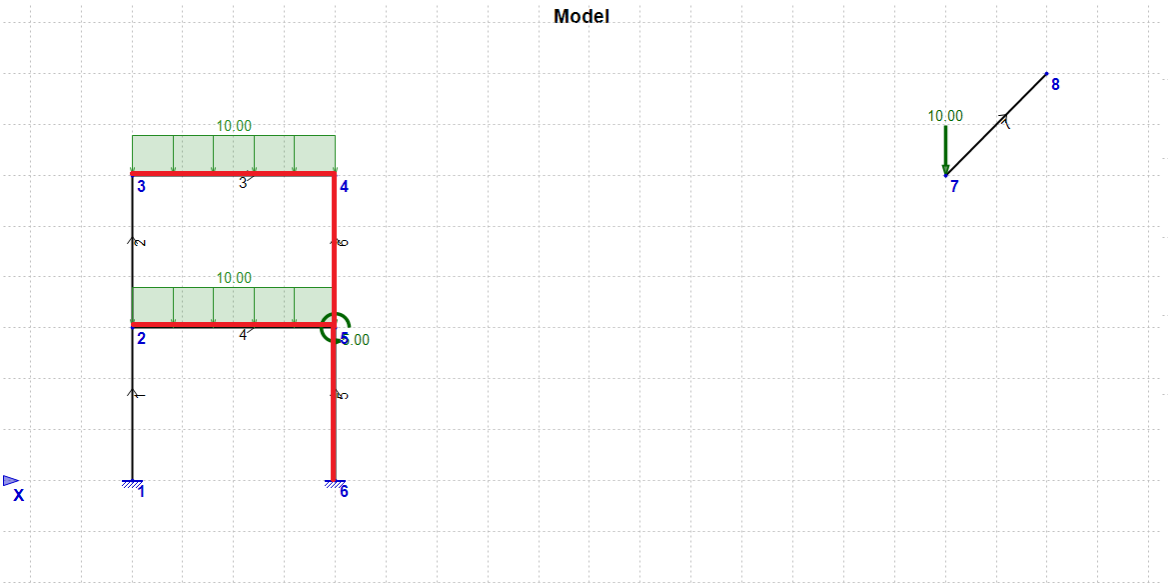
- Multiple Elements (one or more) can be selected to be copied at once.
- Copy Elements can be applied multiple times (up to 50) with a single click.

(c) 2022 EngiLab PC

- 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.

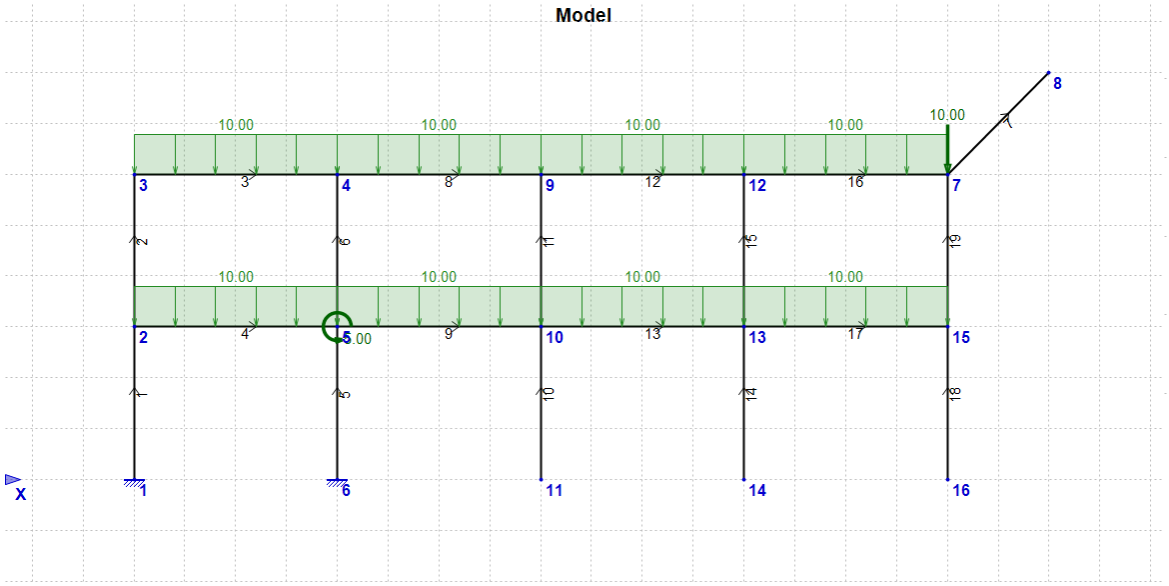
### Example 3 – Copying Elements

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:

(c) 2022 EngiLab PC



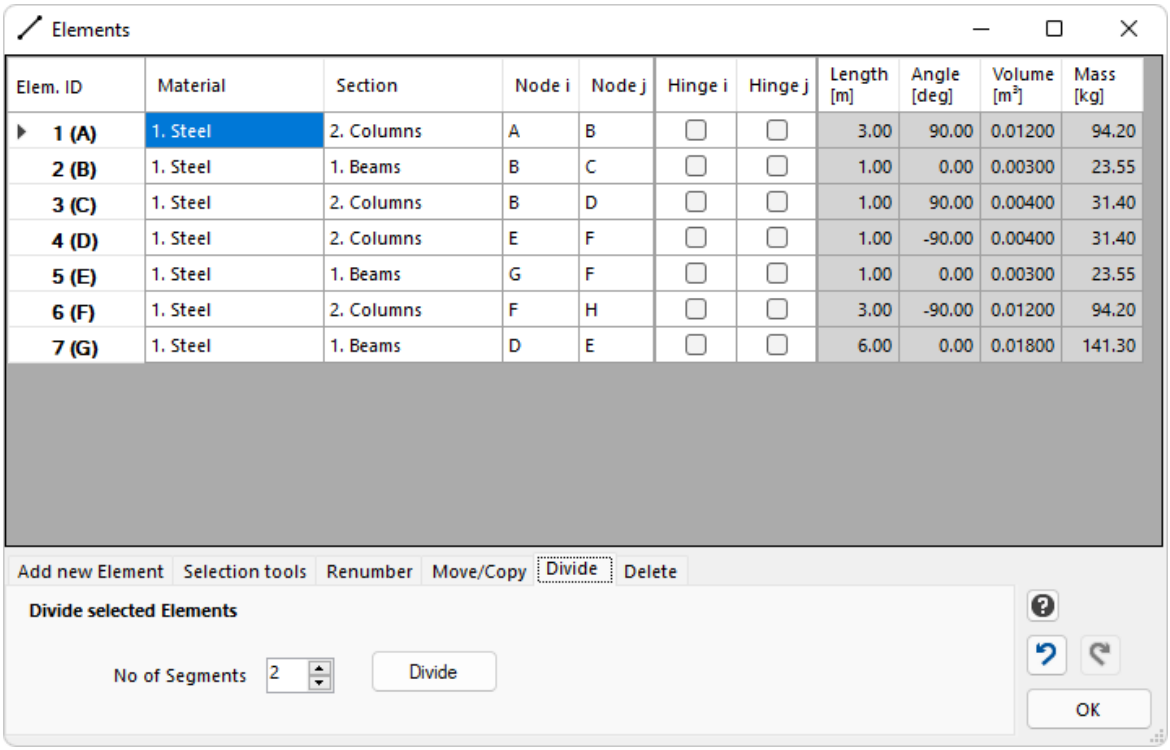

- New Elements and Nodes are created and we now have a fully connected frame structure.
- Node 7 was already at (20, 6) and it is used for the connections of Elements 16 and 19. No new Node is created there, the old Node is instead used.
- Elemental Loads are copied together with the Elements, because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

## Divide Elements

(c) 2022 EngiLab PC

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.

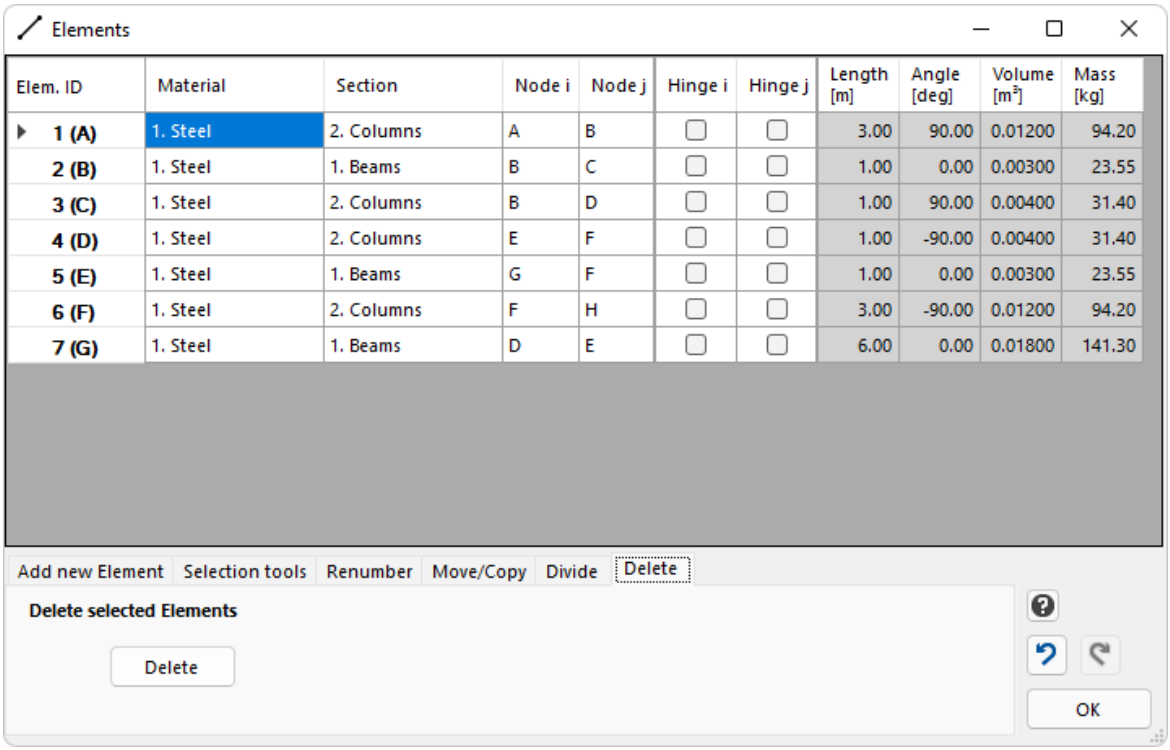
Select the Elements (rows). Type the number of the division segments (up to 40) and then click the "Divide" button. You can Divide more than one Elements at a time.



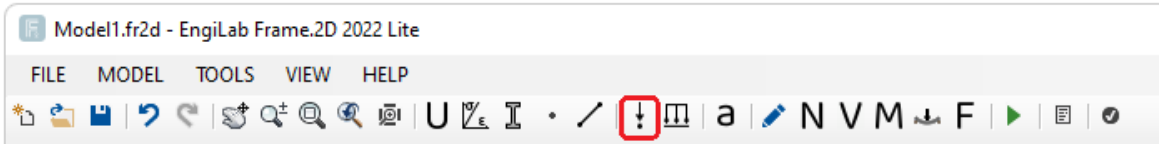

## Delete Elements

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.




## 2.2.6 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.

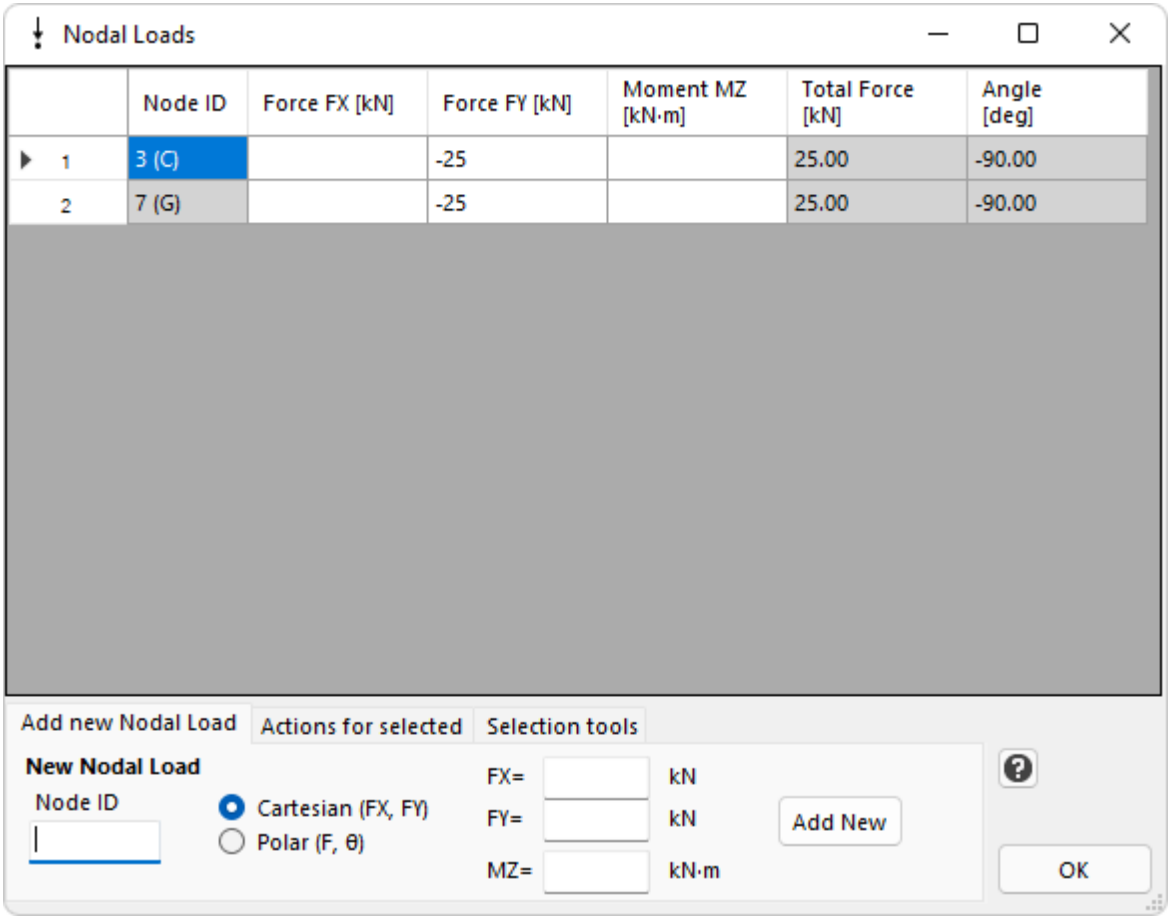
The **Nodal Load** properties are the Loads in each Degree Of Freedom (DOF) of the Node (Sign convention: Global axes):

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

(c) 2022 EngiLab PC

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

- **Total Force**: The magnitude of the Total Force acting on the Node (always positive), which is calculated as TotalForce=sqrt($FX^2$+$FY^2$)
- **Angle**: The Angle defining the orientation of the Total Force vector. Angles are measures counter-clockwise from the horizontal: Angle 0 means that the Total Force points to the right (positive X-Global Axis). Angle 90 deg means that the Total Force points upwards (positive Y-Global Axis), etc. Angle -90 deg means that the Total Force points downwards (negative Y-Global Axis), etc.



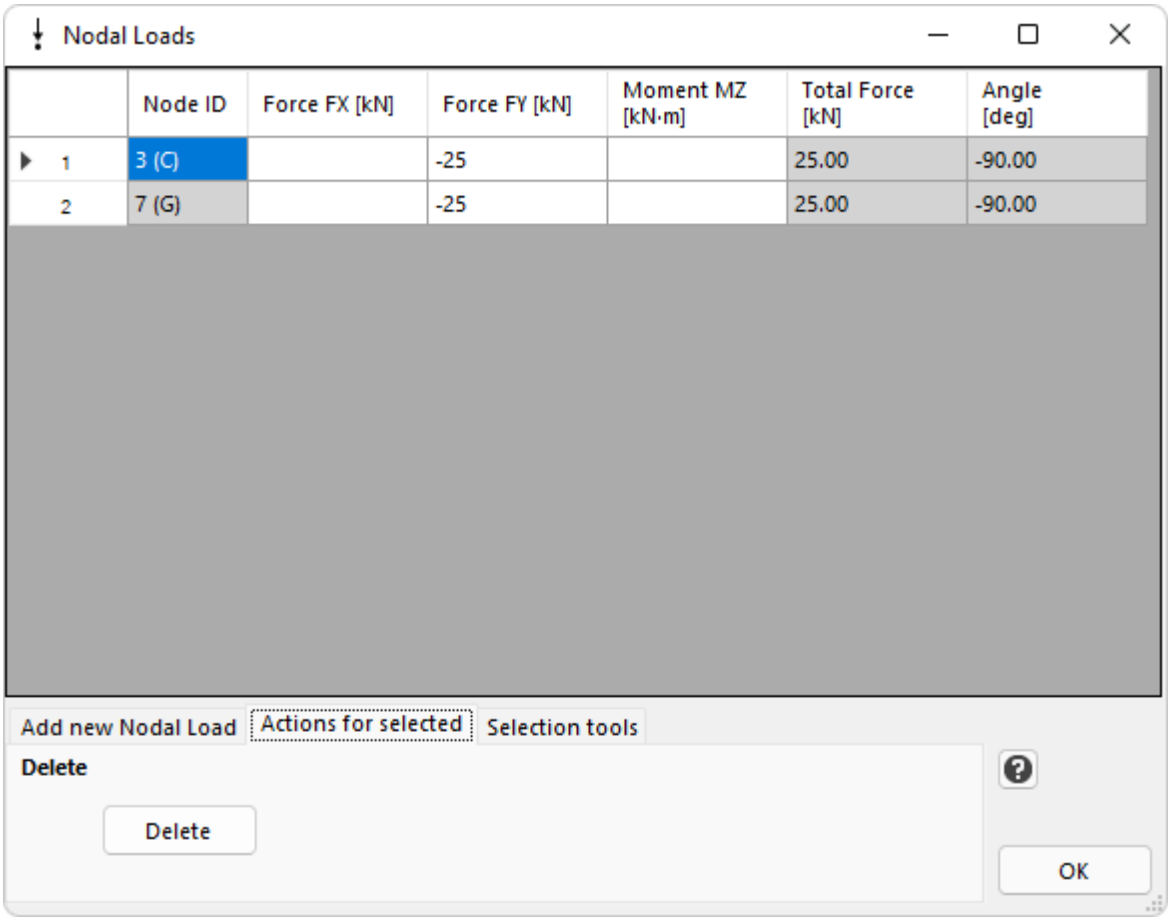

## Add new Nodal Load

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.

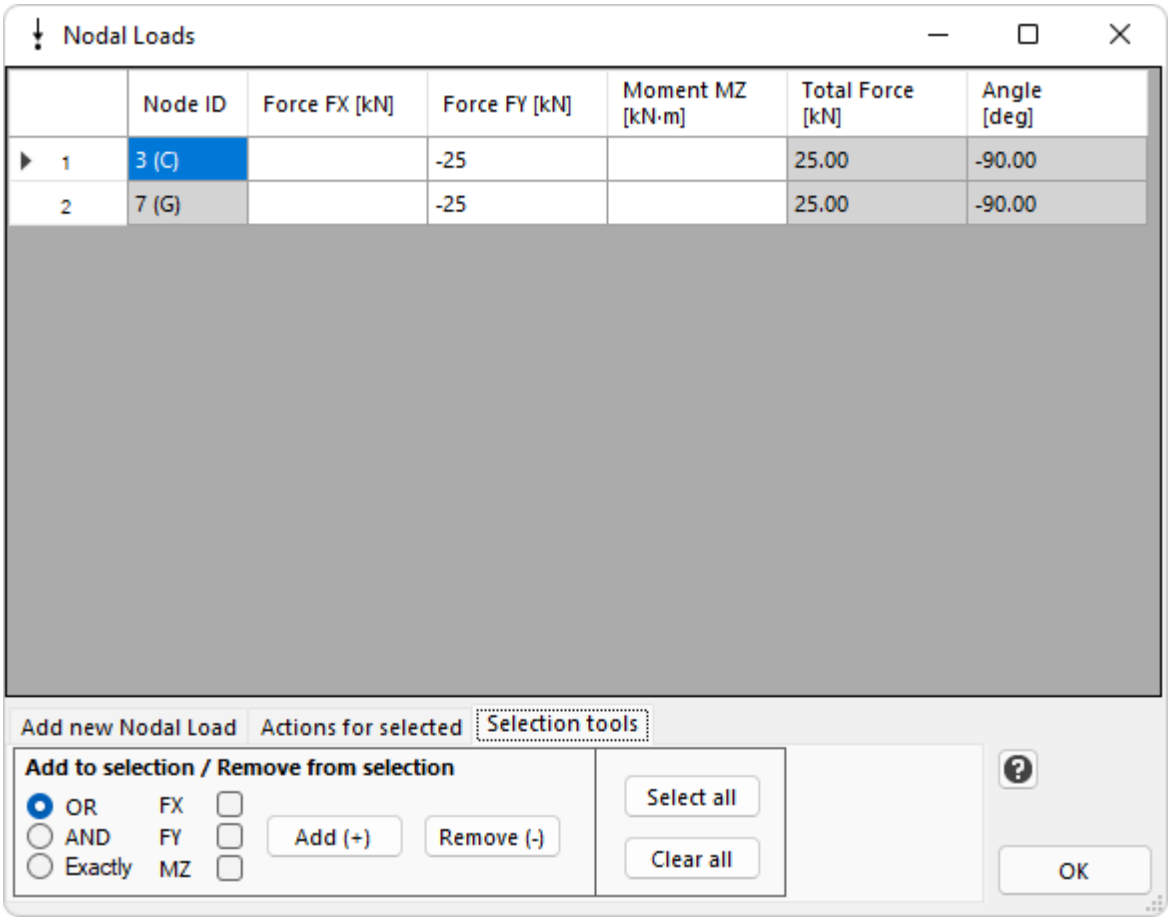
- **Delete Nodal Load(s)**. Click the "Delete" button. You can Delete more than one Nodal Loads at a time.




(c) 2022 EngiLab PC

### 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.




## 2.2.7 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.

(c) 2022 EngiLab PC

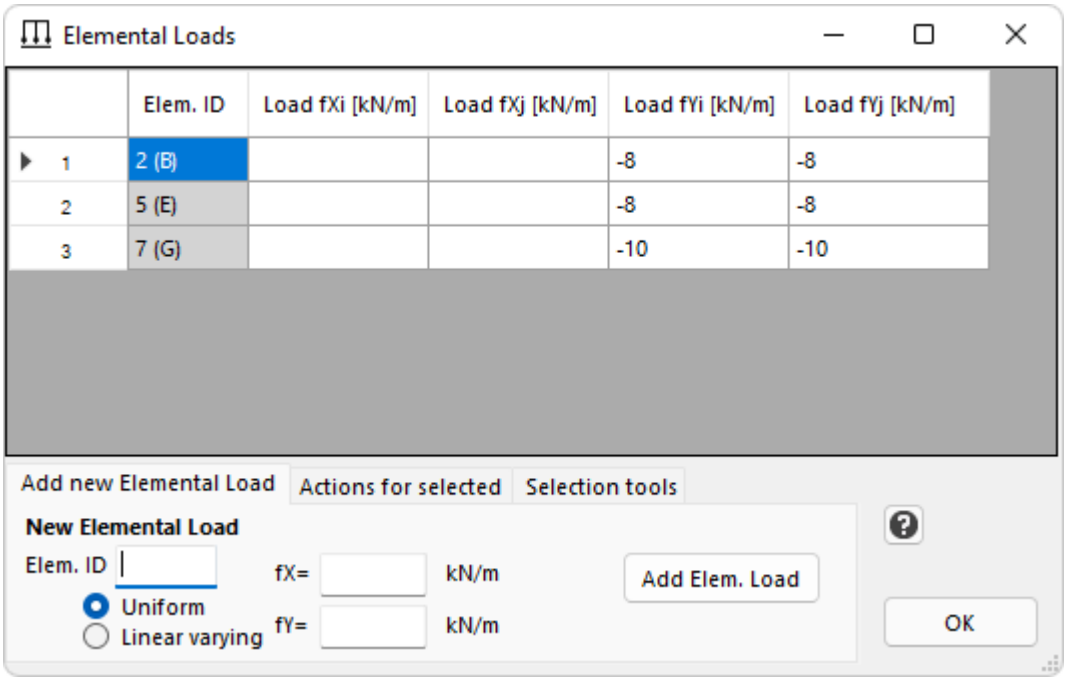
**The Elemental Load properties are the following:**

### Elemental Load in X-Direction

- **Force fXi**: Start value (at start i) of the X-Linear varying Load acting along the element, positive when pointing to the right
- **Force fXj**: 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 FYi**: Start value (at start i) of the Y-Linear varying Load acting along the element, positive when pointing up
- **Force FYj**: End value (at end j) of the Y-Linear varying Load acting along the element, positive when pointing up




## Add new Elemental Load

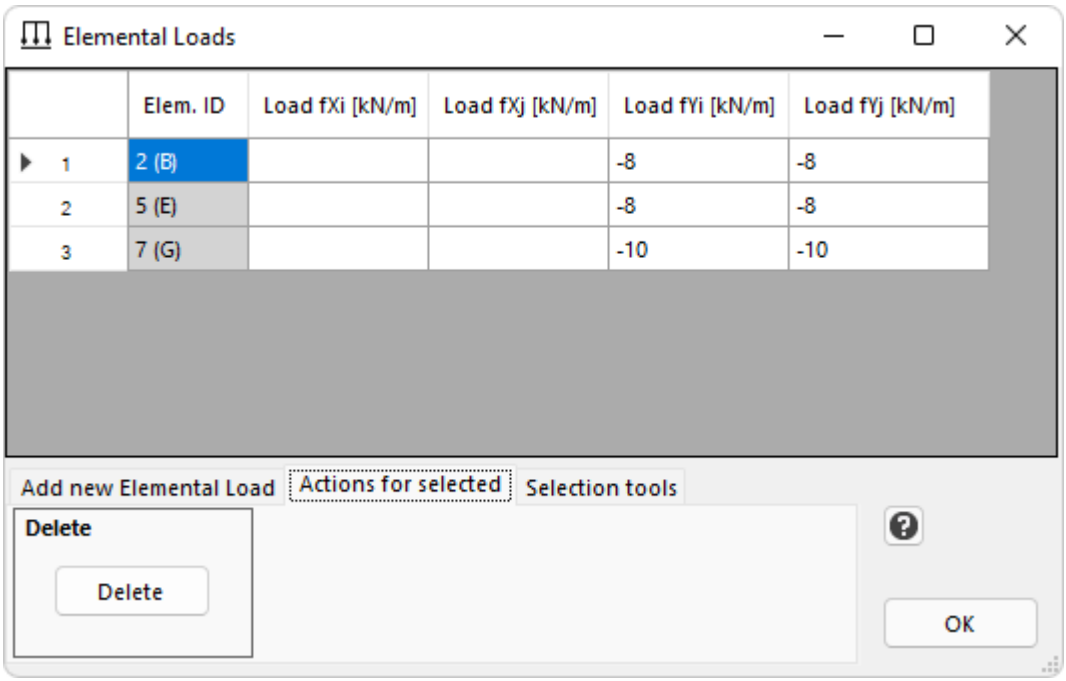
To add a new Elemental Load, first specify the Element ID. Then select Uniform or Linear varying Load. In general, all Elemental Loads are considered as linear varying loads by the program, but if you select a uniform load then the program will ask only for two values fX=fXi=fXj and fY=fYi=fYj. If you select Linear varying load, then the program will ask for all four values, fXi, fXj, fYi and fYj. After you have finished with the loads, then click the "Add Elemental Load" button.

(c) 2022 EngiLab PC

### Actions for selected Elemental Loads

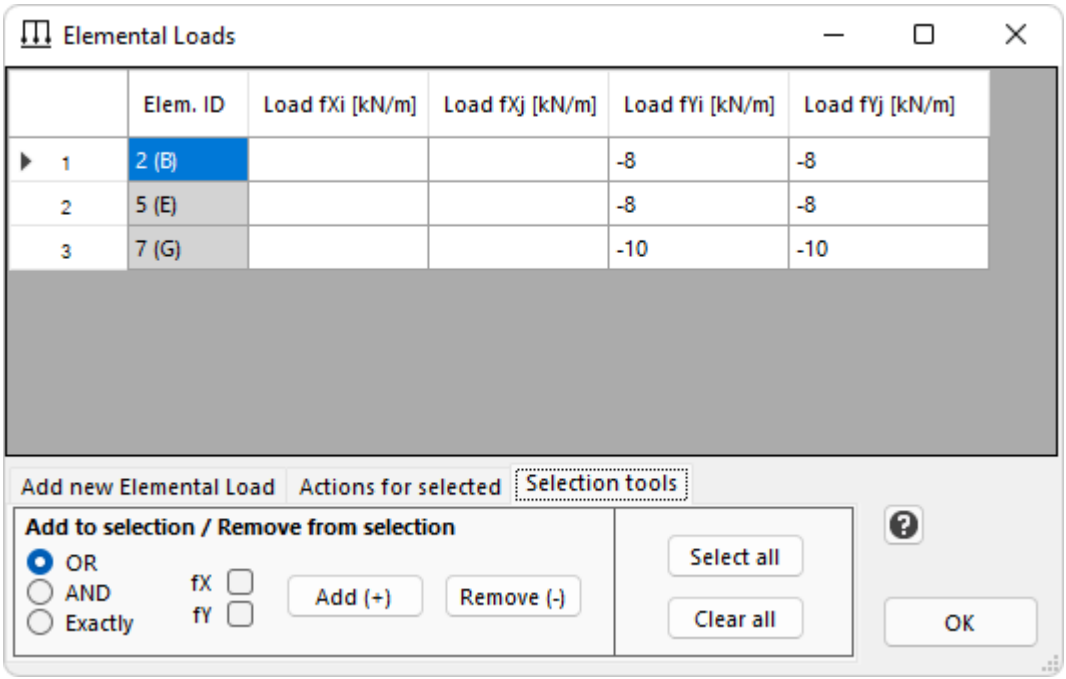
In order to perform these actions, **you have to first select the entire row(s) of the corresponding Elemental Load(s)** and then apply the action.

- **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 fX and/or fY values.



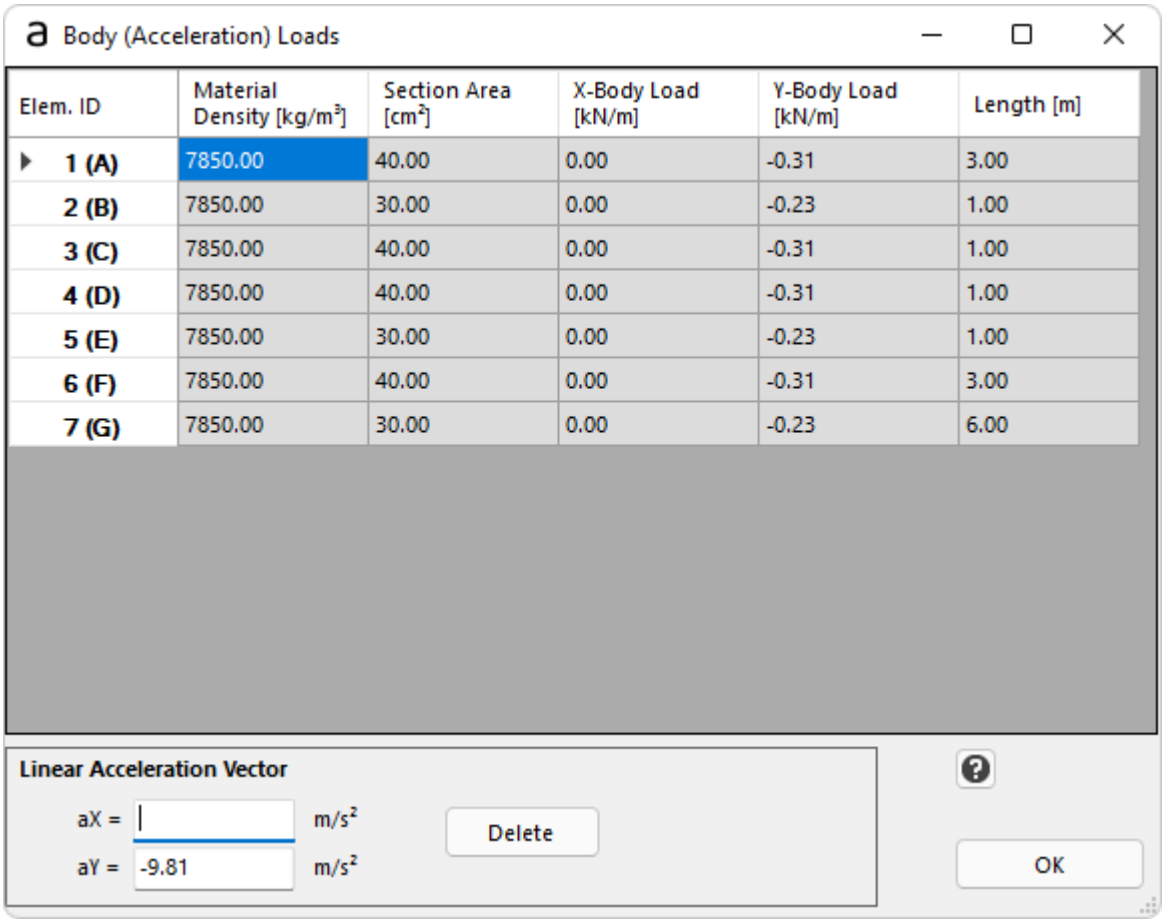

## 2.2.8 Body (Acceleration) Loads




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 aX and aY 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).

(c) 2022 EngiLab PC




## 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** Units.

**Example**: For a consistent system of unis: 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.

(c) 2022 EngiLab PC

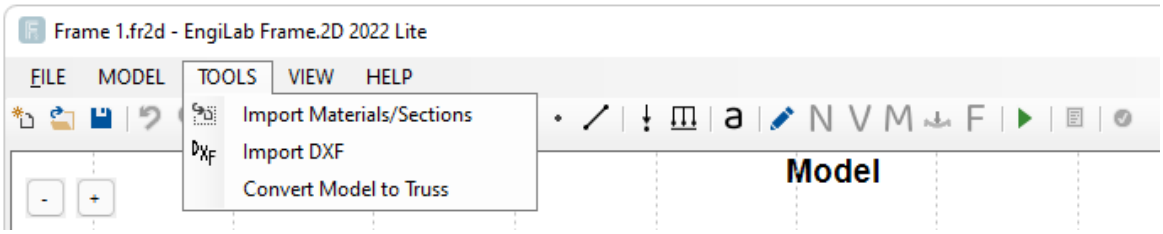
*This page intentionally left blank.*

# Chapter




## Tools

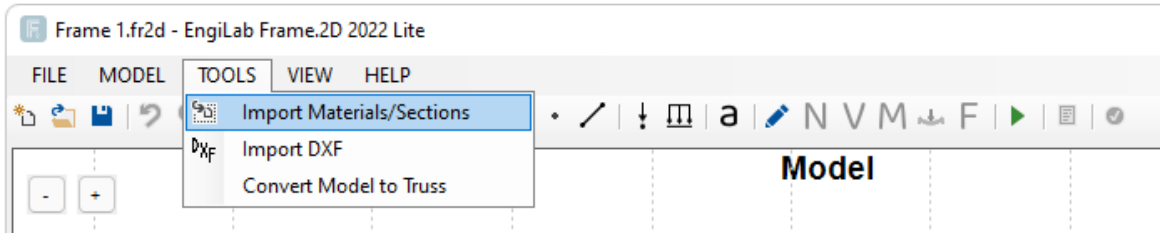
# 3 Tools



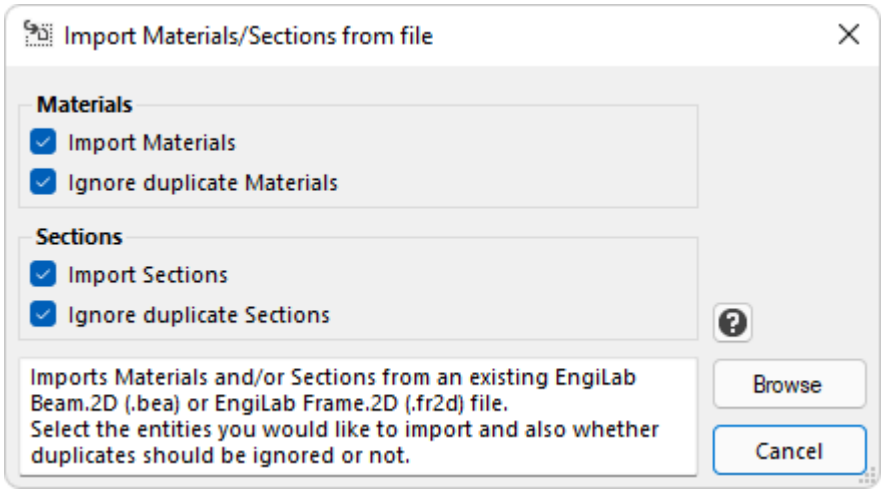

The Tools Menu of the program offers the following options:

- Import Materials/Sections
- Import DXF
- Convert Model to Truss

## 3.1 Import Materials/Sections



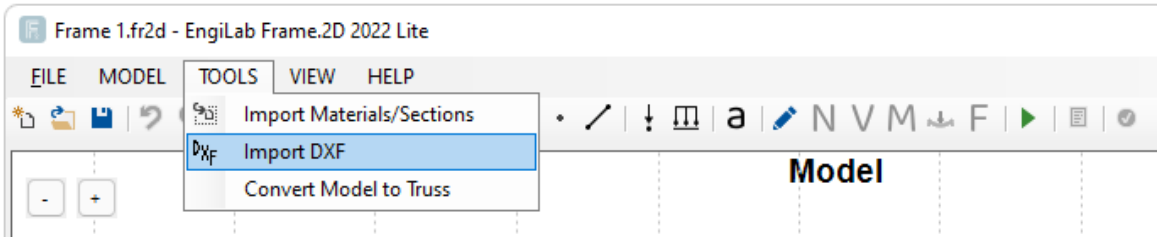

The user can import **Materials** and/or **Sections** from an existing EngiLab Frame.2D file. You can import Materials/Sections from an older EngiLab Beam.2D file (**.bea** file) or from a newer EngiLab Frame.2D file (**.fr2d** file).



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

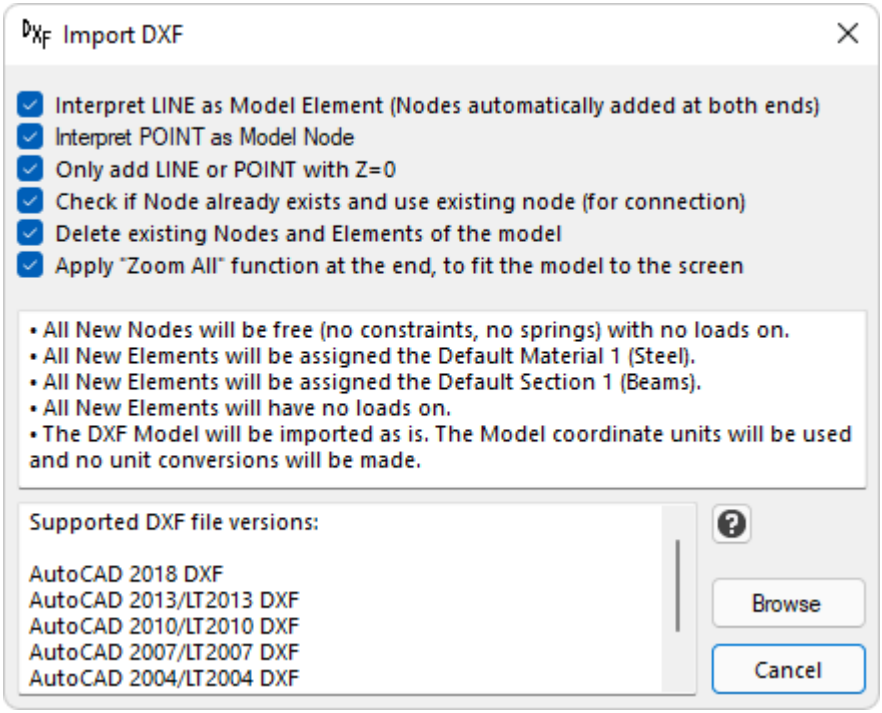
The Materials and Sections are normally imported with their units. If the unit system used is different between the imported file and the current Model, the program will ask the user whether it should perform unit conversions before importing.

## 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 for Elements (and optionally POINTS for Nodes) 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 <u>at least one Material and one Section</u>**. All the LINES of the DXF file will be converted to Model Elements assigned the Default Material and the Default Section.




(c) 2022 EngiLab PC

**Notes:**

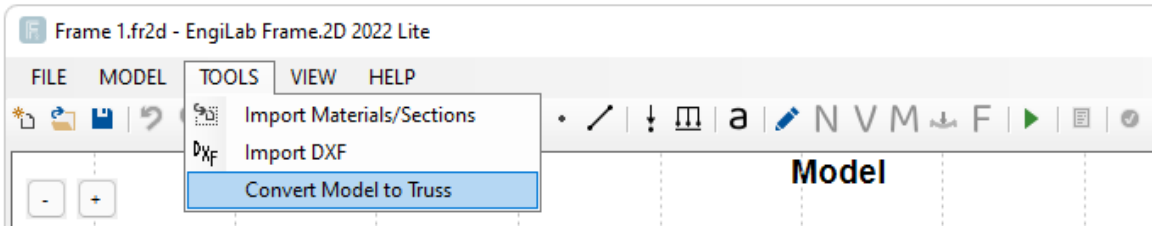
- All new Nodes will be free (no constraints, no springs) with no loads on them.
- All new Elements will be assigned the Default Material.
- All new Elements will be assigned the Default Section.
- All new Elements will have no hinges, and no Elemental Loads on them.

**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**. This is done by adding hinges to the ends of some Elements of the Model. 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 in EngiLab Frame.2D 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 distributed load on it. The same is true if there are any Elemental Loads (distributed loads) acting on the Elements.

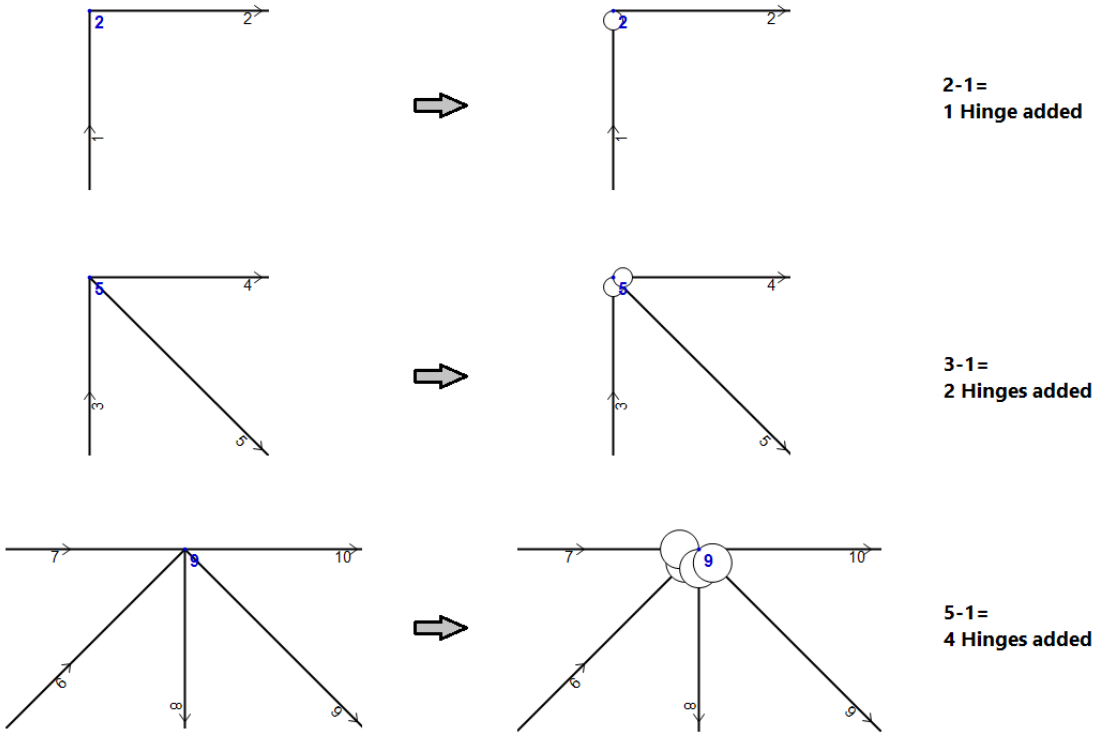
For dedicated 2D Truss analysis, EngiLab also offers **EngiLab Truss.2D** software, which works with 2 Degrees of Freedom per Node and is dedicated to the analysis of 2D

(c) 2022 EngiLab PC

Trusses, offering more pre- and pro-processing features for this type of structures, such as stress and strain diagrams, color plots and others.

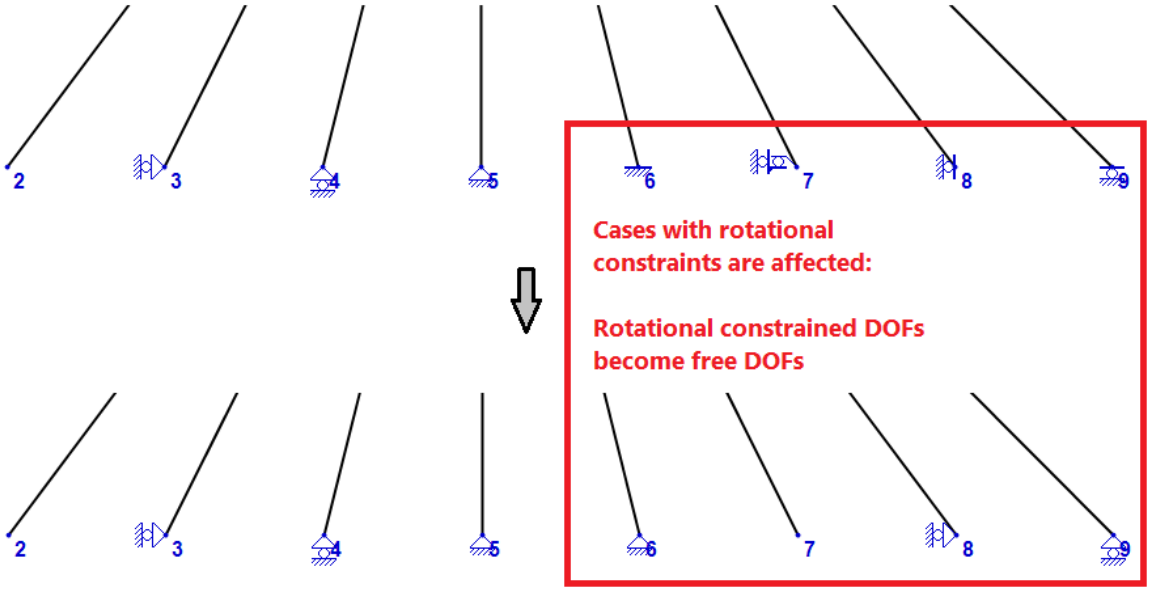
In EngiLab Frame.2D, in order to convert a **Frame model** to a **Truss model**, the program automatically makes the following changes to the Model:

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.



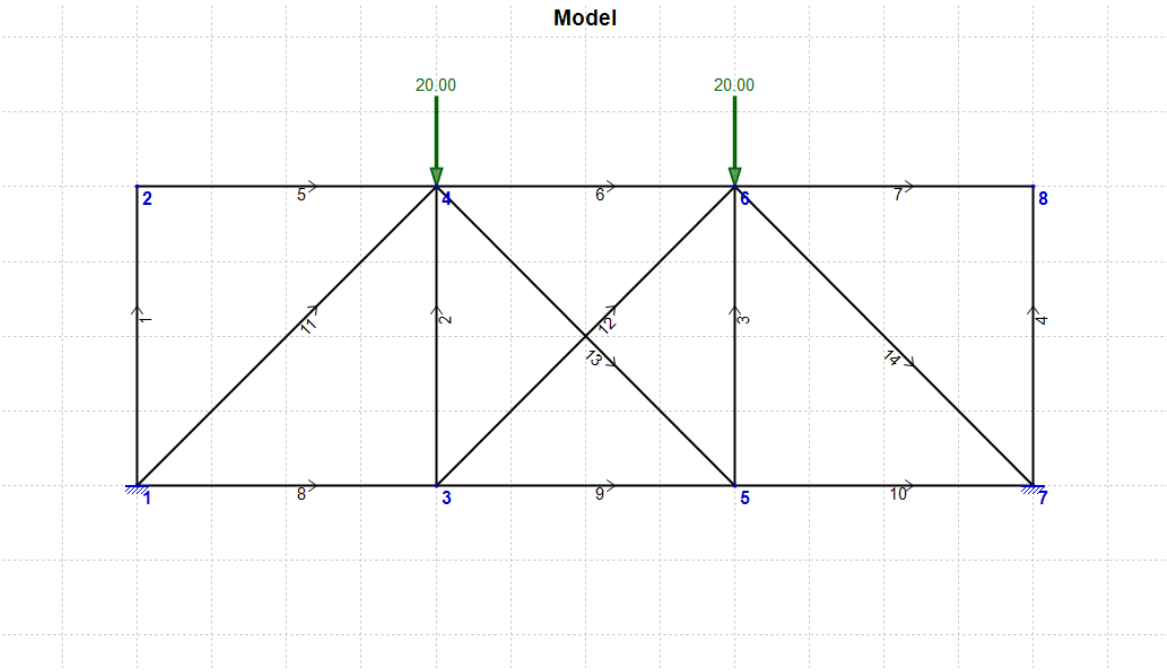

2. **Supports are converted to Truss-Model supports**, by releasing any constraints on the rotational Degree of Freedom (**RZ-Con**). The other two DOFs of each node are not affected. The picture below explains this conversion for every support case in EngiLab Frame.2D.

(c) 2022 EngiLab PC

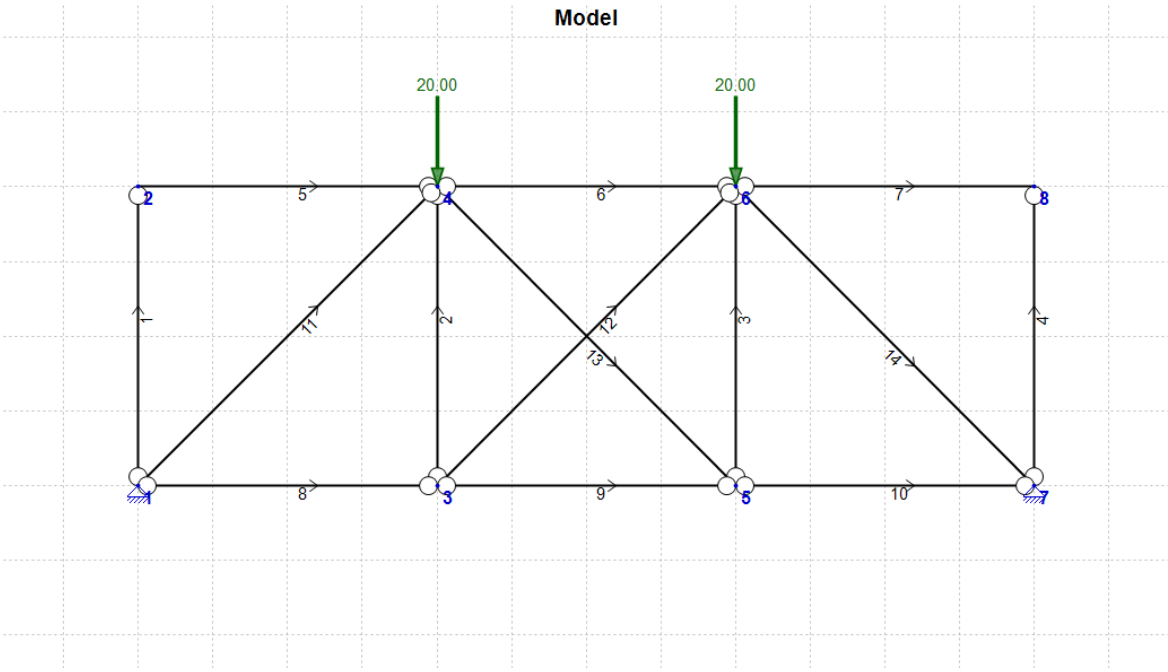



**See the example below.**

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.




(c) 2022 EngiLab PC

*This page intentionally left blank.*

# Chapter




## Analysis, Results and Validation

# 4 Analysis, Results and 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**]$^{-1}$·{**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

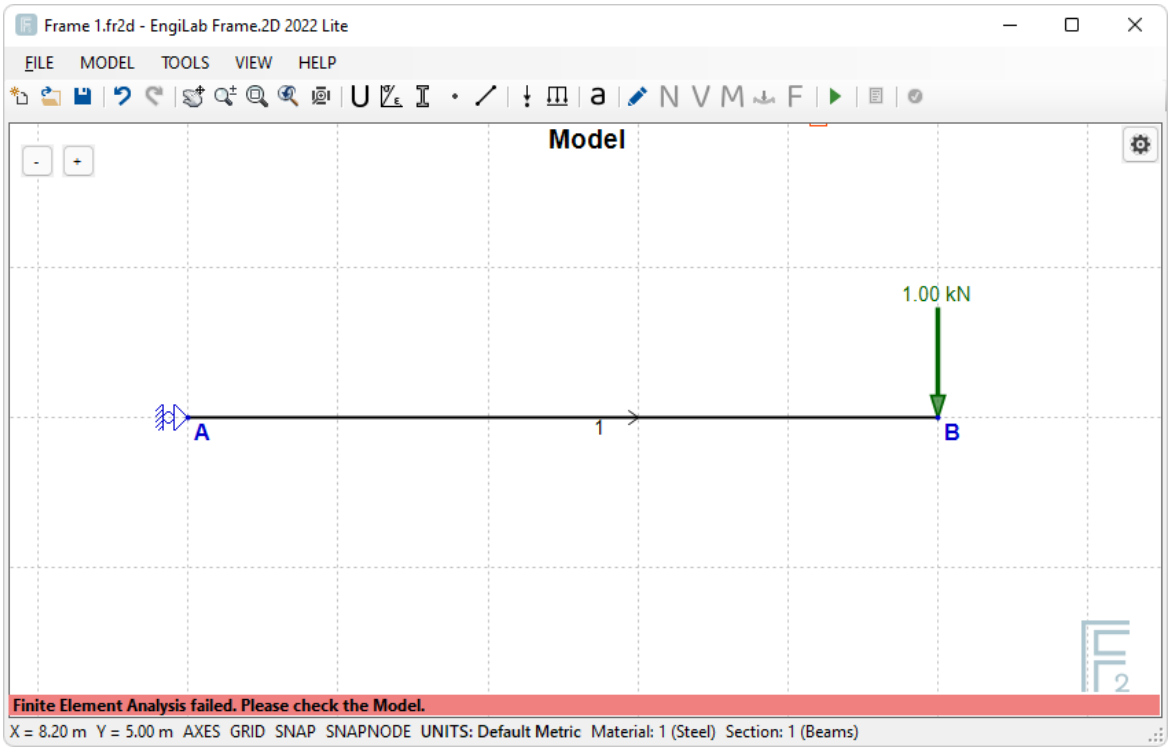
Computational errors may occur during the analysis process. In that case, the program warns the user with one of the following messages:

#### 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.

(c) 2022 EngiLab PC

**Example**: An under-constrained model, such as a Cantilever beam where instead of a fixed support, there is a Roller support as shown below. The model can freely move and 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.




(c) 2022 EngiLab PC

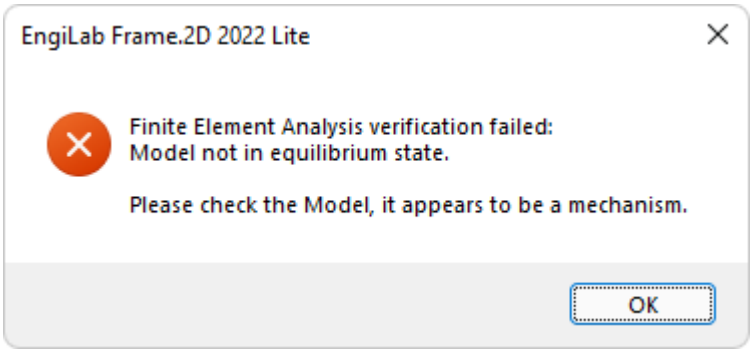
**2. 'Finite Element Analysis verification failed: Model not in equilibrium state. Please check the Model, it appears to be a mechanism.'**

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, and no numerical errors occurred. Yet, after the Analysis is finished, EngiLab Frame.2D performs several verification tests to make sure that the Analysis results are valid and trustworthy. In particular, the program calculates the following, for the entire Model (taking also into account the calculated support reactions):

- Sum of Forces in the X- direction (ΣFX)
- Sum of Forces in the Y- direction (ΣFY)
- Sum of Moments about the (0, 0) point on the plane (ΣM(0,0)).

These calculations take into account the external forces and the calculated (from the Analysis) support reactions. In a well-defined Model that has been successfully analyzed, all three values must be equal to zero, i.e. ΣFX=0, ΣFY=0, ΣM(0,0)=0 so that the Model is in equilibrium state after the external loads have been applied.

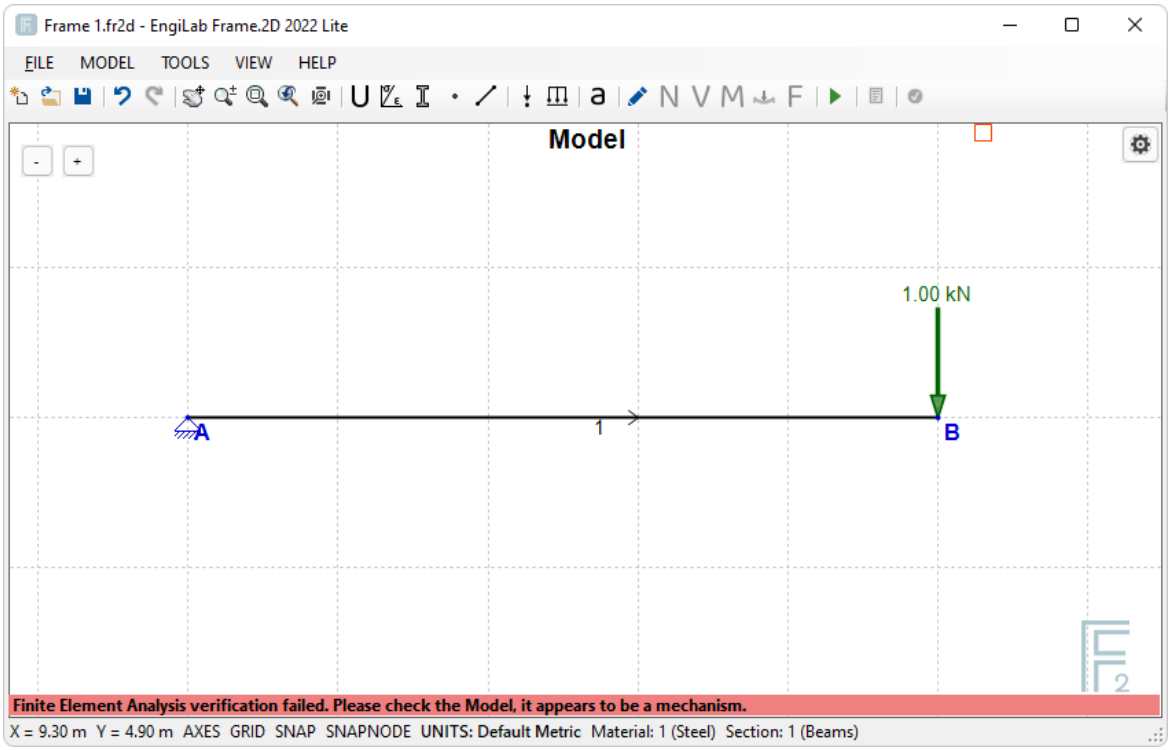
If, after the analysis, any of these values is not equal to zero (given a tolerance value of $10^{-5}$, due to rounding errors), then this is a clear indication that something is wrong with the Model and the program will show the following error message.



This means that most 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 rounding errors. As a result, the Analysis solver finds a solution (with no practical interest) to the linear equations system, yet the resultant displacements have excessive values and the Model is NOT in equilibrium state.

This, for example, can happen in the Model shown below. The Model is obviously a mechanism (only two X-Rollers as supports), as was the one above, but this time the computer gives us the second error message, instead of the first.

This for example can happen in the Model shown below. The model is obviously a mechanism (only one pinned support), 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.

(c) 2022 EngiLab PC

- 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.

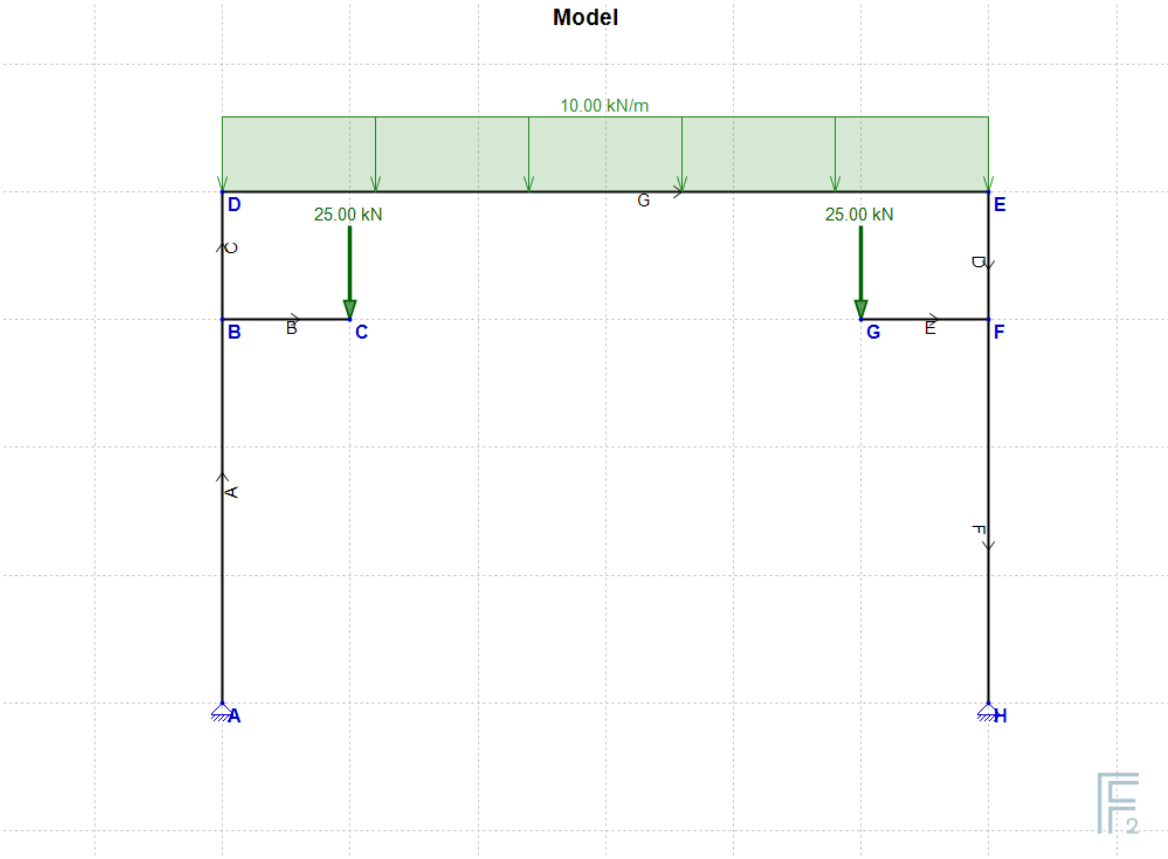
- N: Axial Force Diagram
- V: Shear Force Diagram
- M: Bending Moment Diagram

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

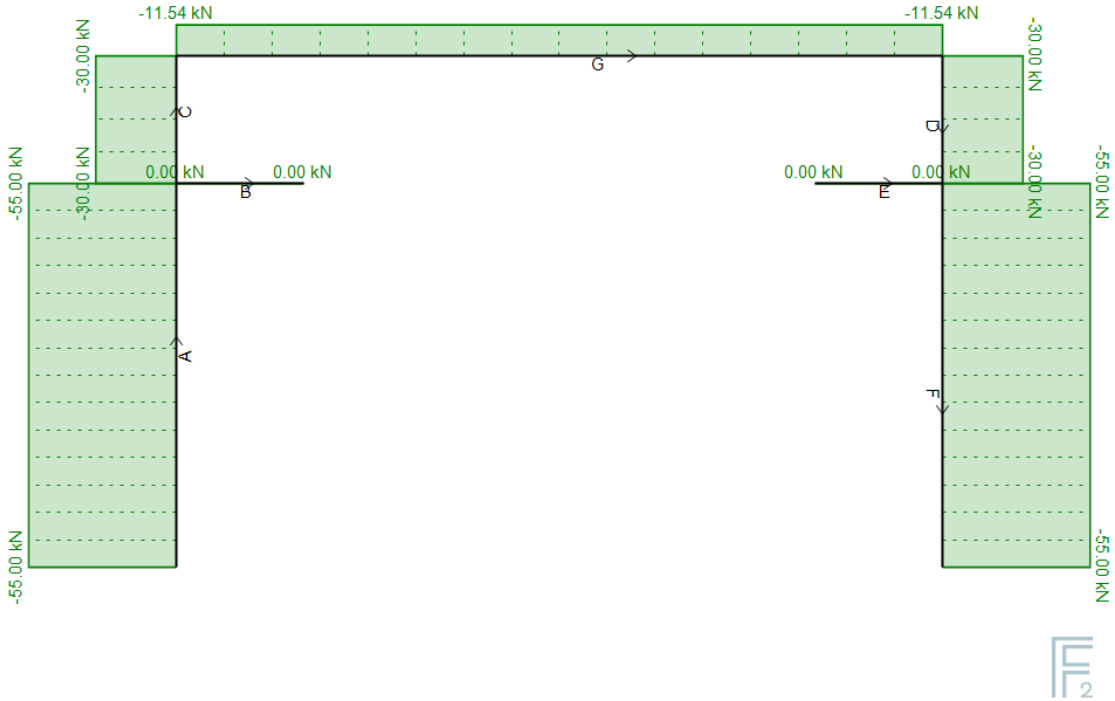
**See also**: NVM Diagrams (Settings).

(c) 2022 EngiLab PC

## Example

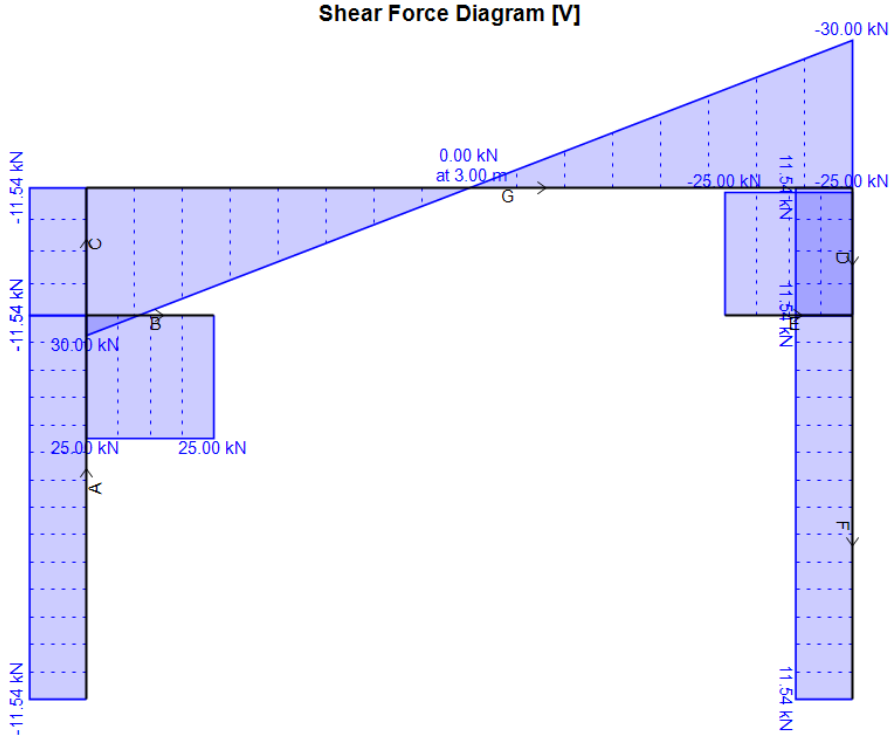



Axial Force Diagram [N]

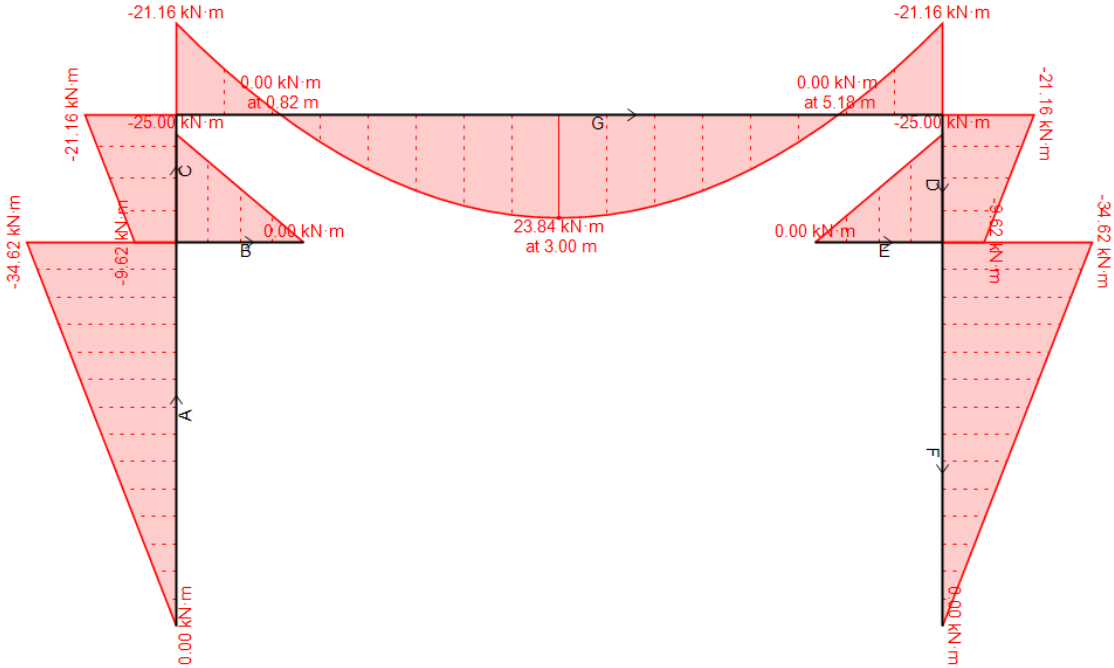



(c) 2022 EngiLab PC

**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.

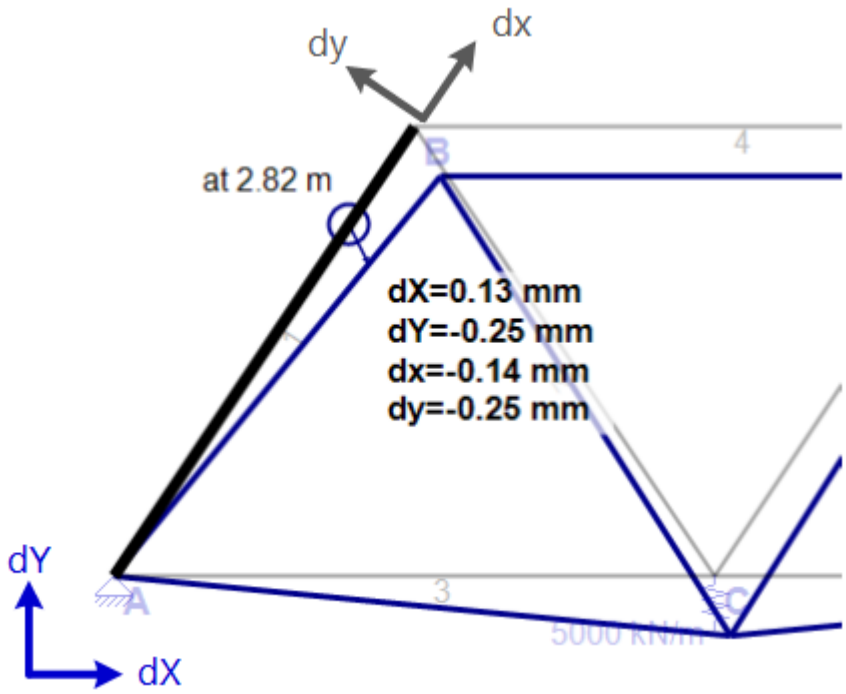
- : Deformation

**Note**: Deformation values are given also on screen, if the mouse pointer hovers over an element. The values that are given on screen are the **dX**, **dY**, **dx** and **dy** displacements of the corresponding point of each element.

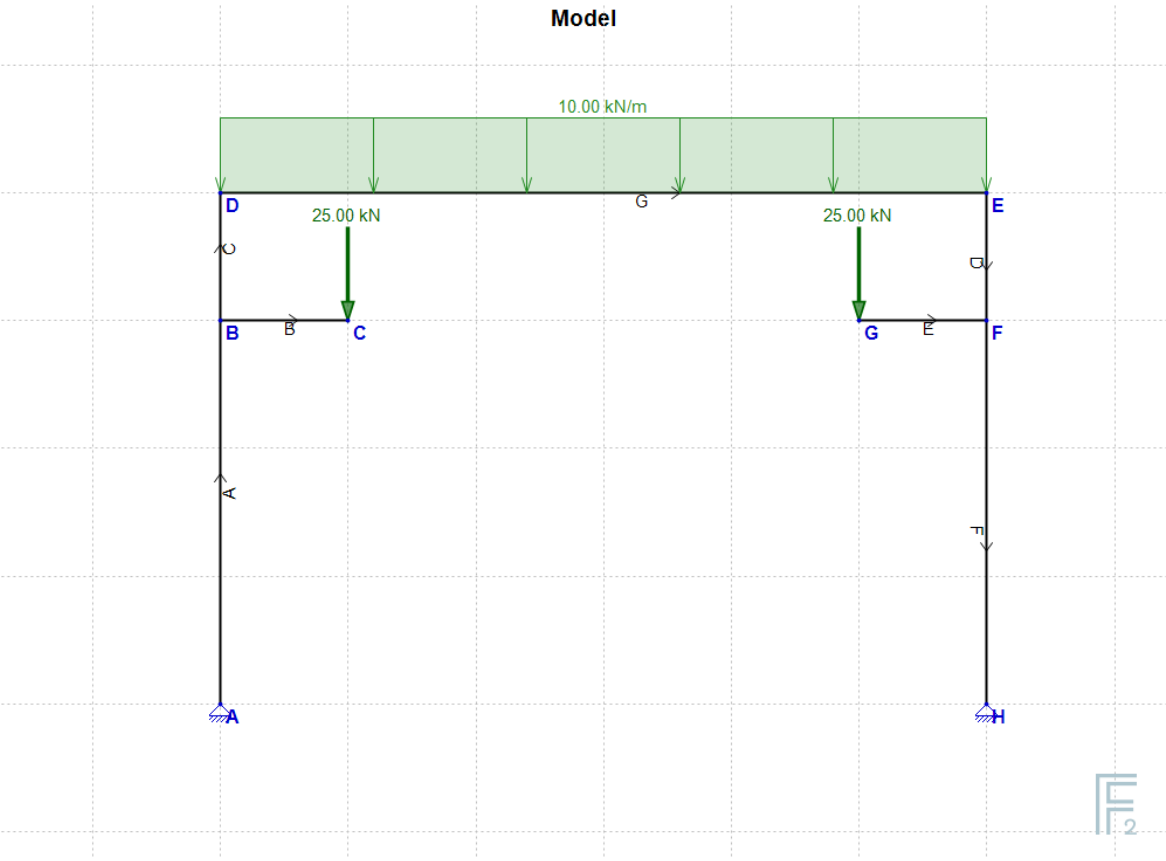
- **dX** and **dY** are the X- and Y- displacements of a point, respectively, in **Global axes**. For example a positive dX means displacement to the right, while a positive dY means displacement upwards.
- **dx** and **dy** are the x- and y- displacements of a point, respectively, in **Local Element Axes**, where x is defined from the Start Node i and the End Node j of the Element and y is perpendicular to it.

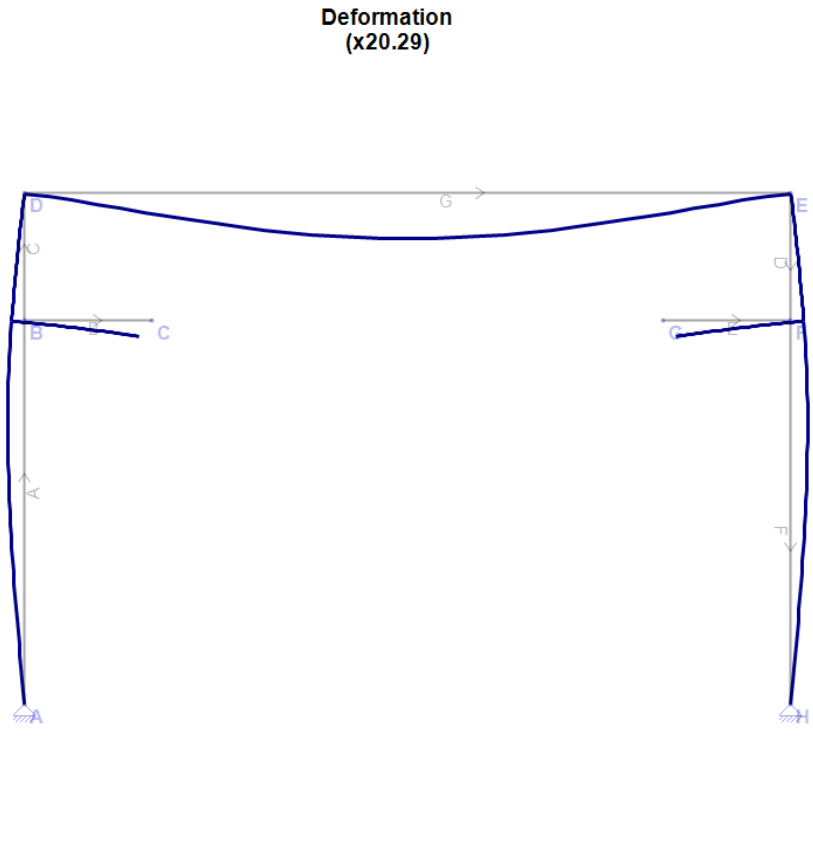
See the illustrative example below which explains this convention.

(c) 2022 EngiLab PC


dy
dx
at 2.82 m
B
4
dX=0.13 mm
dY=-0.25 mm
dx=-0.14 mm
dy=-0.25 mm
dY
dX
A
3
C
5000 kN/m

## Example


Model
10.00 kN/m
D
G
E
25.00 kN
25.00 kN
C
D
B
B
C
G
E
F
A
F
A
H




## 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.

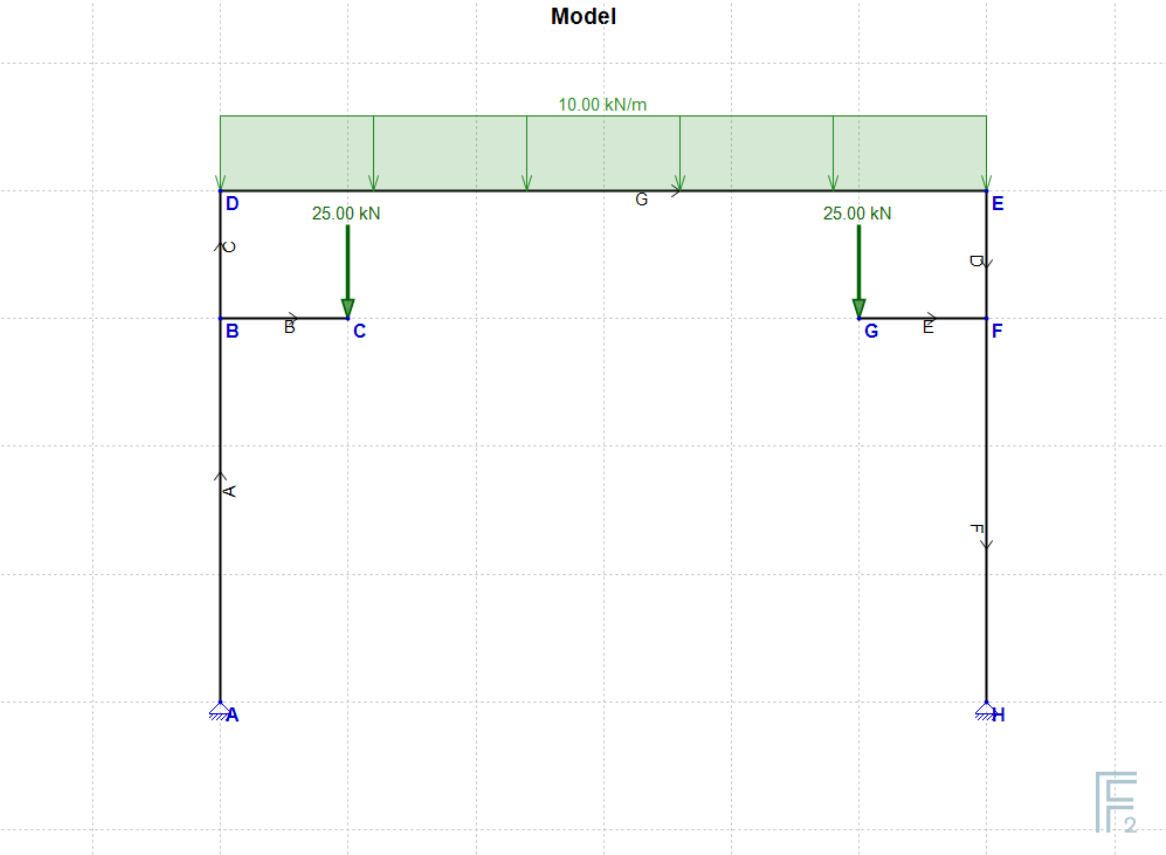
- F: Free Body Diagram

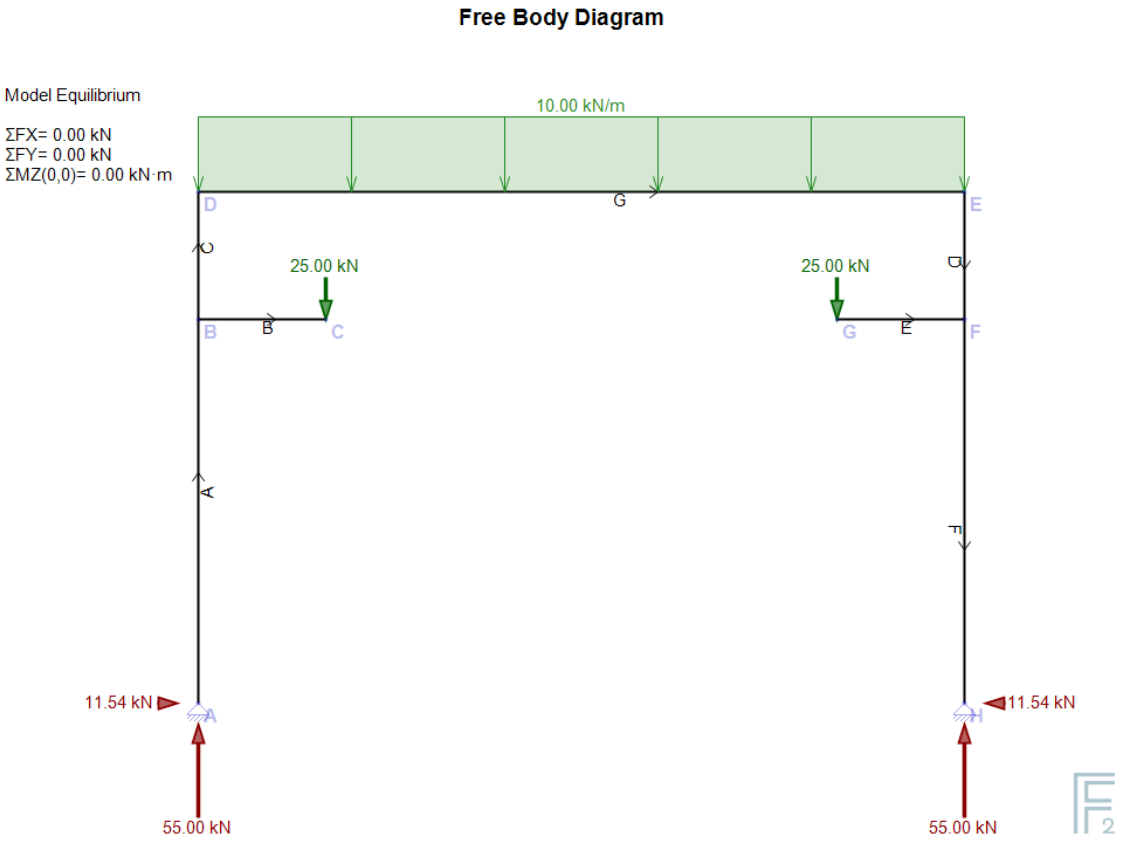
The Free Body Diagram shows the support reactions on screen and also the calculations of the equilibrium of the Model.

**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.

(c) 2022 EngiLab PC

## Example






## 4.5 Analysis Results Form



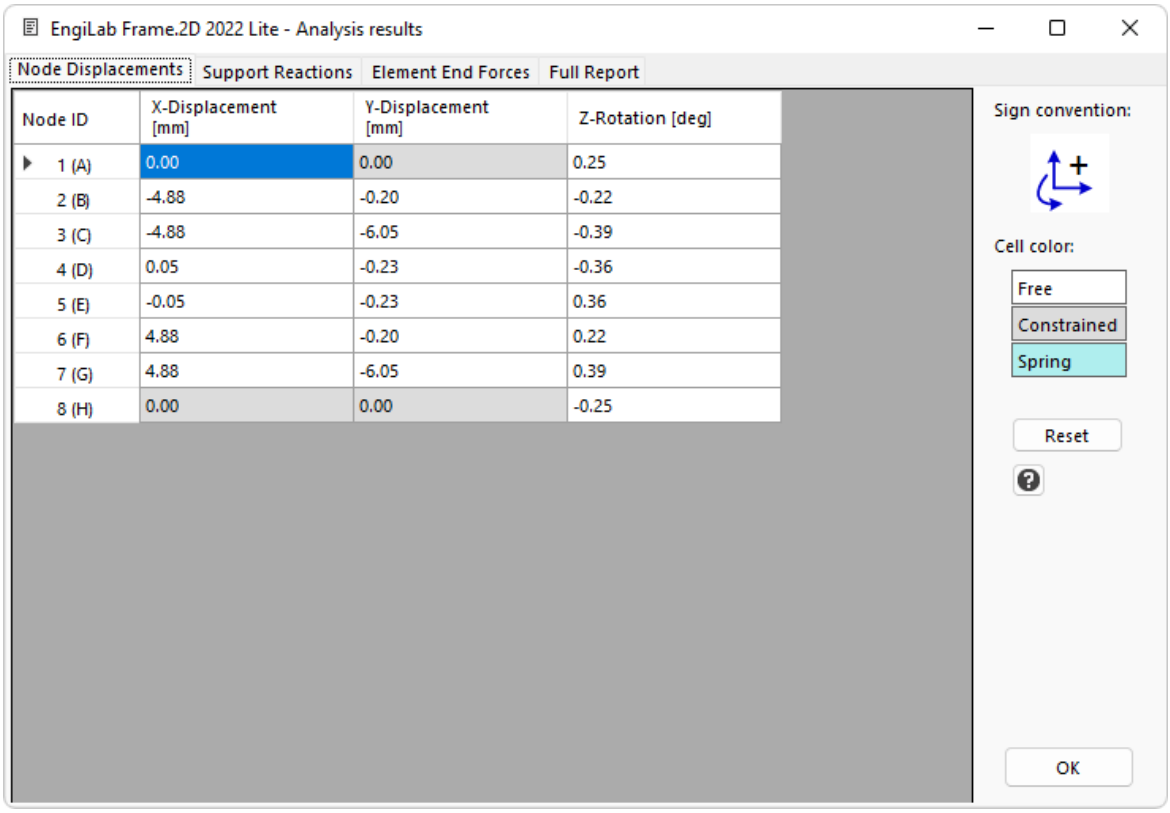

By clicking the Analysis results button, you get a new form where you can see the following:

- Node Displacements
- Support Reactions
- Element End Forces
- Full Report

(c) 2022 EngiLab PC

### 4.5.1 Node Displacements

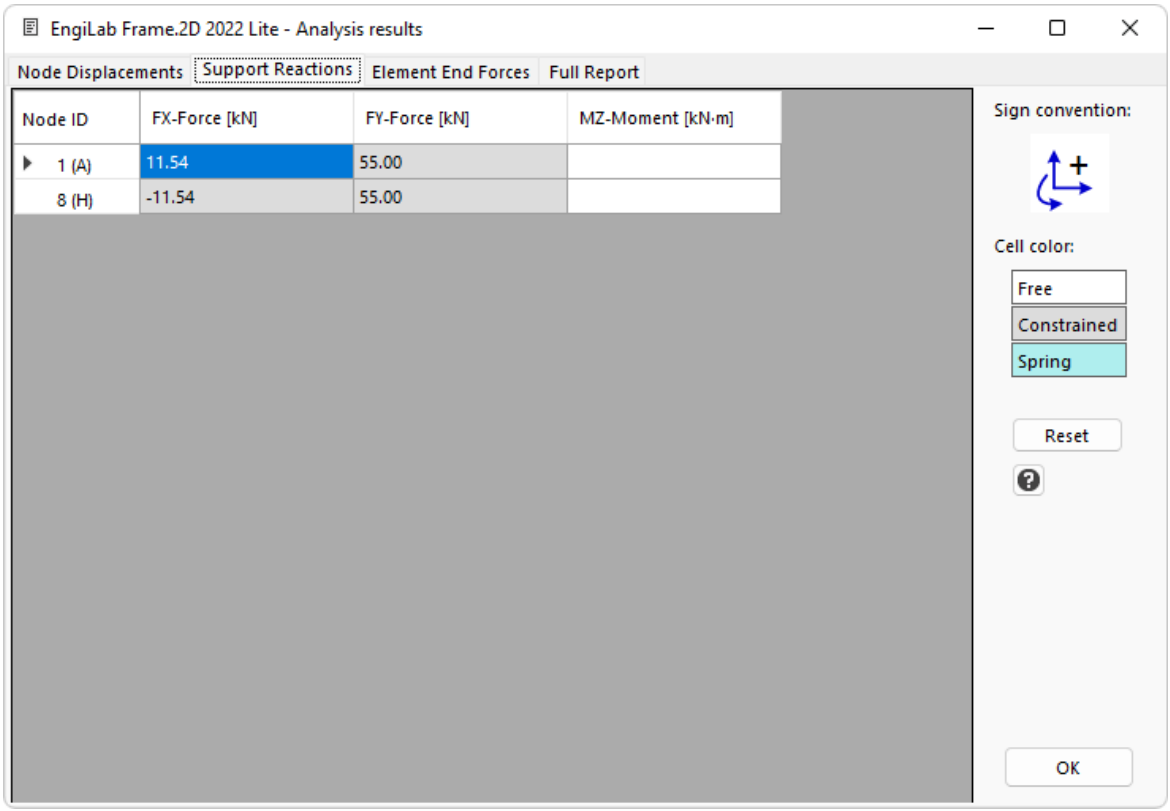
Shows the Node Displacements of the Model. The sign convention is according to the Global Axes.



### 4.5.2 Support Reactions

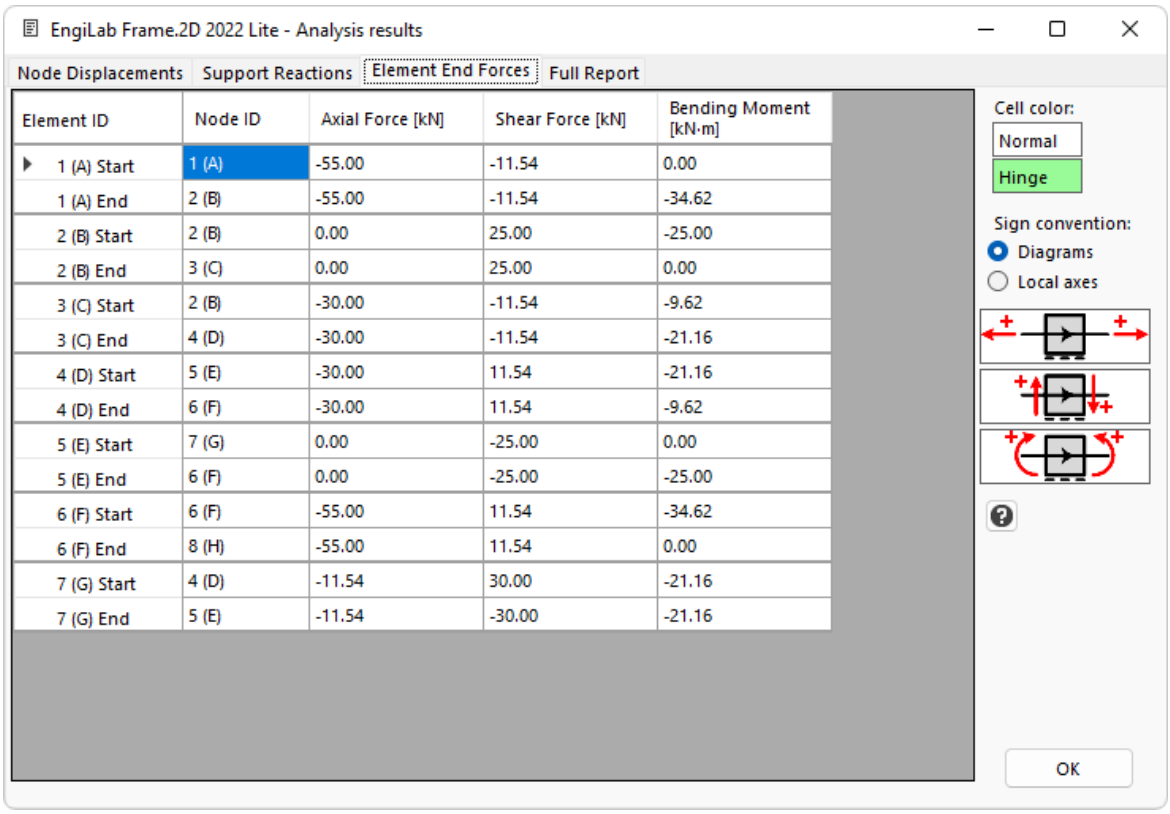
This tab shows the Support Reactions (Constraints or Springs) of the Model. The sign convention is in accordance with the Global Axes.

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



### 4.5.3 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.

 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.

(c) 2022 EngiLab PC




(c) 2022 EngiLab PC

### 4.5.4 Full Report

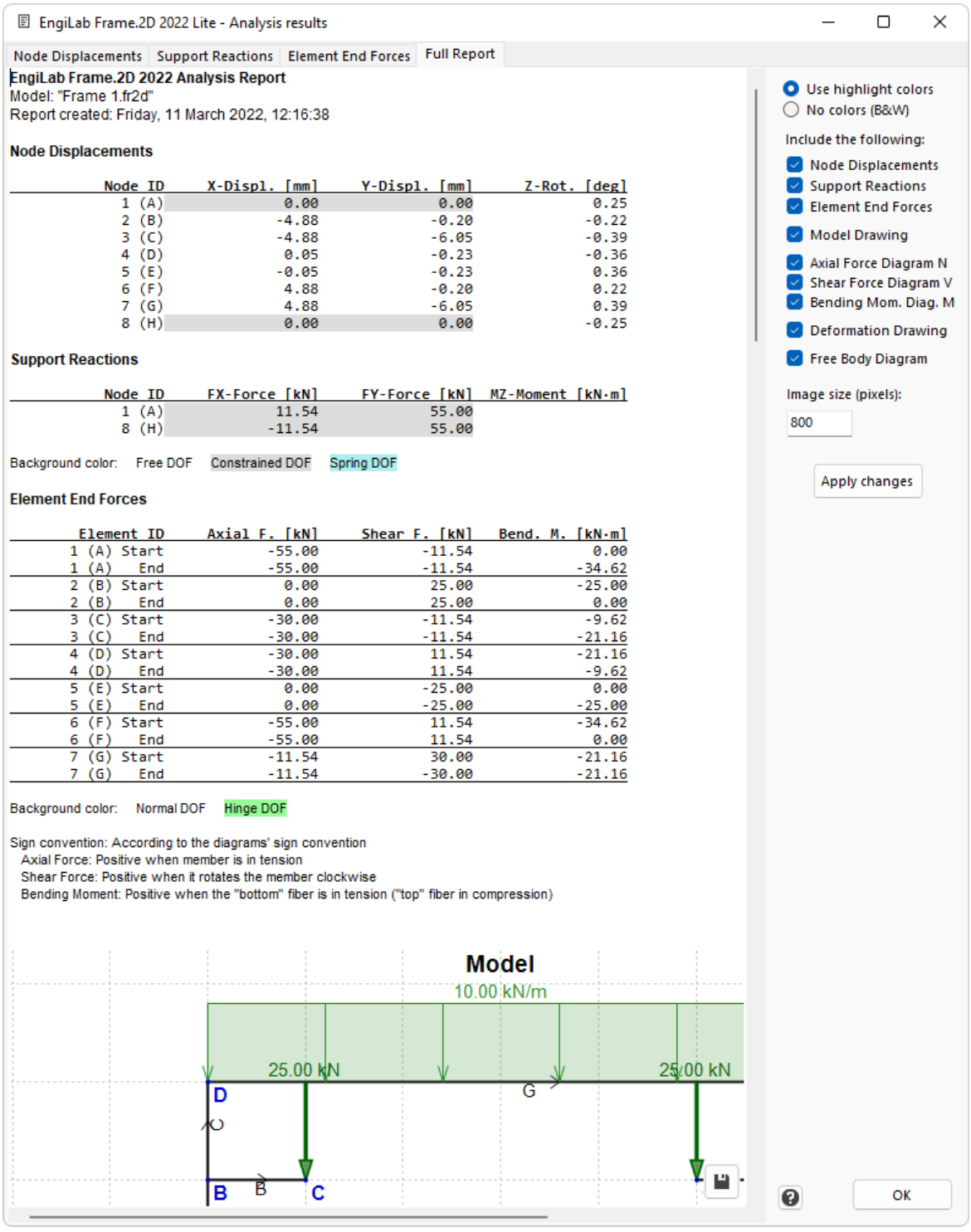
This tab shows all the Analysis results in Rich Text Format (RTF). You can easily copy the document and paste it anywhere (for example in MS Word), or you can click the "Save" button and save the document in RTF format which can be opened with a text editor, such as MS Word.

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.

(c) 2022 EngiLab PC




## 4.6 Analysis Validation Form




(c) 2022 EngiLab PC

By clicking the Analysis Validation button , you get a new form where you can see the Analysis Validation results.

If the Analysis results are correct, then the Model and all its parts (joints and members) should be in equilibrium.

The following tabs are available in the form:

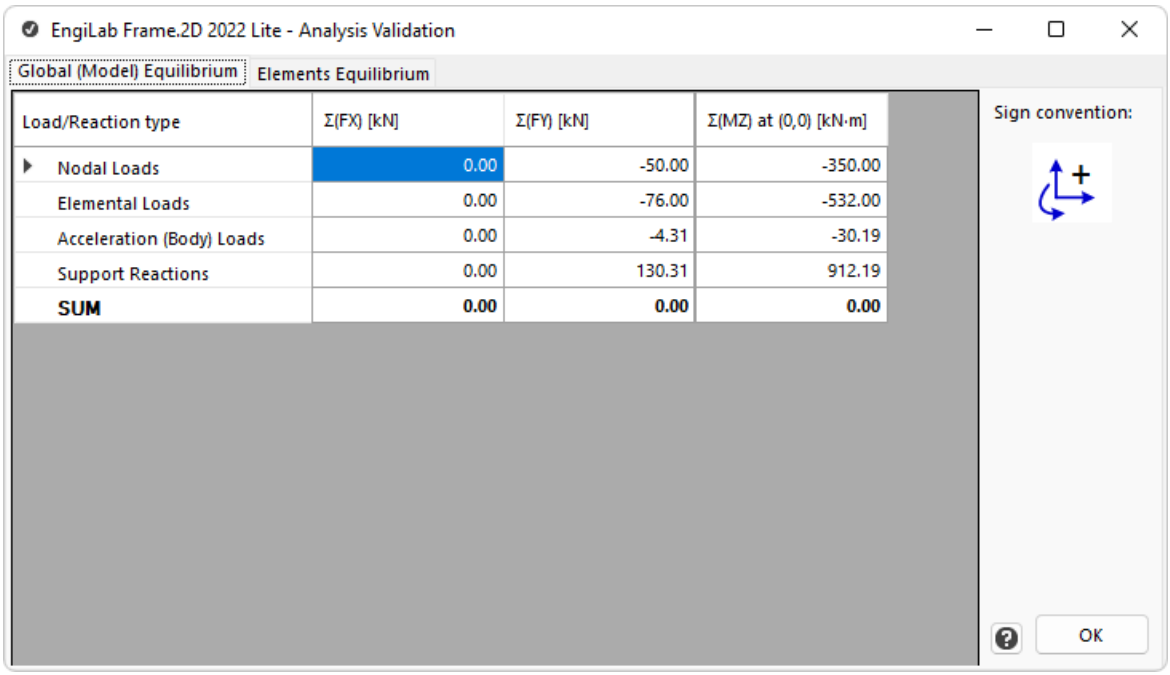
- Global (Model) Equilibrium
- Elements Equilibrium

### 4.6.1 Global (Model) Equilibrium

The program calculates the **Sum of X-Forces Σ(FX)**, the **Sum of Y-Forces Σ(FY)** and the **Sum of Bending Moments Σ(MZ)** (at Point X=0, Y=0) for the whole Model, for the Nodal Loads, Elemental Loads, Acceleration (Body) Loads and the Support Reactions. All three Sums (see the 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. This is already checked by the program, every time an Analysis is conducted, so theoretically it will never happen. But the calculations are there for you to check and verify.

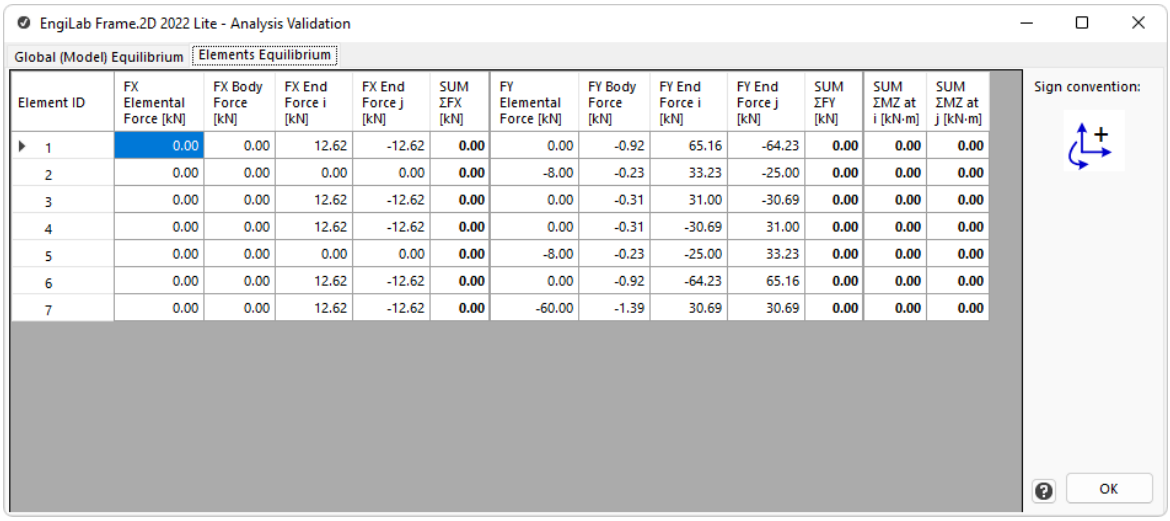
**Note**: The sign convention is according to the Global Axes, as shown in the figure.



### 4.6.2 Elements Equilibrium

This tab shows the calculations related to the **Equilibrium of Elements**. 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.




| Element ID | FX Elemental Force [kN] | FX Body Force [kN] | FX End Force i [kN] | FX End Force j [kN] | SUM ΣFX [kN] | FY Elemental Force [kN] | FY Body Force [kN] | FY End Force i [kN] | FY End Force j [kN] | SUM ΣFY [kN] | SUM ΣMZ at i [kN·m] | SUM ΣMZ at j [kN·m] |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 0.00 | 0.00 | 12.62 | -12.62 | **0.00** | 0.00 | -0.92 | 65.16 | -64.23 | **0.00** | **0.00** | **0.00** |
| 2 | 0.00 | 0.00 | 0.00 | 0.00 | **0.00** | -8.00 | -0.23 | 33.23 | -25.00 | **0.00** | **0.00** | **0.00** |
| 3 | 0.00 | 0.00 | 12.62 | -12.62 | **0.00** | 0.00 | -0.31 | 31.00 | -30.69 | **0.00** | **0.00** | **0.00** |
| 4 | 0.00 | 0.00 | 12.62 | -12.62 | **0.00** | 0.00 | -0.31 | -30.69 | 31.00 | **0.00** | **0.00** | **0.00** |
| 5 | 0.00 | 0.00 | 0.00 | 0.00 | **0.00** | -8.00 | -0.23 | -25.00 | 33.23 | **0.00** | **0.00** | **0.00** |
| 6 | 0.00 | 0.00 | 12.62 | -12.62 | **0.00** | 0.00 | -0.92 | -64.23 | 65.16 | **0.00** | **0.00** | **0.00** |
| 7 | 0.00 | 0.00 | 12.62 | -12.62 | **0.00** | -60.00 | -1.39 | 30.69 | 30.69 | **0.00** | **0.00** | **0.00** |

(c) 2022 EngiLab PC

*This page intentionally left blank.*

# Chapter

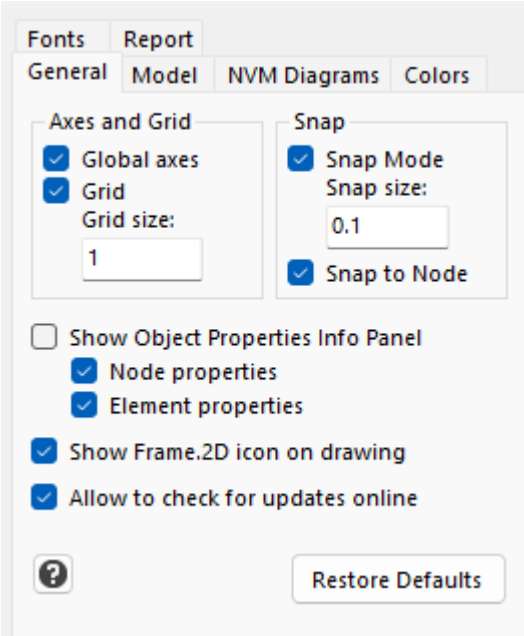



## Settings

# 5 Settings


- General
- Model
- NVM Diagrams
- Colors
- Fonts
- Report


## 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

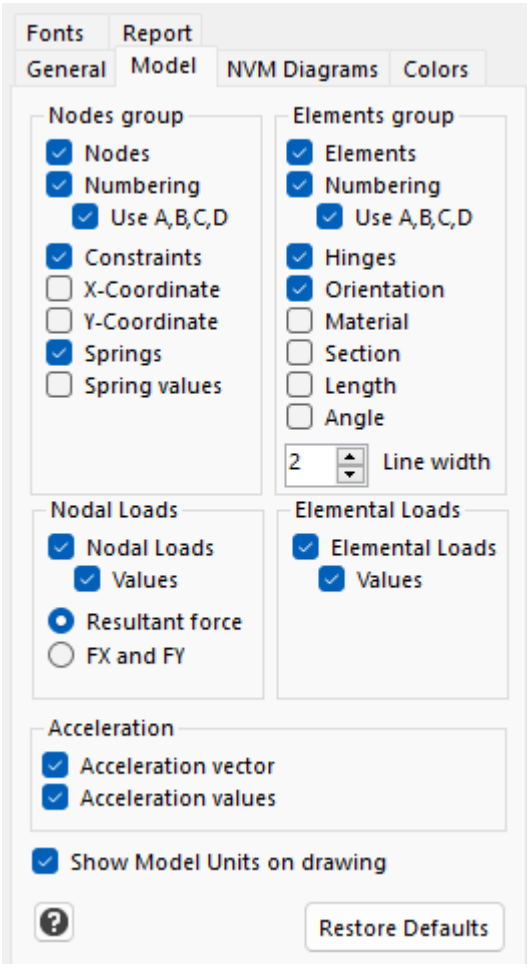
- **Snap Mode**: Enables/Disables the Snap Mode. If Snap Mode is enabled (Default=True), then you can only draw Nodes and Elements at increments defined by the Snap Size setting
- **Snap Size**: Sets the size of the Snap (Default=0.1)

- **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.

- **Show Object Properties Info Panel**: Shows/Hides the Object Properties Information Panel. It is used to show to the user the properties of Nodes and Elements highlighted on the screen with hovering over a Node or an Element.
- **Node properties** / **Element properties**: The user selects if Node and/or Element properties will be shown on the Information Panel.

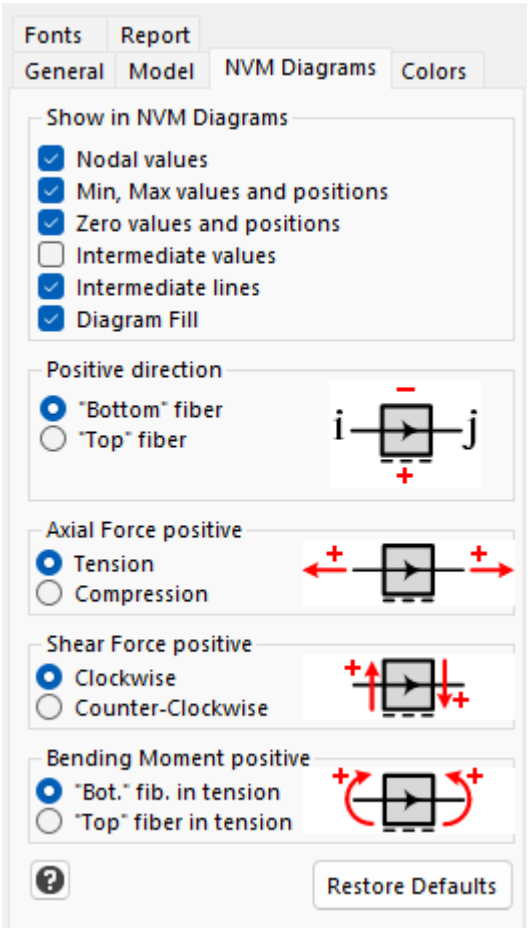
- **Show Frame.2D icon on drawing**: Shows/Hides the Frame.2D (**$F_2$**) icon. When shown, it appears near the bottom right corner of the screen.
- **Allow to check for updates online**: Enables/Disables the automatic update feature. If disabled, the program will not check for updates online.

## 5.2 Model (Settings)



Each setting Shows/Hides the corresponding object on screen.

## 5.3 NVM Diagrams (Settings)



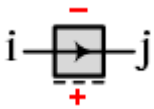

**Available settings:**

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

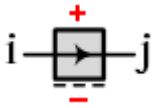
(c) 2022 EngiLab PC

### 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:




Positive towards the "Bottom" fiber (**Default**)




Positive towards the "Top" fiber

### Axial Force positive

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



Axial Force is Positive when member is in Tension (**Default**)



Axial Force is Positive when member is in Compression

### Shear Force positive

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



Shear Force is Positive when member is rotated clockwise (**Default**)



Shear Force is Positive when member is rotated counter-clockwise

### Bending Moment positive

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



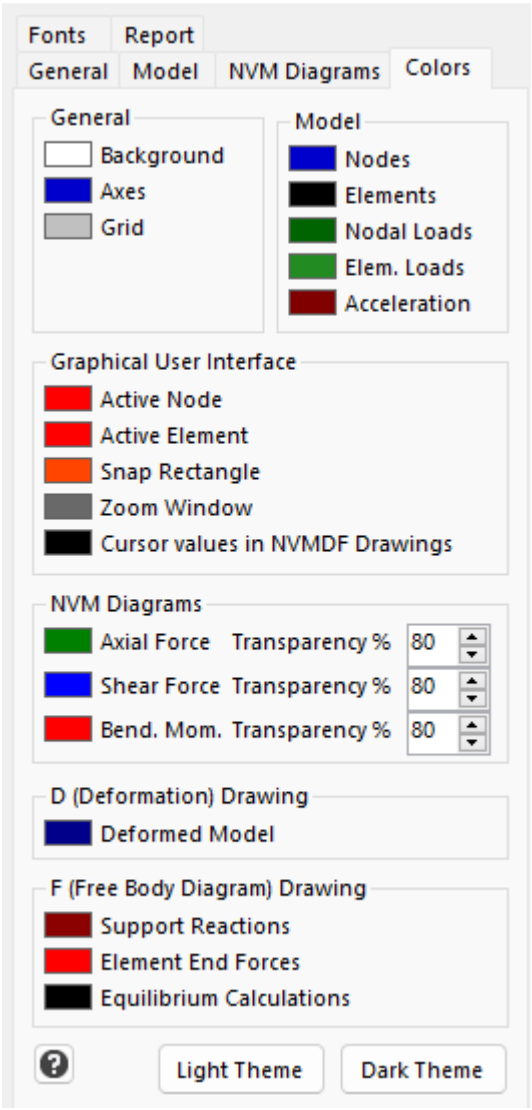
Bending Moment is Positive when the "Bottom" fiber is in tension (**Default**)



Bending Moment is Positive when the "Top" fiber is in tension

(c) 2022 EngiLab PC

## 5.4 Colors (Settings)



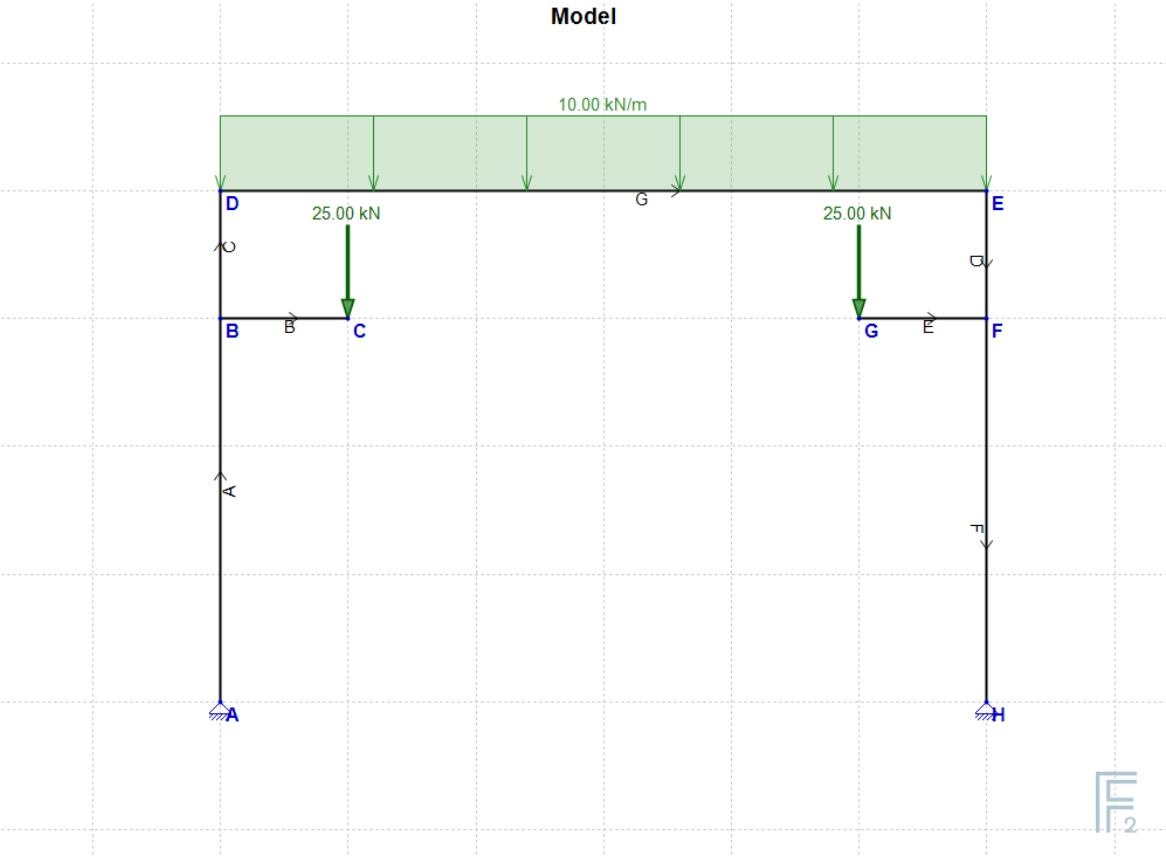

Each setting controls the color of the corresponding object on screen.

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

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

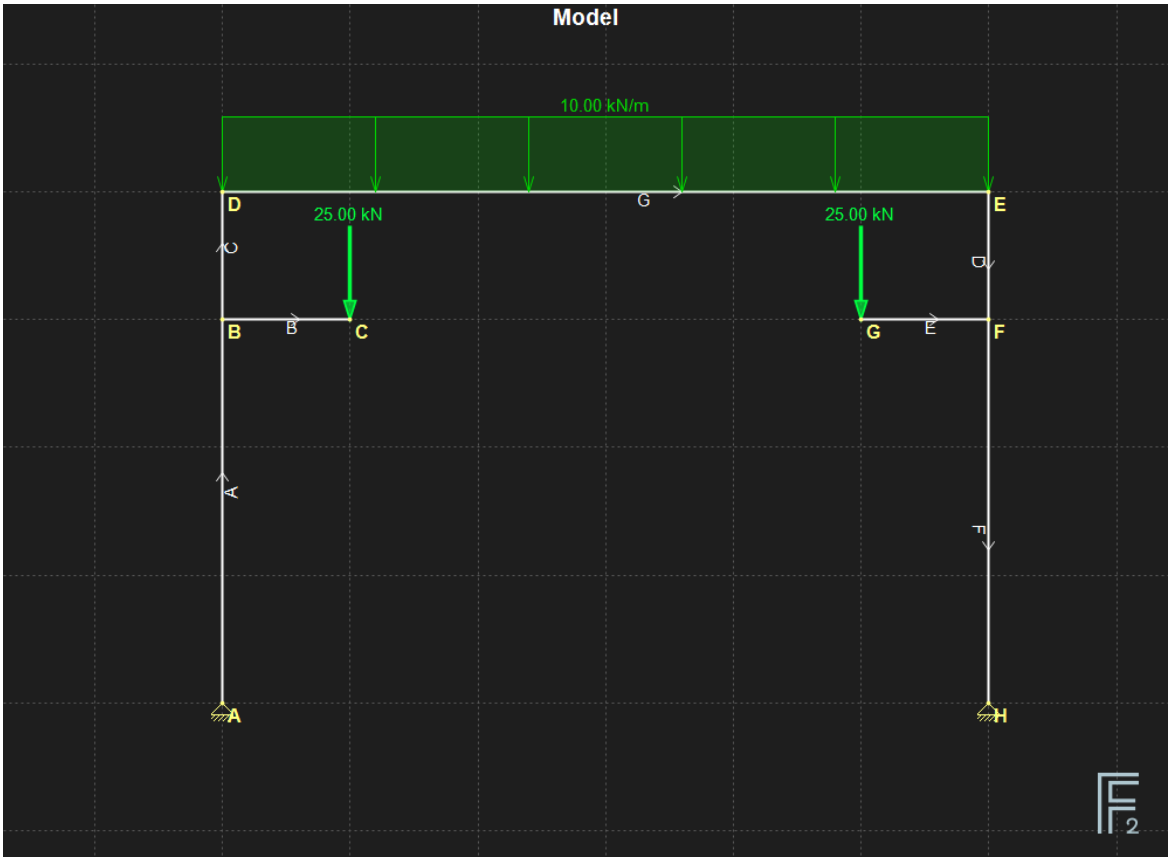
(c) 2022 EngiLab PC

## Light Theme (Default)

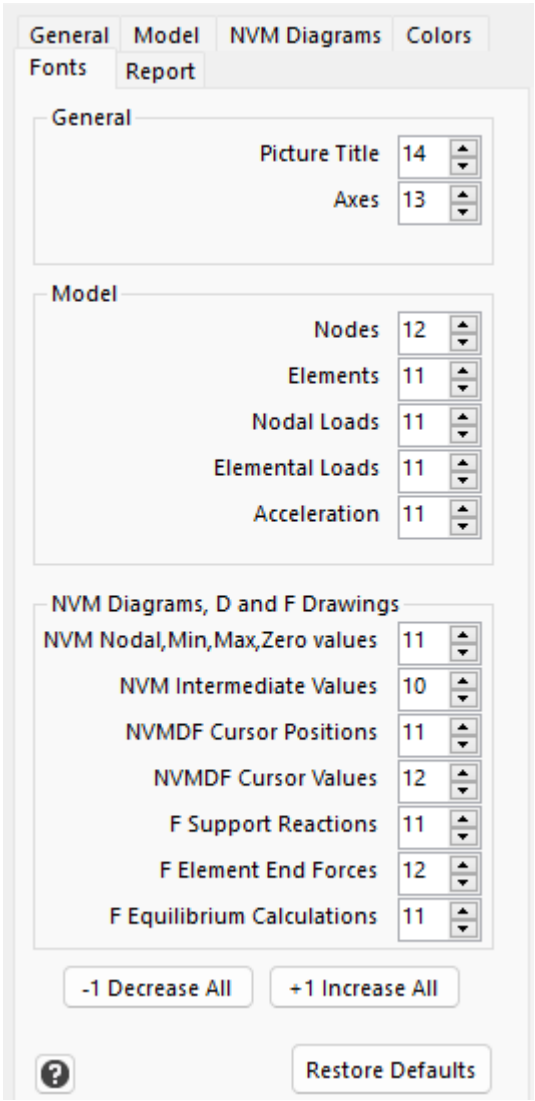



(c) 2022 EngiLab PC

## Dark Theme



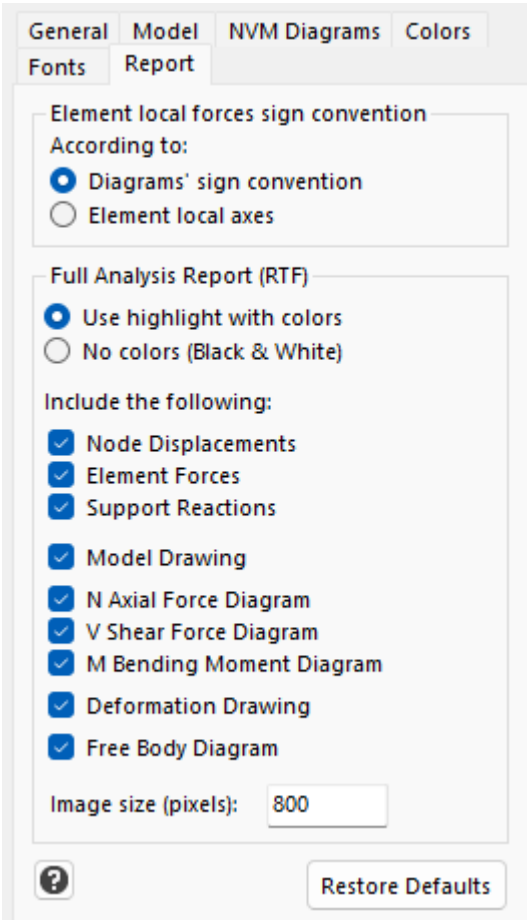
## 5.5 Fonts (Settings)




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

(c) 2022 EngiLab PC

## 5.6 Report (Settings)

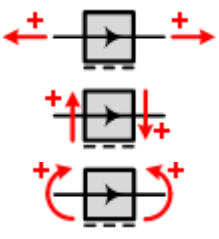



### Available settings:

#### 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

(c) 2022 EngiLab PC

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. 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
- **Image size (pixels)**: You can select the (max) size of the images, in pixels.

(c) 2022 EngiLab PC

*This page intentionally left blank.*

# Chapter




## Useful information

# 6 Useful information


- Tips on Modeling hinges
- Tips on Modeling symmetric structures


## 6.1 Tips on Modeling hinges

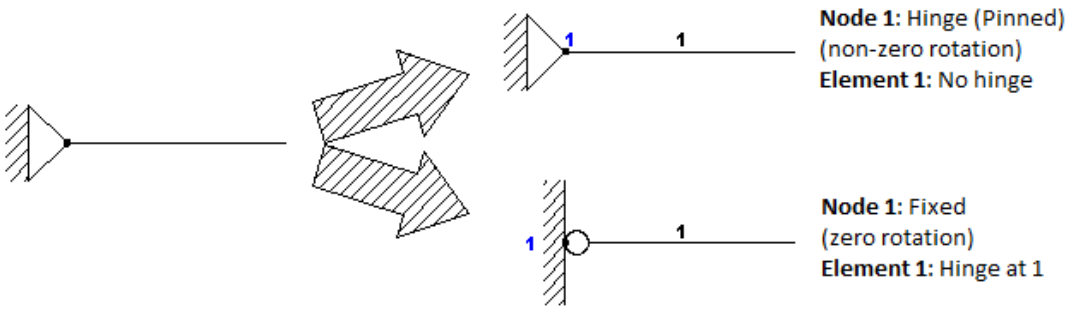
The hinges (rotational releases) of a model is an issue which requires our attention. There are two kinds of hinges: **External hinges (Pinned constraints)** and **Internal hinges**.

### External hinges (Pinned constraints)

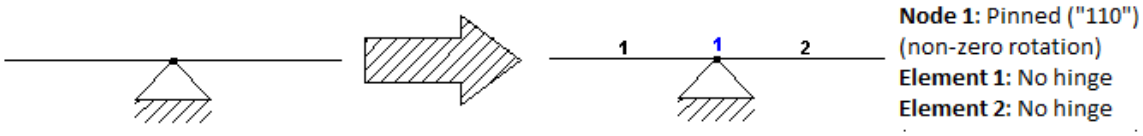
**1**. An external **hinge to which only ONE element is connected** can be given in a model using two possible ways (the result of the analysis should be the same):

- **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.



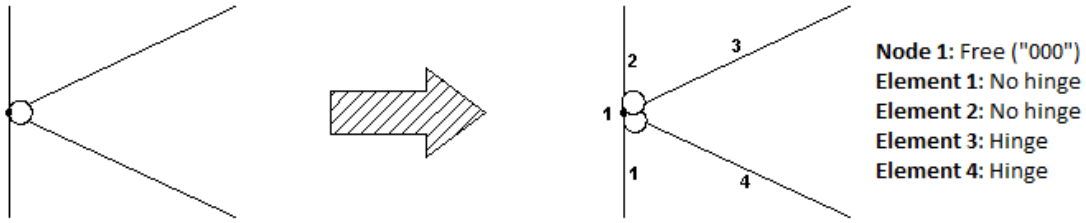
**2**. An **external hinge to which more than one elements are connected** must be given in the Model as follows:




## Internal hinges

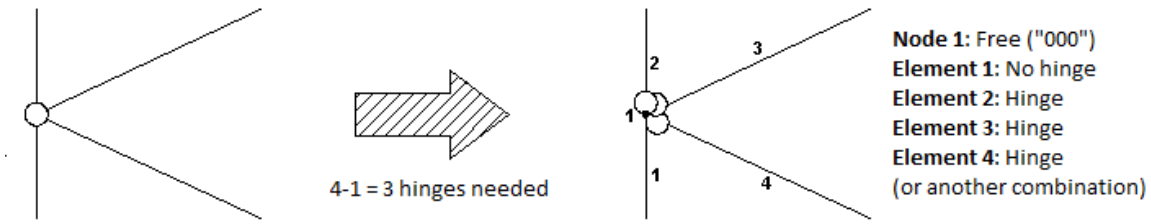
An **internal hinge** must be given always as a hinge of one or more elements as follows:

**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

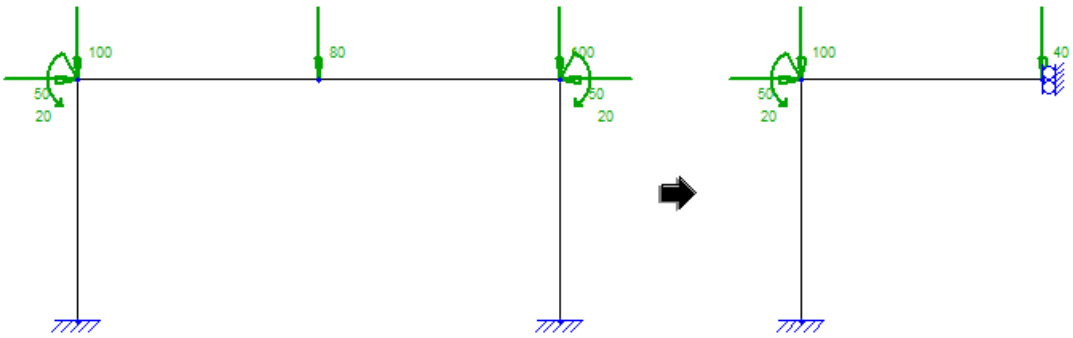
(c) 2022 EngiLab PC

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.

## 6.2 Tips on Modeling symmetric structures

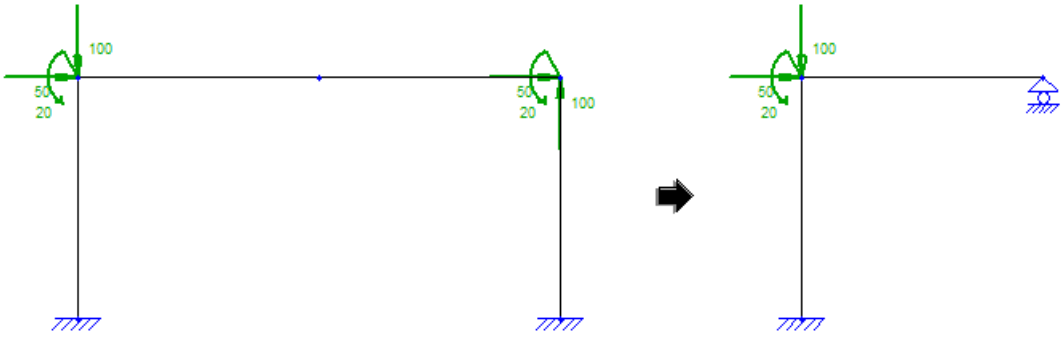
If a structure is symmetric (Symmetric structure with symmetric / anti-symmetric loads), then the size of the finite element model can be reduced, which, in turn, reduces the time and cost of the analysis. For each plane of symmetry in the model, the model size can be reduced by a factor of approximately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

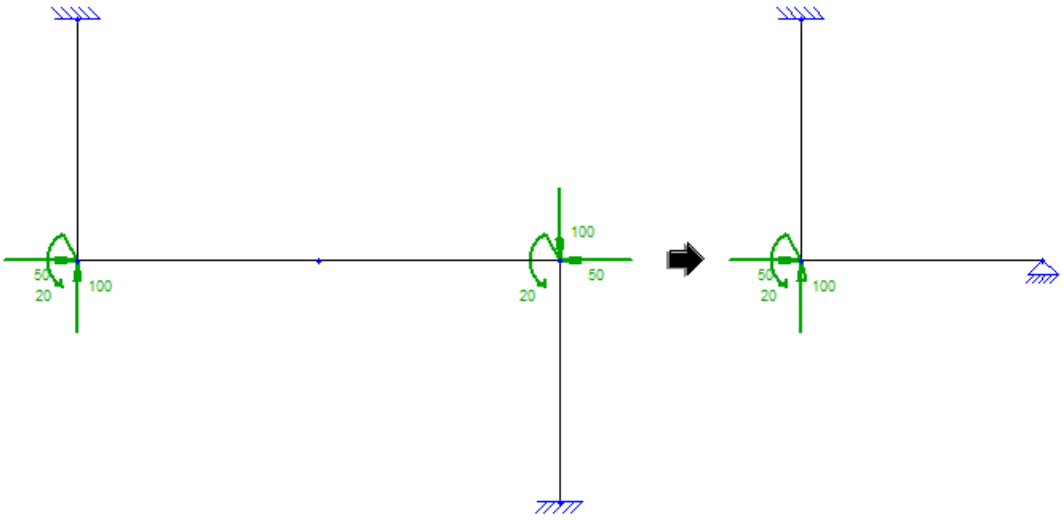
### Case 2: Symmetric structure (axis) + Anti-Symmetric loads (axis)




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

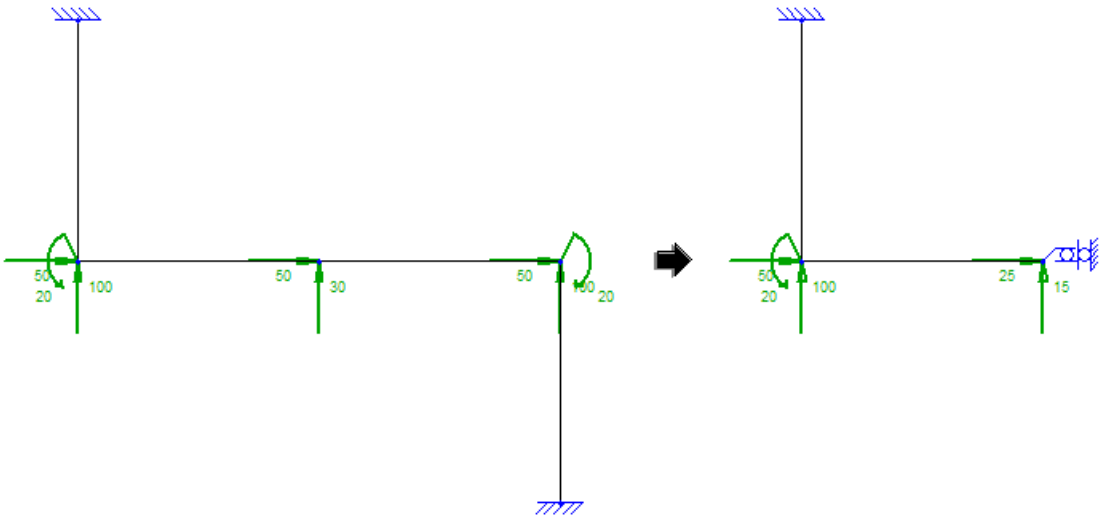
(c) 2022 EngiLab PC

### 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**.

(c) 2022 EngiLab PC

*This page intentionally left blank.*

# Chapter




## Example Problems

# 7 Example Problems

## 7.1 Example Problem 1 (based on the 2015 version)


- 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

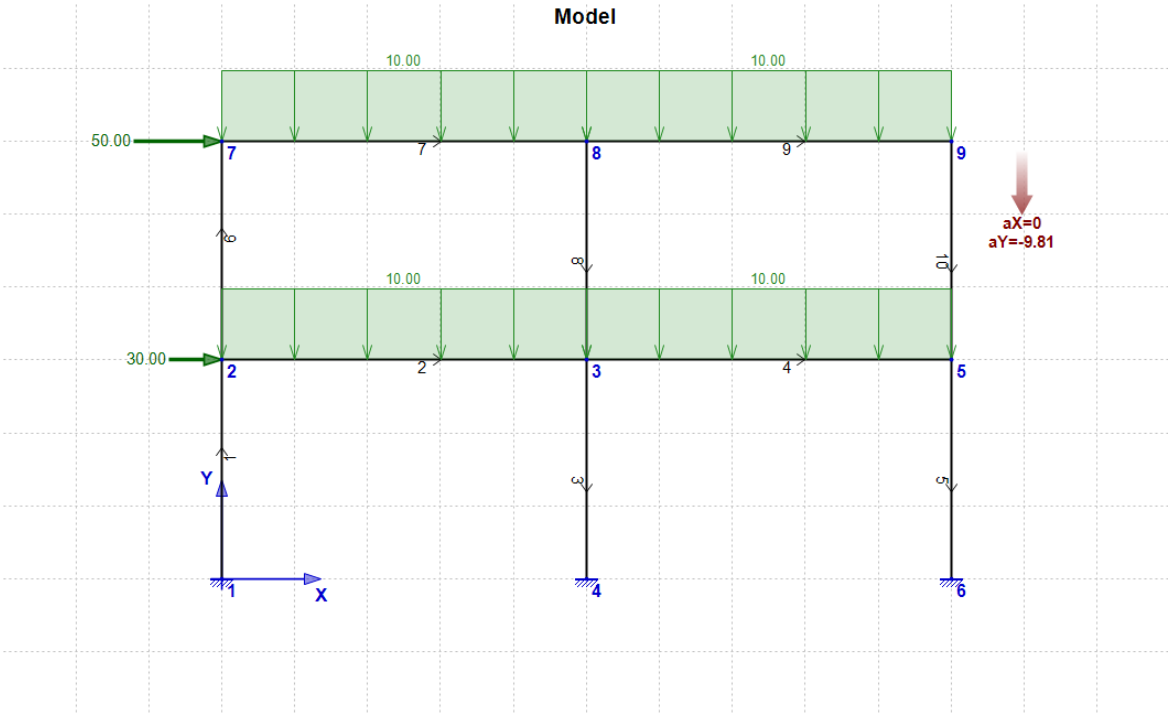

Note that this first example in based on the 2015 version of EngiLab Frame.2D. The second example is based on the 2022 version. There are some differences between the two versions of the program, the most important being the introduction of Units in the 2022 version. Before that, only consistent units were supported by the program.

Although the first example is based on an older version of the software, the user can still follow the steps described and generate this Model also in newer versions of the software such as EngiLab Frame.2D 2022.

### 7.1.1 Overview - Example 1

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

(c) 2022 EngiLab PC




(c) 2022 EngiLab PC

**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).

(c) 2022 EngiLab PC

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

**Note:**

**This first example was made before the release of EngiLab Frame.2D 2022 (v4.0)**. The 2022 version introduced Units in the model. Before that the program did not support individual units directly and was working only with a consistent system of units. This example is based on this. Although the problem itself is described with specific units, these units are converted to a consistent system of units and the program works with this system. In other words, in this example we convert all quantities to equivalent quantities in a consistent units and then they are given to the program.

Another approach (using EngiLab Frame.2D 2022 or later) would be to specify the units of the problems exactly as they are given by the problem and then give quantities needed, without any conversions.

**In this demonstration, we will use the consistent system of units as this specific example is compatible with version 2015 of EngiLab Frame.2D.**

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 (**in versions earlier that 2022**). 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 the Consistent 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 2022 (v3.5) supports 715 units in 22 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/T²** (1 Length unit) / (1 time unit)². In our example: **m/s²**
- The **Mass unit** is given by: **F·T²/L** (1 force unit) / (1 acceleration unit). In our example: kN/(m/s²) = Mg = **t** (metric ton) (**IMPORTANT**: t has to be used instead of kg!)
- The **Density unit** is given by: **F·T²/L⁴** (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/L²** (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**

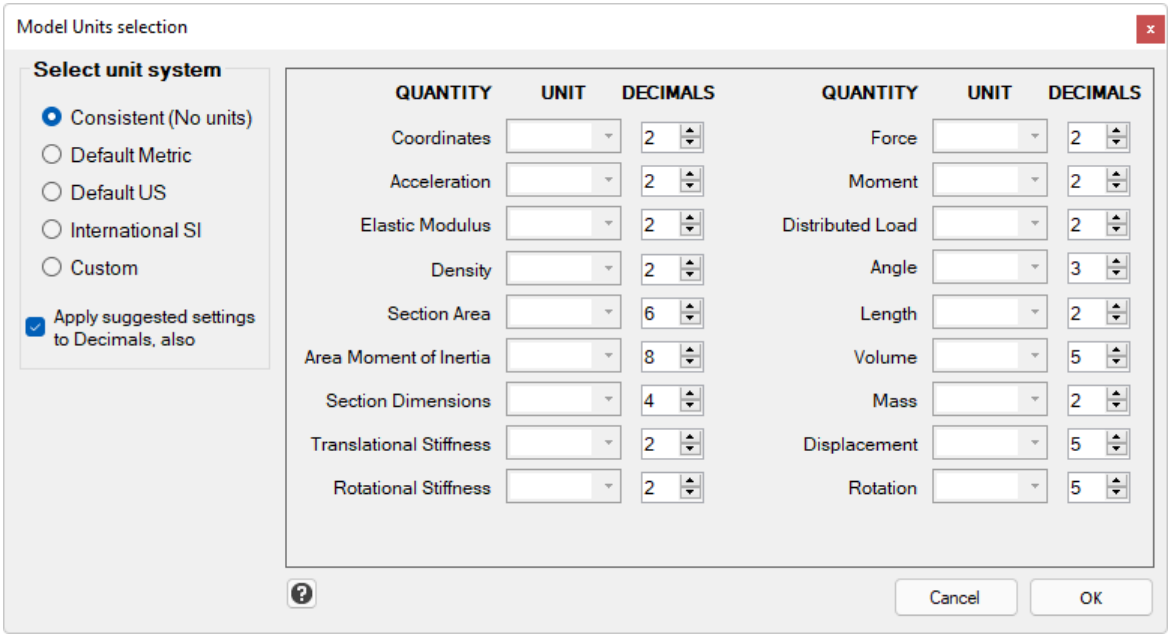
In EngiLab Frame.2D before 2022 (v4.0), there are no units so there is nothing to define in the program as the Consistent System of Units is always used.

In EngiLab Frame.2D 2022 and later, we need to define the System of Units that we use, among many options. For this example, we will use the Consistent System, as follows:




(c) 2022 EngiLab PC

Click U to define the **System of Units**. In this example, we will use the **Consistent** System of units. For details, see Units




Select the **Consistent** System of Units. Click OK.

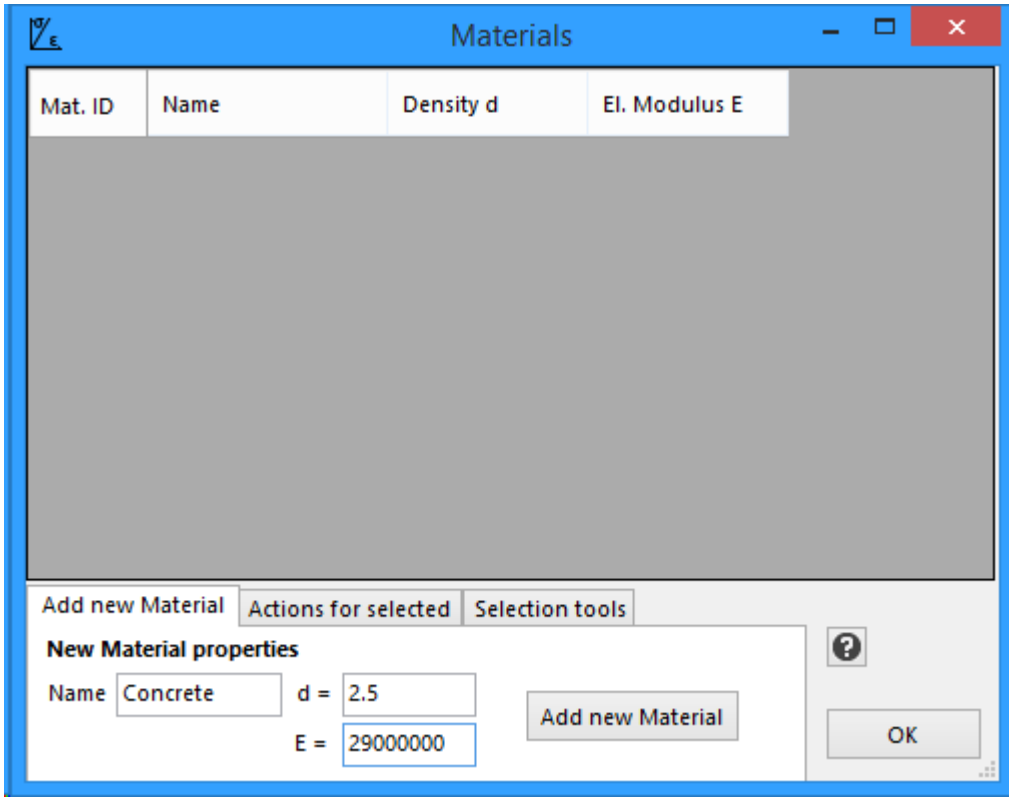
(c) 2022 EngiLab PC

### 7.1.3 Step 2. Define Materials

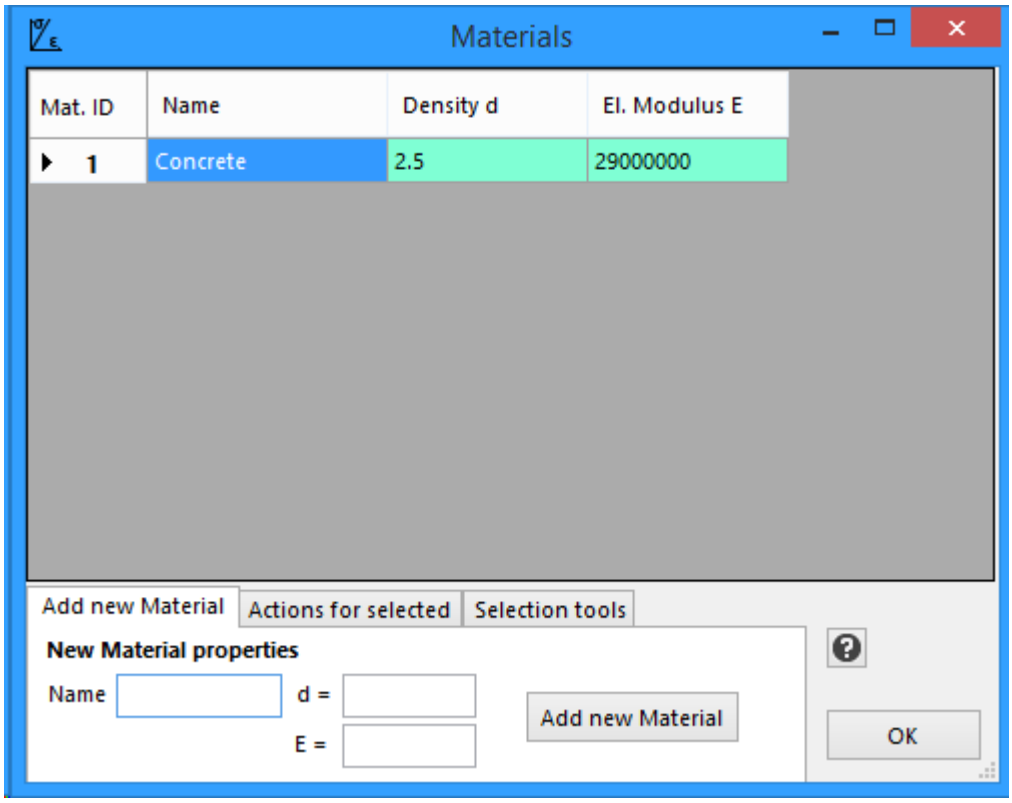



After the preparation of the input data, it is time to start working with the program. Click to define **Material**(s). For details, see Materials.

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.



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**.

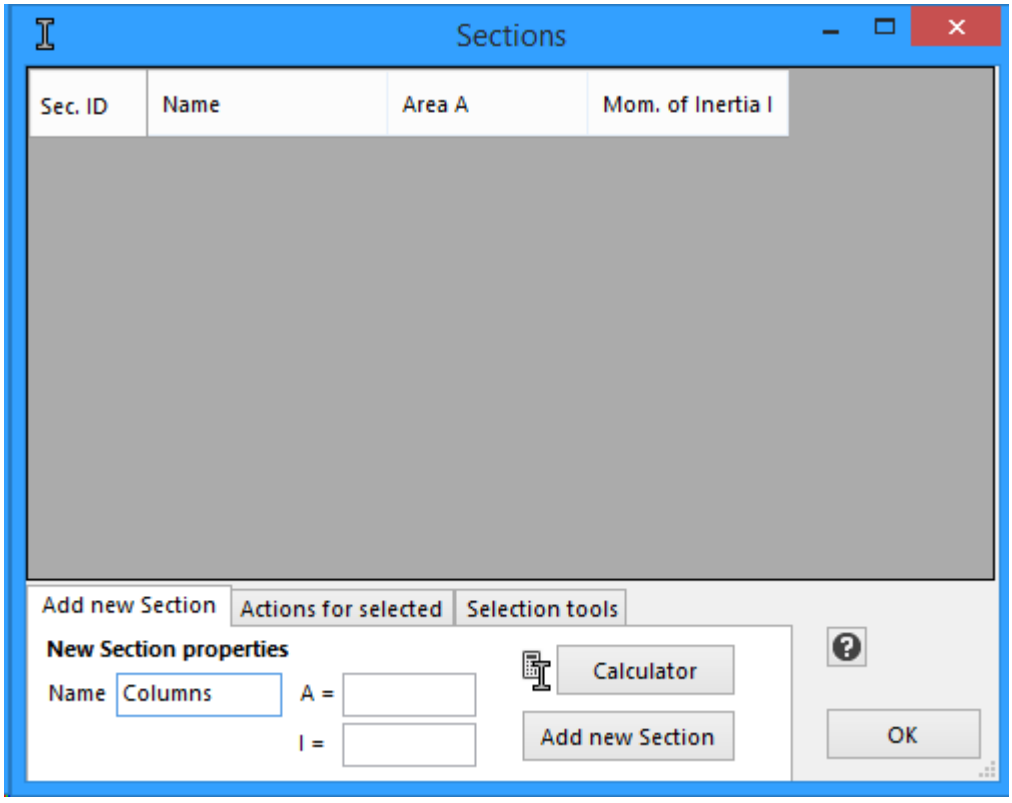
(c) 2022 EngiLab PC

### 7.1.4 Step 3. Define Sections

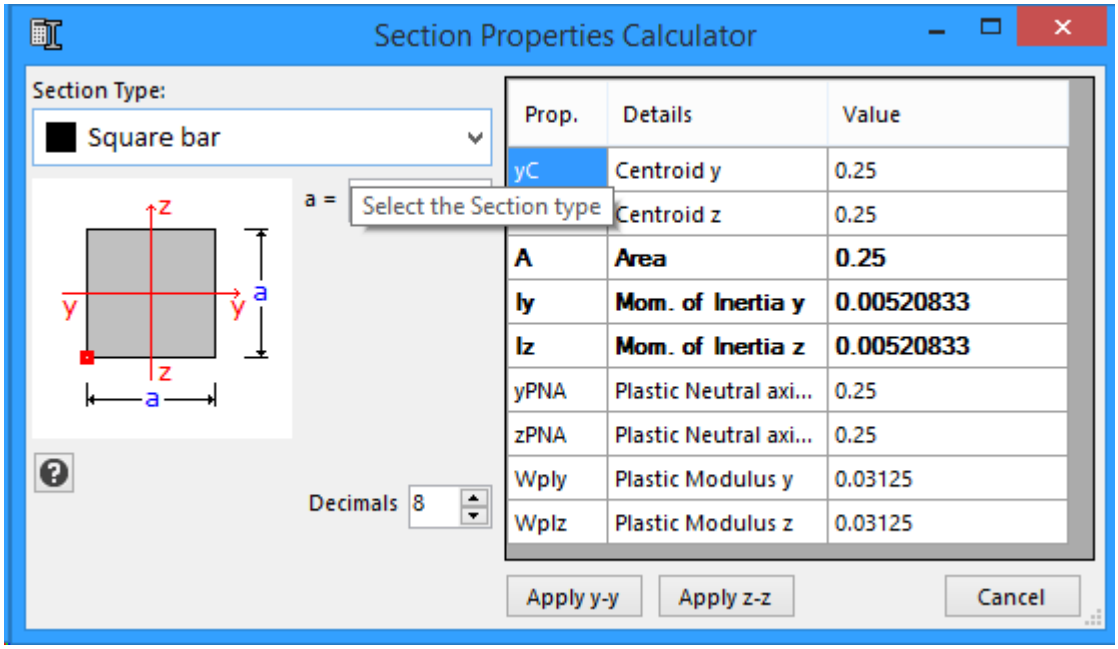



After the definition of the Materials, you need to define the Sections of the Model. Click I to define **Sections**(s). For details, see Sections.

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.

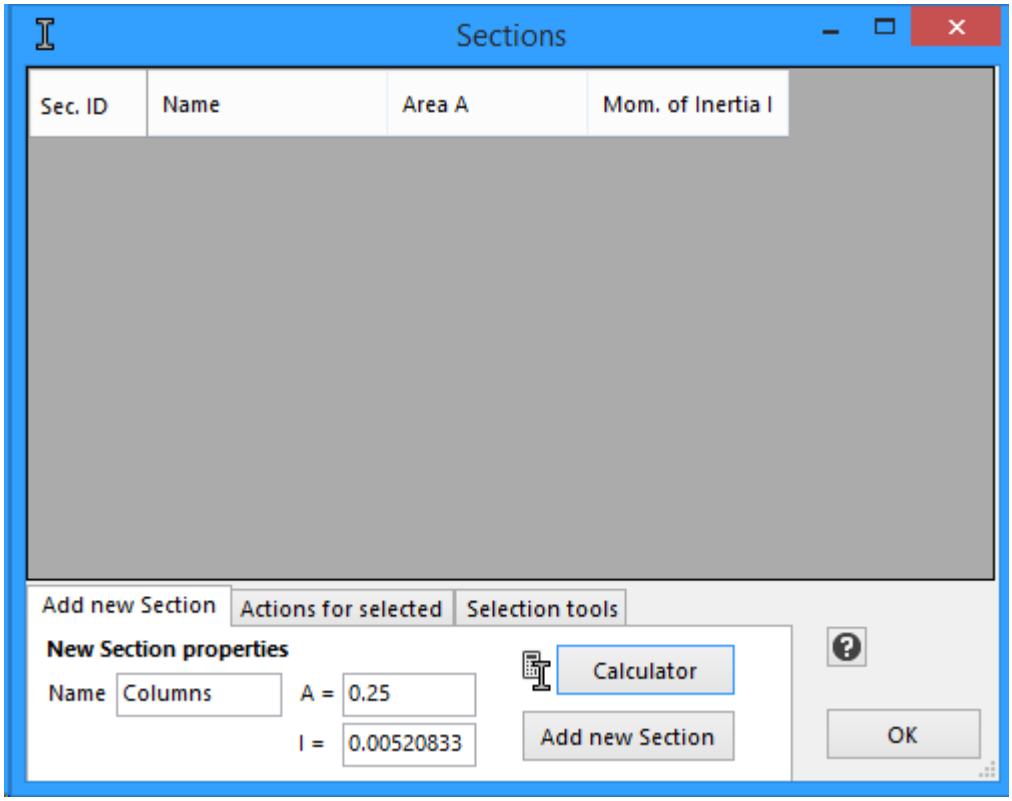


You can calculate the Section properties (A and I) yourself, but the easiest way is to use the "Section Properties Calculator" that is built in EngiLab Frame.2D. **Click the "Calculator" button**. The Section Properties Calculator form appears. Select the Square Bar Section and type "0.5" for the a dimension,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

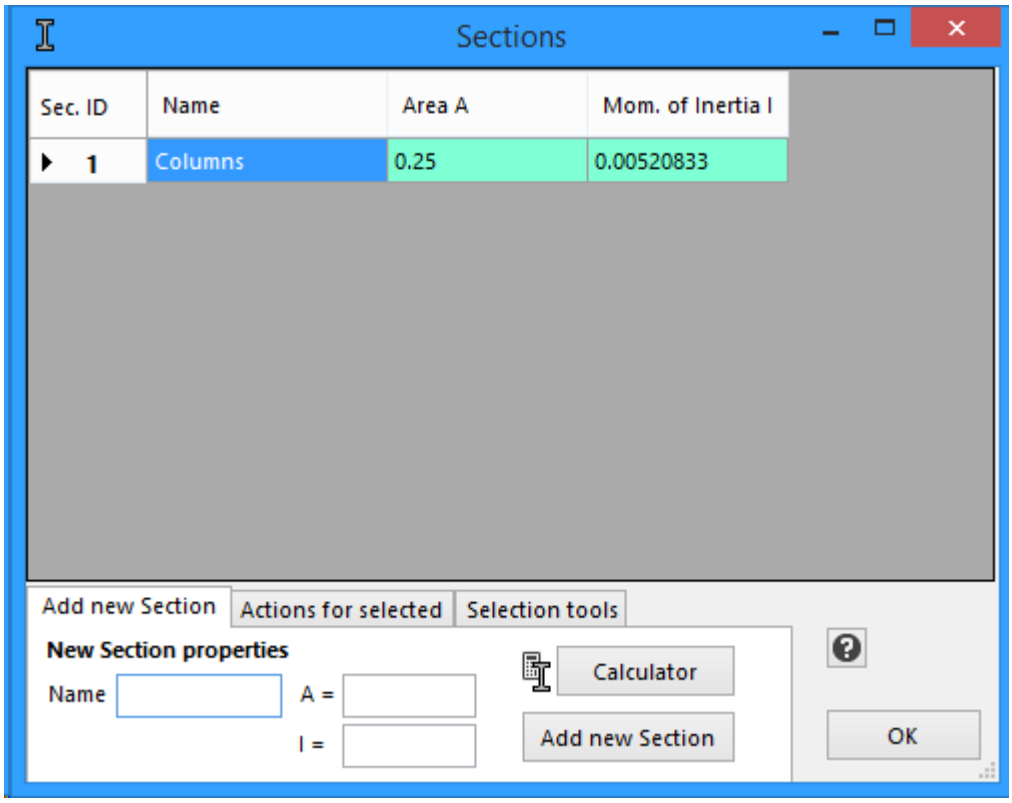



(c) 2022 EngiLab PC

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

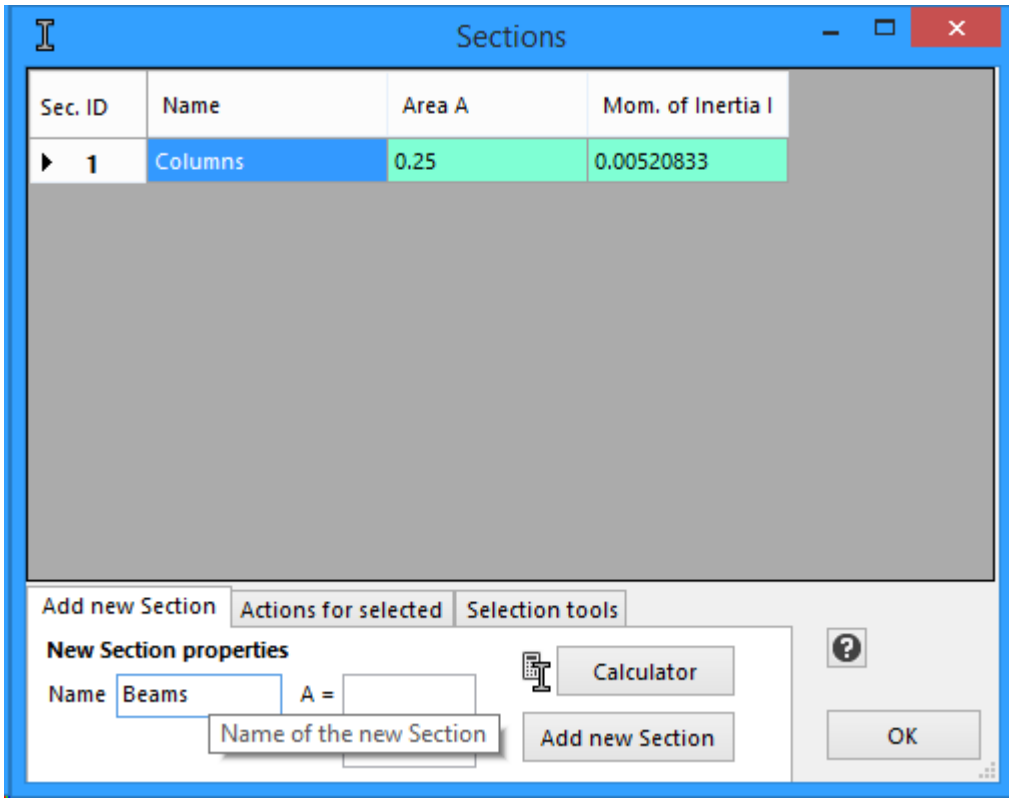


Now **Click the "Add new Section" button**. The new Section will be added to the table that shows the Model Sections.

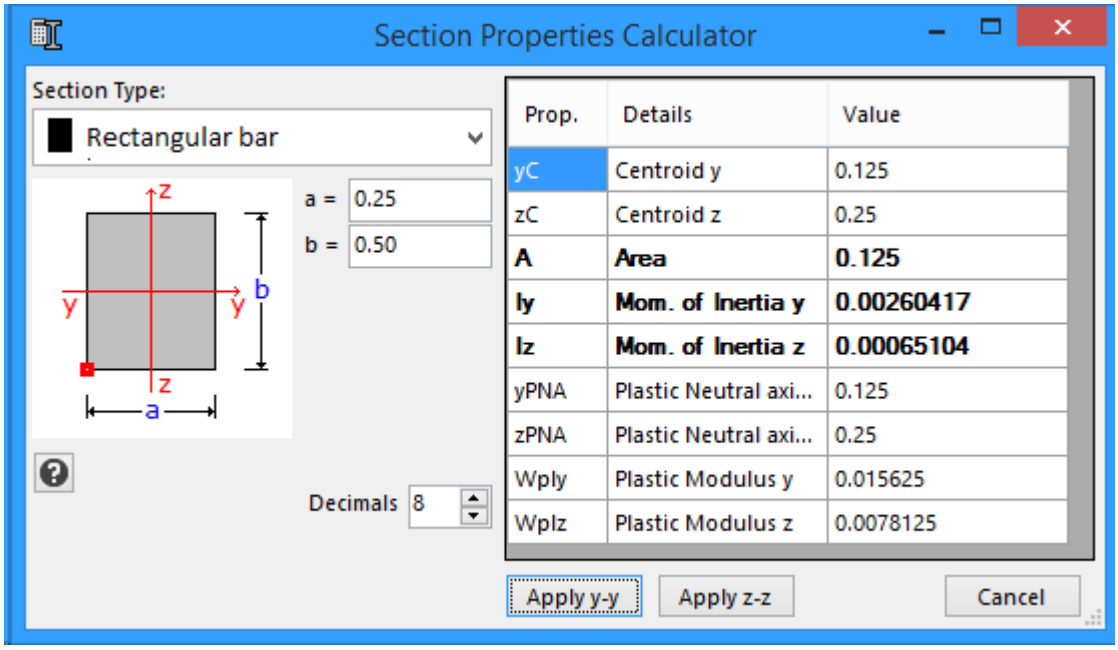



(c) 2022 EngiLab PC

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

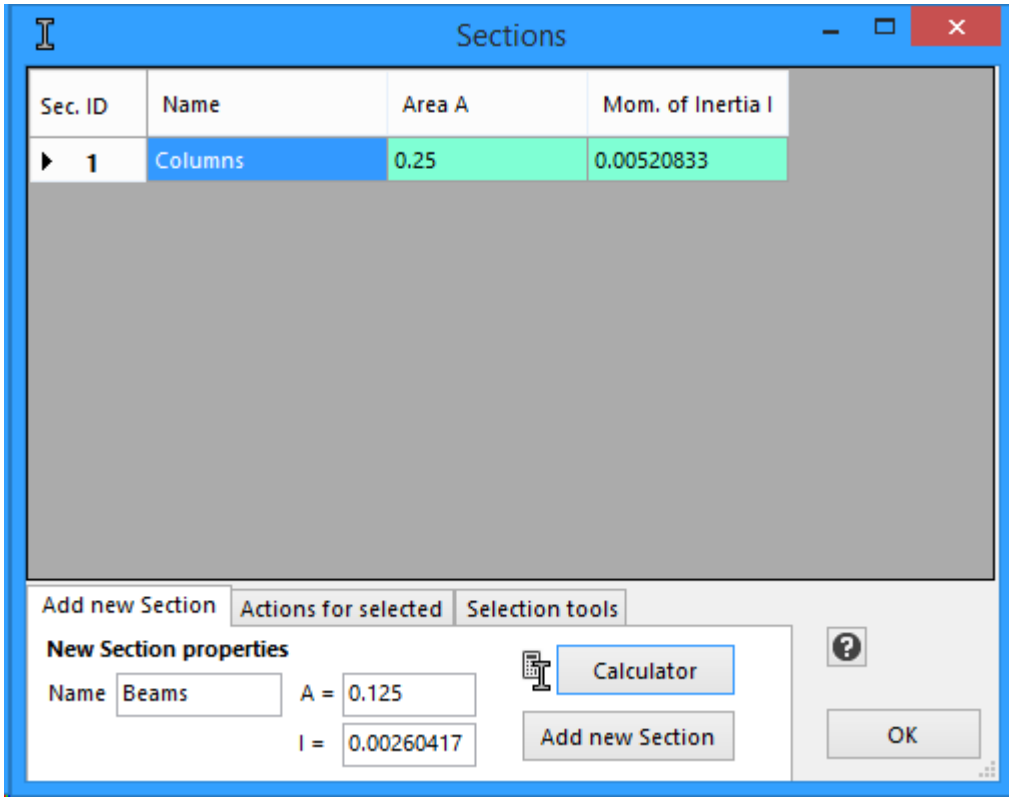


You can use the Section Properties Calculator again. **Click the "Calculator" button**. The Section Properties Calculator form appears. Select the **Rectangular Bar Section** and type "0.25" for the a dimension and "0.50" for the b dimension, as shown below. Press ENTER after you have finished, in order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

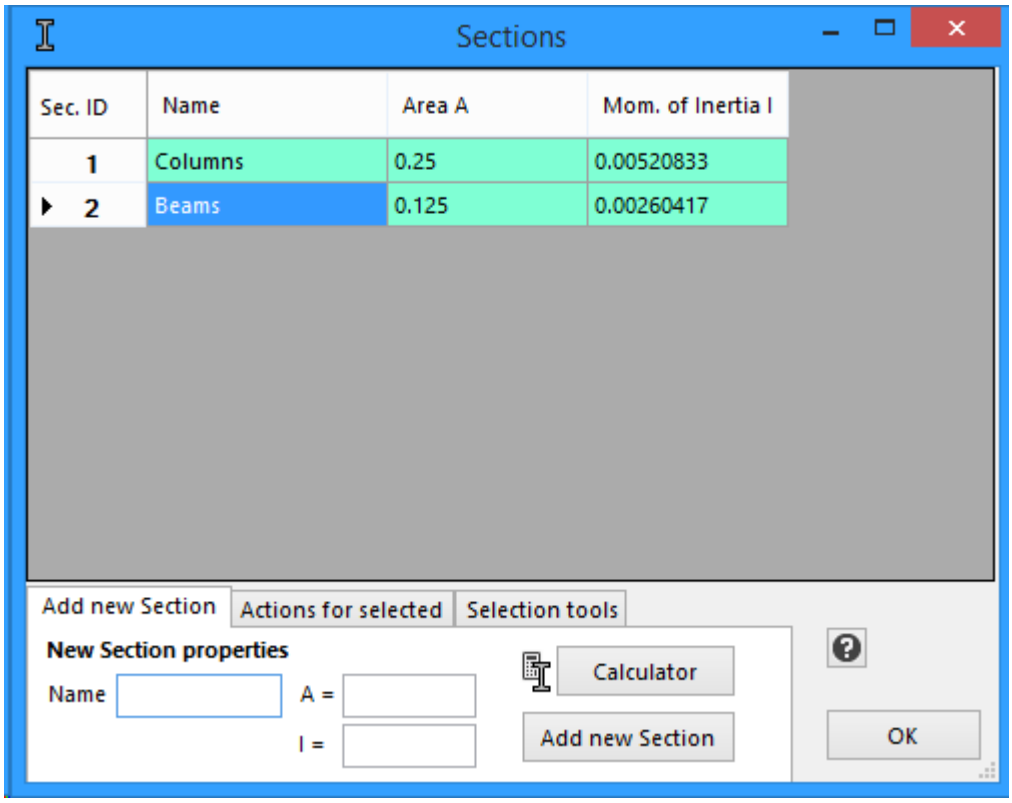



(c) 2022 EngiLab PC

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



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




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

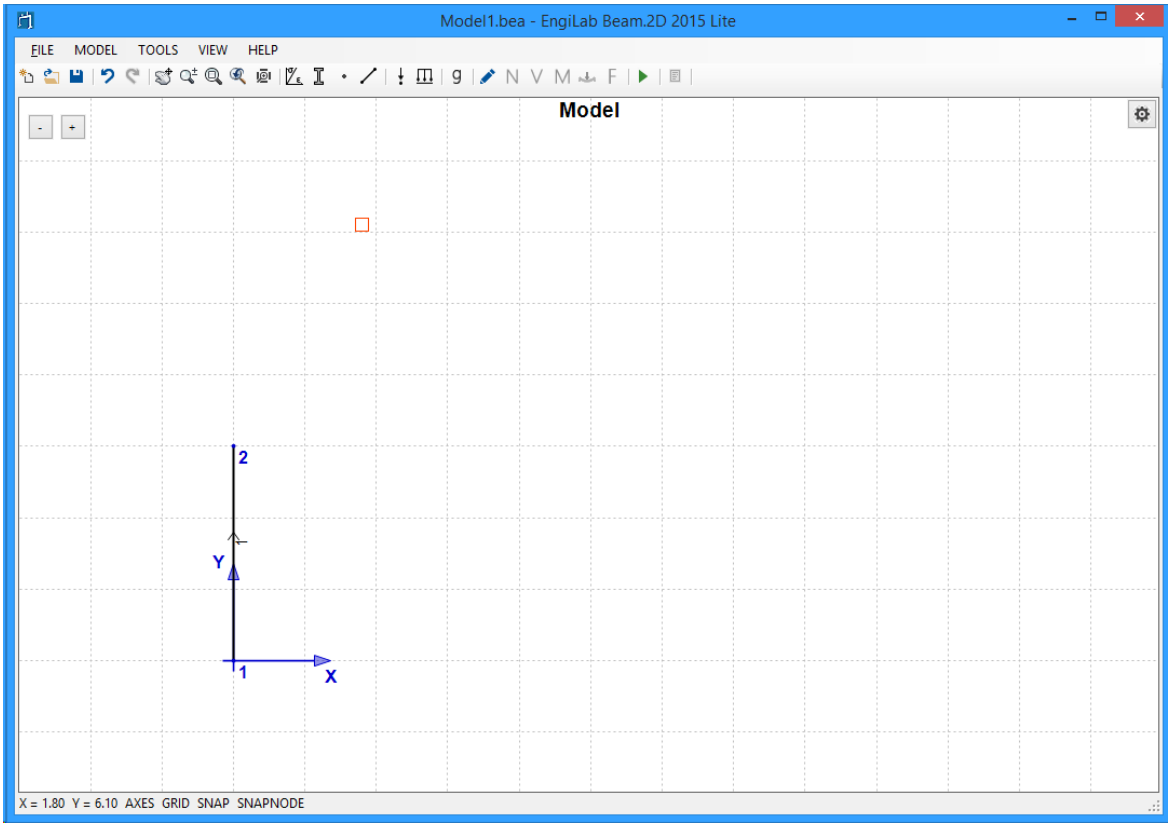
(c) 2022 EngiLab PC

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

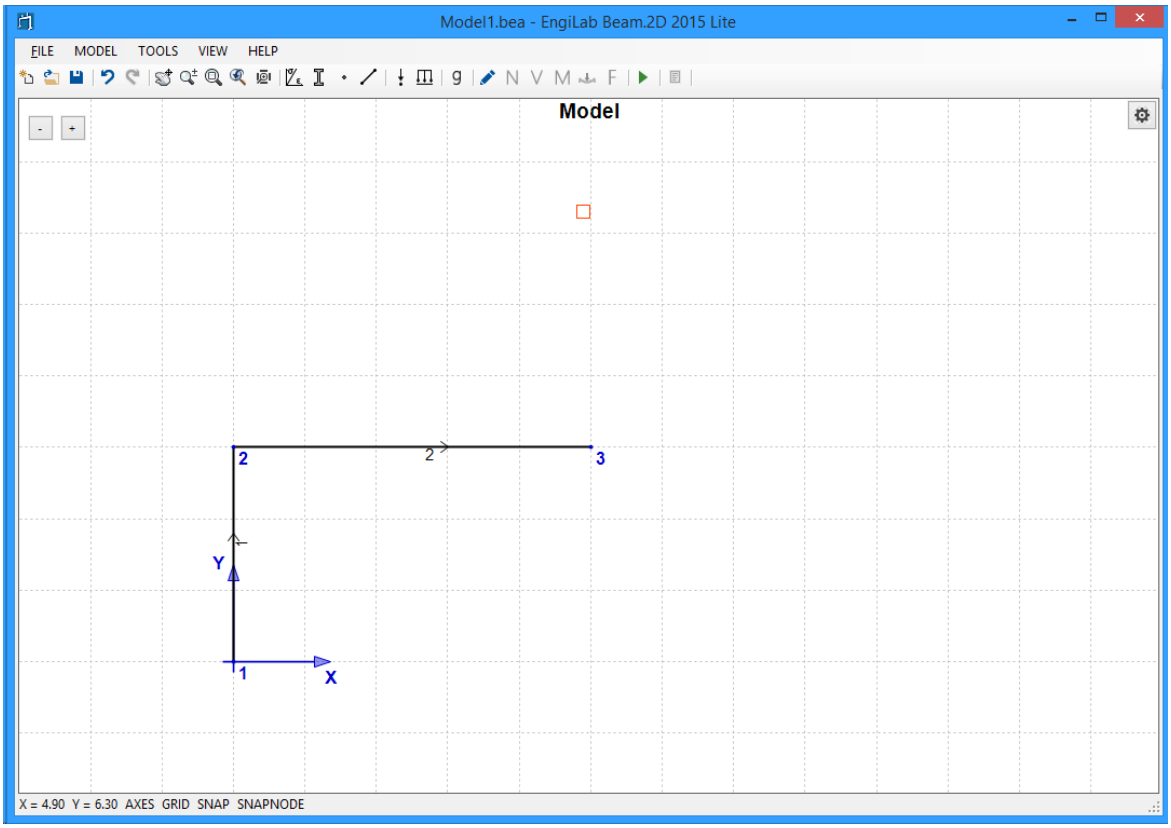
After you have defined the Material and the Sections, you can **start drawing your Model on screen**, as follows:

- 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.

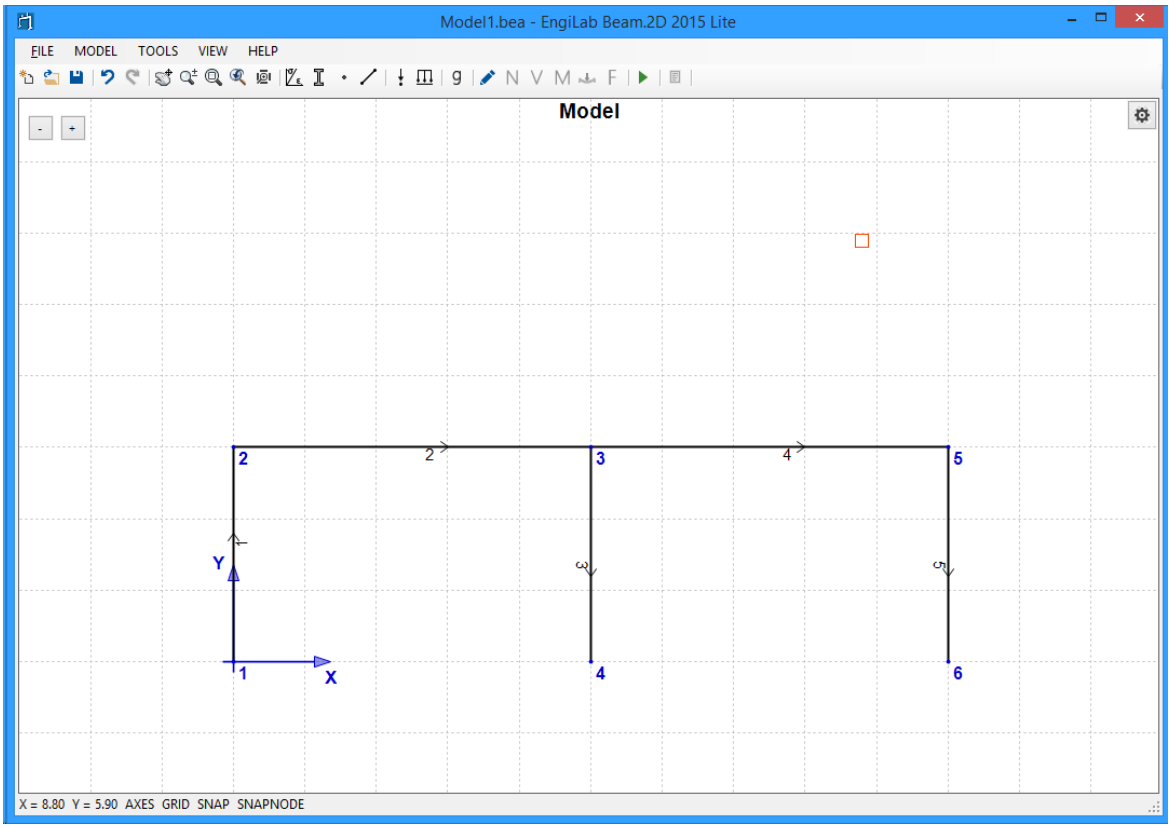


Continue with the Beam. Hold down the left button starting at Node 2 and release it at distance 5 in the X Direction. The picture should be as follows.

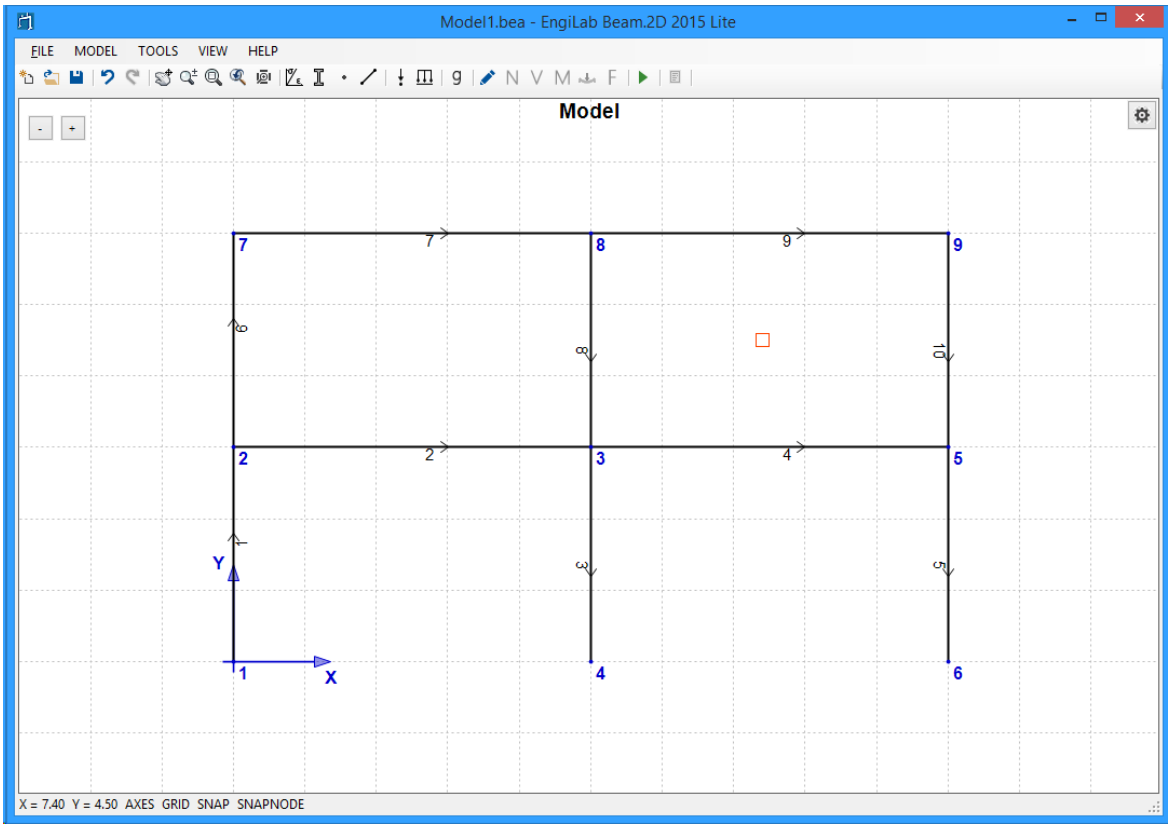



(c) 2022 EngiLab PC

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.

(c) 2022 EngiLab PC

### 7.1.6 Step 5. Edit Nodes

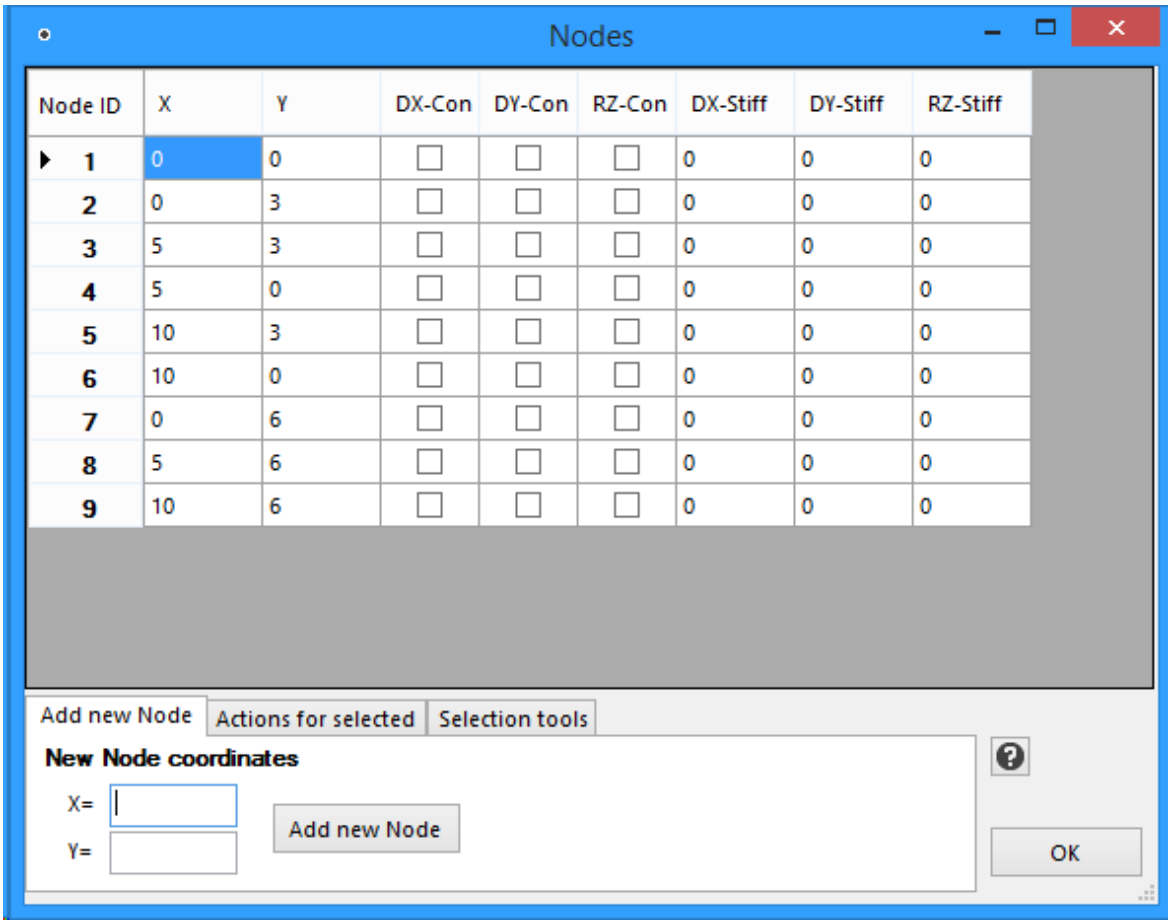



After the preparation of the Model on screen, you will need to do some corrections on the Nodes that have been generated.

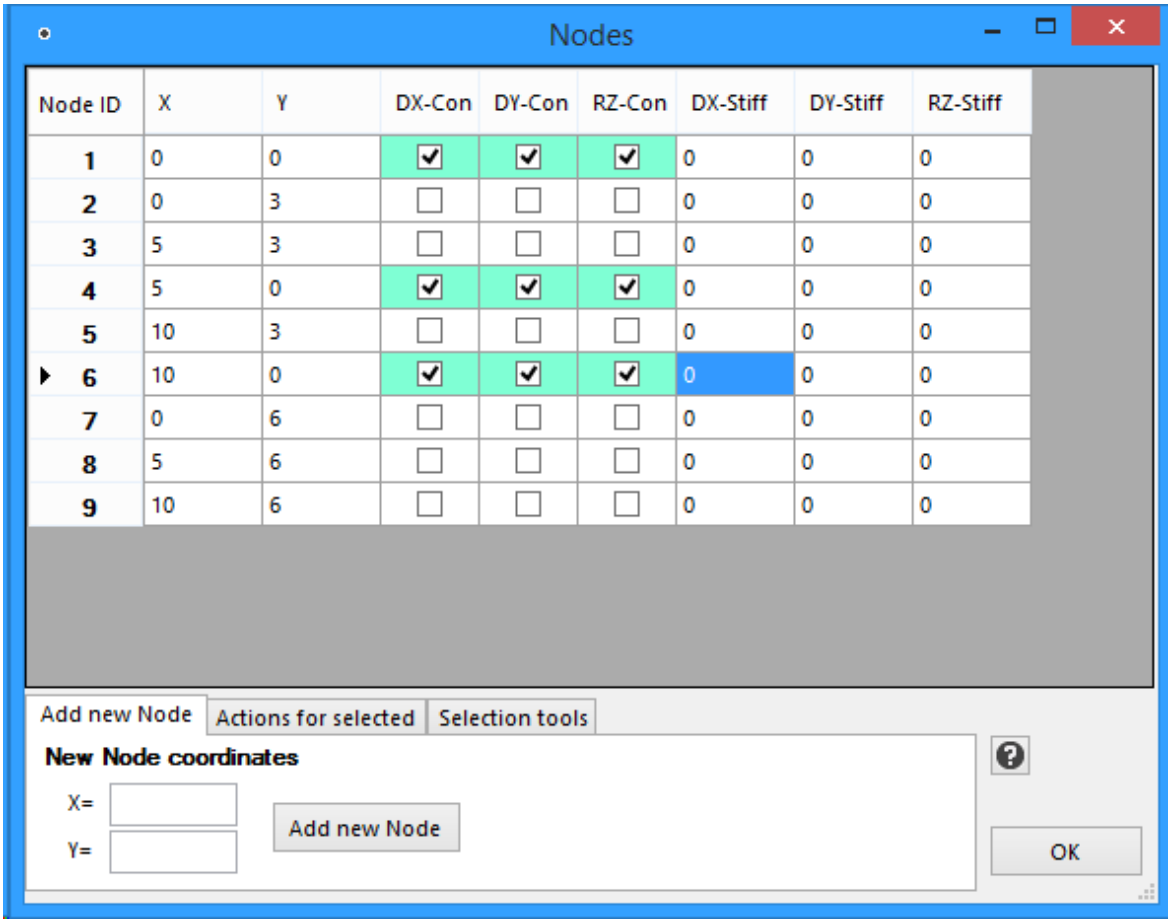
For example you may want to:

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

Click to edit **Nodes**. For details, see Nodes.

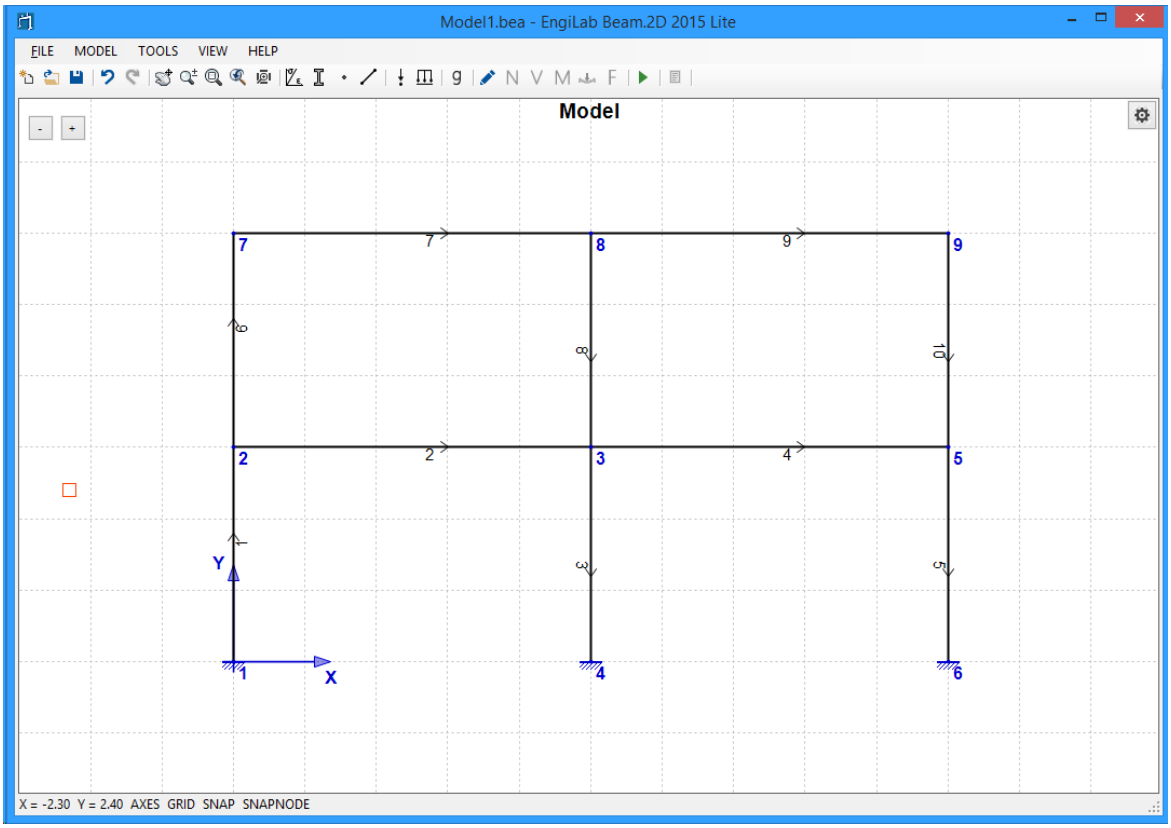


In our example, there is no need to move Nodes, as they are already in their correct positions. What we need to do is to add Constraints (Supports) to the Model. **Set the Constraints for Nodes 1, 4 and 6** as follows.




(c) 2022 EngiLab PC

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




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

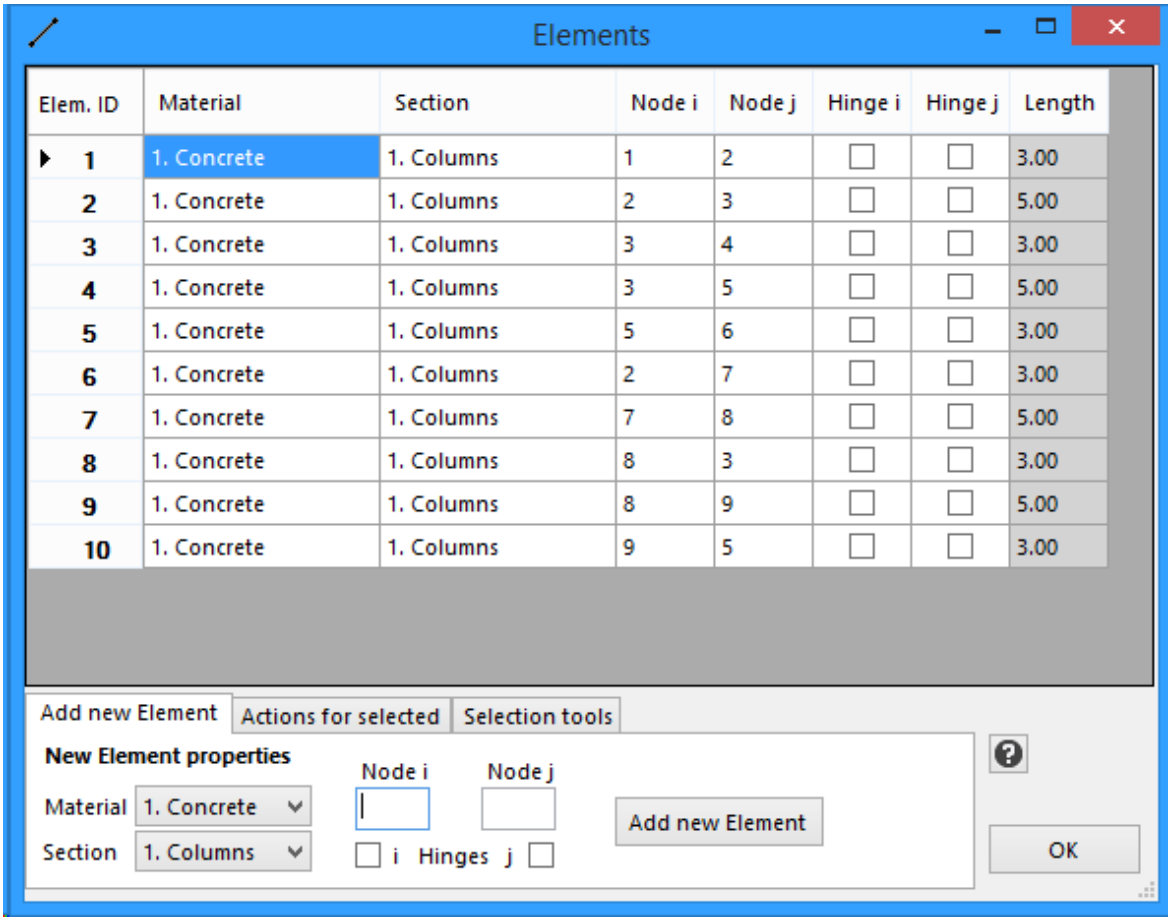
### 7.1.7 Step 6. Edit Elements




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

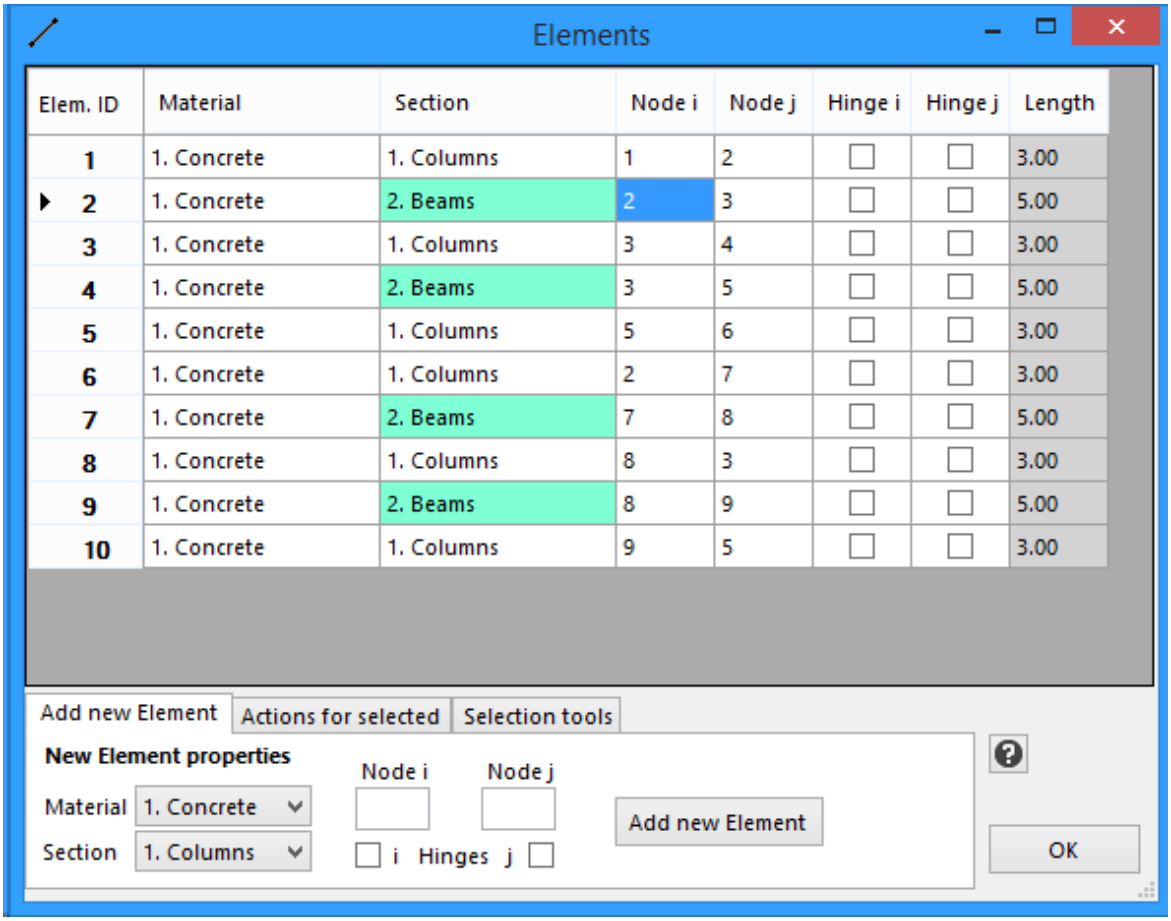
- **Assign the right Material and Section to every Element**, if needed. All Elements that are defined on screen are assigned Material 1 and Section 1.
- Define Hinges at Element ends. Each Element can have a hinge at either end (Start-i or End-j or both).

Click to edit **Elements**. For details, see Elements.




(c) 2022 EngiLab PC

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.

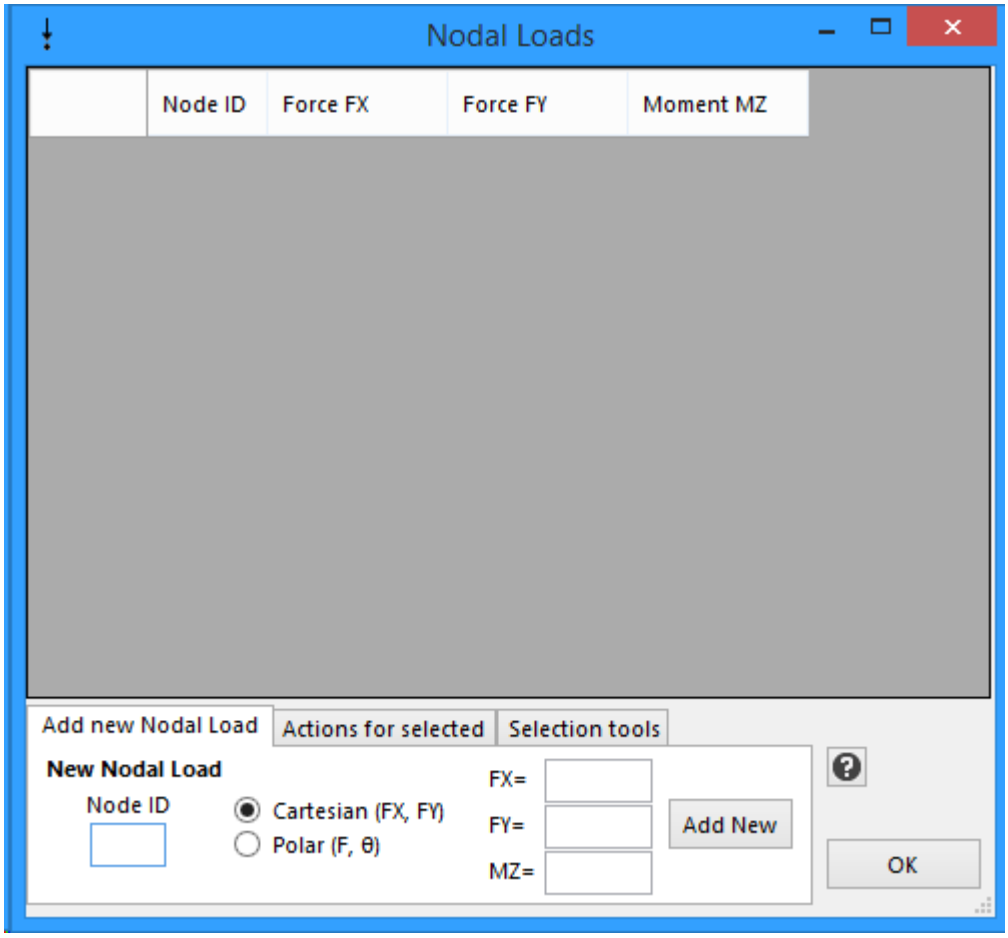
(c) 2022 EngiLab PC

### 7.1.8 Step 7. Define Nodal Loads



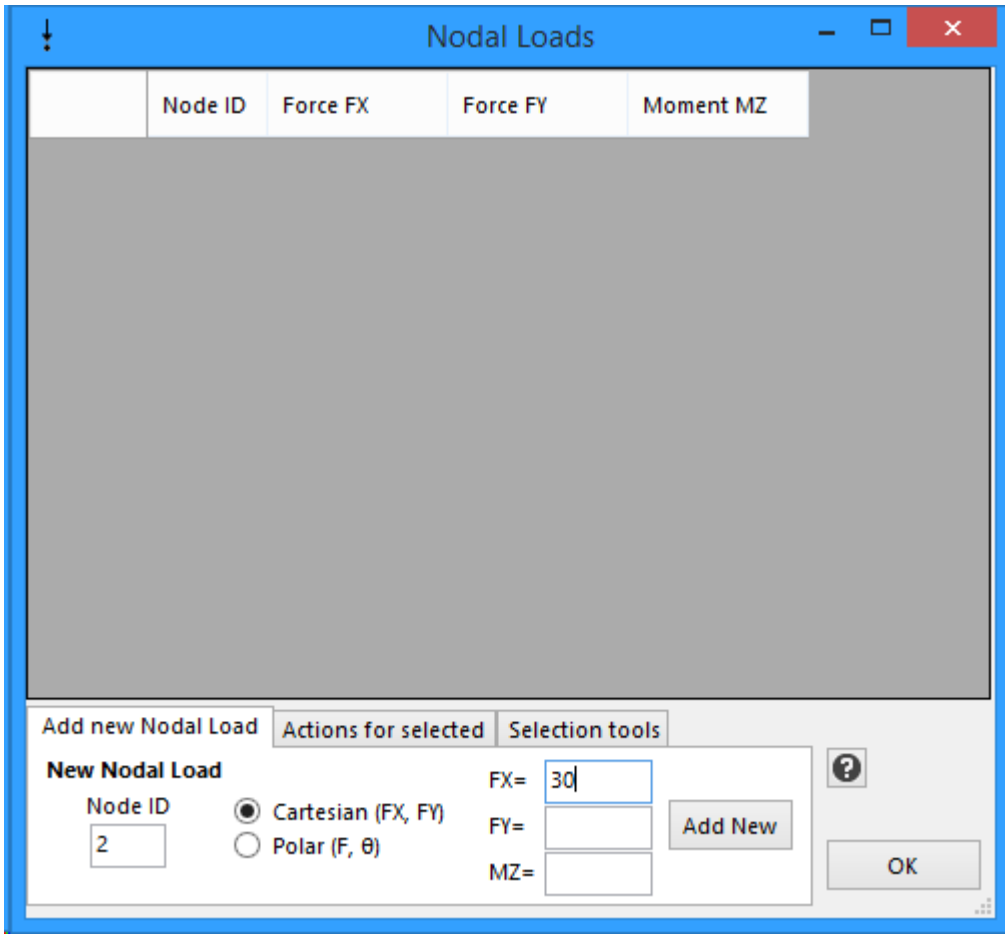

Click to define **Nodal Loads**. For details, see Nodal Loads.

The Nodal Loads form appears.

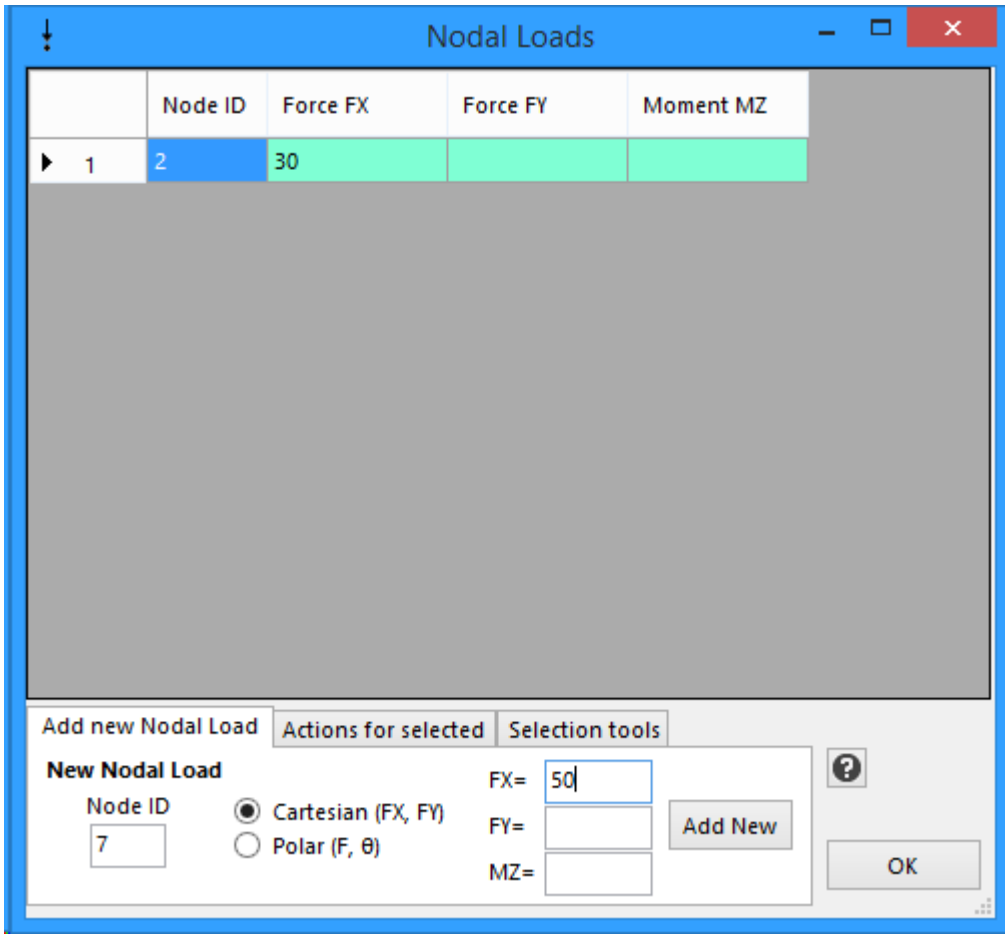



(c) 2022 EngiLab PC

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

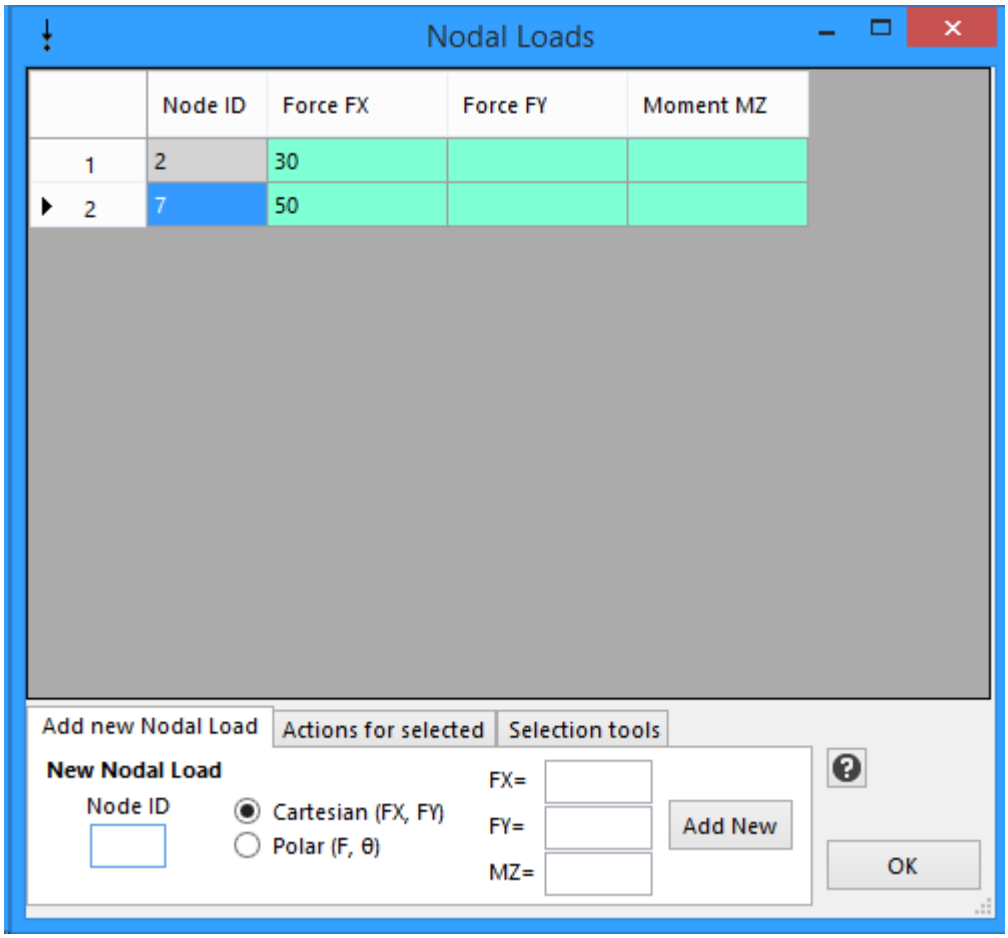


**Click "Add Nodal Load"**. Then add a Nodal Load **FX=50 for Node 7**.

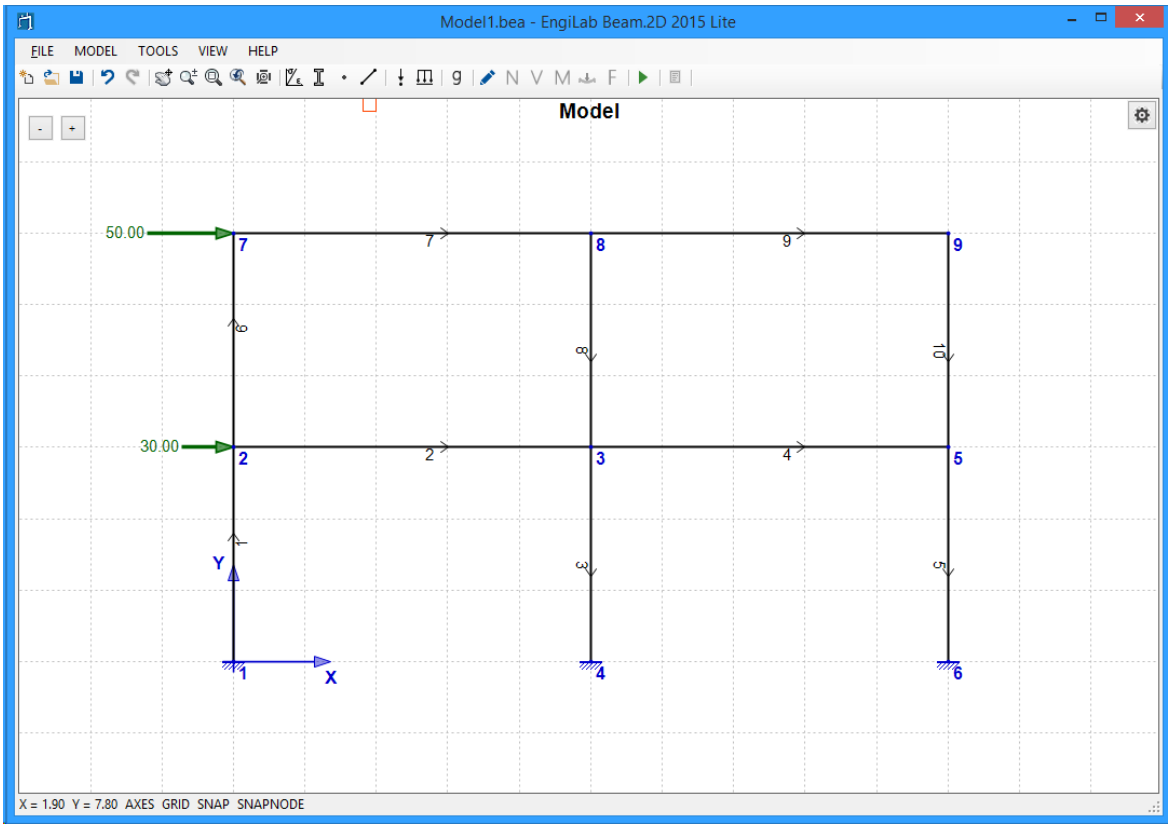



(c) 2022 EngiLab PC

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



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

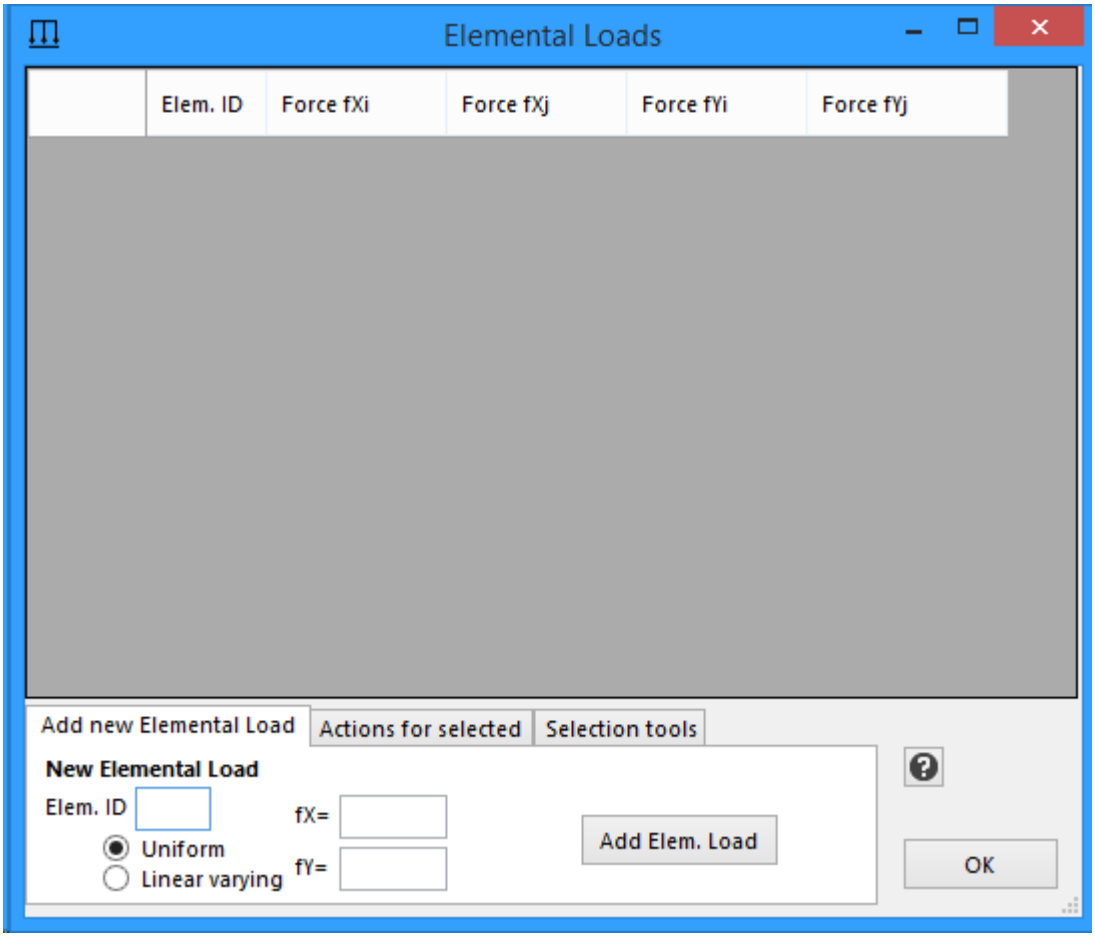


### 7.1.9 Step 8. Define Elemental Loads

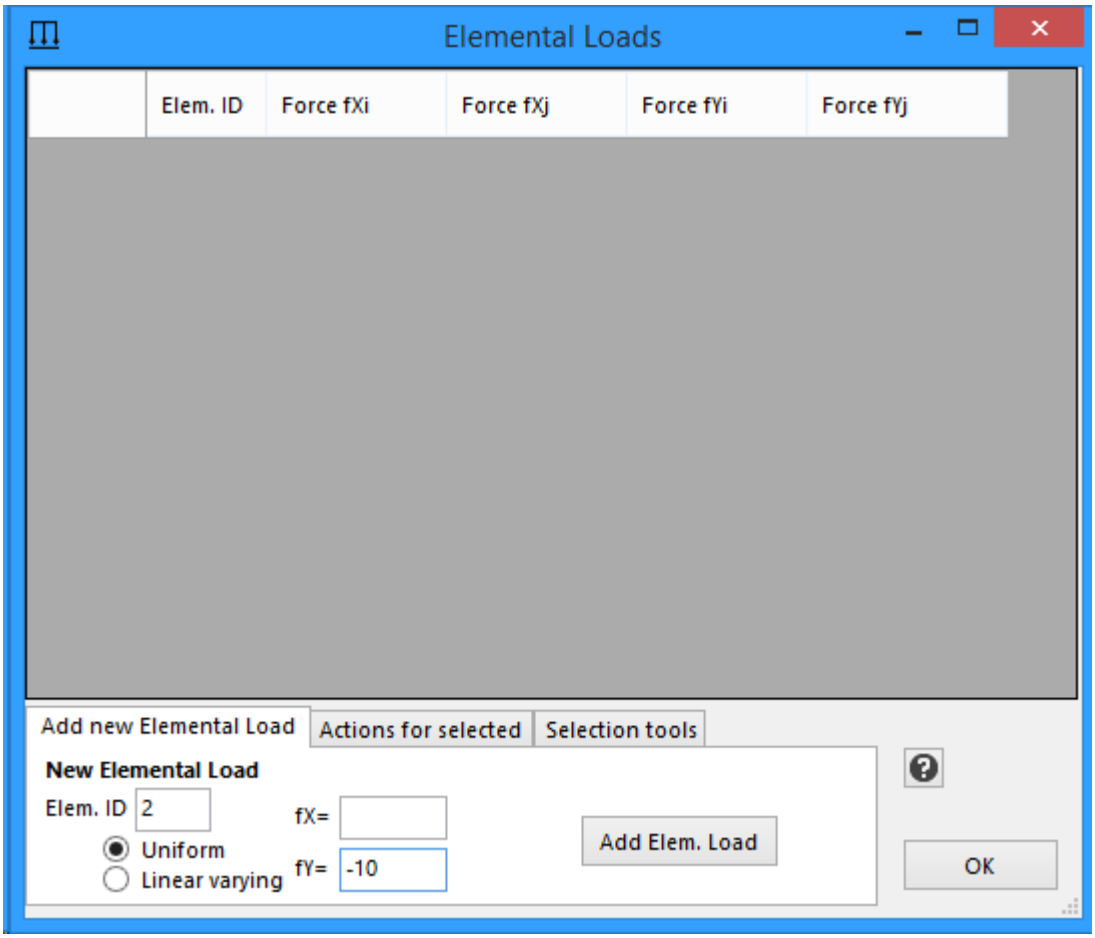



Click to define **Elemental Loads**. For details, see Elemental Loads.

The Elemental Loads form appears.

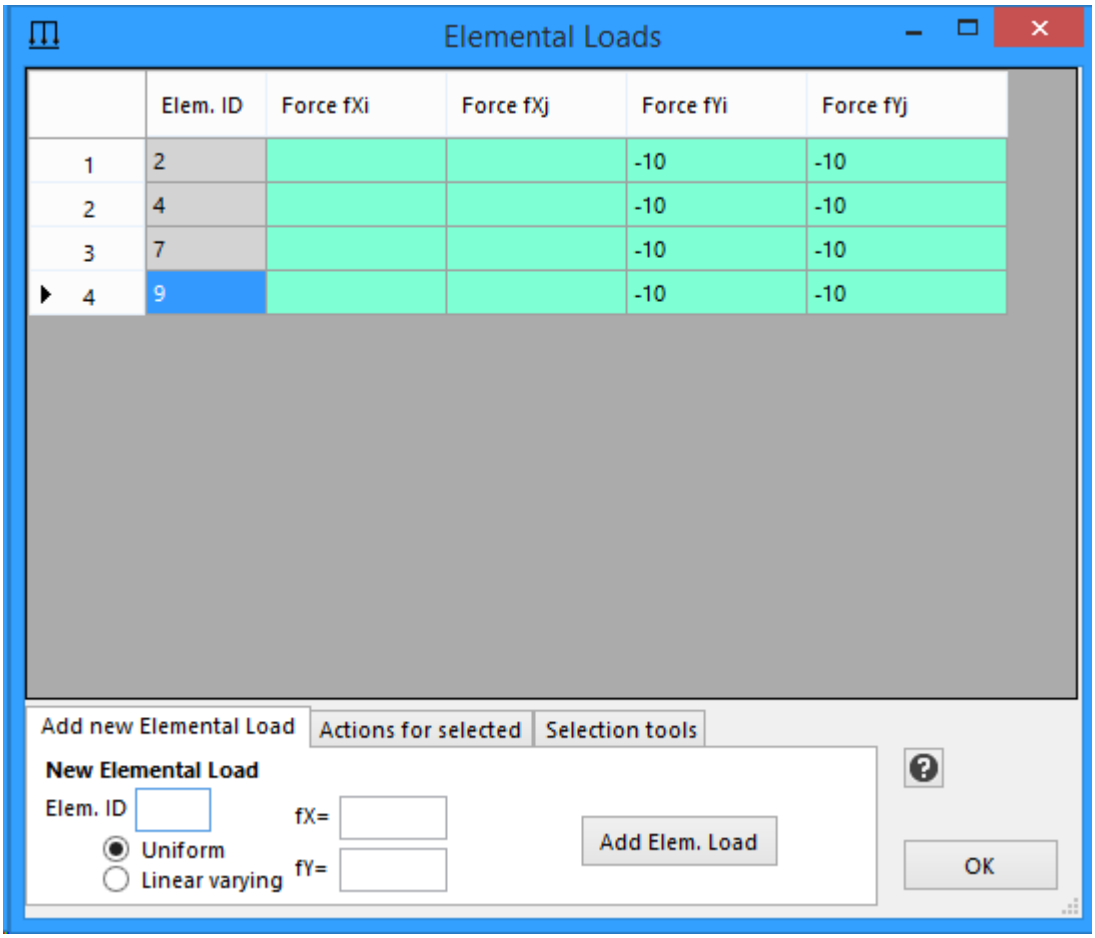


Add a **Uniform Elemental Load fY=-10 for Element 2**.

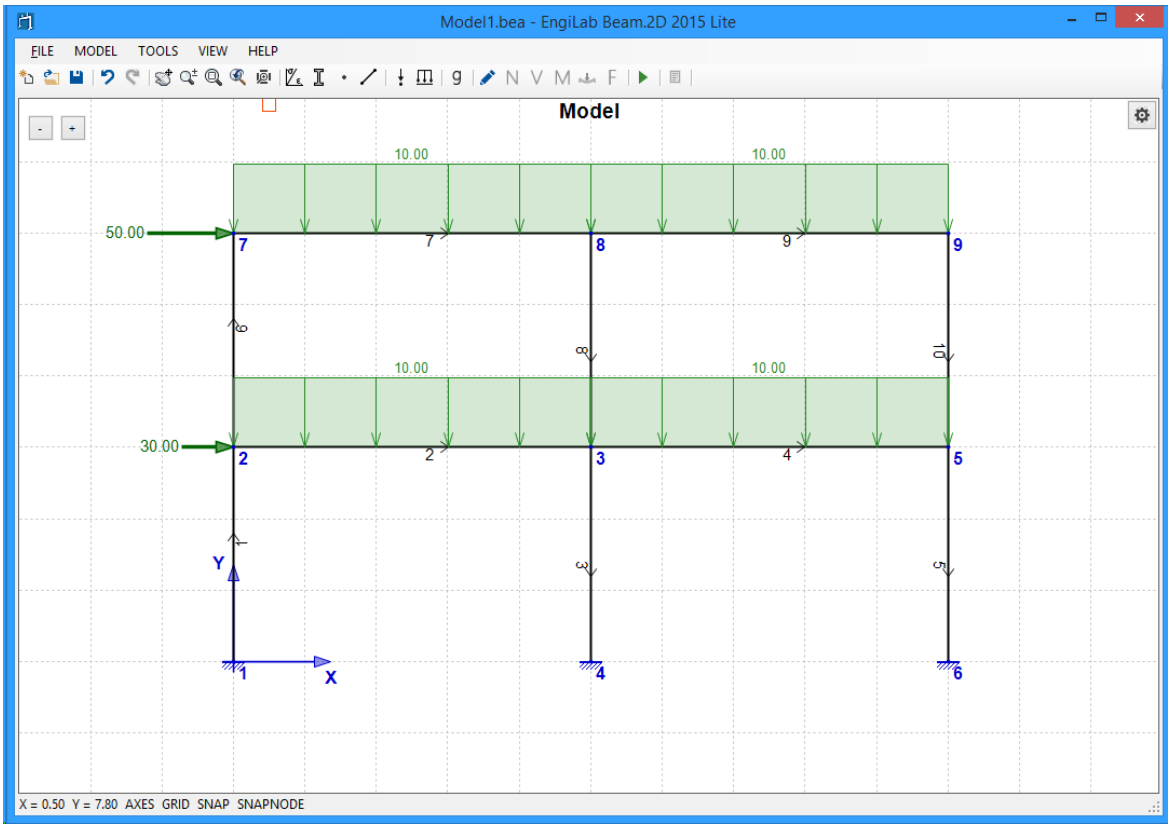



(c) 2022 EngiLab PC

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



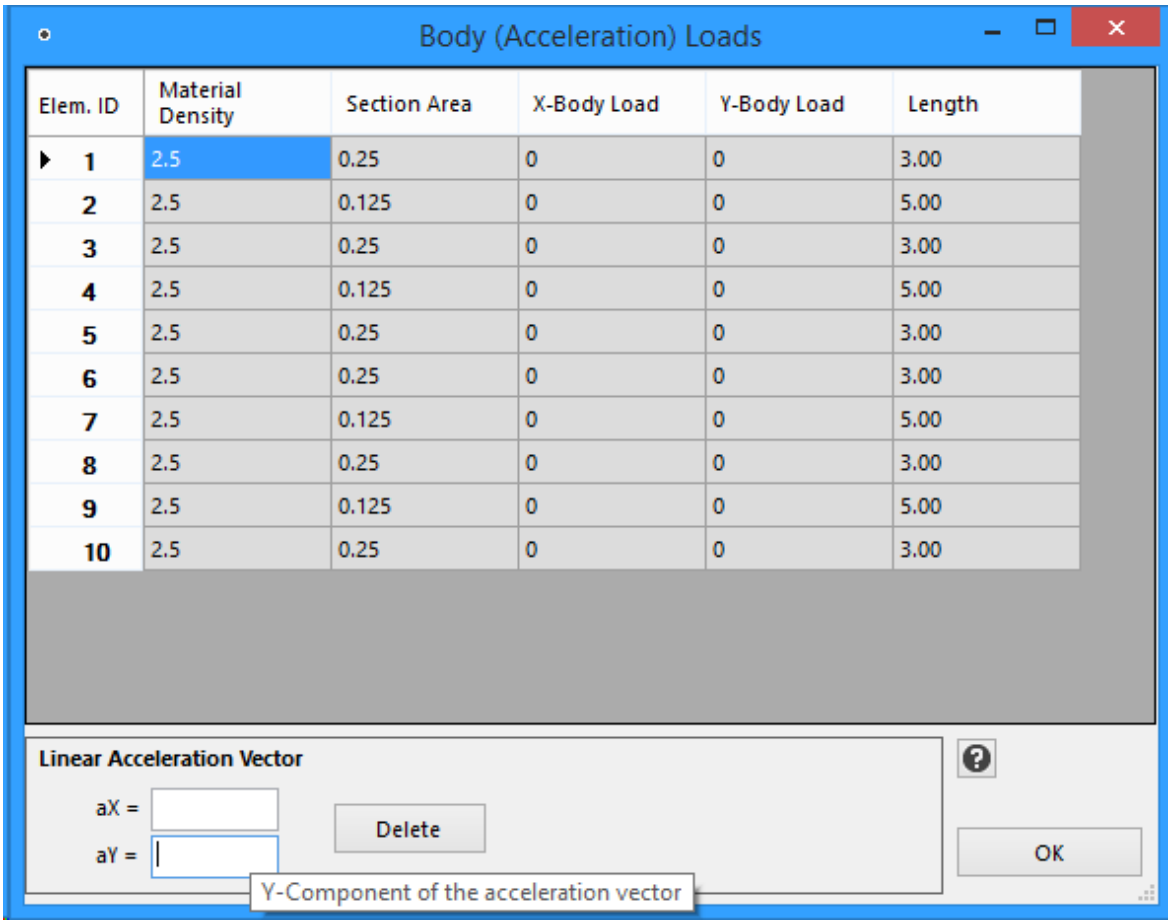
**Click OK to exit**. Now the Model should look like this.



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



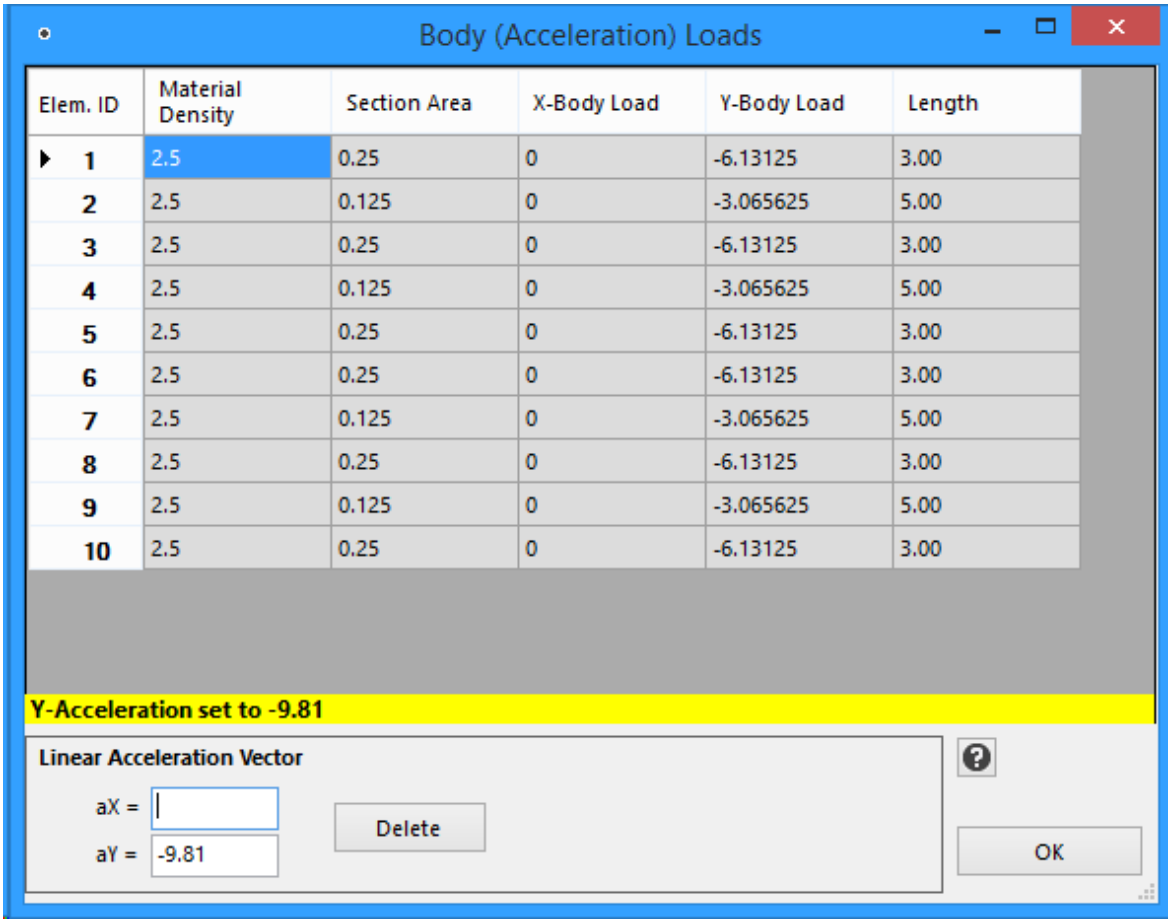

Click to define **Body (Acceleration) Loads**. For details, see Body (Acceleration) Loads.




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

(c) 2022 EngiLab PC

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




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

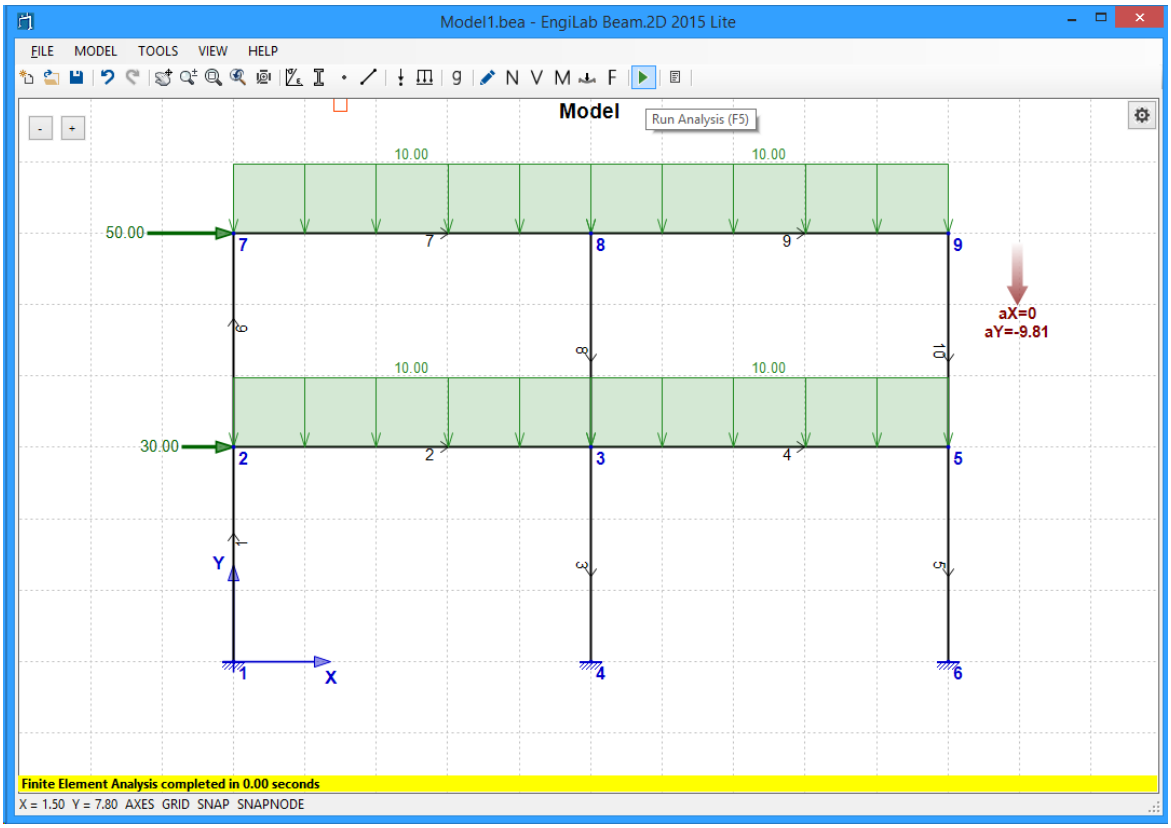
(c) 2022 EngiLab PC

### 7.1.11 Step 10. Run the Analysis




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

You should see the following picture.



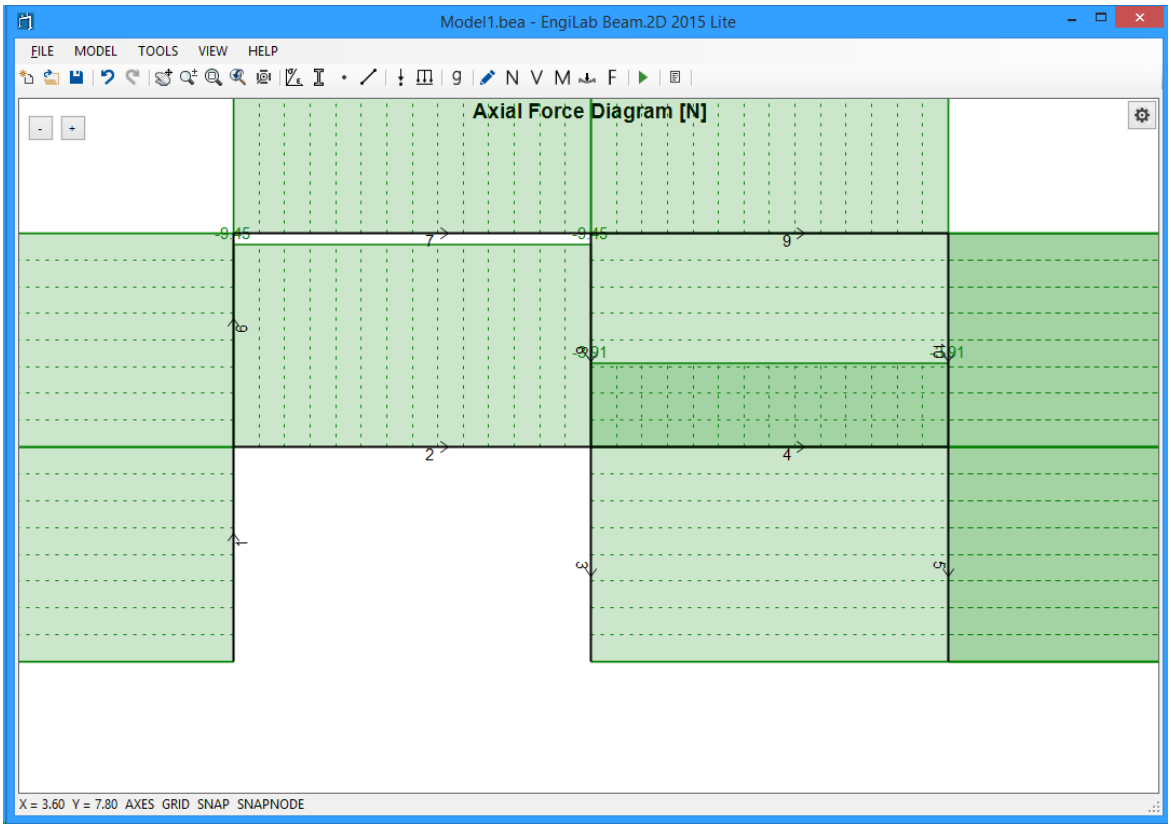

The Finite Element Analysis has been completed successfully.

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



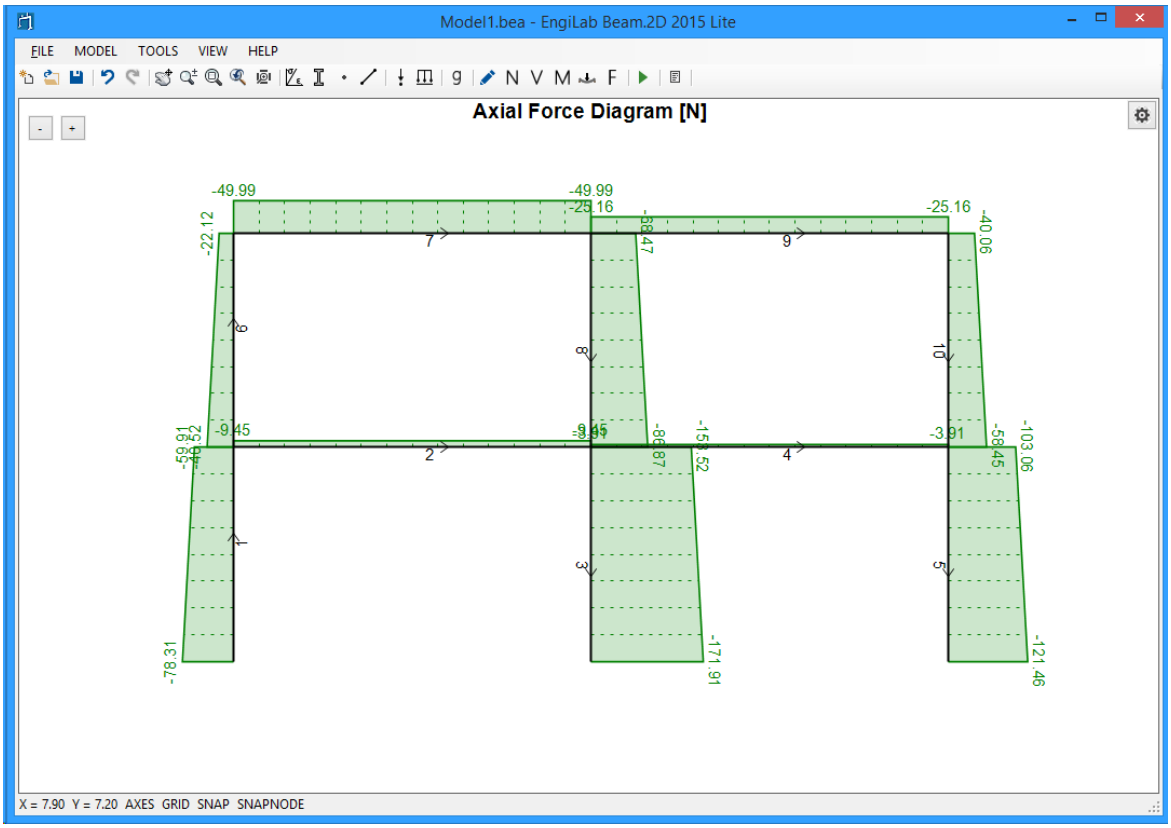

#### 1. Axial Force Diagram

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



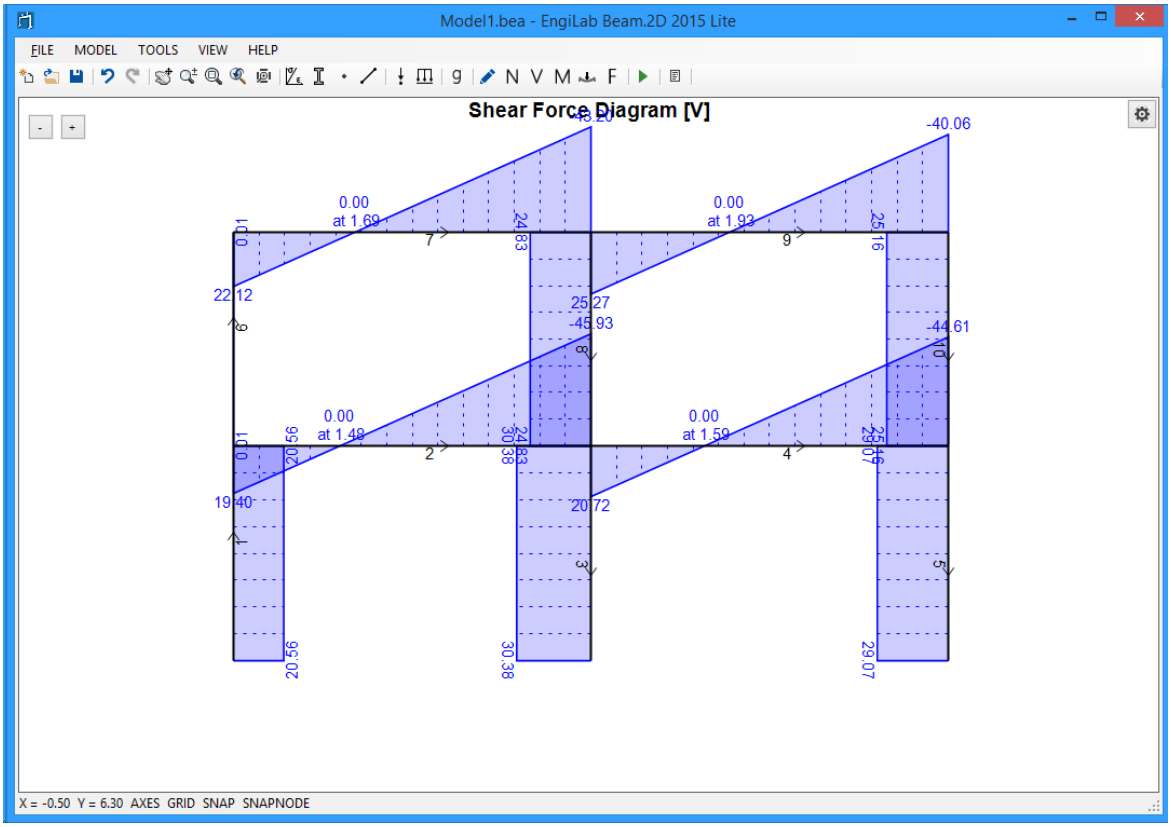

The Axial Force Diagram is out of Scale. **Click the "Zoom All" button** to automatically scale the Diagram.

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



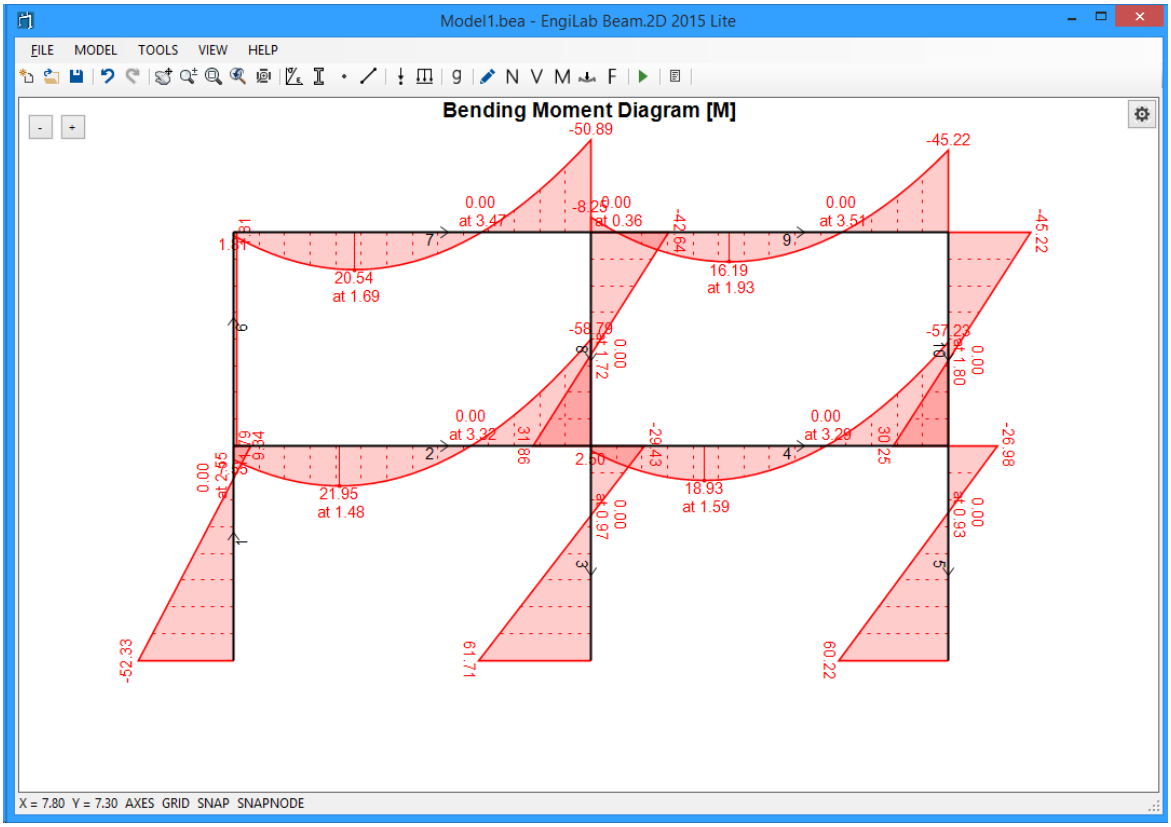
## 2. Shear Force Diagram

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

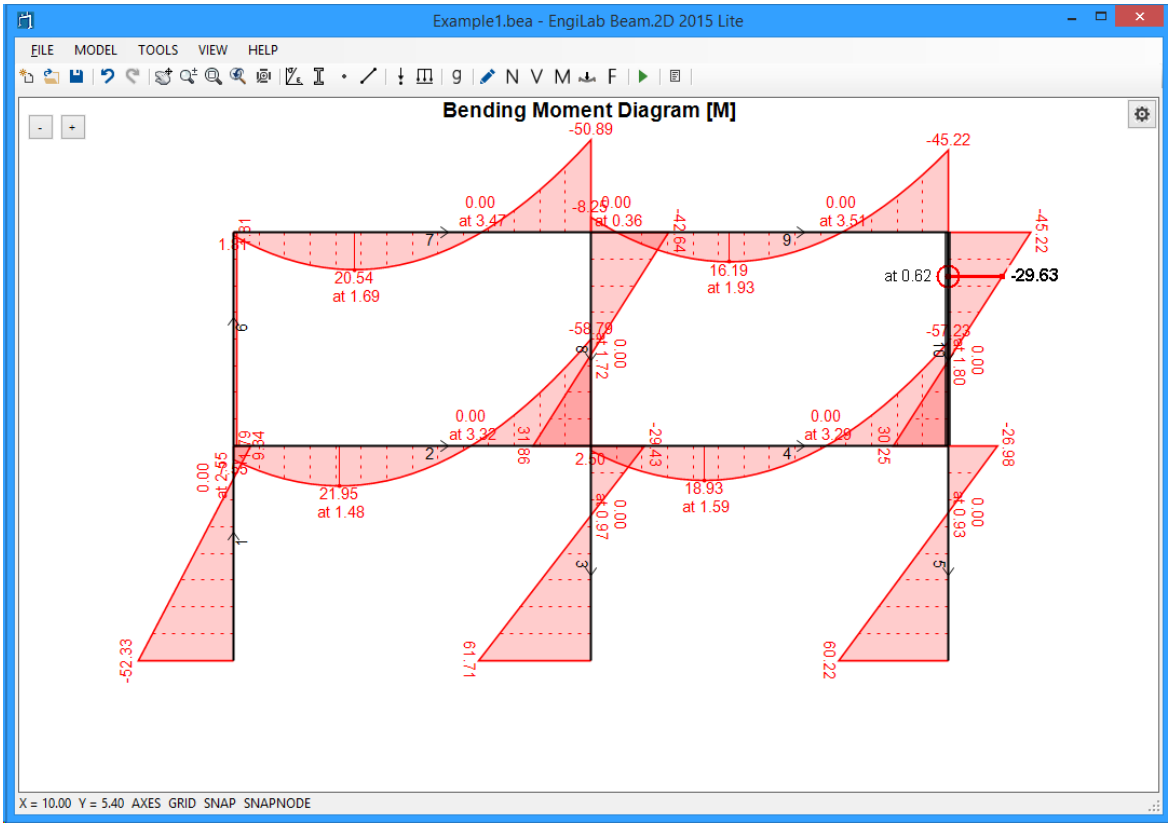


### 3. Bending Moment Diagram

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

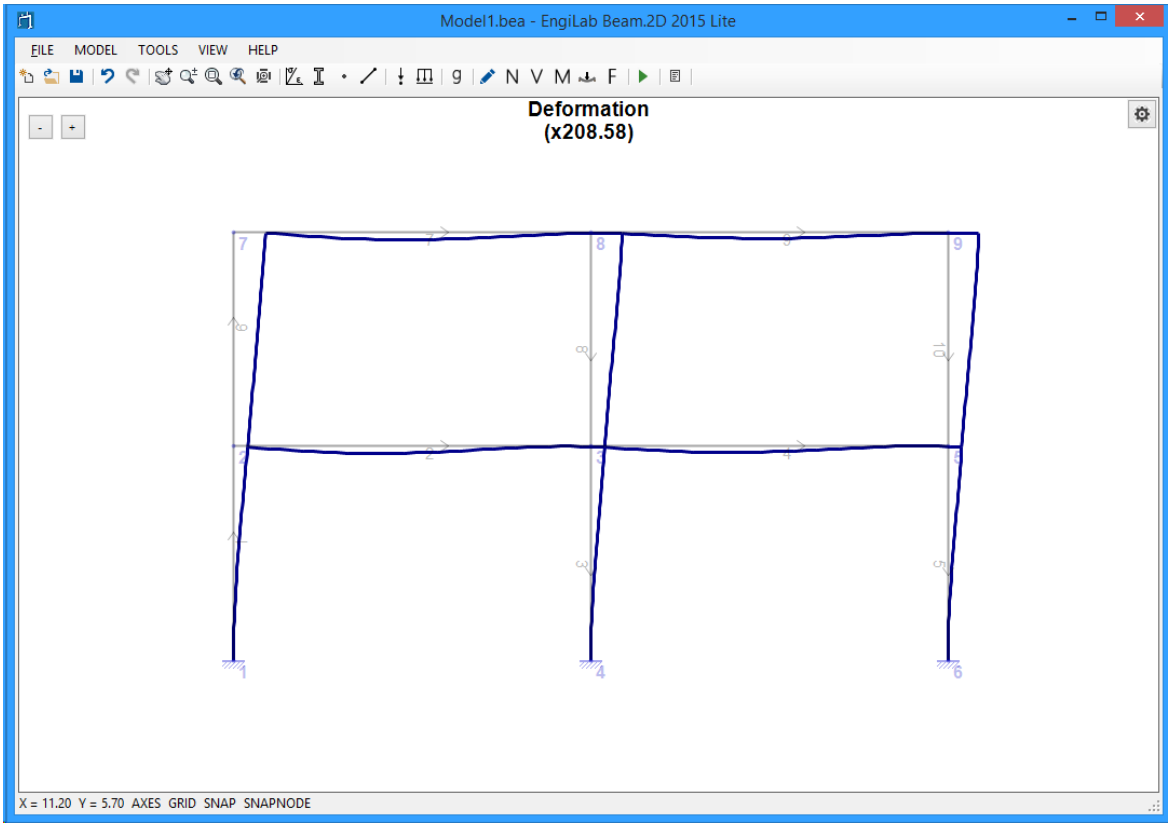


Note that if you move the pointer over an Element, you can read the corresponding value of the Diagram, as shown below for the Bending Moment Diagram case. This happens for all Diagrams and also for the Deformation and the Free Body Diagram.

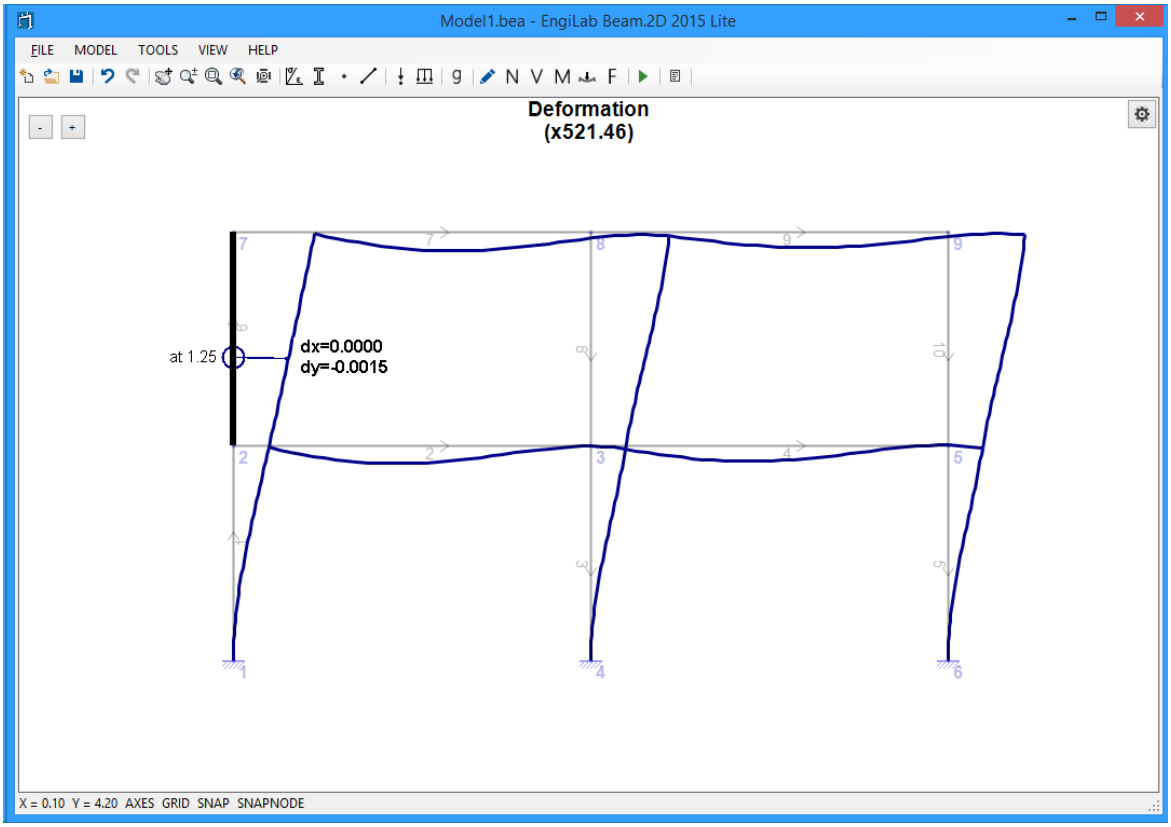


## 4. Deformation

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



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




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

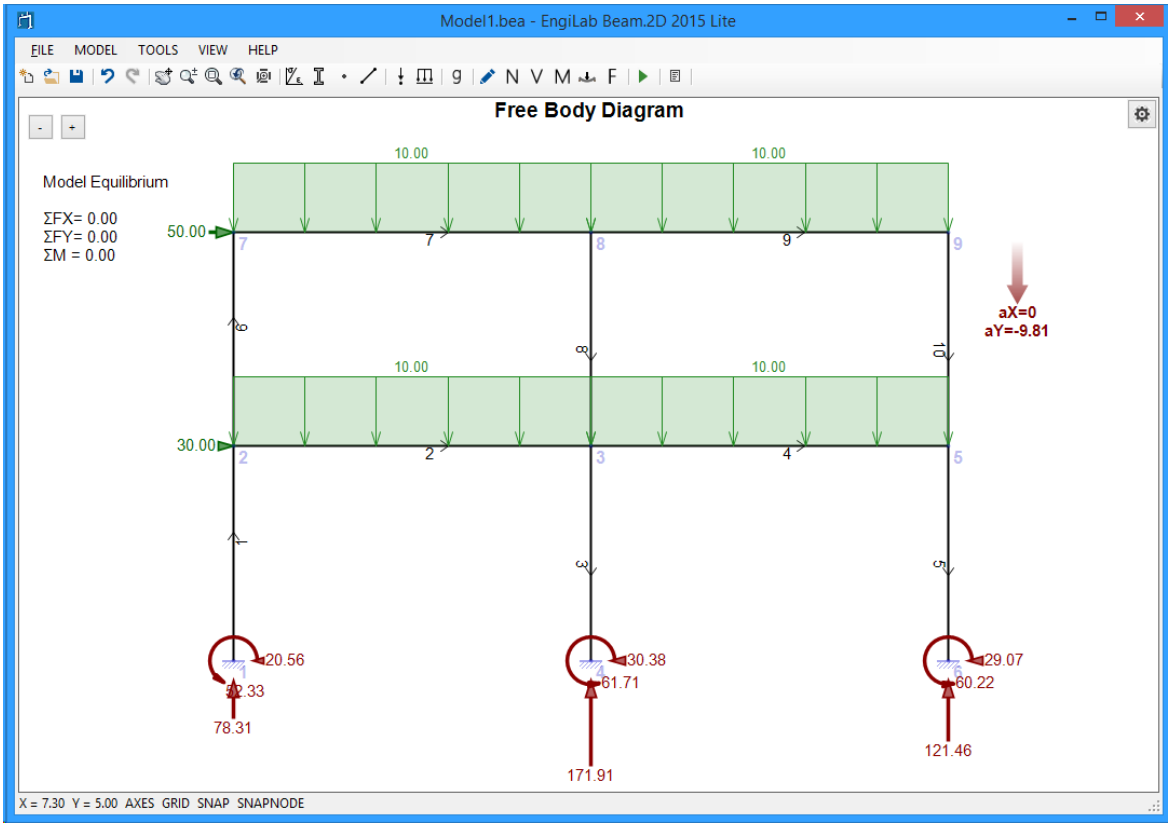
**IMPORTANT**: The values that are given on screen are the x and y displacements of the corresponding point of each element **in Local Element Axes**. For example, in the picture below Element 6 goes upward, which means that the Element x-Axis is pointing upwards and the Element y-Axis is pointing to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

**Note**: EngiLab Frame.2D 2022 (v4.0) and later, shows the deformations in both Local Element Axes and Global Axes. See for example Example 2 - Step 11.

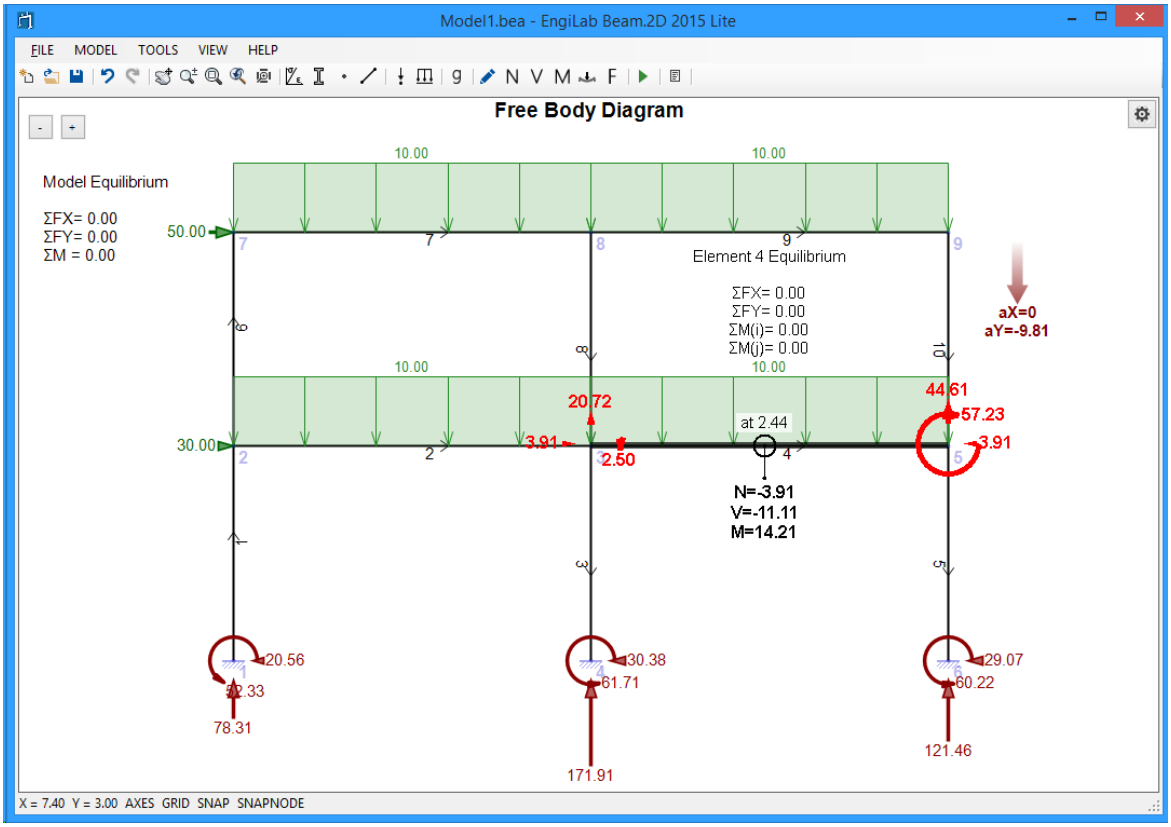
(c) 2022 EngiLab PC

## 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 (ΣFX, ΣFY, ΣM(i), ΣM(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 ΣFY. There is an external uniform load with value 10, acting along Element 4 (Length = 5 m), which gives an external load of 10*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 * 5 * 2.5 * 9.81 = 15.33 kN.

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

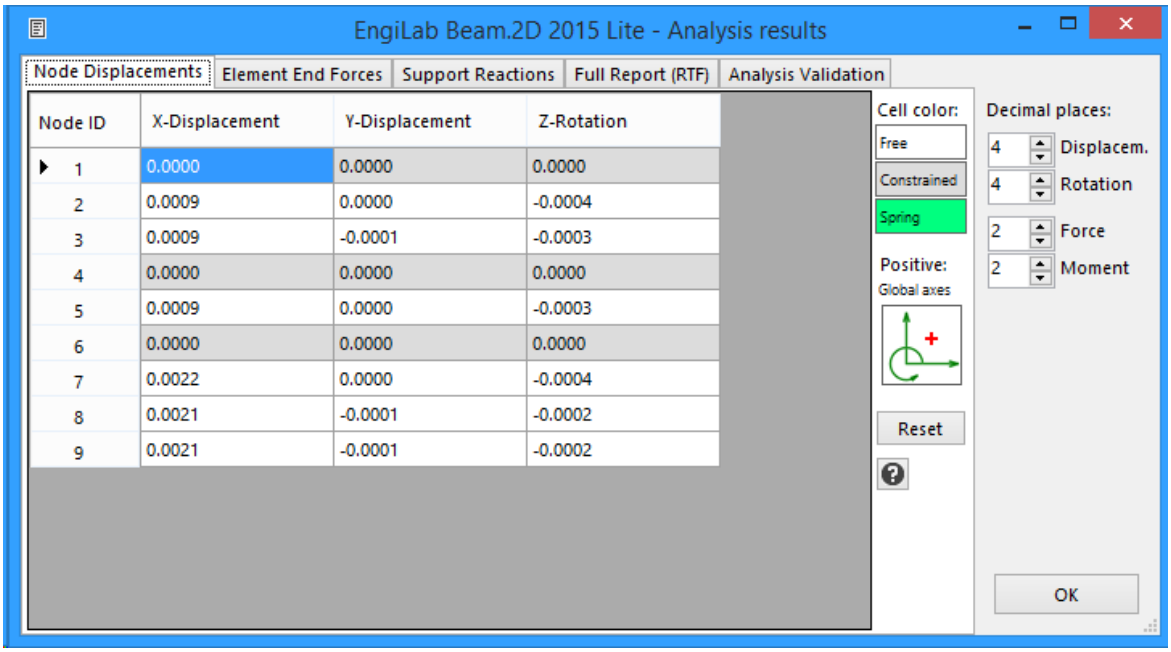
(c) 2022 EngiLab PC

### 7.1.13 Step 12. View the analytical results

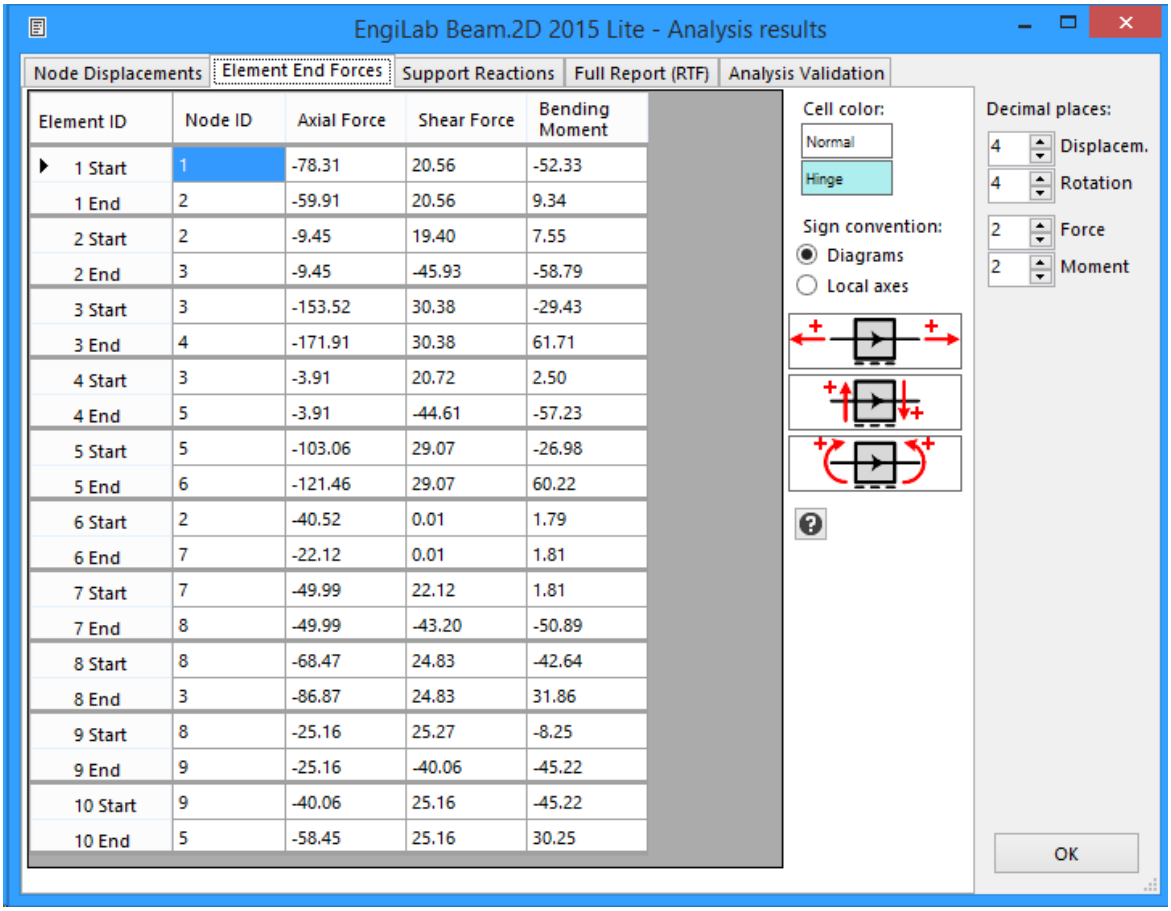



Click  to see the analytical results.

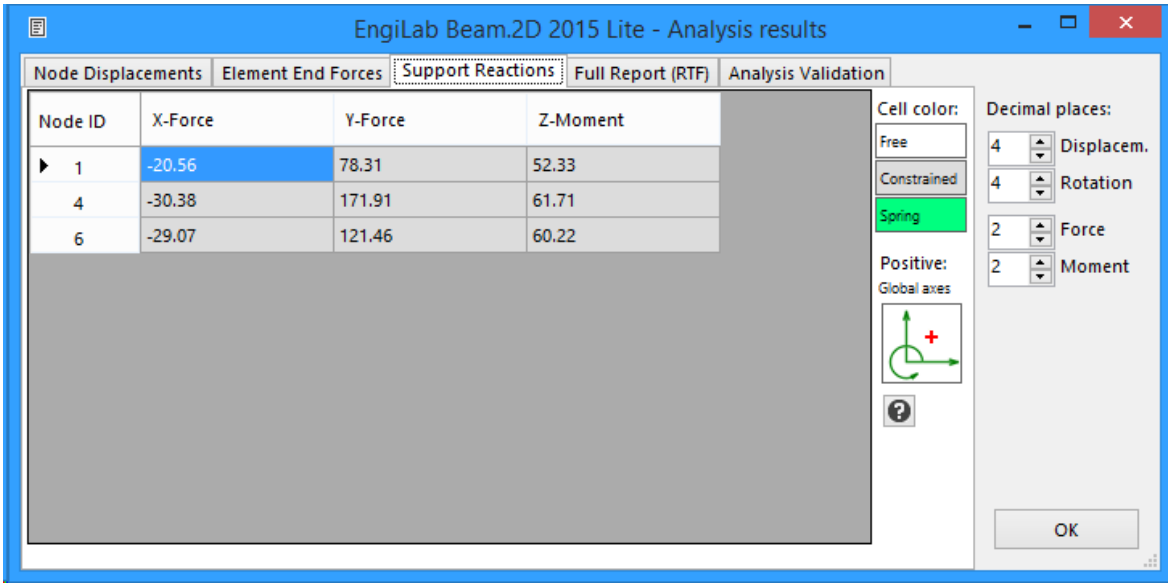
#### 1. Node Displacements



## 2. Element Forces



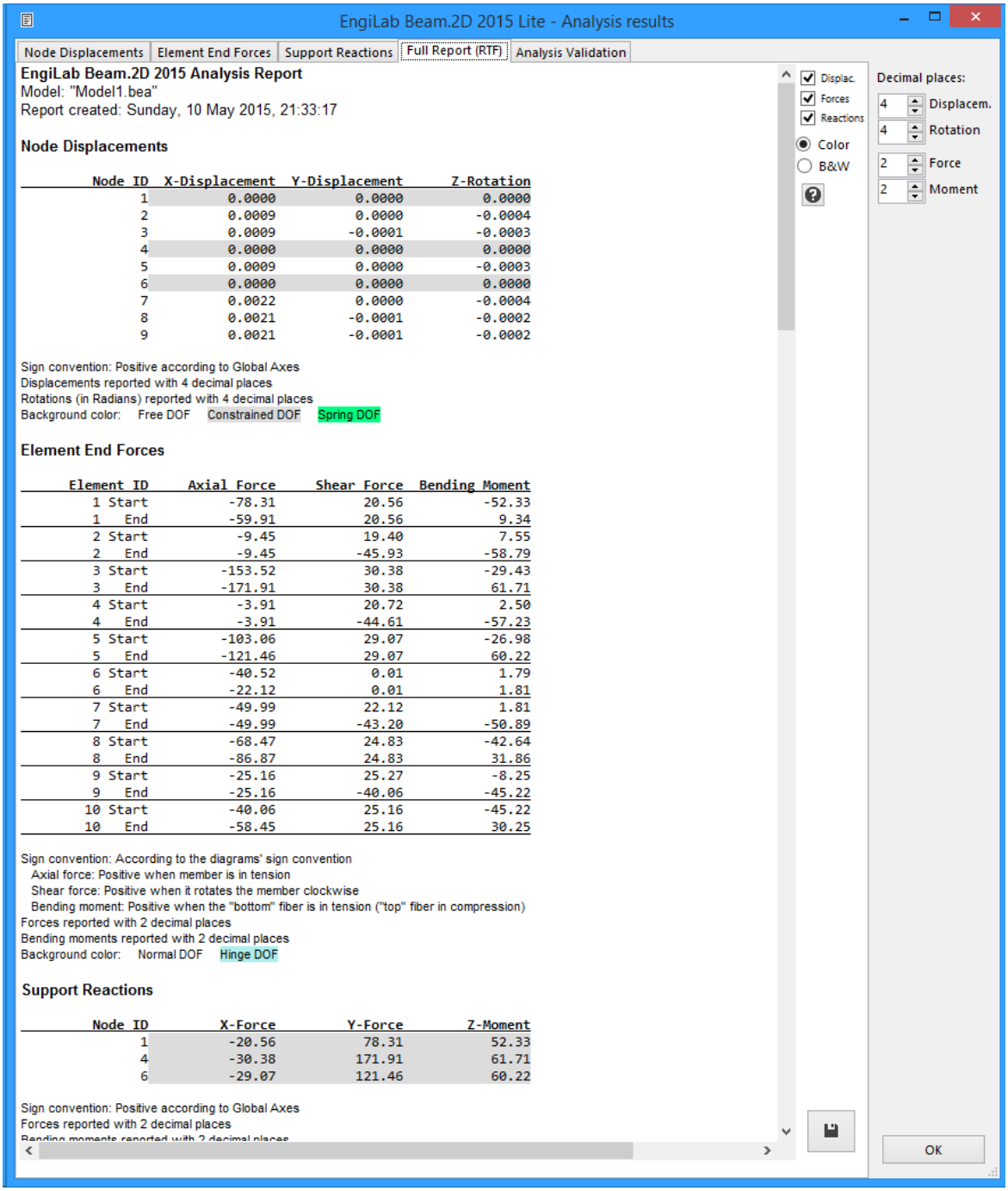

## 3. Support Reactions




(c) 2022 EngiLab PC

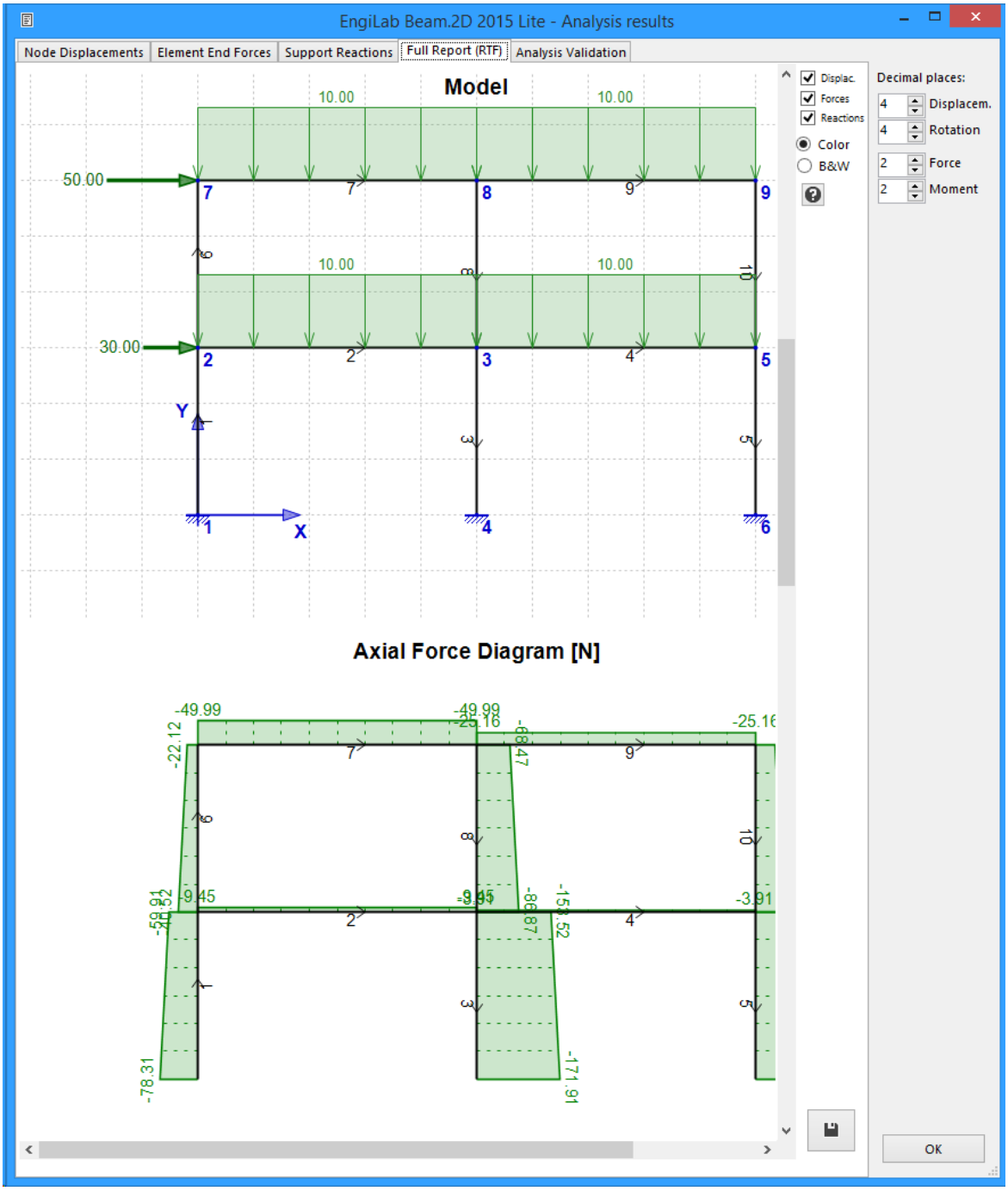
(c) 2022 EngiLab PC

## 4. Full Analysis Report (RTF)



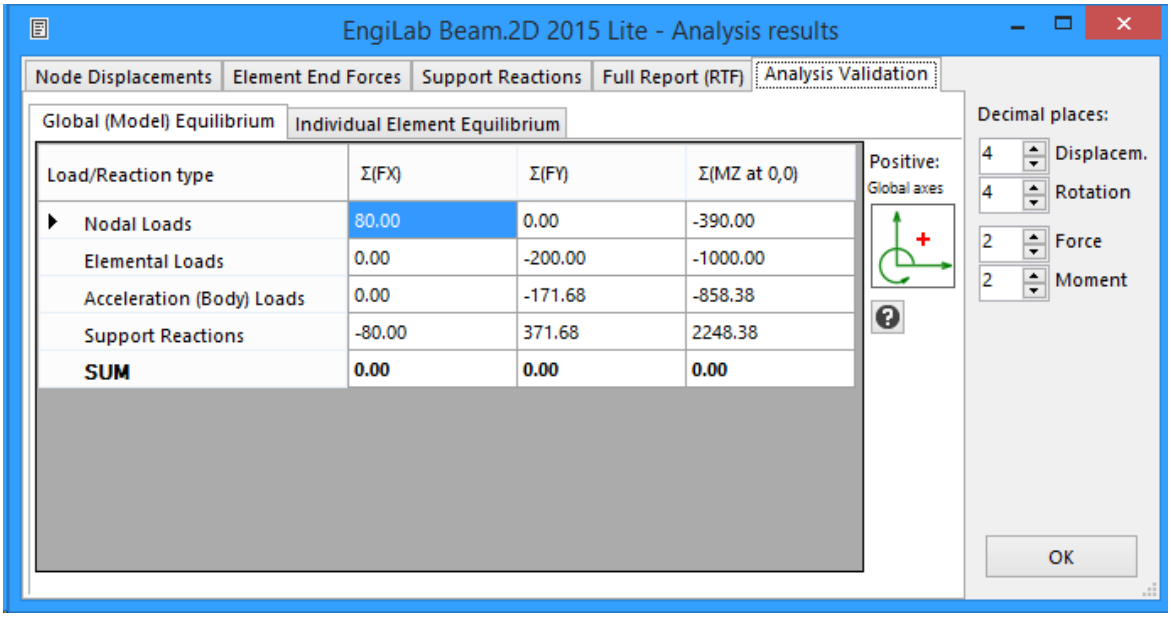

The Full Analysis Reports includes also the pictures of the Model, N, V, M Diagrams, Deformation and Free Body Diagram, as shown below.

(c) 2022 EngiLab PC

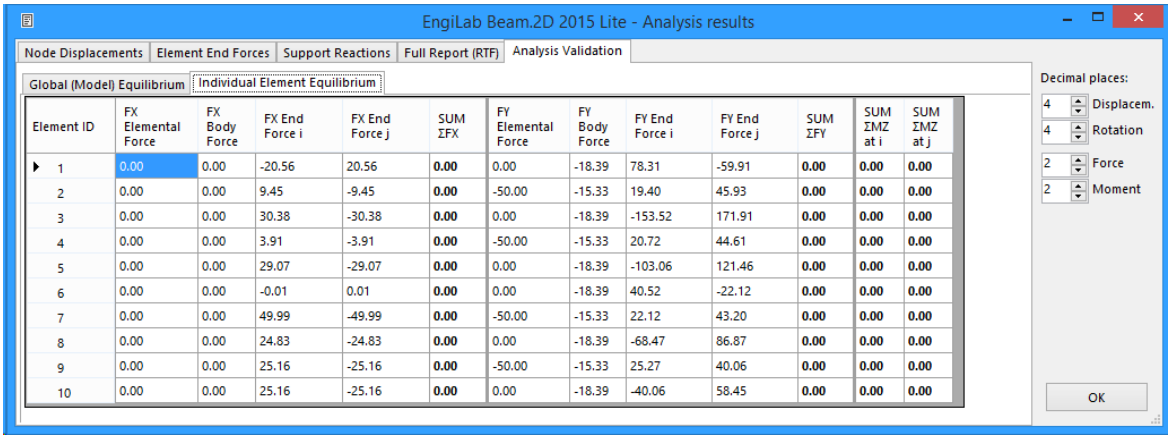

EngiLab Beam.2D 2015 Lite - Analysis results
Node Displacements
Element End Forces
Support Reactions
Full Report (RTF)
Analysis Validation
Model
10.00
10.00
50.00
30.00
Displac.
Forces
Reactions
Color
B&W
Decimal places:
Displacem.
Rotation
Force
Moment
Axial Force Diagram [N]
-49.99
-25.16
-22.12
-9.45
-78.31
-171.91
-3.91
OK

# 5. Analysis Validation

## 5a. Global (Model) Equilibrium




## 5b. Individual Element Equilibrium

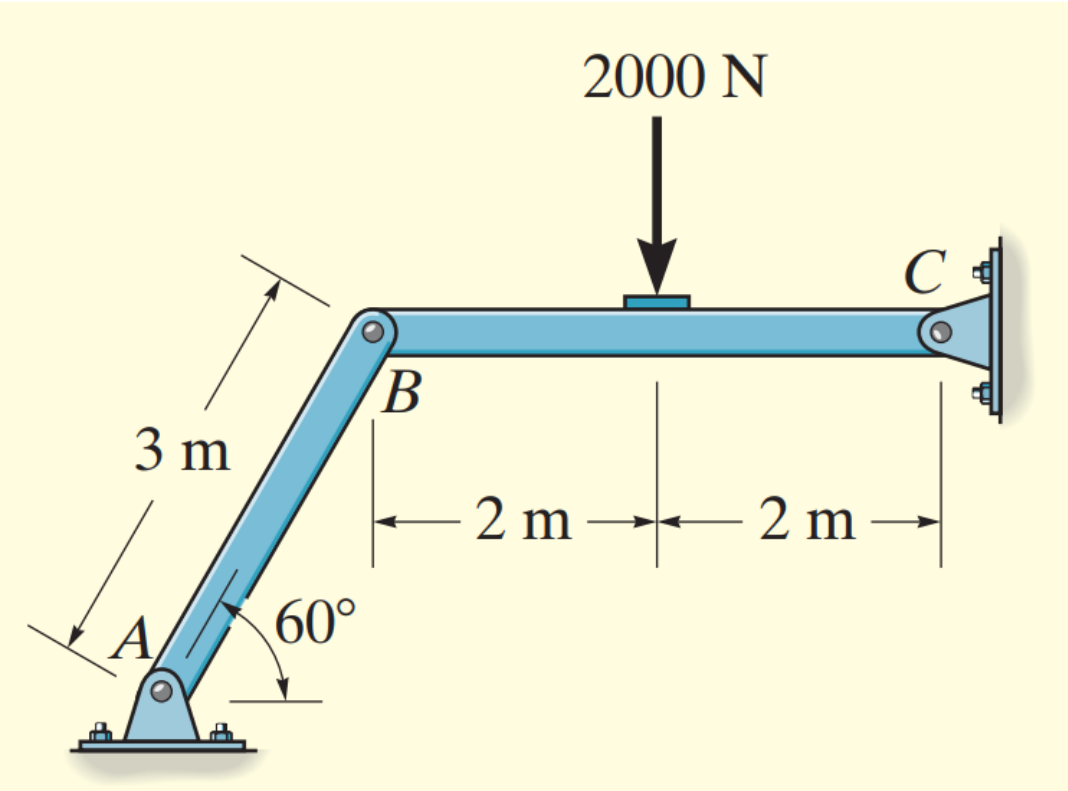



(c) 2022 EngiLab PC

## 7.2 Example Problem 2 (based on the 2022 version)

The second example is the 2D Frame shown in the figure. The figure and the problem are taken from the book "**Engineering Mechanics: Statics**" from **Russell C. Hibbeler** [1]. The problem, as described in [1] is stated as:

"Determine the **horizontal and vertical components of force** which the pin at C exerts on member BC of the frame in the figure."




Note that this is a **statically determinate 2D Frame**. It has two pinned supports, so the number of unknown reactions is 2*2=4. We have 3 equilibrium equations. But there is also an internal hinge in the model, which offers an extra equilibrium equation. In this case, there are 4 unknown reactions and 4 equilibrium equations. In statically determinate frames, all support reactions and member forces can be determined only by taking into account the equilibrium equations of the Model. This means that the Material Elastic Modulus (E), the Section Area (A) and the Section Moment of Inertia (I) properties of each member are not important in determining these quantities (although they do affect the Model displacements!). Any E, A and I, for any member, will lead to the same results, when forces and support reactions are concerned. The same is not true for displacements. Displacements do depend on the Elastic Modulus, the Area and the Moment of Inertia of each member, but they are not asked for in this problem, at least in the book.

For our calculations in %PROGRAM NAME%>, we need to assign a Material and a Section to each Element, as the program is general and can analyze any 2D Frame,

(c) 2022 EngiLab PC

statically determinate or statically indeterminate, while it also calculates the displacements of all Nodes, among many others.

To draw a structure quickly without the need to set the Material and Section properties manually, EngiLab Frame.2D offers the "**QuickStart**" option, where the program asks the user if he/she agrees to assign a predefined Material and Section to each Element drawn on screen. The predefined Material and Section properties that EngiLab Frame.2D assigns to each member when drawn on screen are the following:

### Material

The predefined Material is "**Steel S 235**" with the following properties:

- Elastic Modulus **E = 210 GPa**
- Density **d = 7850 kg/m$^3$**

### Section

The predefined Section is "**IPE 200**" with the following properties:

- Area **A = 28.5 cm$^2$**
- Moment of Inertia **I = 1943 cm$^4$**

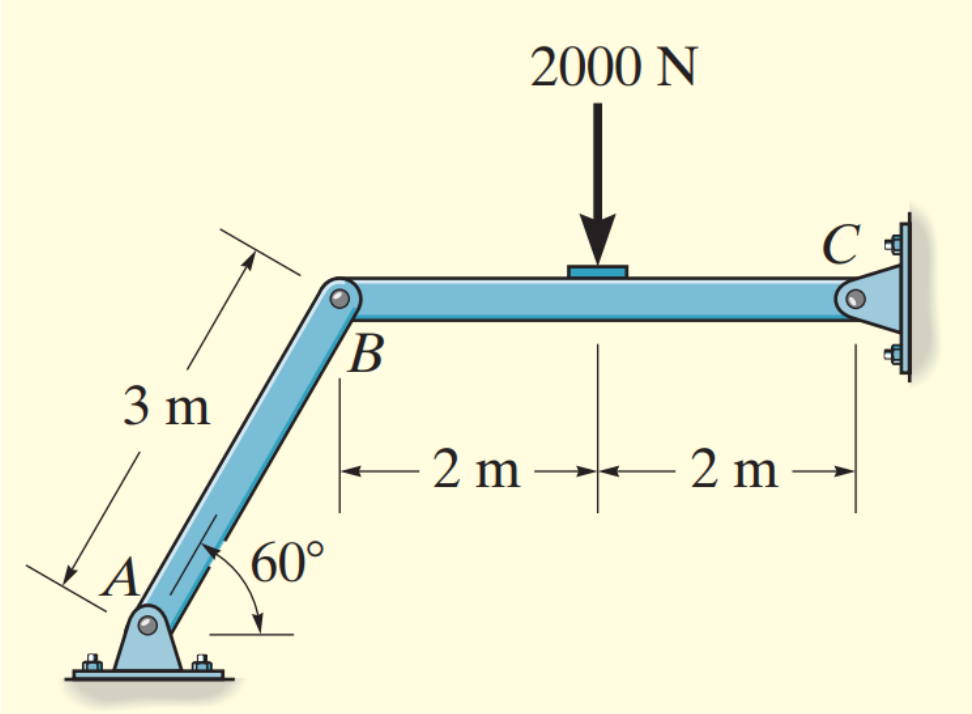
---

### References

[1] Engineering Mechanics: Statics, R.C. Hibbeler, 14th Edition, Pearson Prentice Hall, ISBN-13: 978-0133918922, 2016.

(c) 2022 EngiLab PC

### 7.2.1 Example 2 - Hand Calculations




The solution, from the book [1], is the following, using hand calculations:

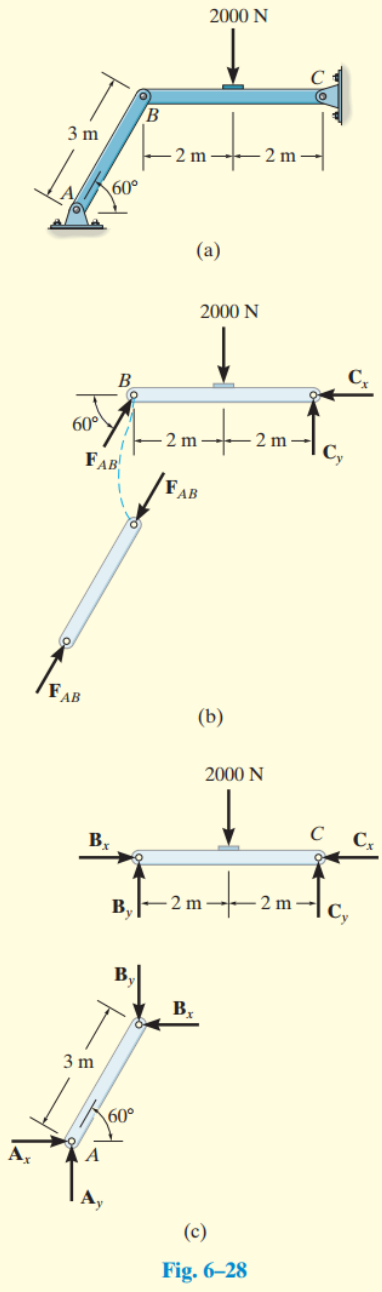



Fig. 6–28

Determine the horizontal and vertical components of force which the pin at $C$ exerts on member $BC$ of the frame in Fig. 6–28*a*.

## SOLUTION I

**Free-Body Diagrams.** By inspection it can be seen that $AB$ is a two-force member. The free-body diagrams are shown in Fig. 6–28*b*.

**Equations of Equilibrium.** The *three unknowns* can be determined by applying the three equations of equilibrium to member $BC$.

$$\zeta + \Sigma M_C = 0;\ 2000\ \text{N}(2\ \text{m}) - (F_{AB}\sin 60°)(4\ \text{m}) = 0 \quad F_{AB} = 1154.7\ \text{N}$$

$$\pm \Sigma F_x = 0;\ 1154.7\cos 60°\ \text{N} - C_x = 0 \quad C_x = 577\ \text{N} \qquad \textit{Ans.}$$

$$+\uparrow \Sigma F_y = 0;\ 1154.7\sin 60°\ \text{N} - 2000\ \text{N} + C_y = 0$$

$$C_y = 1000\ \text{N} \qquad \textit{Ans.}$$

## SOLUTION II

**Free-Body Diagrams.** If one does not recognize that $AB$ is a two-force member, then more work is involved in solving this problem. The free-body diagrams are shown in Fig. 6–28*c*.

**Equations of Equilibrium.** The *six unknowns* are determined by applying the three equations of equilibrium to each member.

*Member AB*

$$\zeta + \Sigma M_A = 0;\quad B_x(3\sin 60°\ \text{m}) - B_y(3\cos 60°\ \text{m}) = 0 \qquad (1)$$

$$\pm \Sigma F_x = 0;\quad A_x - B_x = 0 \qquad (2)$$

$$+\uparrow \Sigma F_y = 0;\quad A_y - B_y = 0 \qquad (3)$$

*Member BC*

$$\zeta + \Sigma M_C = 0;\quad 2000\ \text{N}(2\ \text{m}) - B_y(4\ \text{m}) = 0 \qquad (4)$$

$$\pm \Sigma F_x = 0;\quad B_x - C_x = 0 \qquad (5)$$

$$+\uparrow \Sigma F_y = 0;\quad B_y - 2000\ \text{N} + C_y = 0 \qquad (6)$$

The results for $C_x$ and $C_y$ can be determined by solving these equations in the following sequence: 4, 1, 5, then 6. The results are

$$B_y = 1000\ \text{N}$$

$$B_x = 577\ \text{N}$$

$$C_x = 577\ \text{N} \qquad \textit{Ans.}$$

$$C_y = 1000\ \text{N} \qquad \textit{Ans.}$$

By comparison, Solution I is simpler since the requirement that $F_{AB}$ in Fig. 6–28*b* be equal, opposite, and collinear at the ends of member $AB$ automatically satisfies Eqs. 1, 2, and 3 above and therefore eliminates the need to write these equations. ***As a result, save yourself some time and effort by always identifying the two-force members before starting the analysis!***

### References

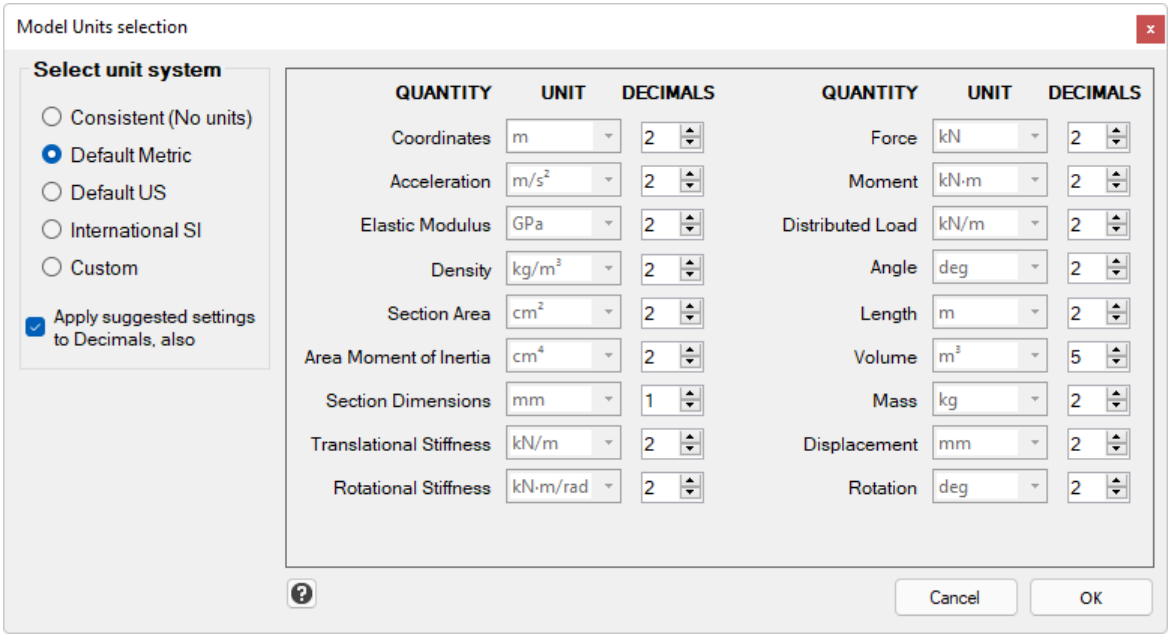
[1] Engineering Mechanics: Statics, R.C. Hibbeler, 14th Edition, Pearson Prentice Hall, ISBN-13: 978-0133918922, 2016.

## 7.2.2 Step 1. Define the System of Units




(c) 2022 EngiLab PC

Click U to define the **System of Units**. This is a very important step for the Model. In this example, we will use the **Default Metric** System of units. For details, see Units




Select the **Default Metric** System of Units. Click OK.

Note that the external force is 2000 N in this example, i.e. 2 kN in the Default Metric System.

(c) 2022 EngiLab PC

### 7.2.3 Step 2. Define Materials




In our example, we will use the **QuickStart** feature of the program and there is no need to define the Material manually. **You can go directly to Step 4. Draw the Model on screen**.

If on the other hand, you prefer to define the Material manually, then you can follow the instructions on this page.

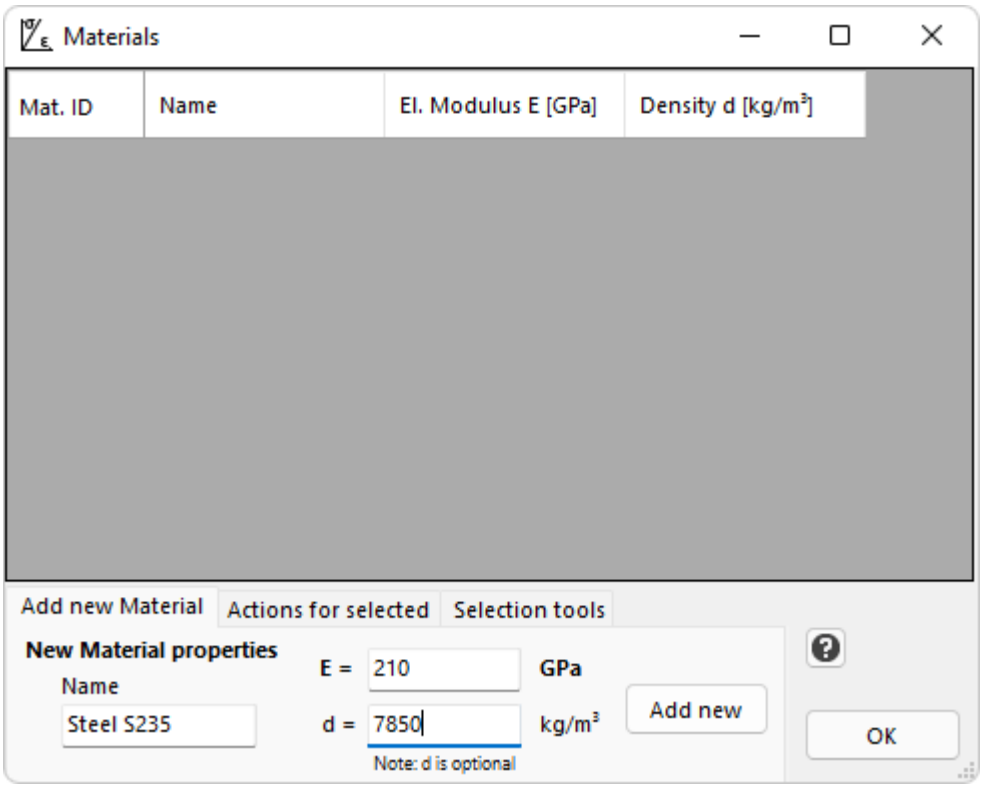
---

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

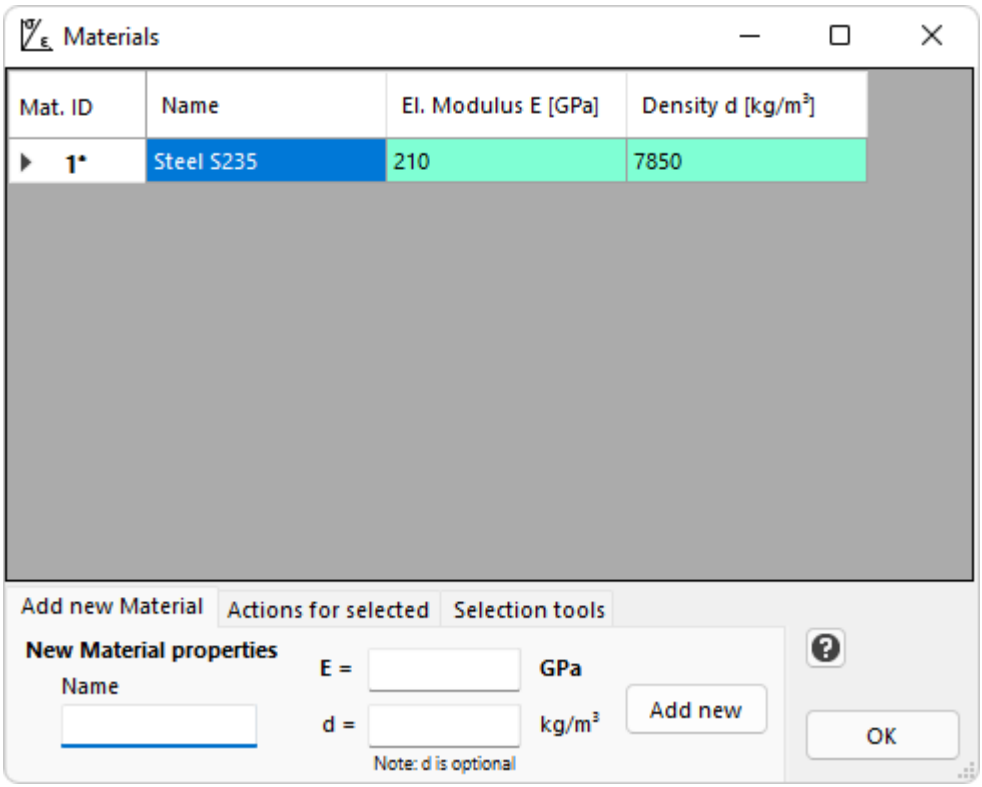
You need to define the Material **Name** (optional), **Material Density d** (optional in general, only needed if you are to define self-weight loads, which is not the case here) and Material **Elastic Modulus E** for each Material. Use the following properties:

- Name: **Steel S 235**
- Elastic Modulus **E = 210 GPa**
- Density **d = 7850 kg/m³**

(c) 2022 EngiLab PC


Materials
Mat. ID
Name
El. Modulus E [GPa]
Density d [kg/m³]
Add new Material
Actions for selected
Selection tools
New Material properties
Name
Steel S235
E =
210
GPa
d =
7850
kg/m³
Note: d is optional
Add new
OK

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




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

(c) 2022 EngiLab PC

### 7.2.4 Step 3. Define Sections




In our example, we will use the **QuickStart** feature of the program and there is no need to define the Section manually. **You can go directly to Step 4. Draw the Model on screen**.

If on the other hand, you prefer to define the Section manually, then you can follow the instructions on this page.

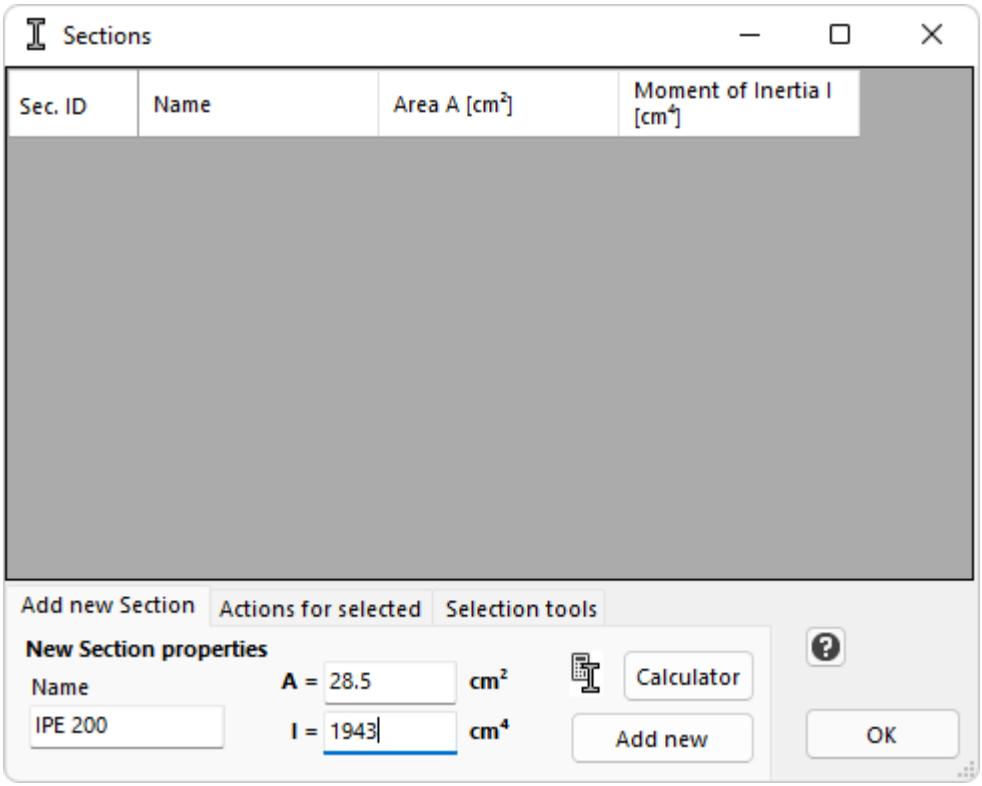
---

Click I to define **Section**(s). For details, see Sections.

You need to define the Section **Name** (optional), **Section Area A** and **Section Moment of Inertia I** (optional in general, only needed if you want to check the Elements for buckling, which is not the case here). Use the following properties:

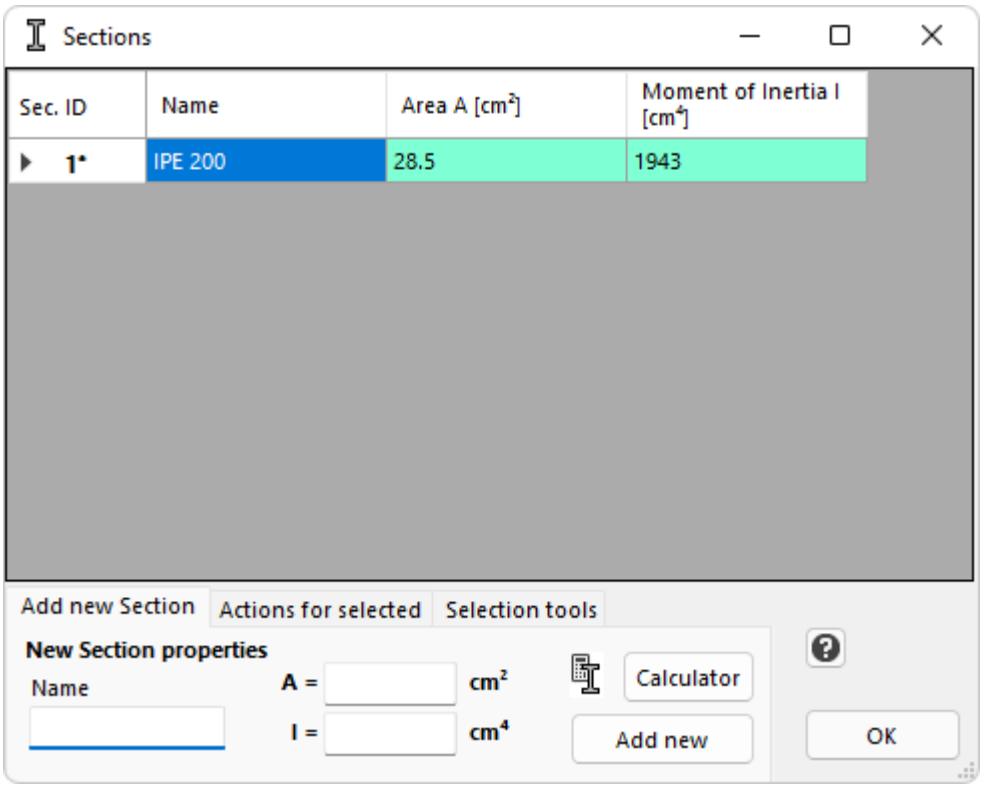
- Name: **IPE 200**
- Area **A = 28.5 cm²**
- Moment of Inertia **I = 1943 cm⁴**

(c) 2022 EngiLab PC




(c) 2022 EngiLab PC

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




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

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

After you have defined the Material and the Section, you can **start drawing your Model on the screen**.

- First make sure that **SNAP** and **SNAPNODE** are both activated (see bottom left 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 the Default Material and the Default Section**. If there is only one Material or Section, then this is the Default Material or Section, otherwise the user can select among the different Materials or Sections.

You can start at any point. Let's start from point 0,0 (Origin of axes). Draw the first member as shown below. Starting from point **(0, 0)**, release the left button after you have reached the **(1.5, 2.6)** point on screen. Note that the Coordinates of Node B are:

X=3*cos(60°) = **1.5 m**

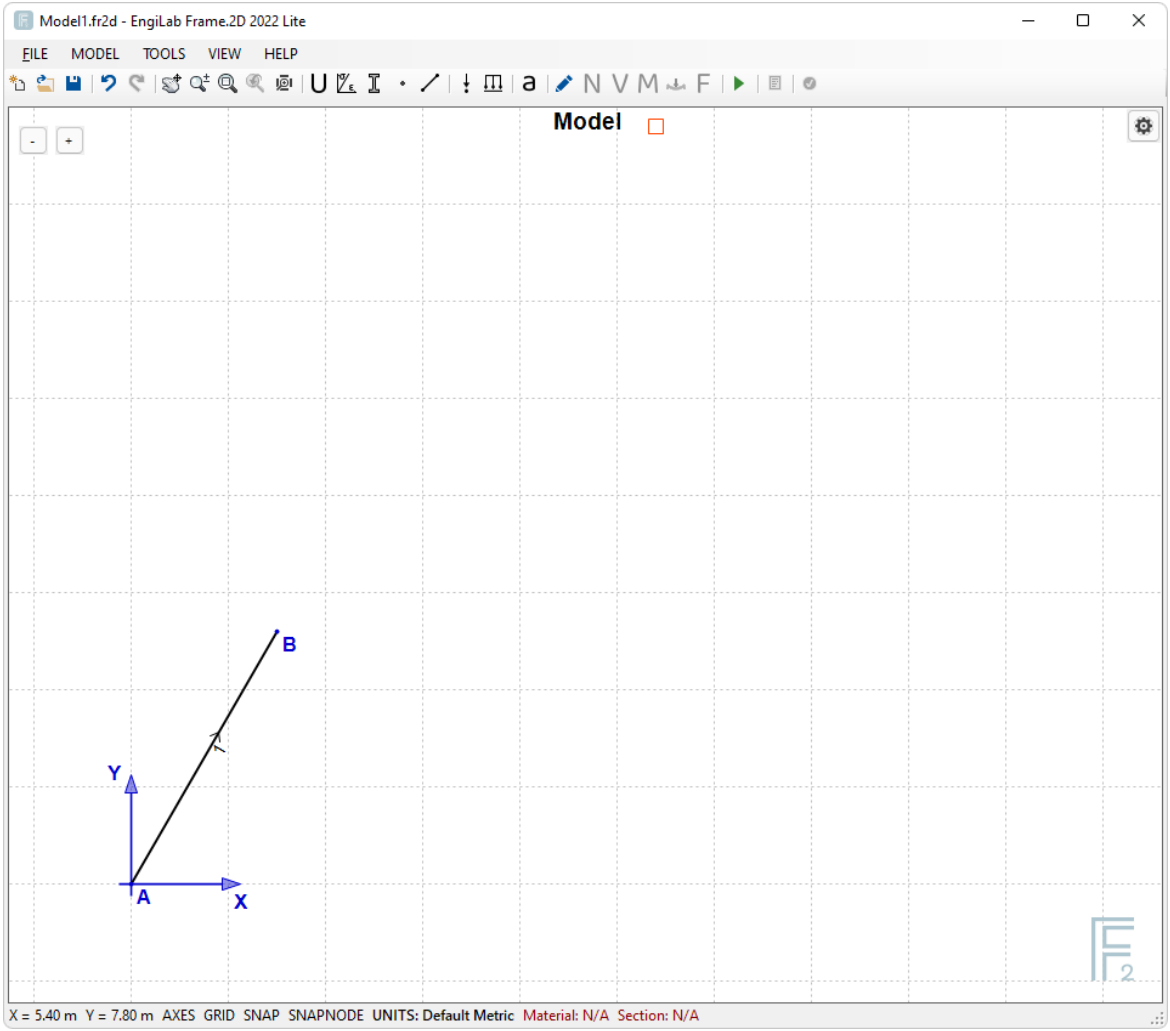
Y=3*sin(60°) = **2.598076211 m**

Later on we will correct the Y value, giving the exact 2.598076211 m in Nodes Coordinates.

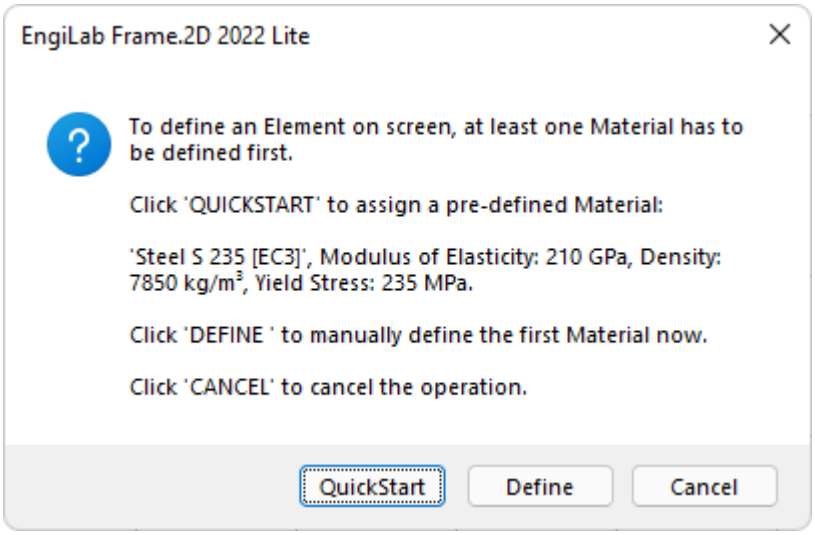
Coordinates are shown on the status bar at the bottom left corner of the window.

**A. In case you have defined the Material and the Section manually** (you have followed Steps 3 and 4), you will get the following screen:

(c) 2022 EngiLab PC

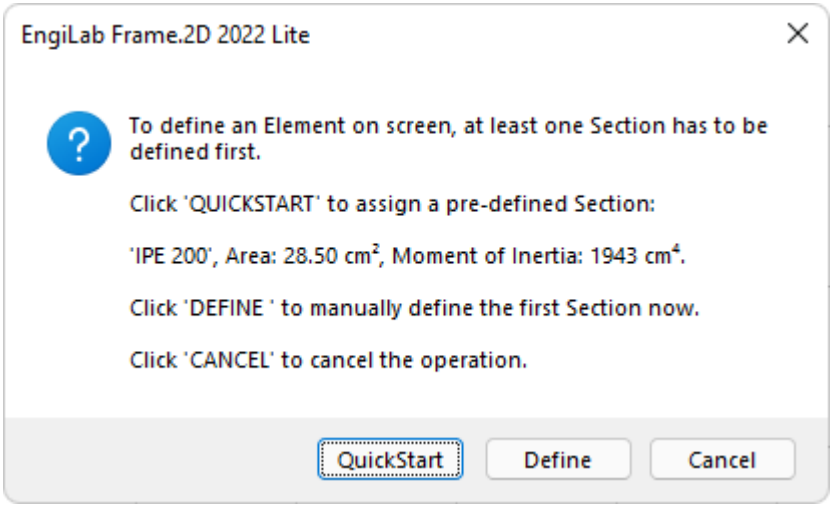



**B. In case you have NOT defined the Material and the Section manually** (you have skipped Steps 3 and 4), you will get the following prompt:




(c) 2022 EngiLab PC

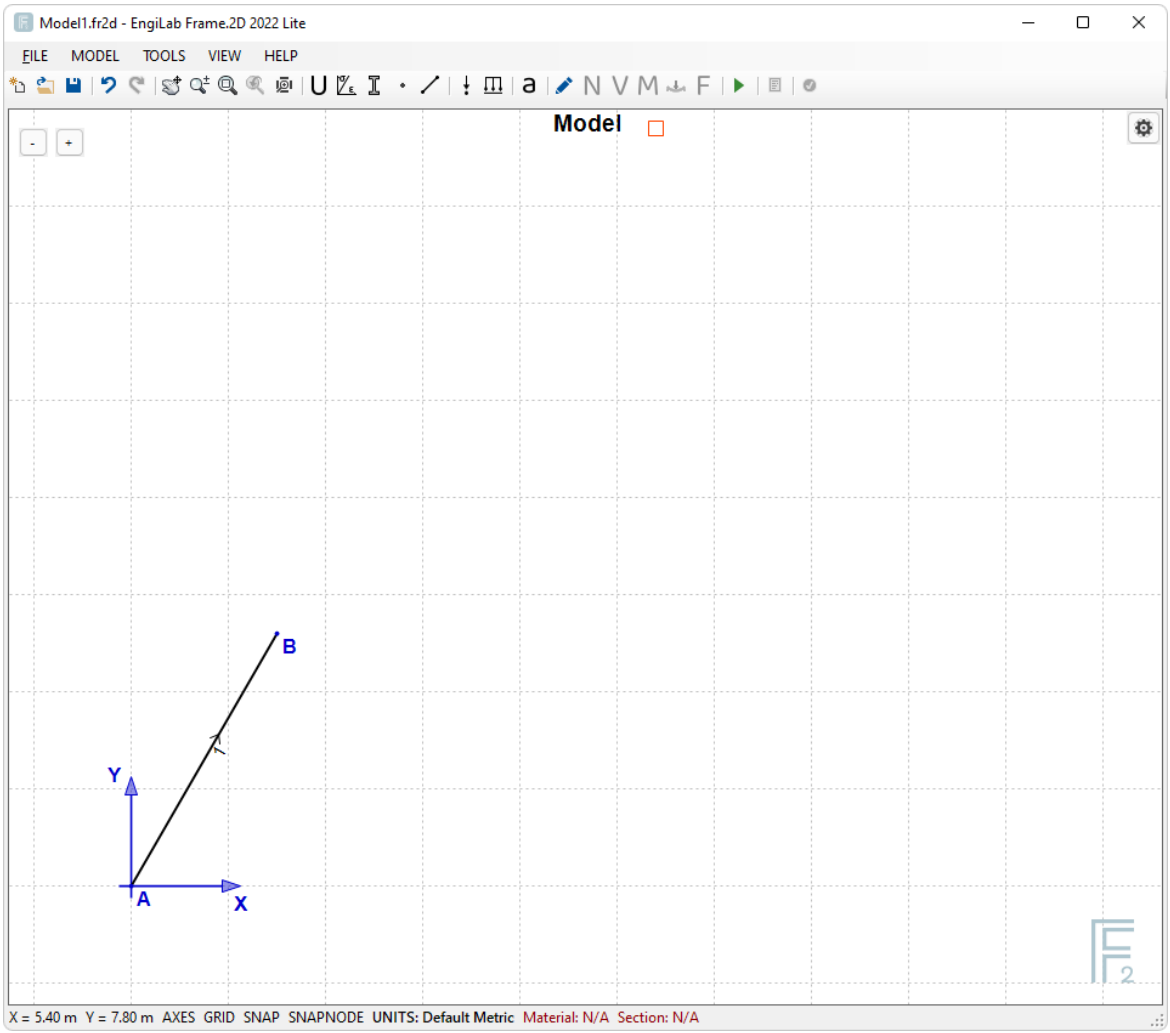
**Click QuickStart** to quickly assign the predefined **Material**. You get another prompt:




**Click QuickStart** again to quickly assign the predefined **Section** and start drawing.

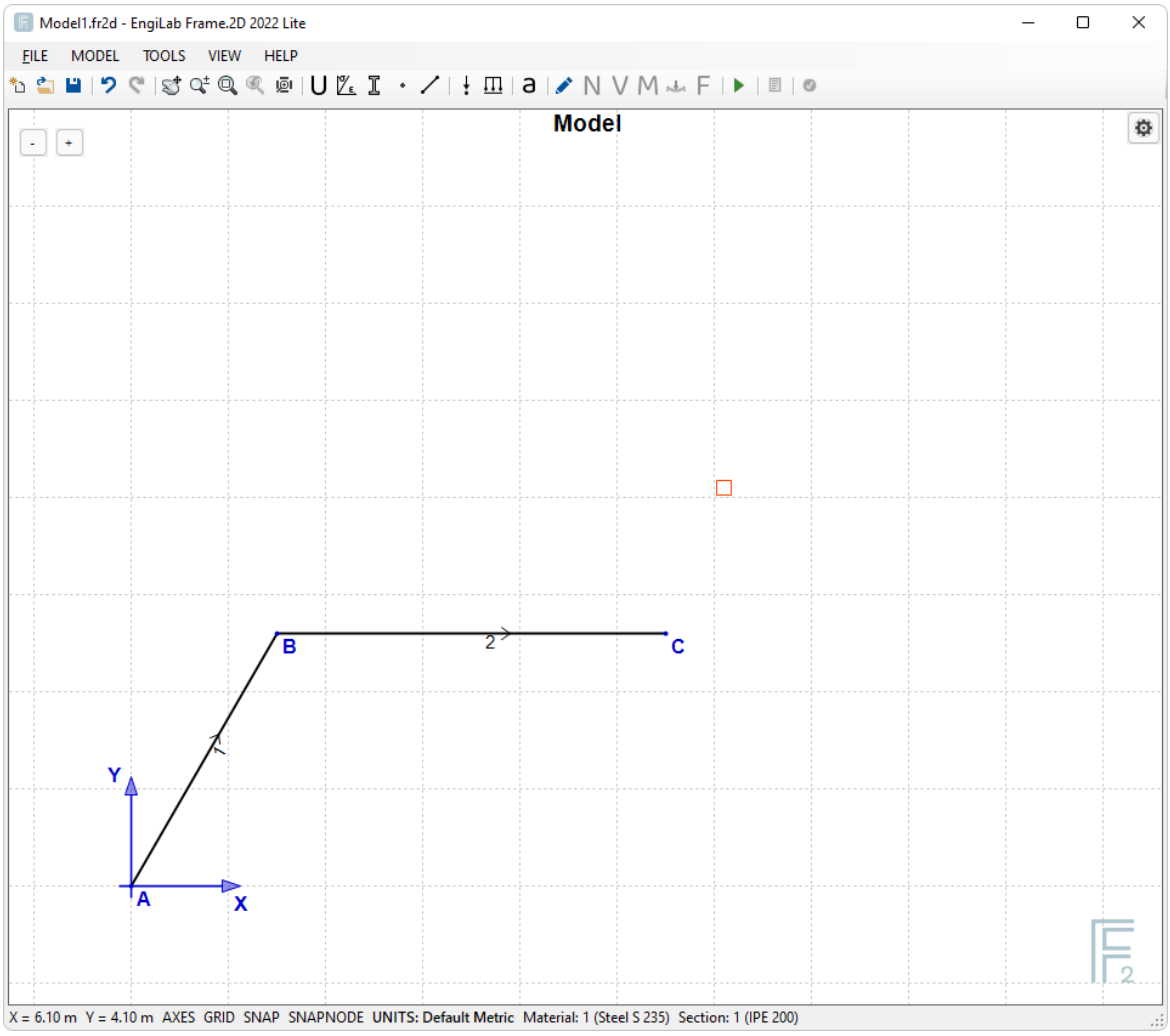
In both cases, you will finally get the first Element and the first two Nodes as follows:

(c) 2022 EngiLab PC




Using the same technique, continue with the other horizontal Element, moving with relative coordinates as **@{4, 0} from Node B**, as shown in the figure below.

(c) 2022 EngiLab PC




Now we have defined Nodes A, B, C and Elements 1 and 2. Note that Nodes B and C are NOT yet in their correct locations. We will fix this in the next step! Then we will apply the Supports and the external loading.

(c) 2022 EngiLab PC

### 7.2.6 Step 5. Edit Nodes




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

For example you may want to:

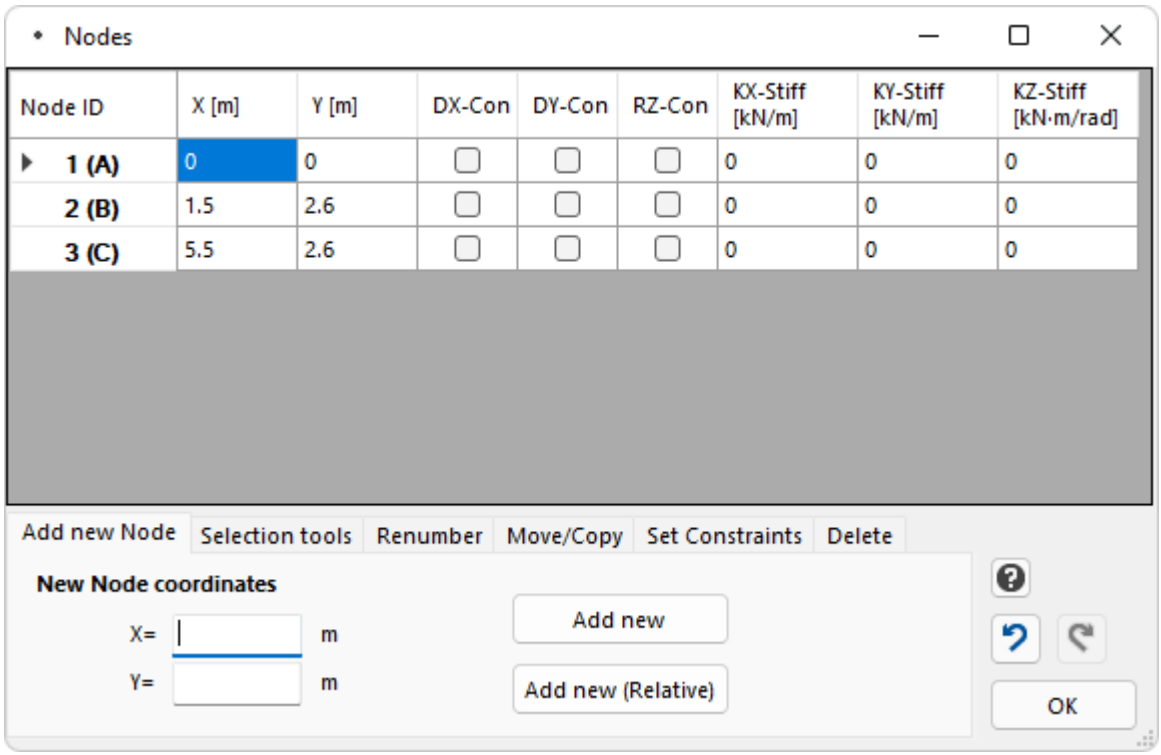
- **Move Nodes to their exact positions, if needed.**
- **Define or change Nodal Constraints (Supports)**.
- Define Springs, if needed.

Click to edit **Nodes**. For details, see Nodes.

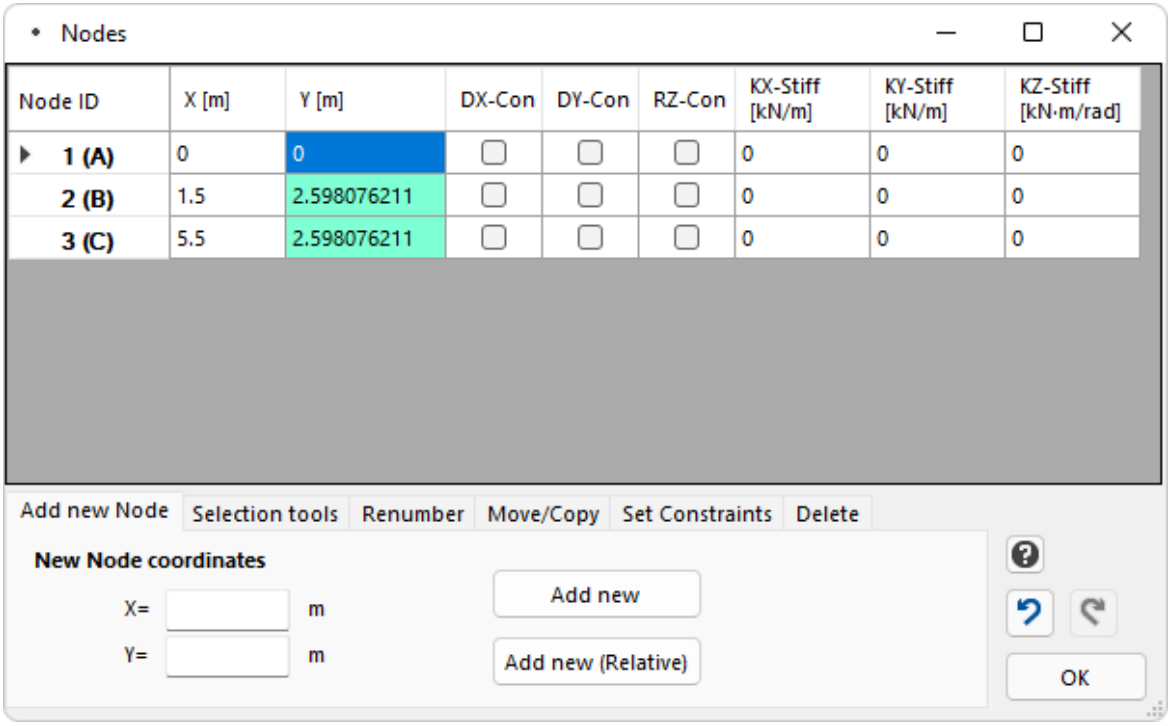



(c) 2022 EngiLab PC

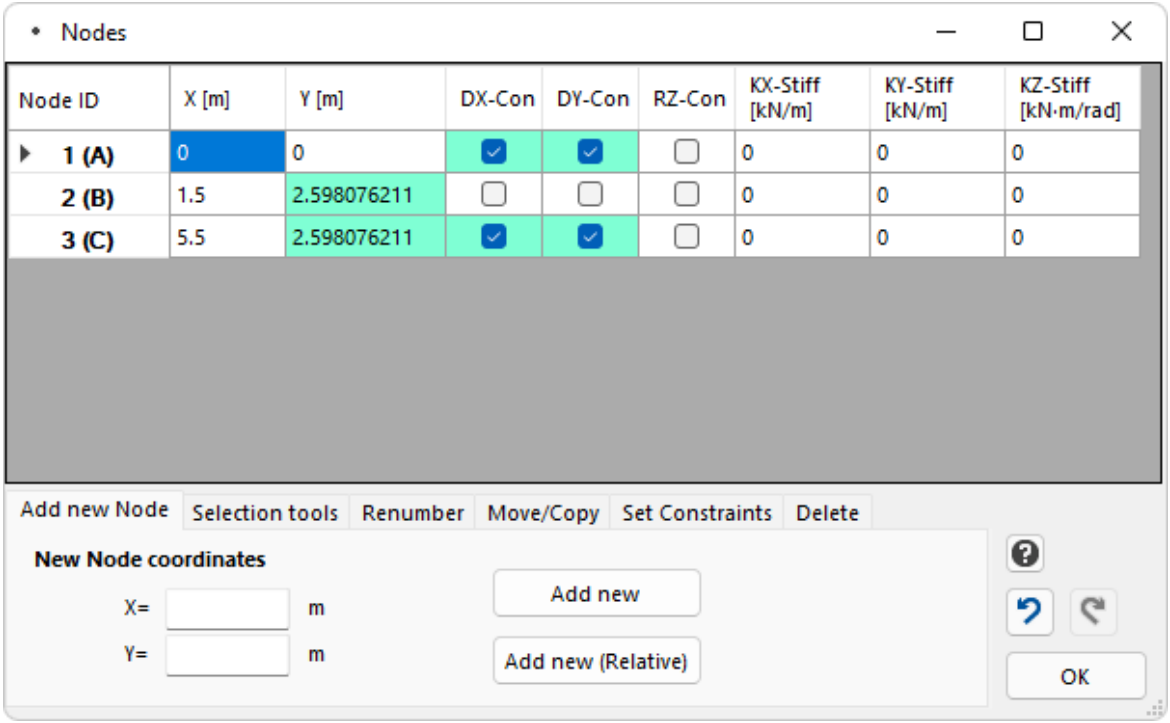
We get the Nodes form as follows:




We see that the Y coordinate of nodes B and C is 2.6 [m], which needs to be corrected to the right value, **2.598076211** [m]. We click on the two cells and we correct the value. Then the table should look like this:

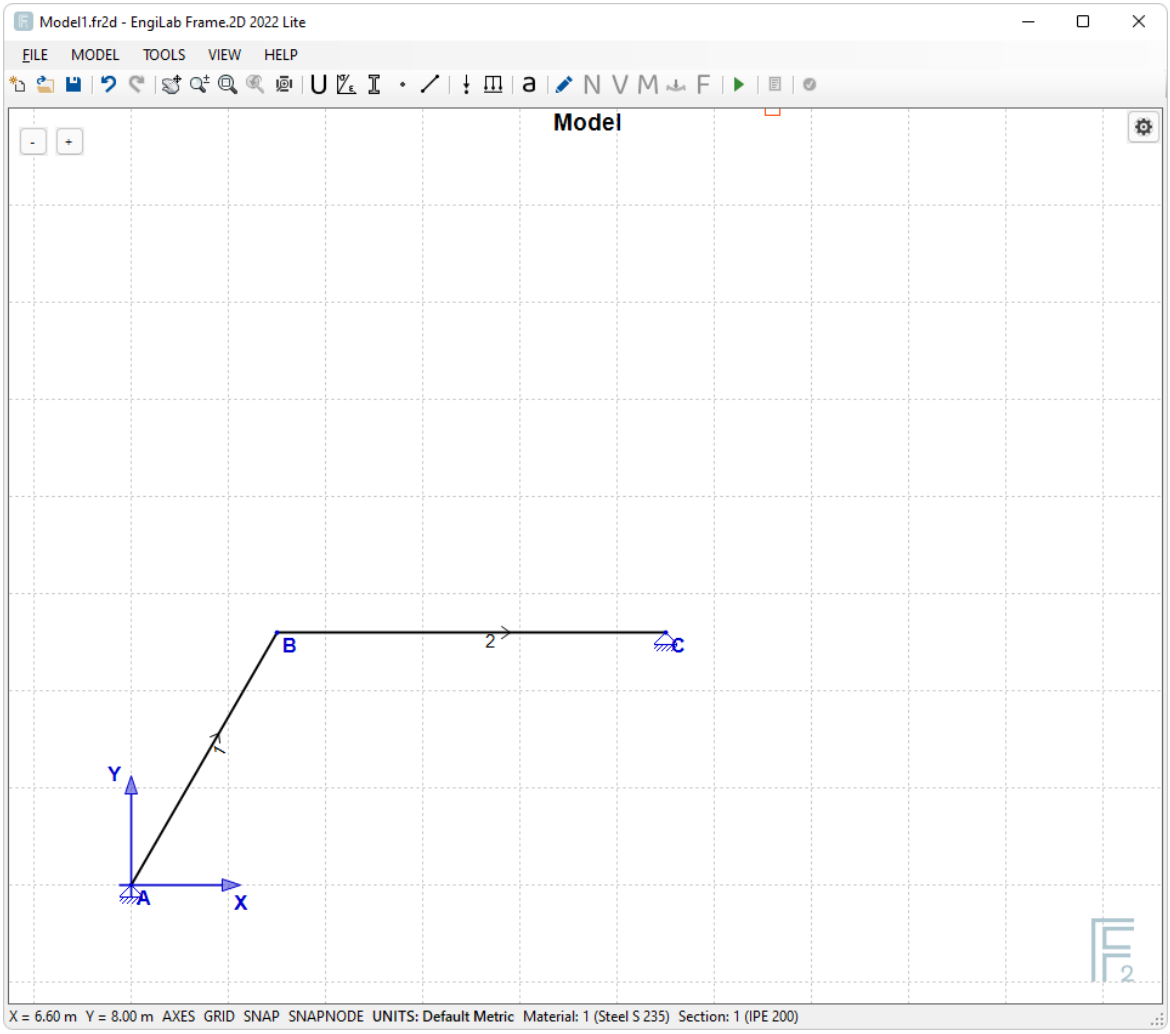



What we need to do is to add Constraints (Supports) to the Model. **Set the Constraints for Nodes A and B** to "pinned", as follows:




(c) 2022 EngiLab PC

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




The supports have been added to Nodes A and C as shown above, and Nodes B and C are now in their correct locations.

### 7.2.7 Step 6. Edit Elements

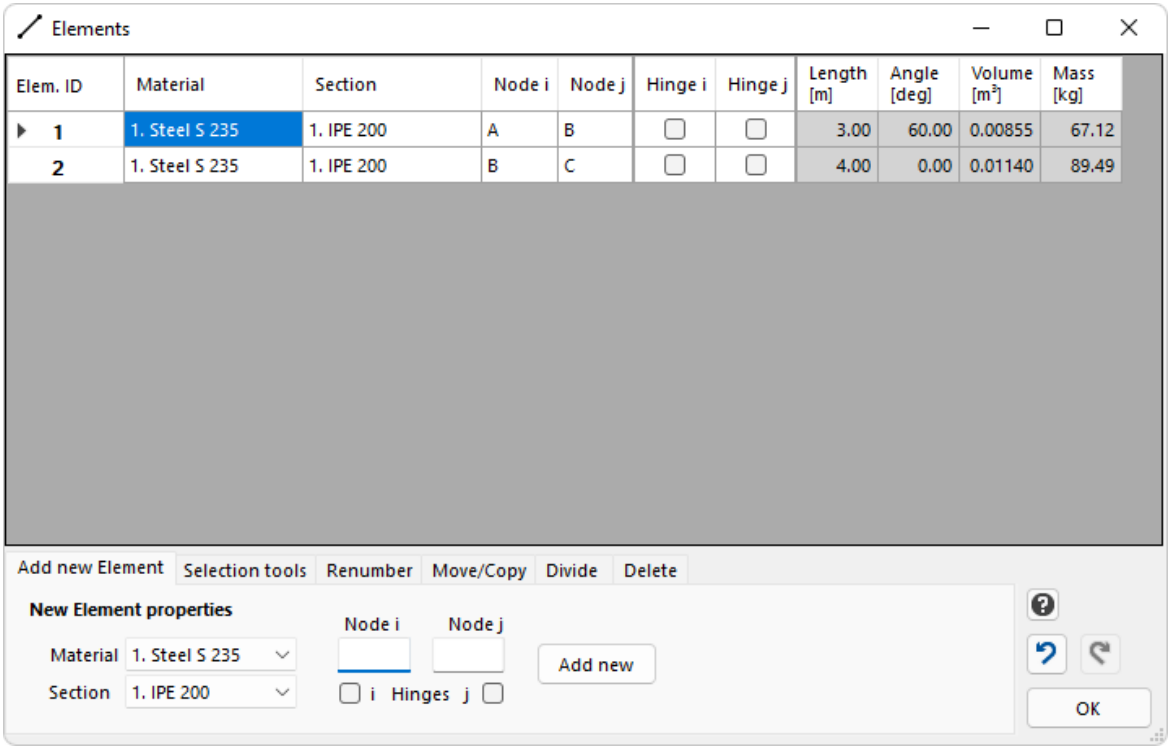



Now let's see the Elements. In our example, the Elements have the correct Material and Section, but we need to do two modifications:

- We need to add the **internal hinge at Node B**. Internal hinges are added in the Element properties. In this example, it has to be added either at the end of Element 1 or at the beginning of Element 2.
- There is a Nodal Load acting along Element 2 (in the middle of it). Since in EngiLab Frame.2D point loads must always be defined as Nodal Loads, this means that **a Node is needed in the middle of Element 2**. Element 2 has to be split in two equivalent Elements.

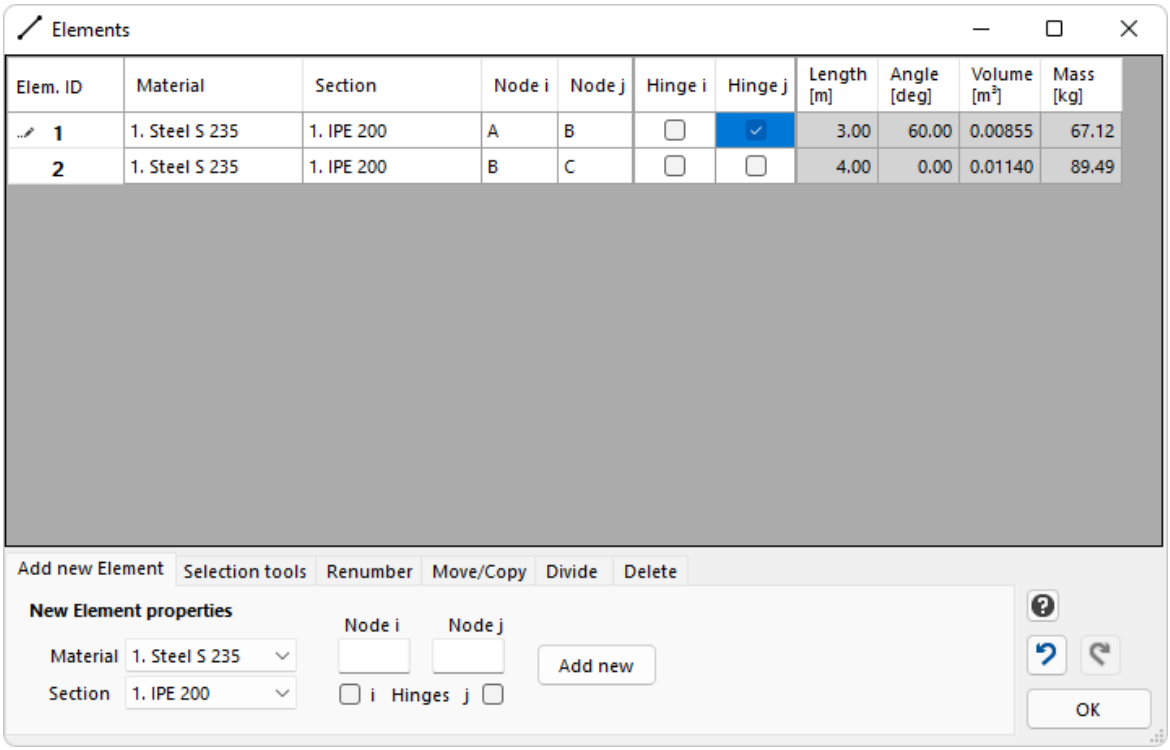
(c) 2022 EngiLab PC

Click to edit **Elements**. For details, see Elements.

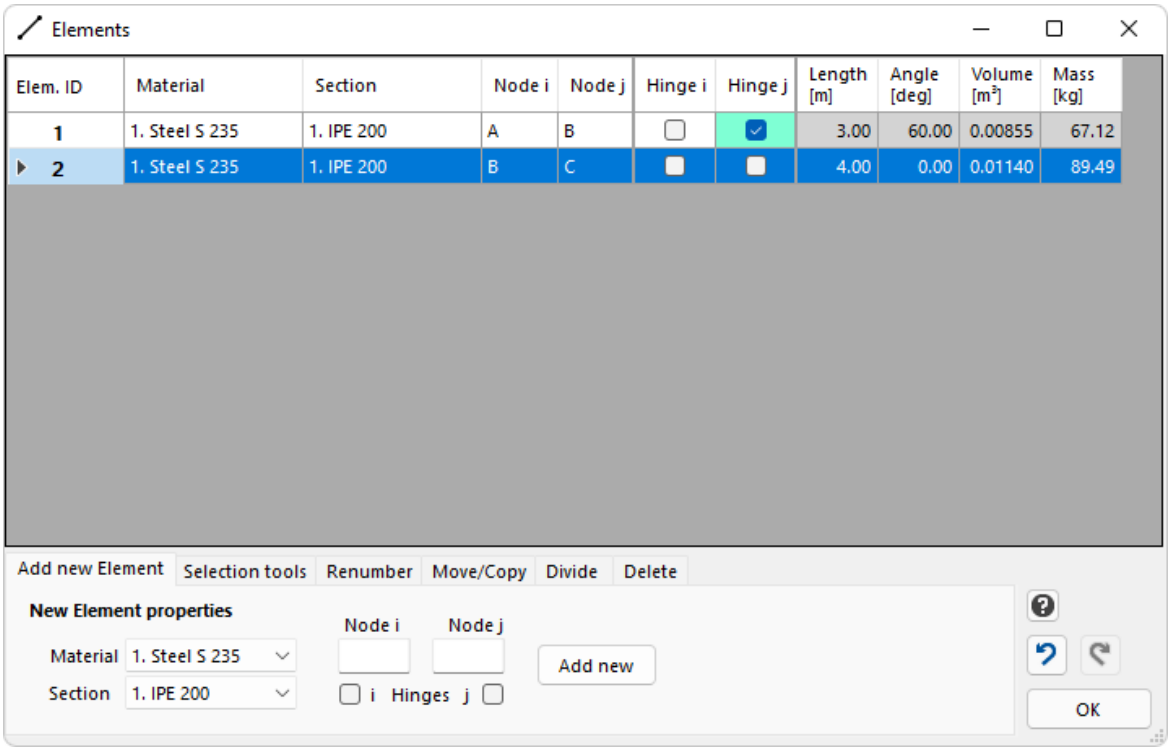



We see the two Elements and their properties. We see that the lengths of the two Elements are 3.00 m and 4.00 m, which is correct. Also, the Angle of Element 1 is 60 deg, which is also correct. We define a hinge at the end j of Element 1 as follows:

(c) 2022 EngiLab PC

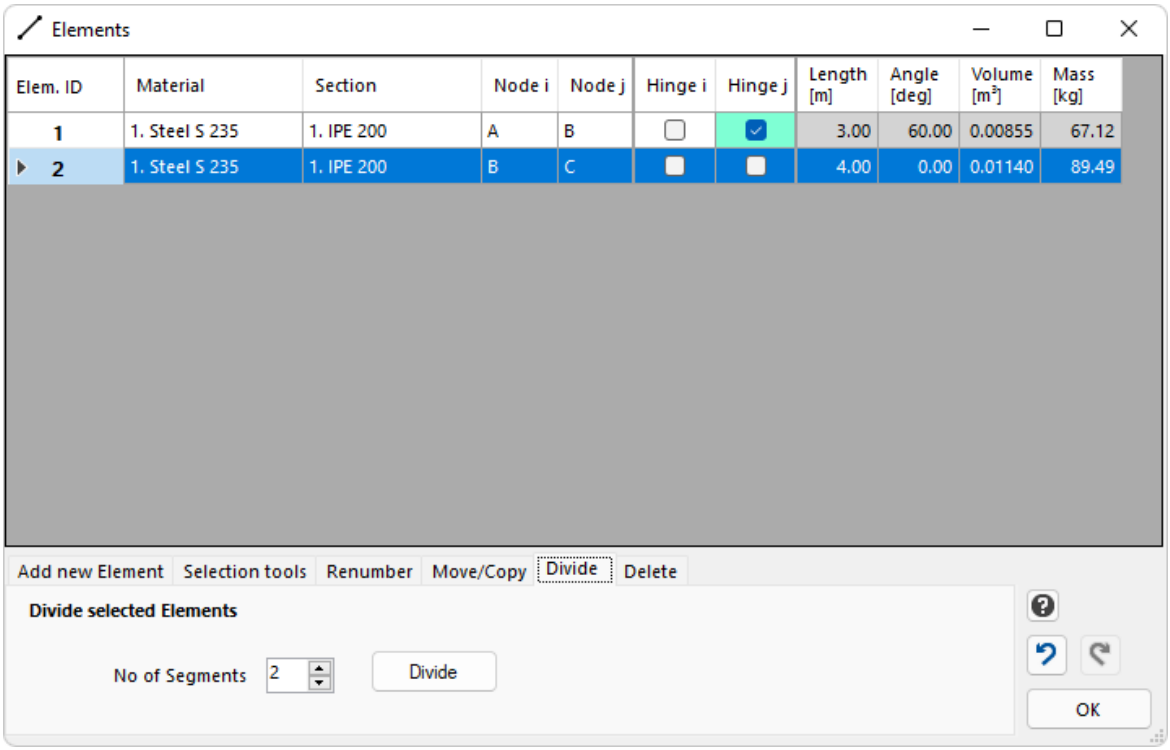



Next we need to divide Element 2 into two Elements. For this we click the "2" of Element 2 and we highlight the entire second row as follows:

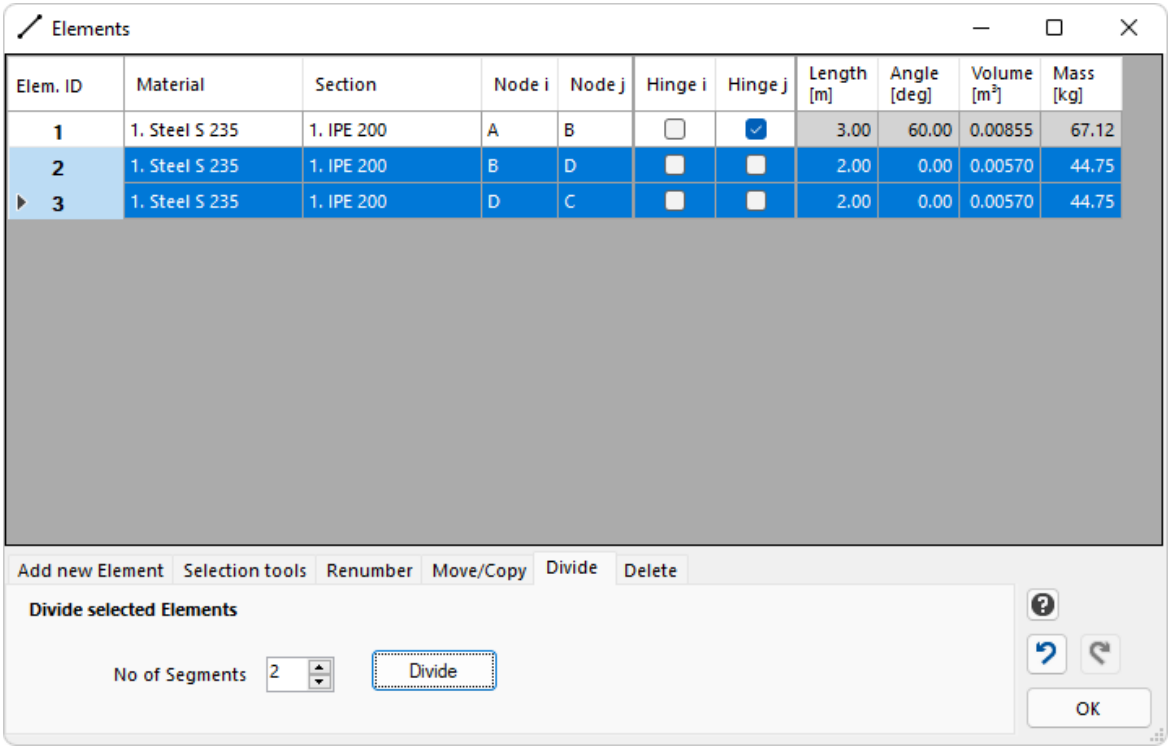



We click the **Divide tab**.

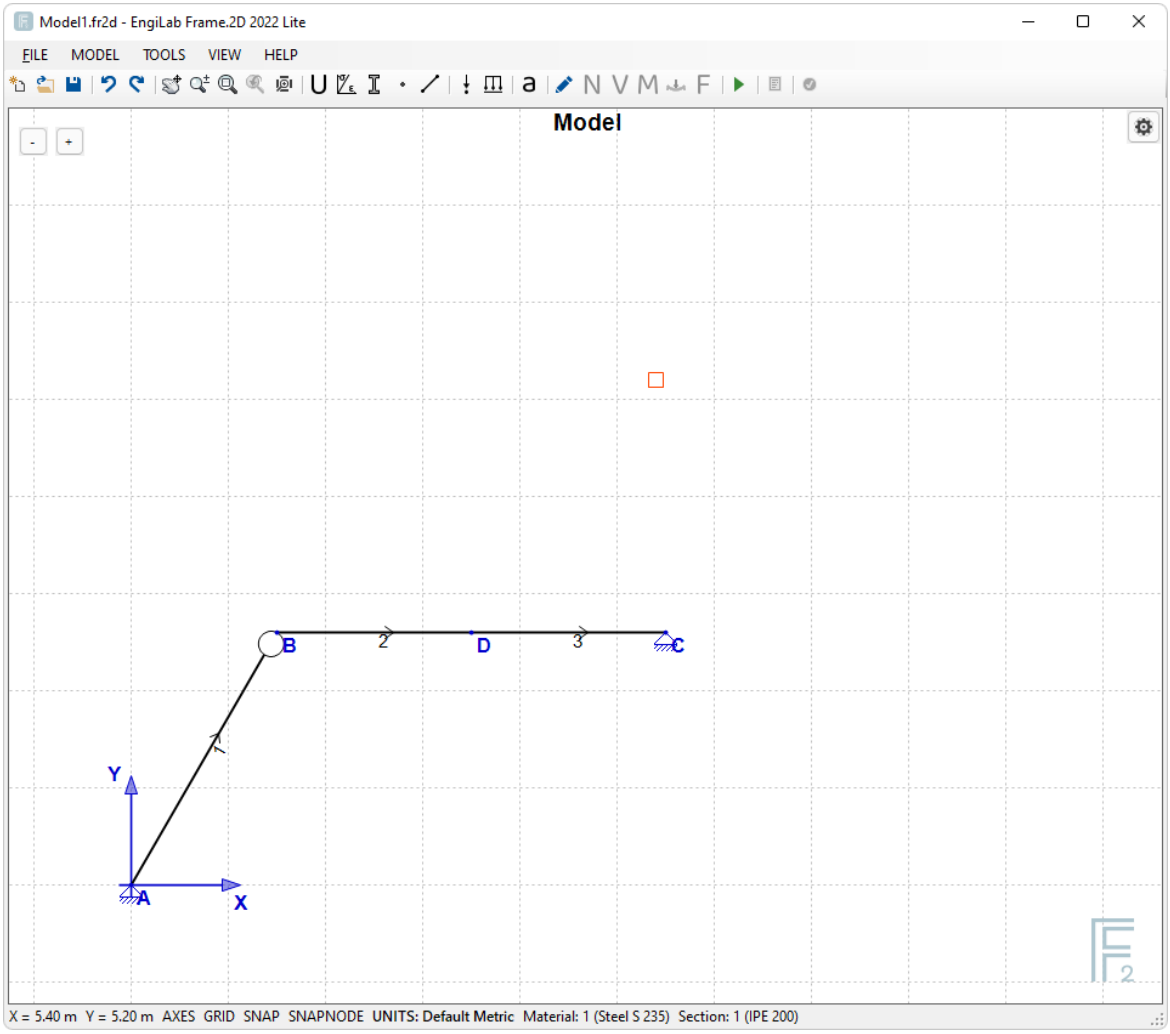
(c) 2022 EngiLab PC




The default Number of Segments is 2 and we can keep it. Having selected Row (Element) 2, we click the "**Divide**" button. Then we get a new Element. Now Element 2 is 2 m and also Element 3 is 2 m long. The previous Element has been divided into 2 equivalent Elements.



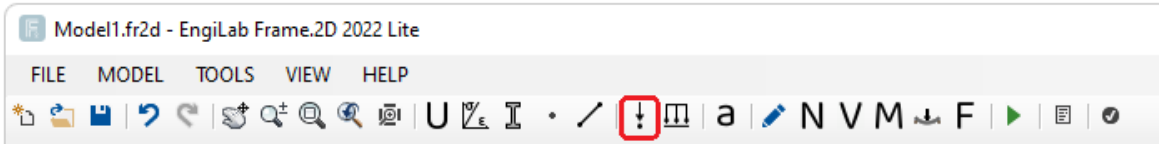
See now our Model. A new Node has been generated in the middle of the previous Element. We can now define our Nodal Load on this new Node D.




Then **Click OK** to exit Elements.

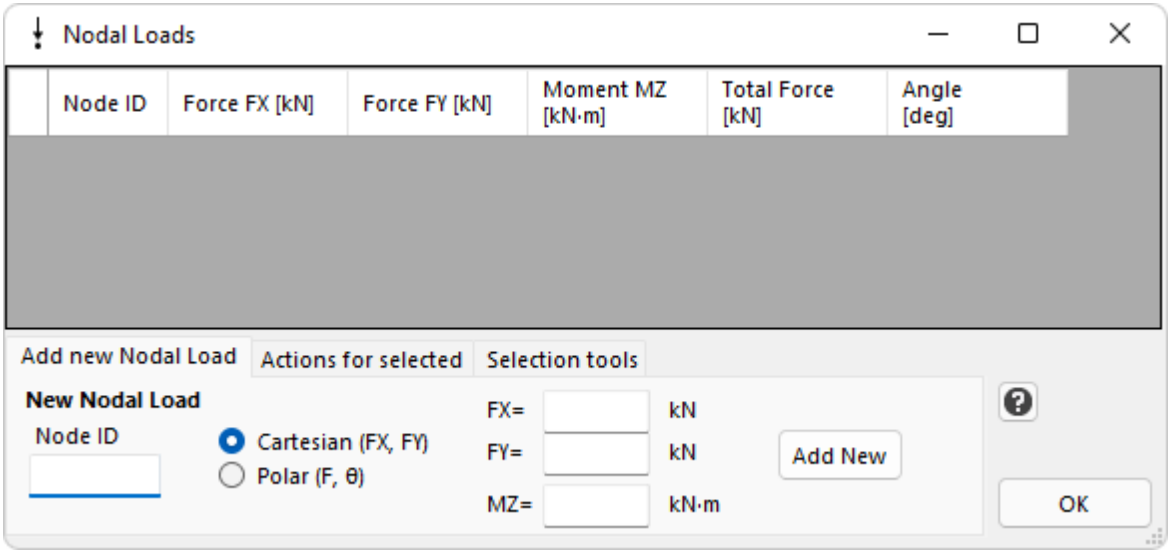
(c) 2022 EngiLab PC

### 7.2.8 Step 7. Define Nodal Loads

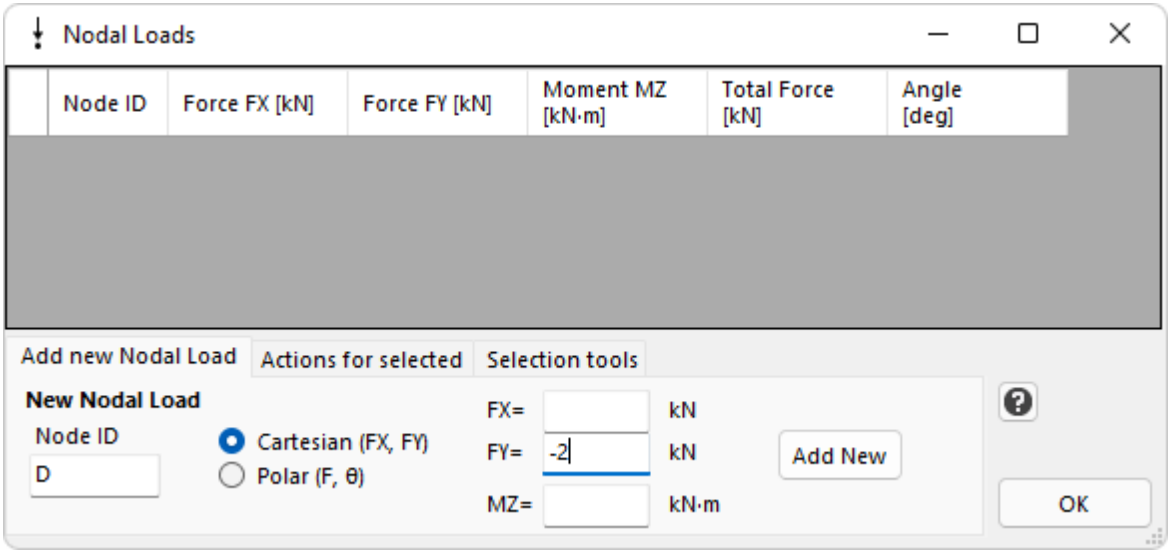



Click to define **Nodal Loads**. For details, see Nodal Loads.

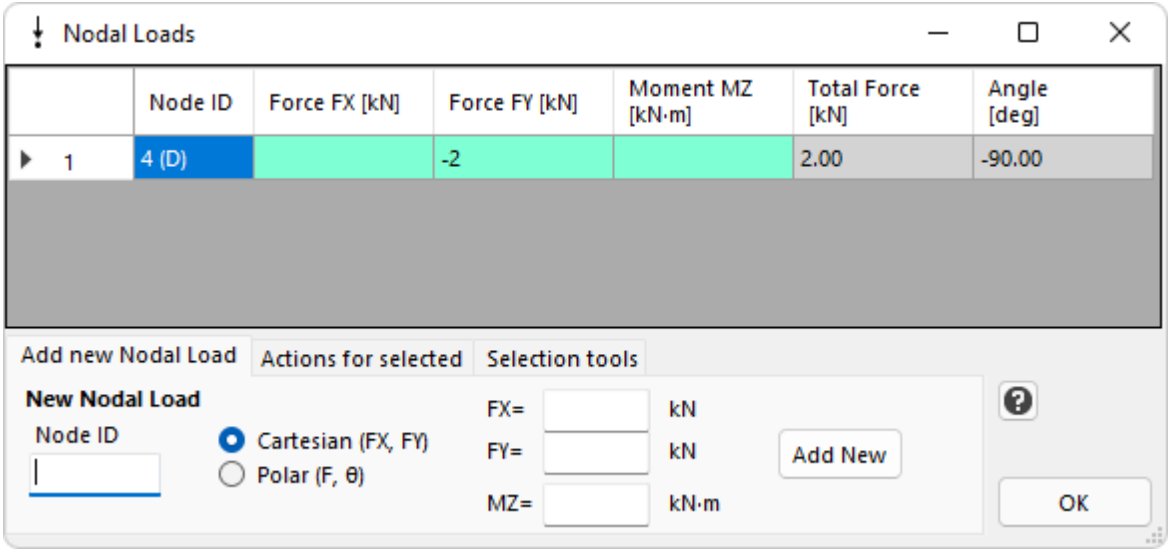
The Nodal Loads form appears.




Add a Nodal Load **FY=-2 [kN] for Node D**.

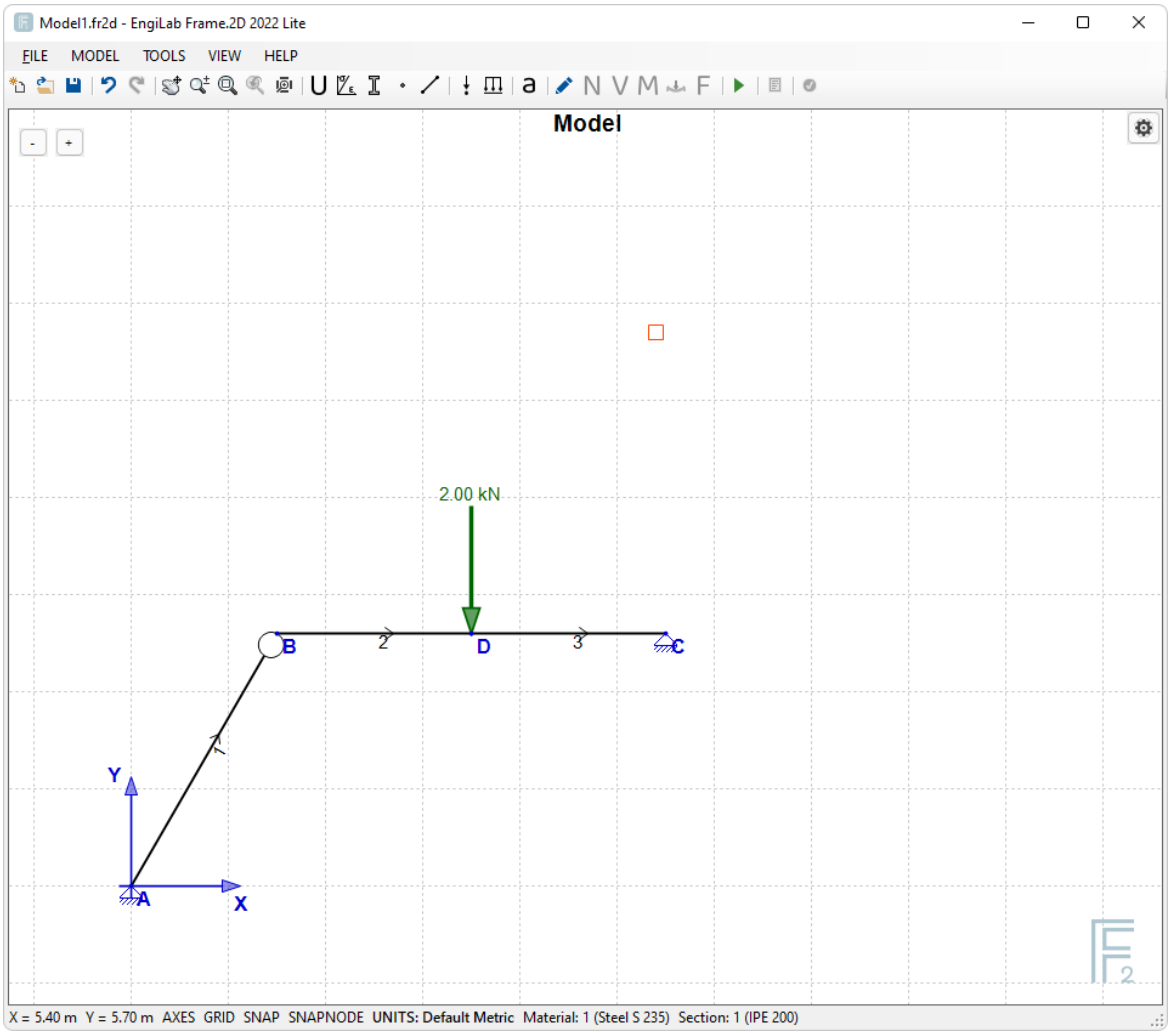


**Click "Add New"**. You get the following.




**Click OK to exit**. Now the Model should look like this. The Nodal Load has been applied!

(c) 2022 EngiLab PC


Model1.fr2d - EngiLab Frame.2D 2022 Lite
FILE MODEL TOOLS VIEW HELP
Model
2.00 kN
B
2
D
3
C
1
Y
A
X
X = 5.40 m Y = 5.70 m AXES GRID SNAP SNAPNODE UNITS: Default Metric Material: 1 (Steel S 235) Section: 1 (IPE 200)

### 7.2.9 Step 8. Define Elemental Loads

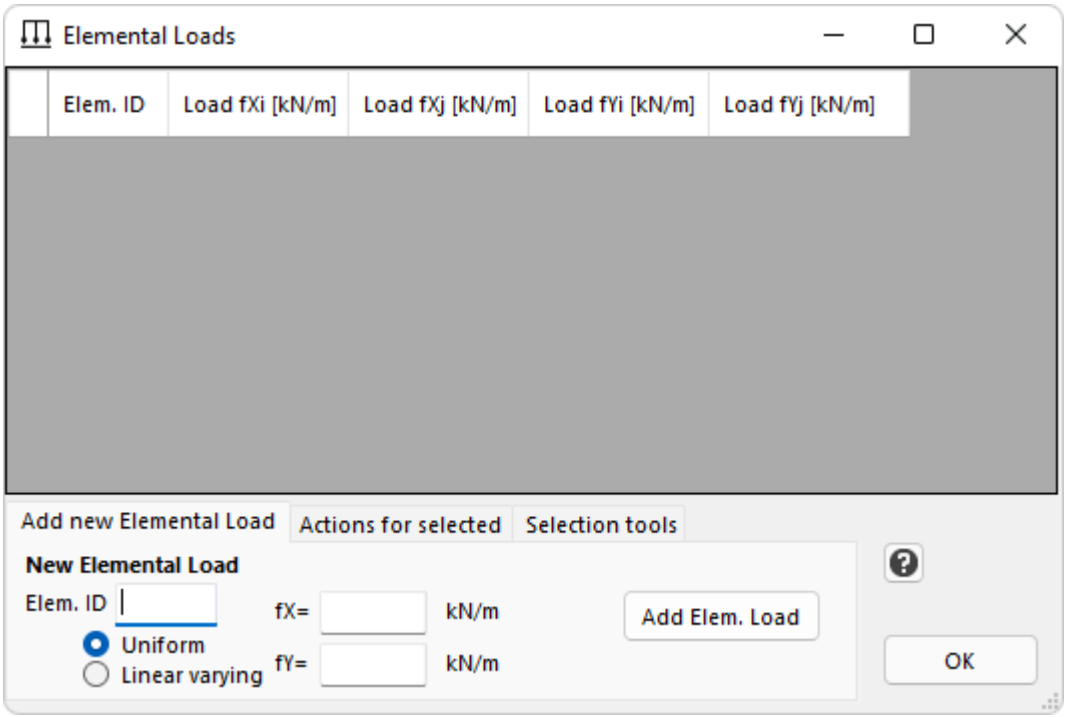



In our example, there are no Elemental Loads, so this step can be skipped.

---

Click to define **Elemental Loads**, in case there are any. For details, see Elemental Loads.

The Elemental Loads form appears.




**Click OK to exit**.

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

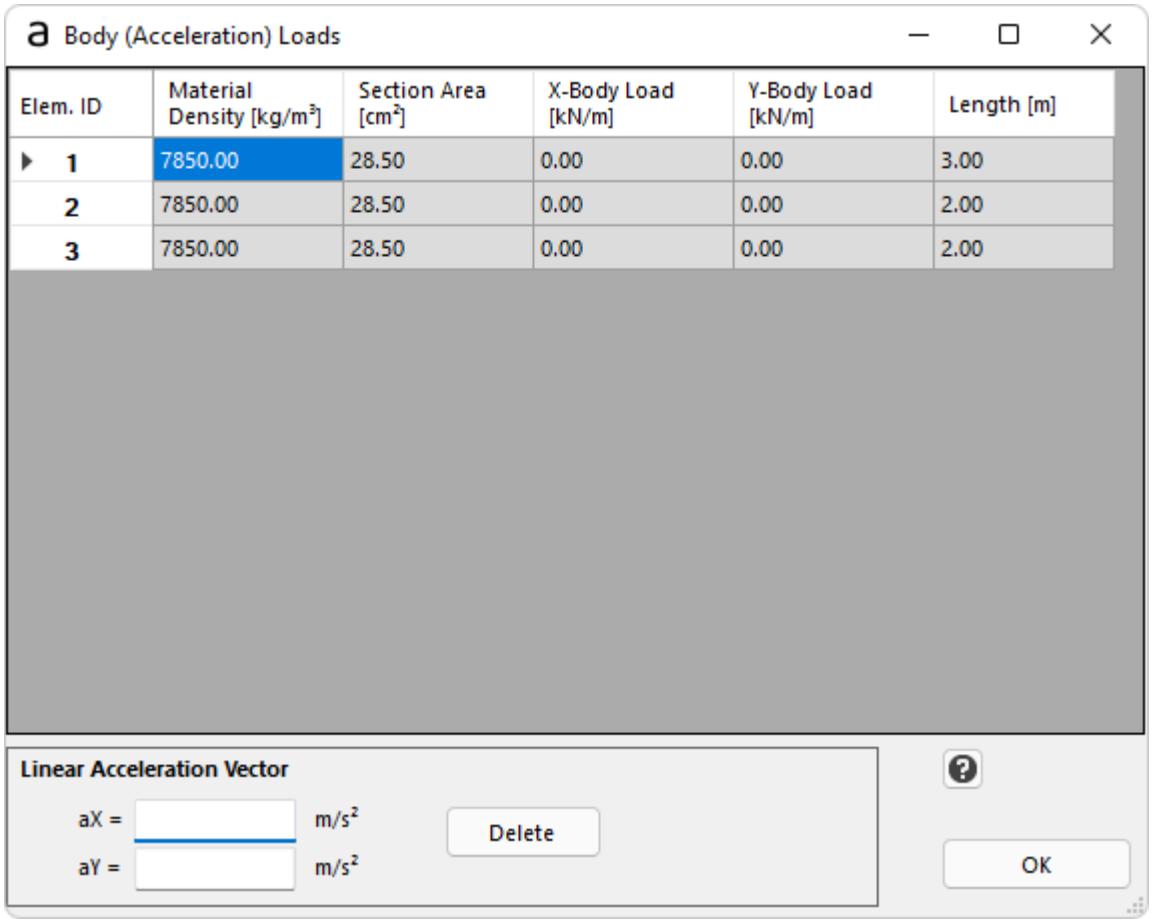



In our example, there are no Body (Acceleration) Loads, so this step can be skipped.

---

Click ⓐ to define **Body (Acceleration) Loads**, in case there are any. For details, see Body (Acceleration) Loads.

The Body (Acceleration) Loads form appears.



**Click OK to exit**.

**Note**: If the want to take into account the self-weight of Elements as an additional elemental load for each Element, we have to provide the **Material Density** for the Material of each Element (we have done that in this example) and also to define a **Linear Acceleration Vector** equal to the **standard earth gravitational acceleration**. We should set the earth gravitational acceleration with a minus (-) sign at the Y direction, i.e. gravity acting towards -Y global axis.

Using the Default Metric System of Units, we need to **input -9.80665, or simply -9.81, at the aY component of the Linear Acceleration Vector** and then press ENTER on the keyboard. The program will automatically calculate the self weight (in kN/m) for each Element, as a Y-Body Load. All properties on the table are read-only properties.

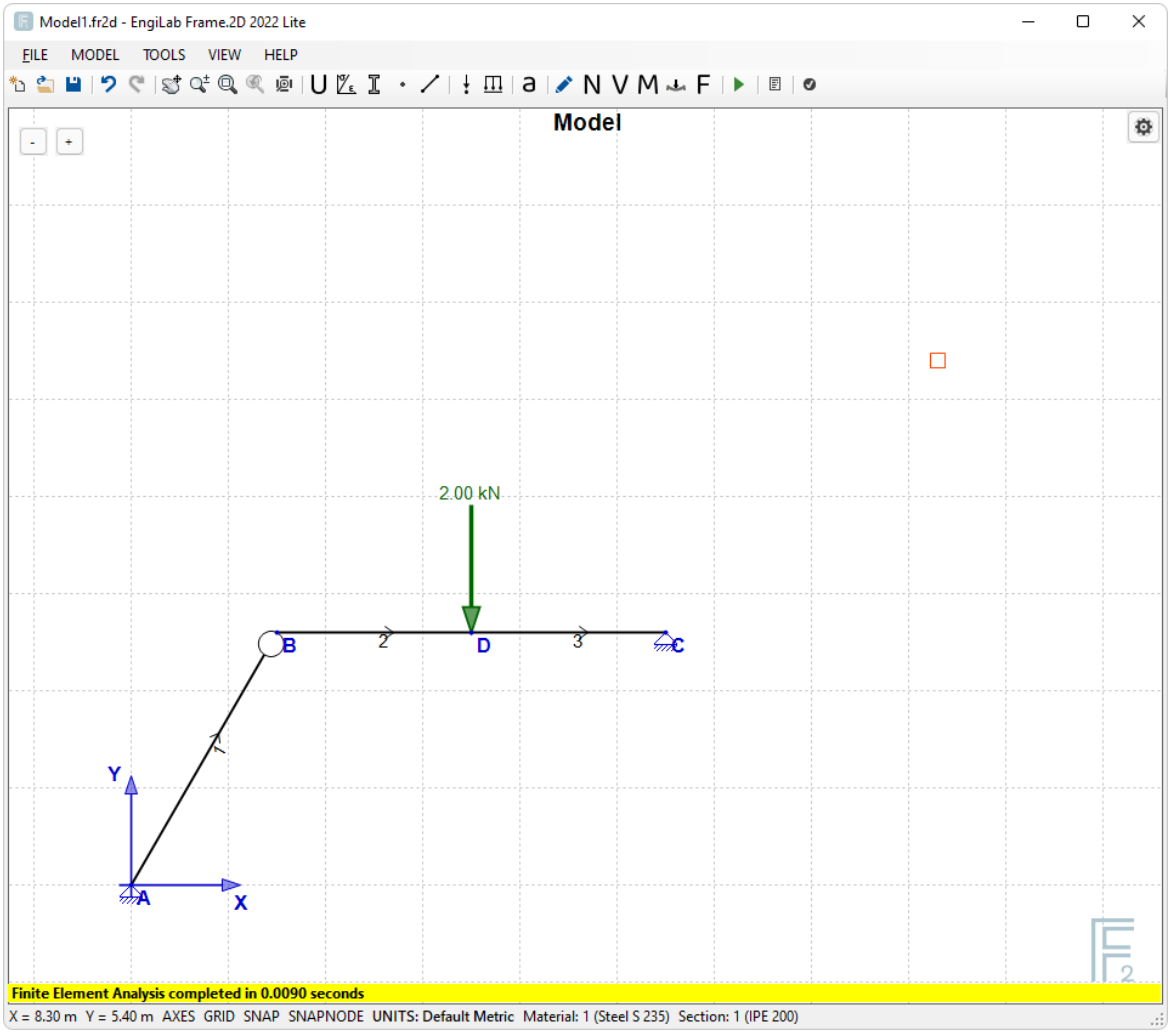
(c) 2022 EngiLab PC

### 7.2.11 Step 10. Run the Analysis




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

You should see the following picture.



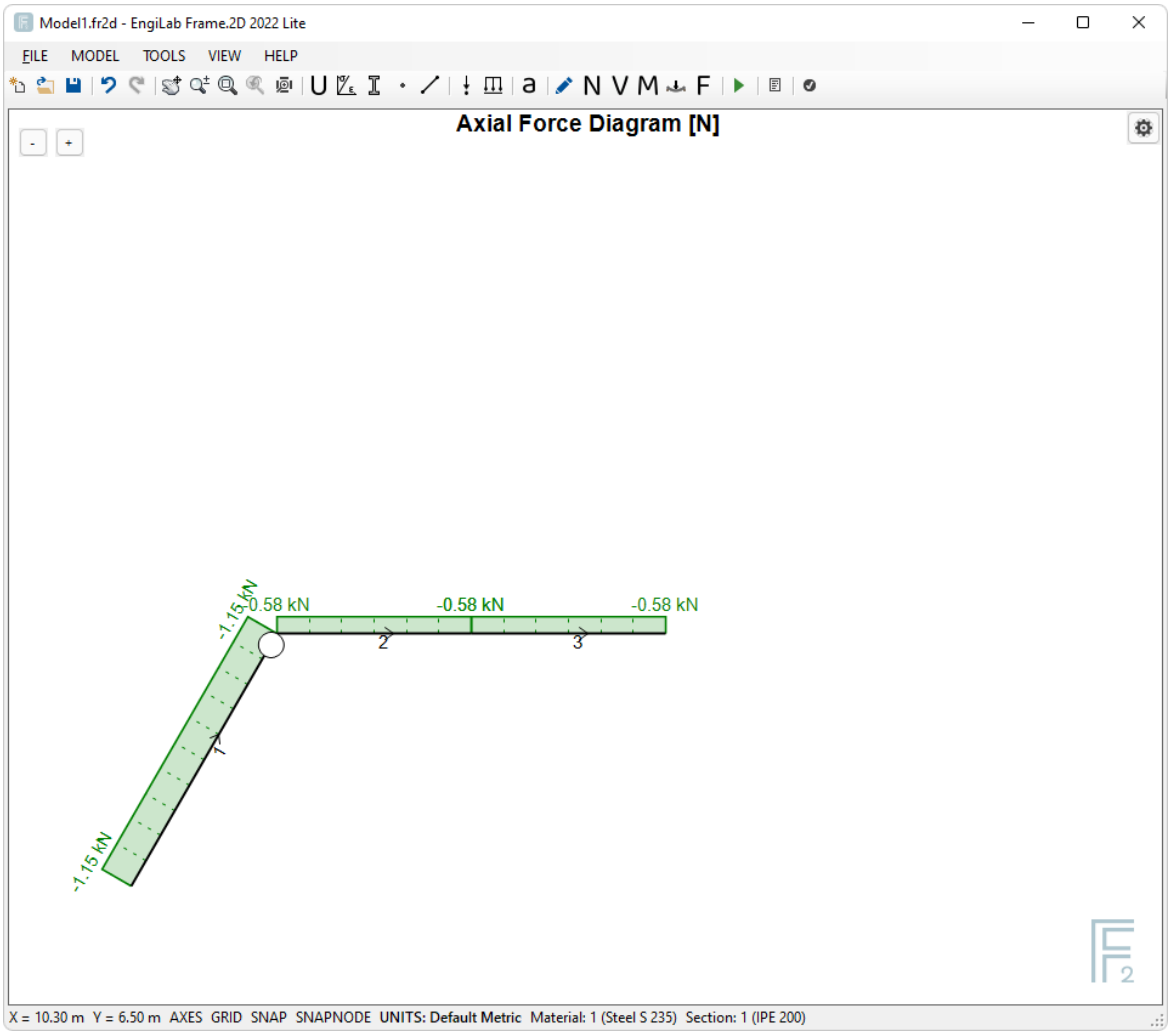

The Finite Element Analysis has been completed successfully.

### 7.2.12 Step 11. View the Analysis Results on screen




#### 1. Axial Force Diagram

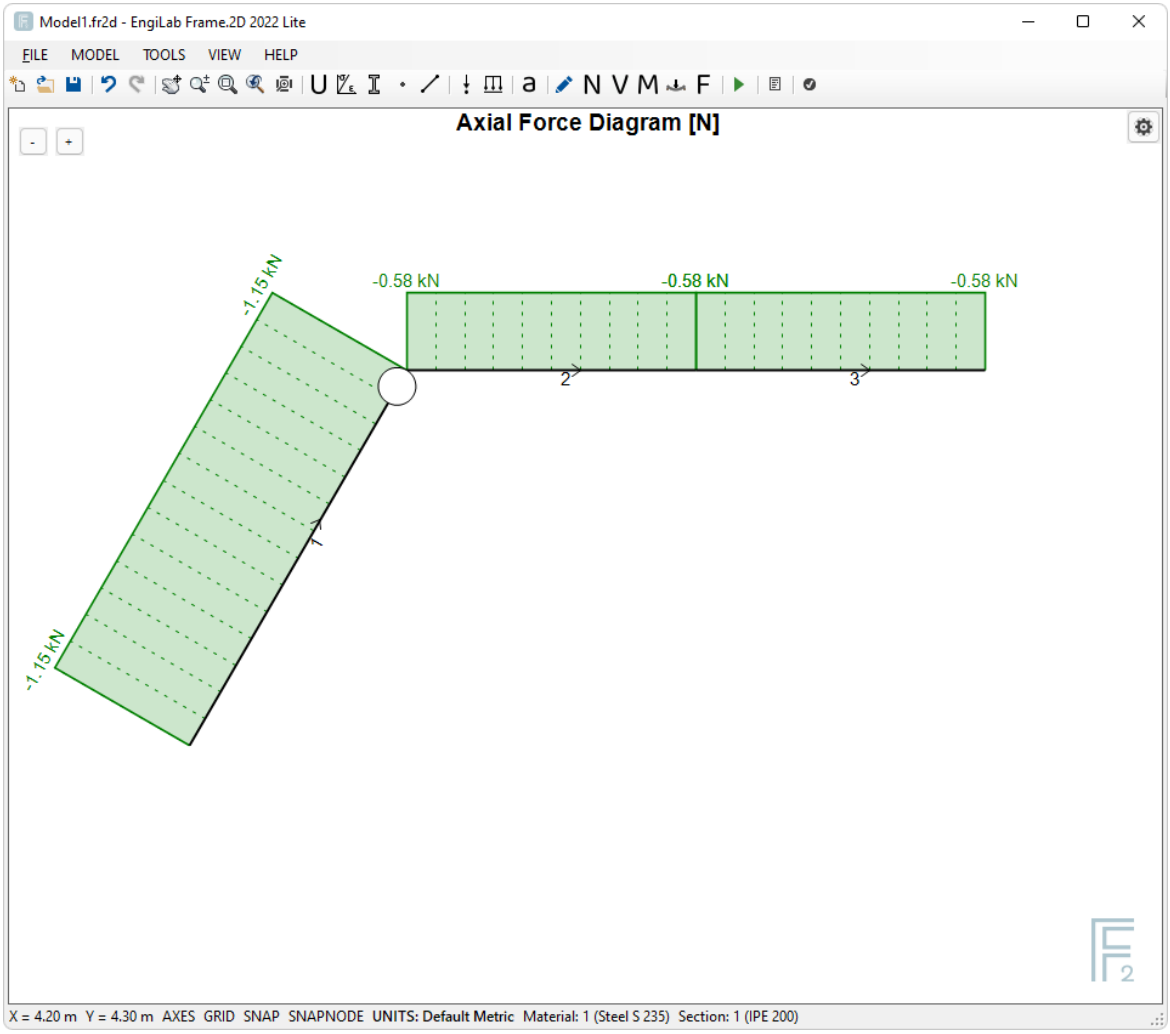
Click N, to see the Axial Force Diagram. You may take the picture below.




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

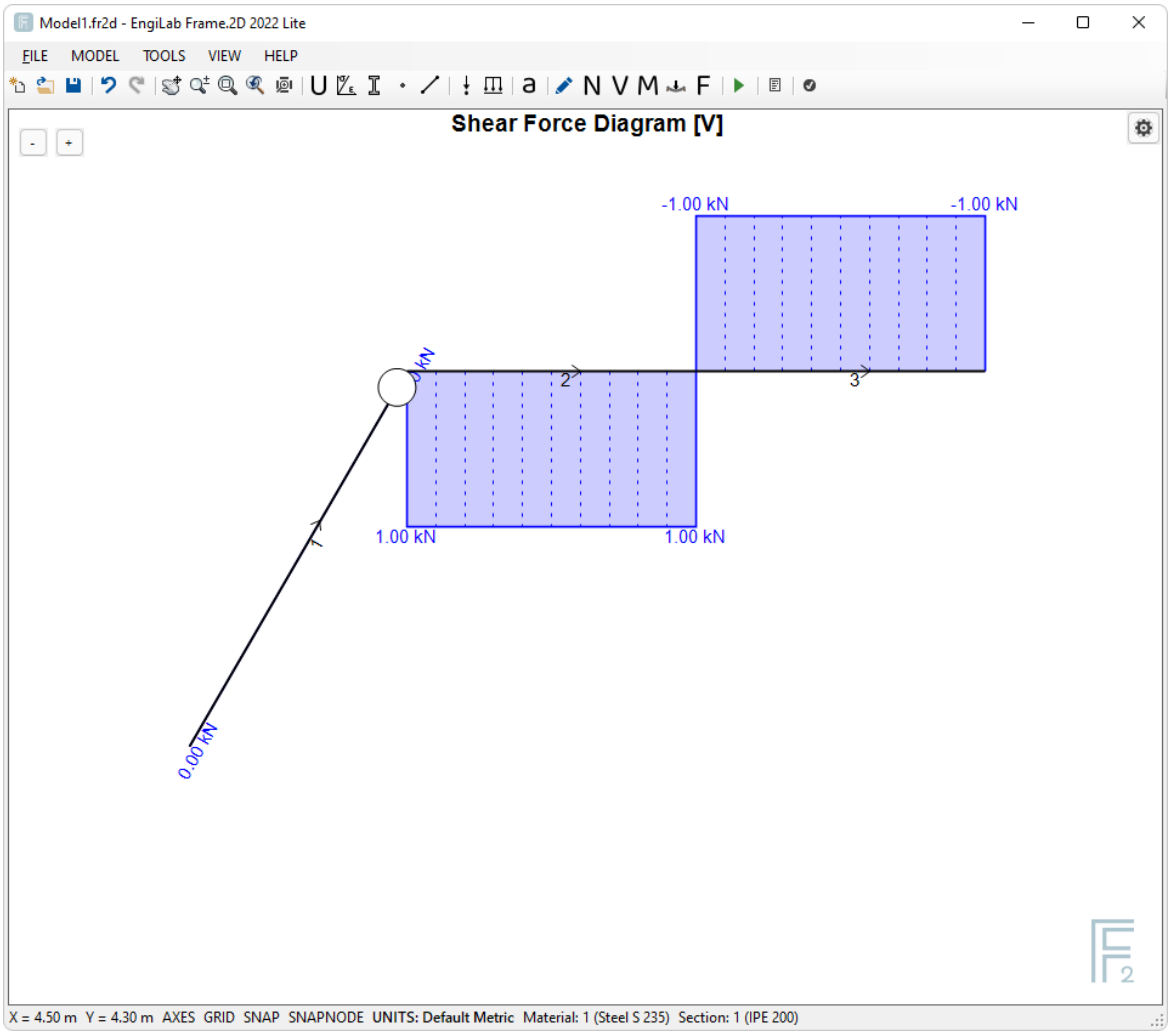
(c) 2022 EngiLab PC

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



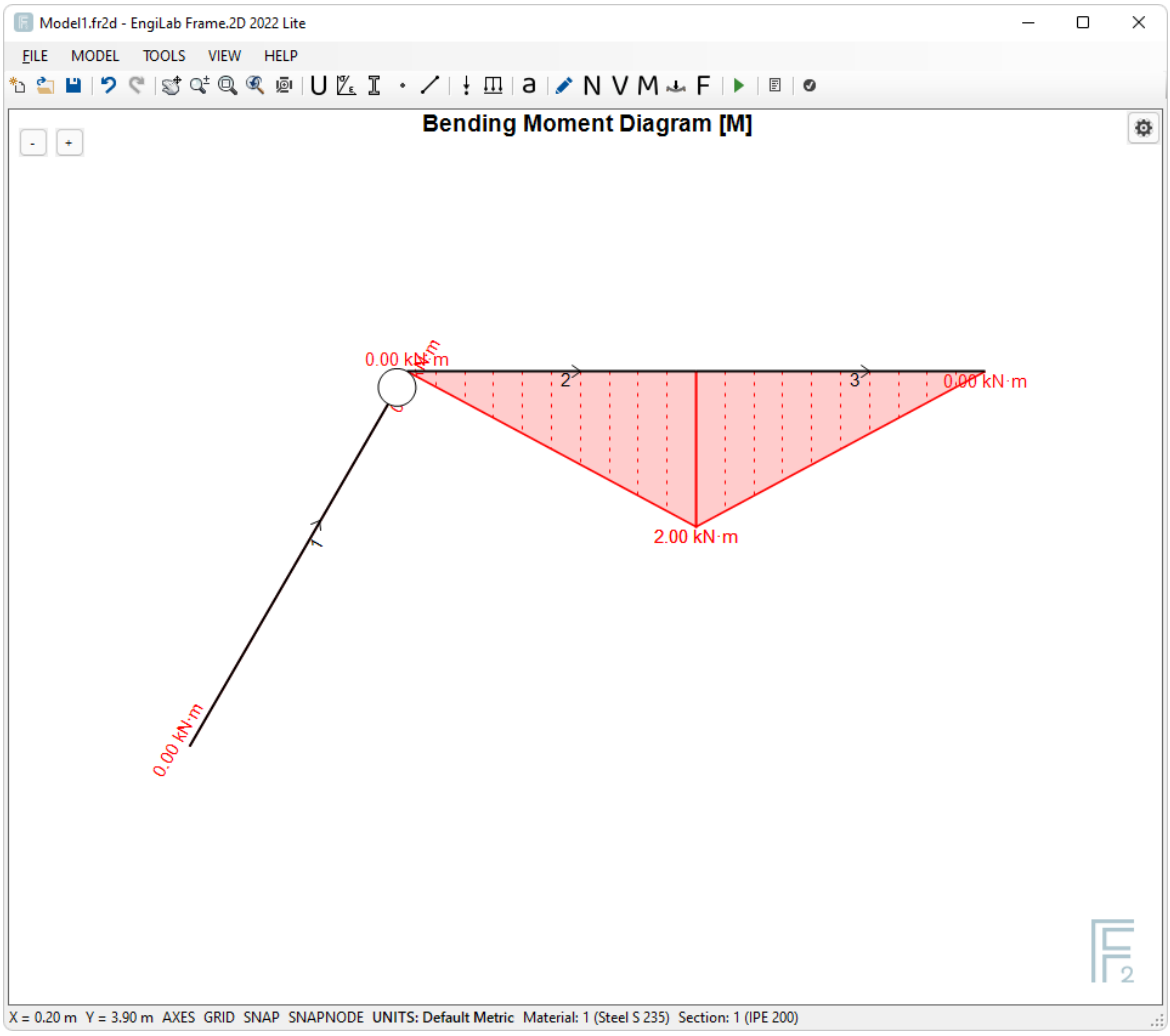
## 2. Shear Force Diagram

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

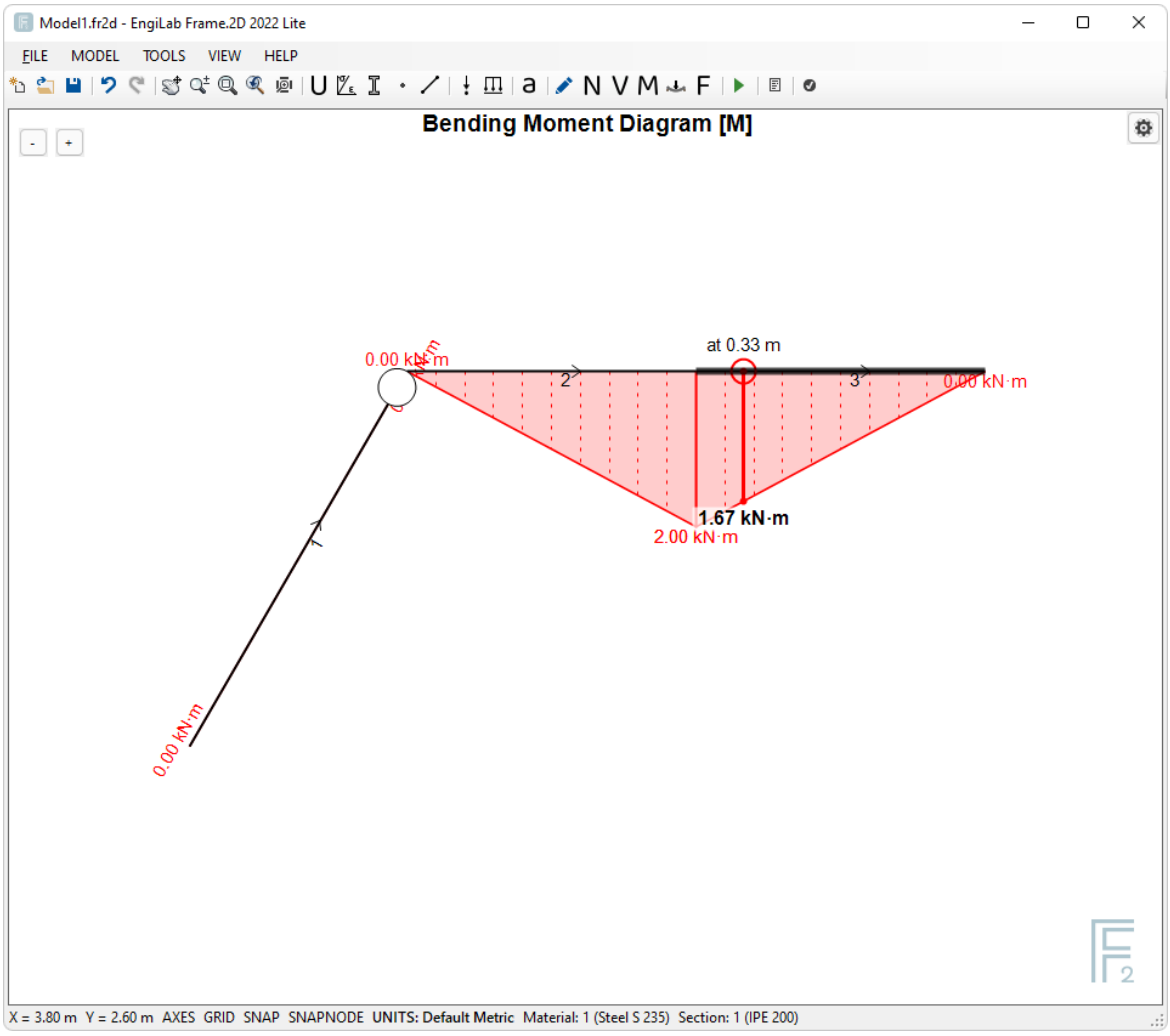


### 3. Bending Moment Diagram

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



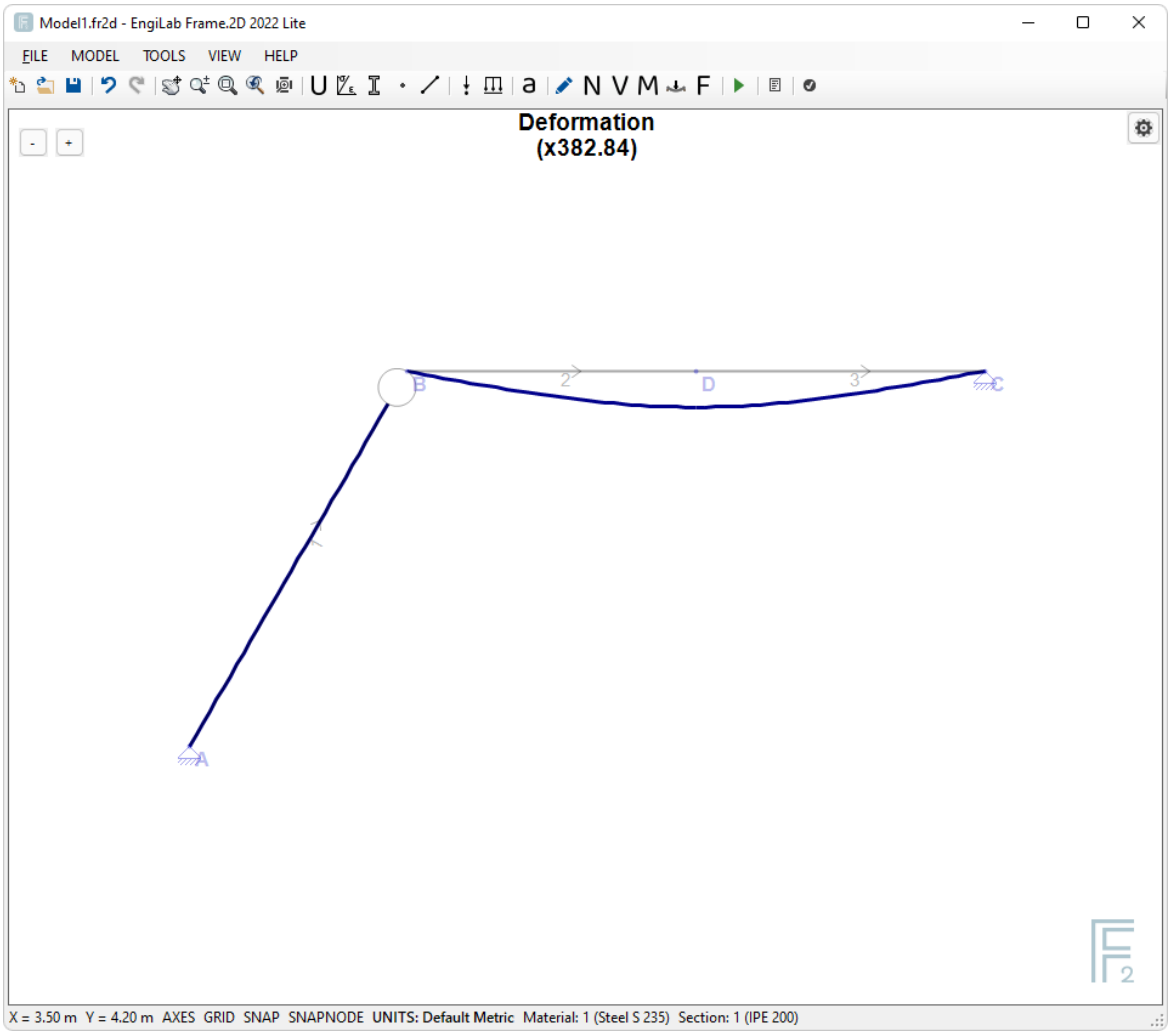
Note that if you move the pointer over an Element, you can read the corresponding value of the Diagram, as shown below for the Bending Moment Diagram case. This happens for all Diagrams and also for the Deformation and the Free Body Diagram.



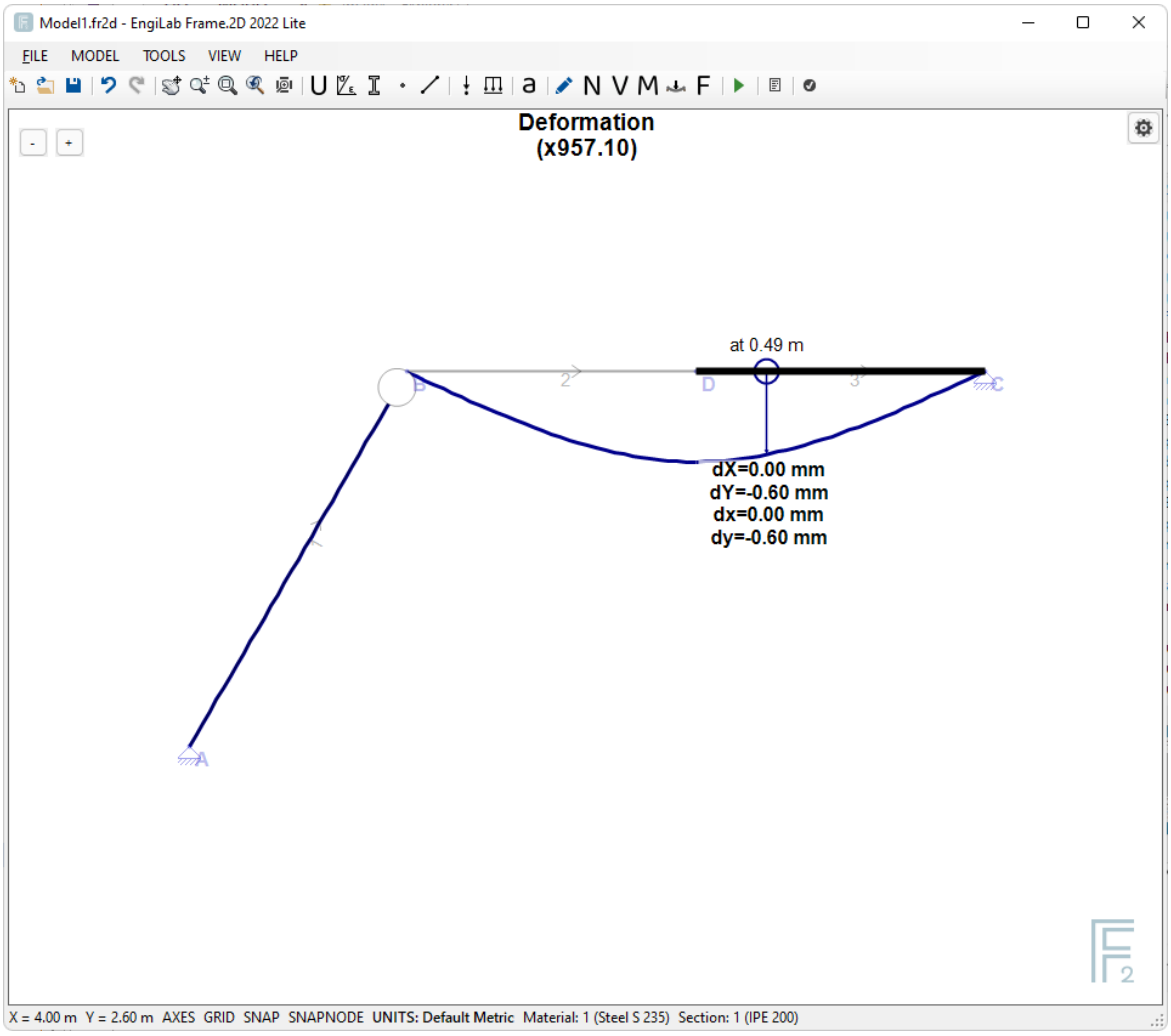

(c) 2022 EngiLab PC

## 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 x382.84.



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 x957.10.




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

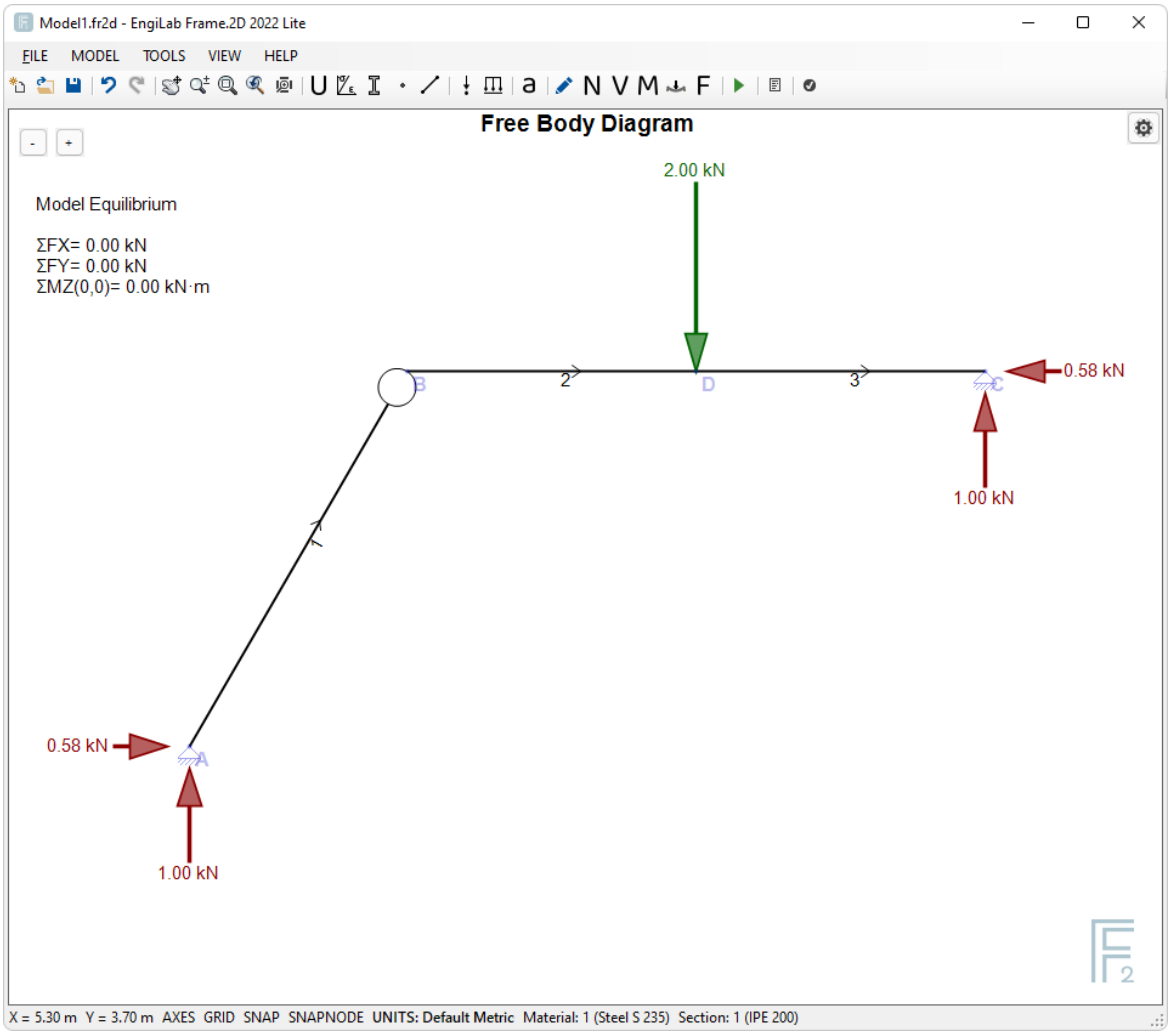
**IMPORTANT**: The values that are given on screen are the **dX**, **dY**, **dx** and **dy** displacements of the corresponding point of each element.

- **dX** and **dY** are the X- and Y- displacements of a point, respectively, in **Global axes**. For example a positive dX means displacement to the right, while a positive dY means displacement upwards.
- **dx** and **dy** are the x- and y- displacements of a point, respectively, in **Local Element Axes**, where x is defined from the Start Node i and the End Node j of the Element and y is perpendicular to it.

(c) 2022 EngiLab PC

## 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.




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.

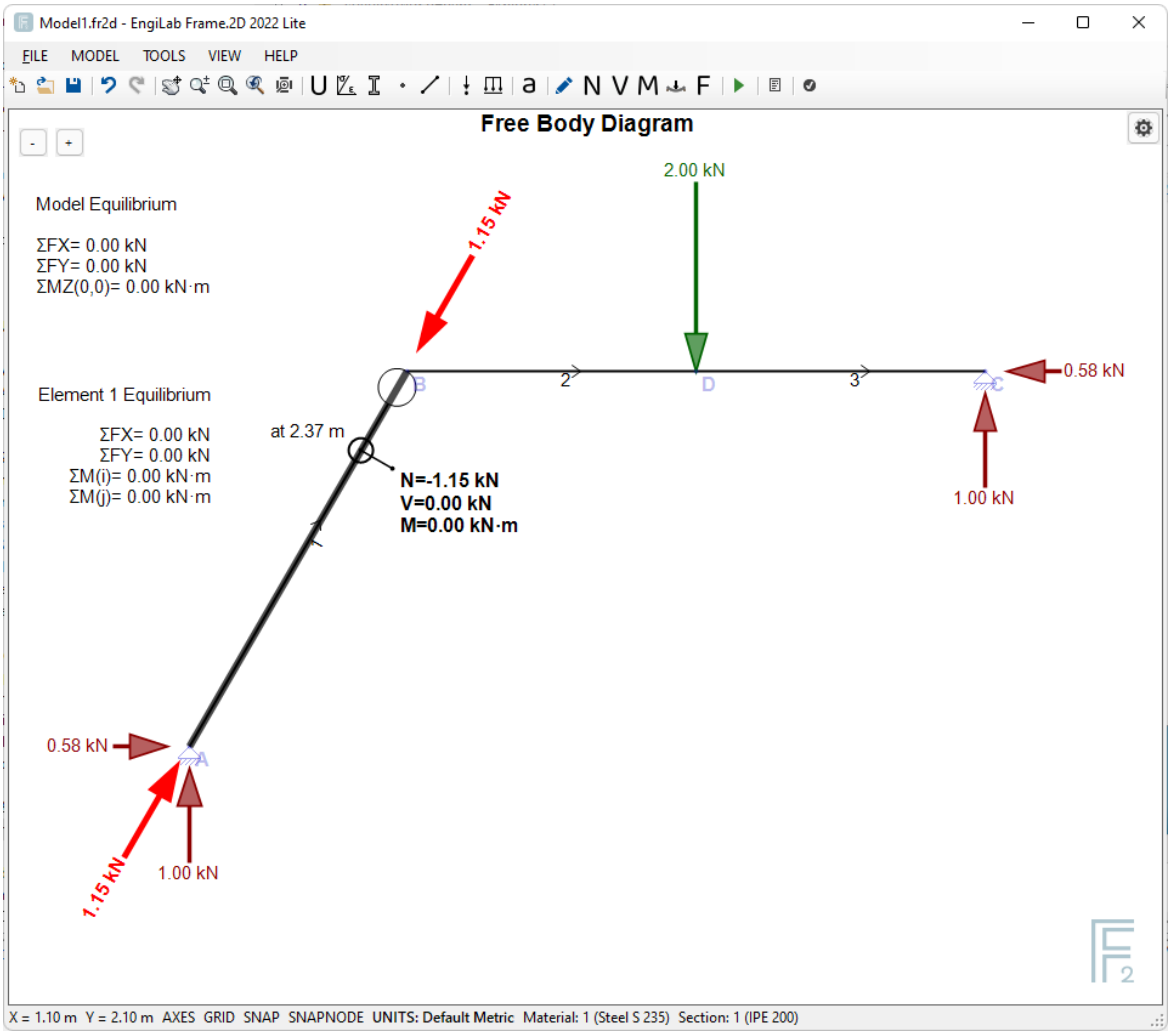
We see that the support reactions are the same as the ones found with Hand Calculations. Based on the hand calculations, the reaction forces were:

- $\mathbf{C_x}$ = 577 N = **0.578 kN**
- $\mathbf{C_y}$ = 1000 N = **1 kN**

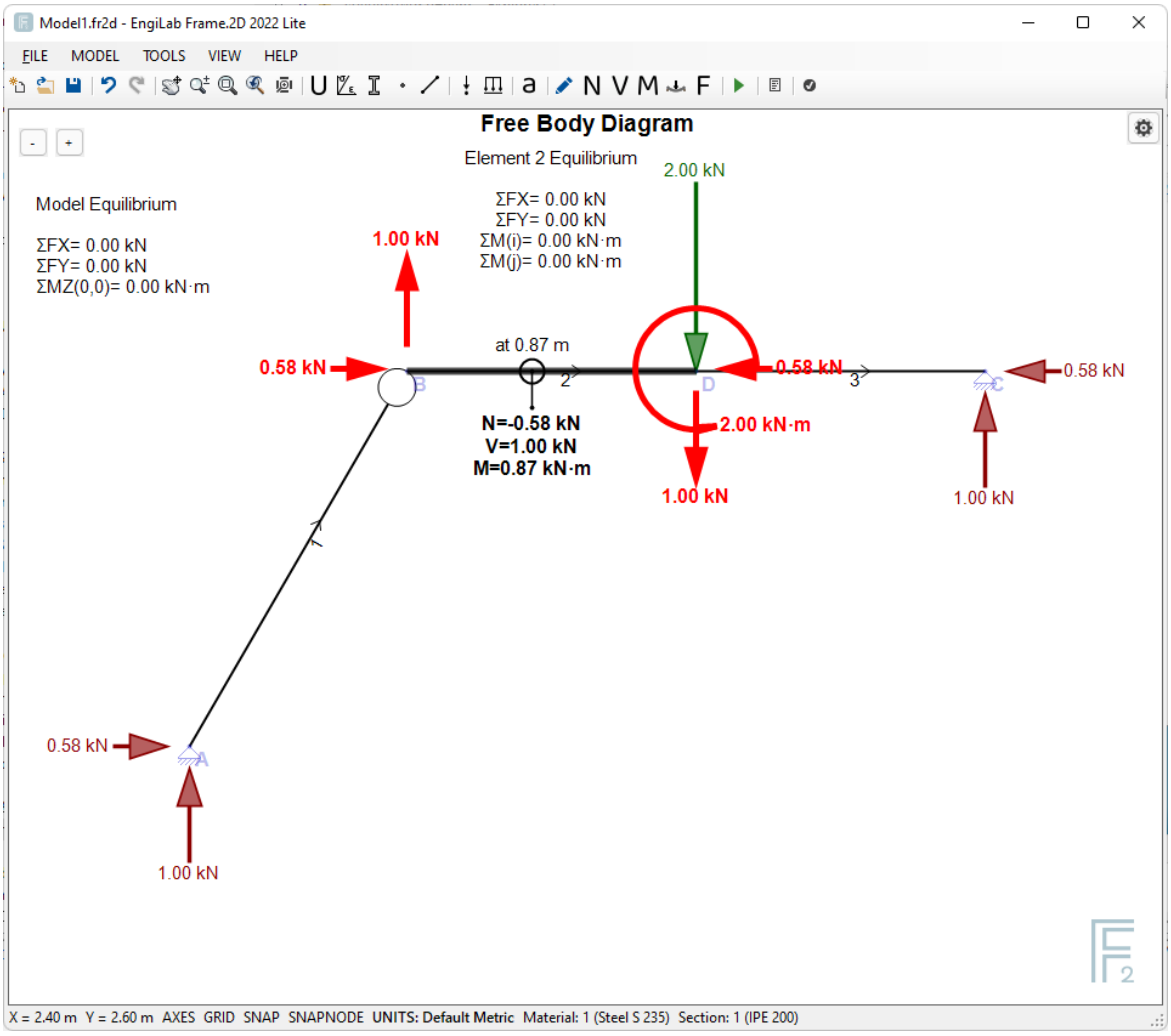
(c) 2022 EngiLab PC

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. For example we see that Element 1 has only axial tension:

(c) 2022 EngiLab PC




While Element 2 has Axial Force, Shear Force and Bending Moment. There is no bending moment on the left edge (at Node B), because of the internal hinge.




**IMPORTANT**: In the example above, we see the calculations of the equilibrium of Elements 1 and 2 of the Model. All equilibrium calculations (ΣFX, ΣFY, ΣM(i), ΣM(j)) are zero, otherwise there would be a problem in our Model or in the analysis results.

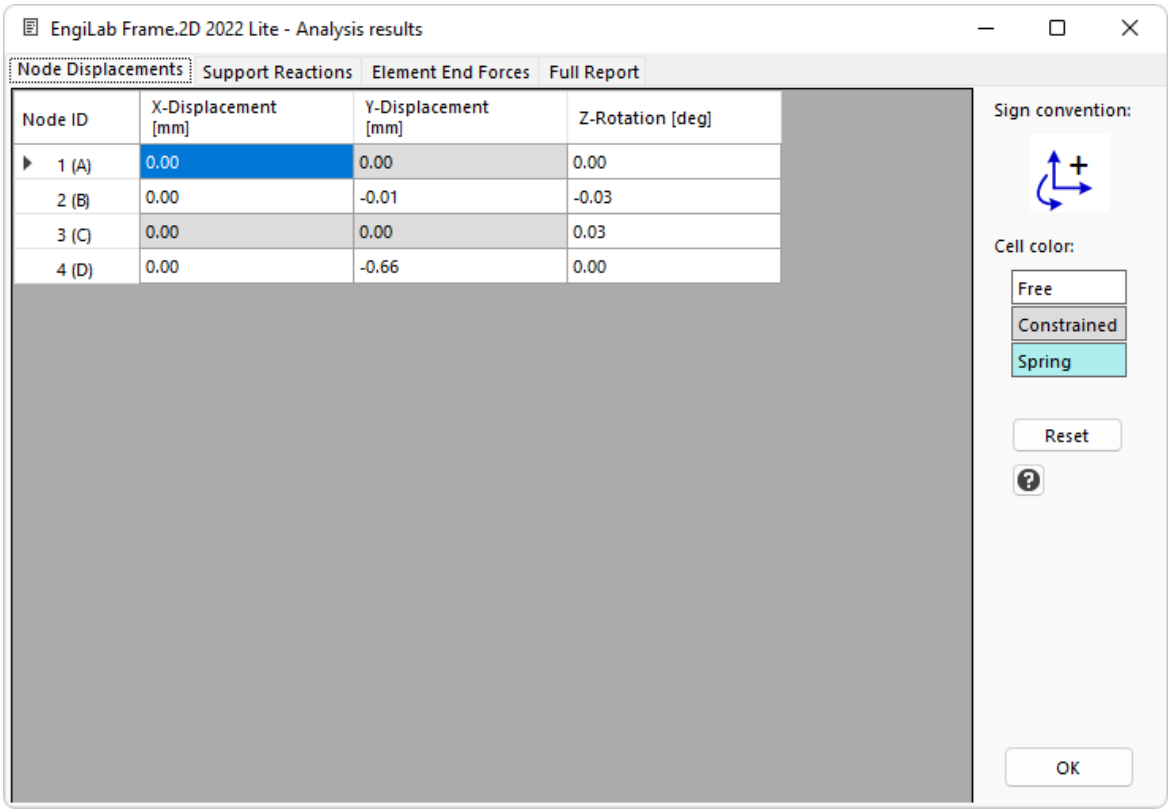
(c) 2022 EngiLab PC

### 7.2.13 Step 12. View the analytical results

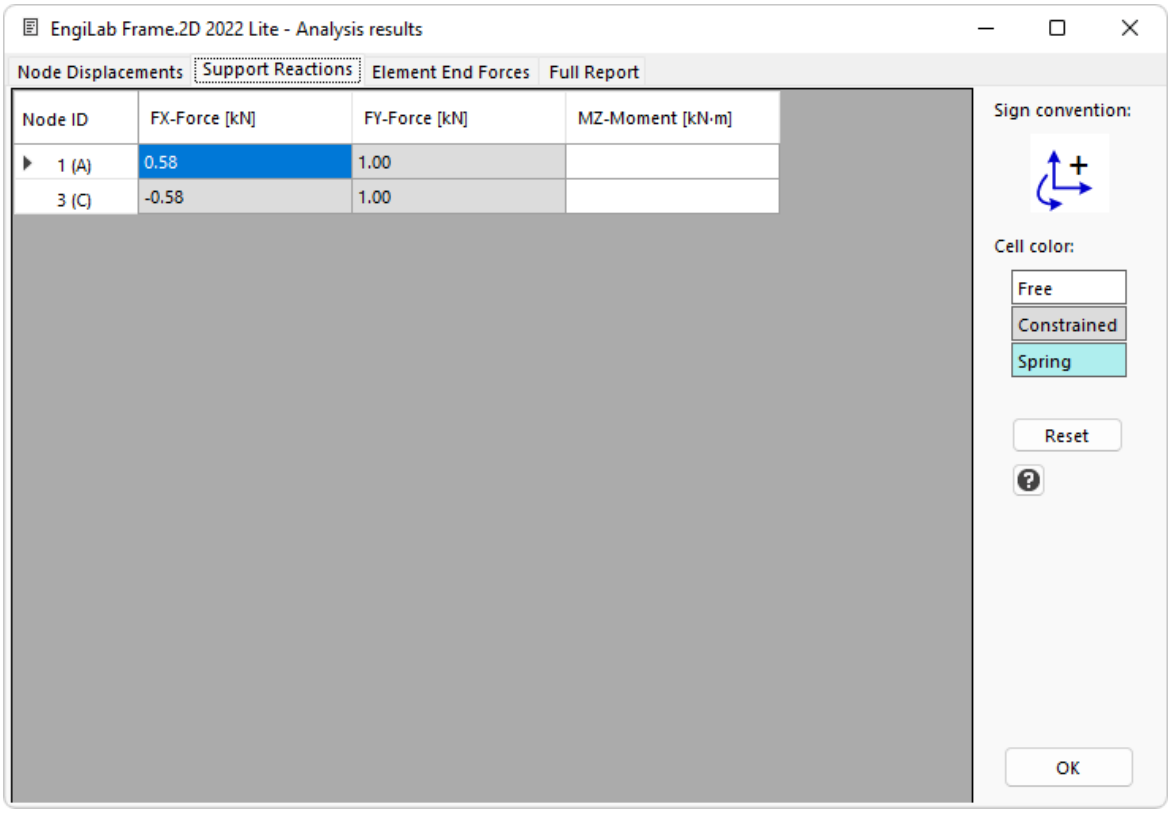



Click to see the analytical results.

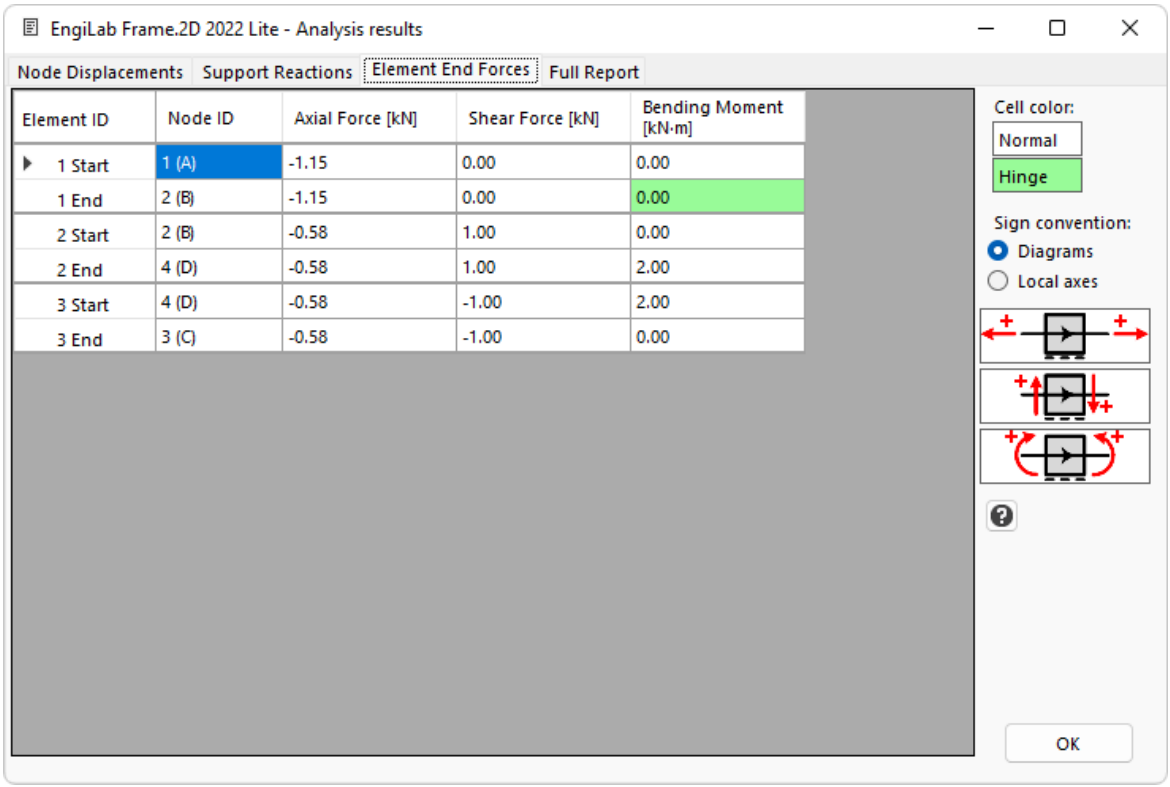
#### 1. Node Displacements



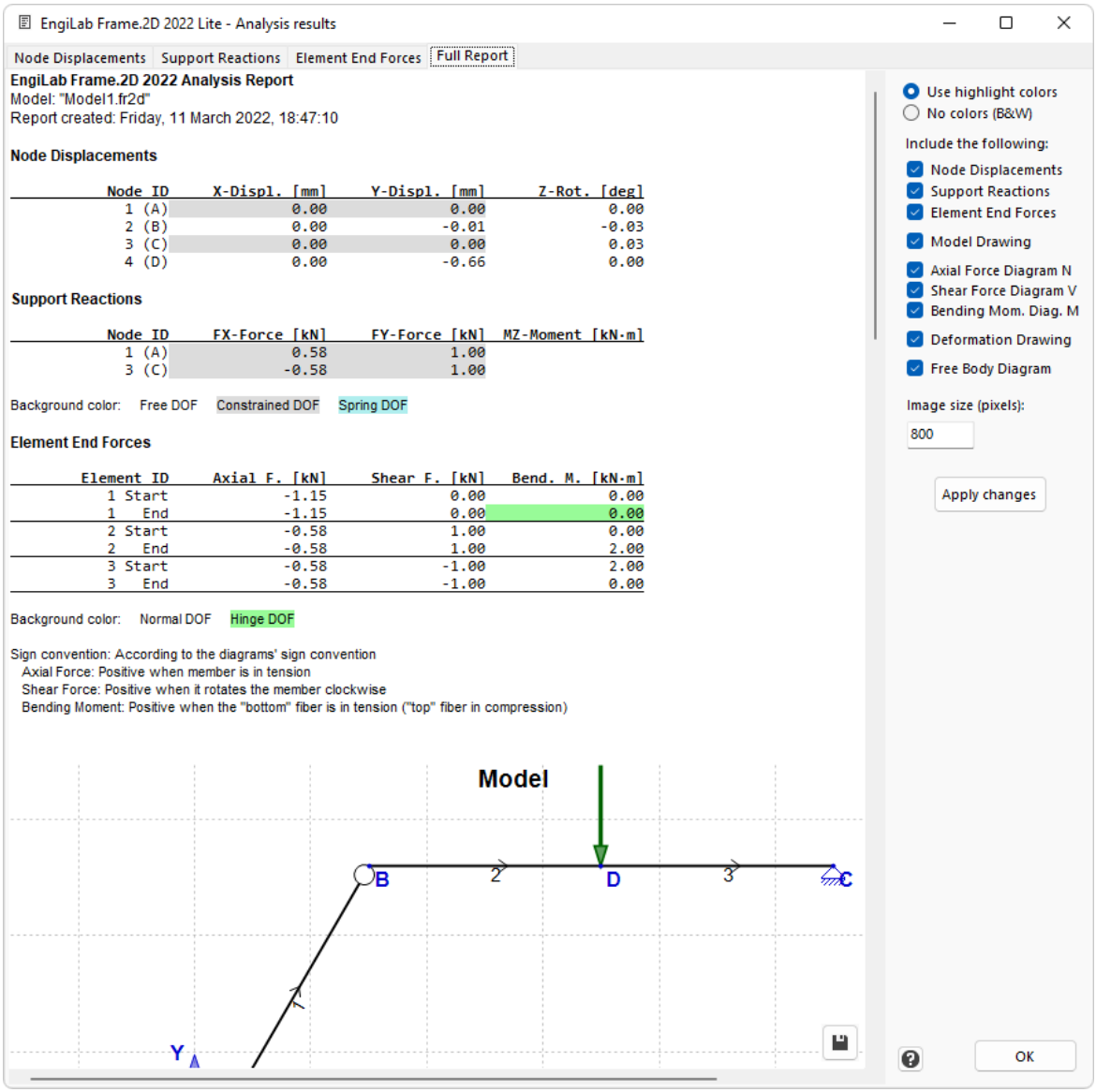
## 2. Support Reactions



## 3. Element Results



### 4. Full Report




The Full Analysis Report may include also the images of the Model, N, V, M Diagrams, Deformation and Free Body Diagram

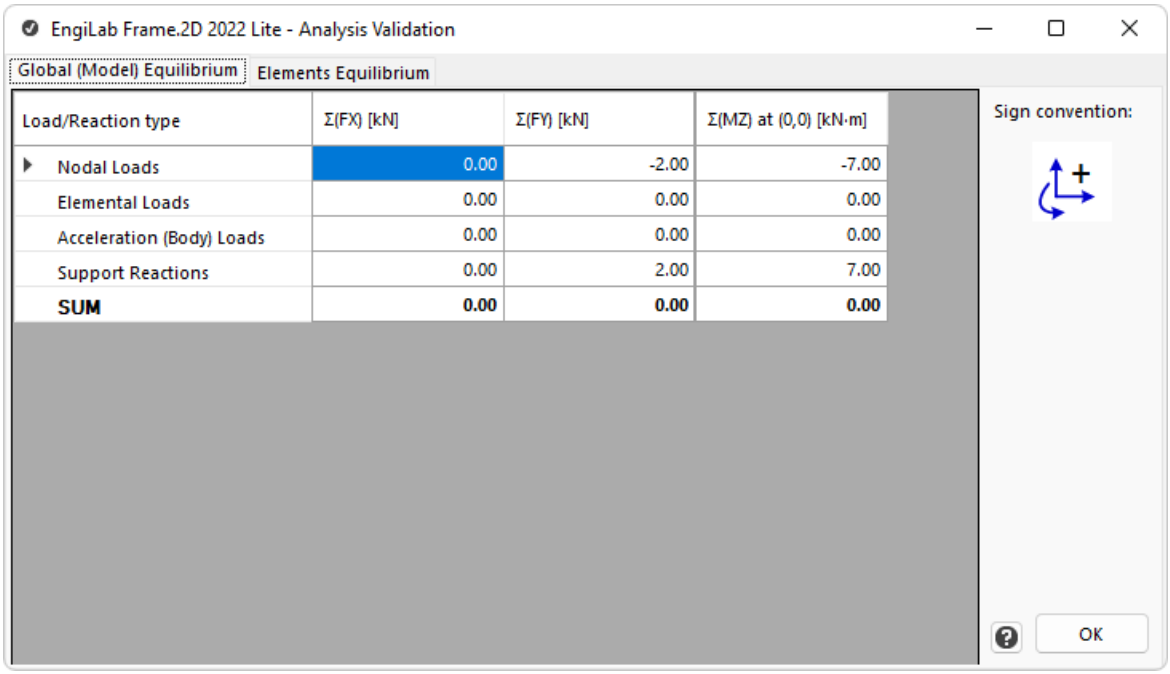
You can select the desired items from the Settings on the right of the Report and then click the "Apply changes" button.

## 7.2.14 Analysis Validation (optional)

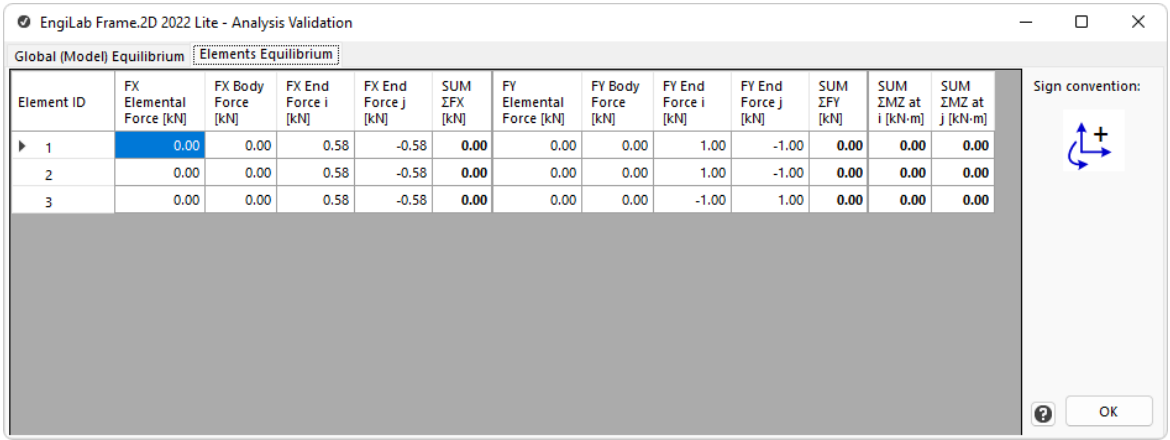


Optionally, you can click to see the Analysis Validation Reports and make sure that everything is OK with the Analysis and its results.

## 1. Global (Model) Equilibrium




## 2. Elements Equilibrium



# 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. ENGILAB WILL NOT, UNDER ANY CIRCUMSTANCES, BE RESPONSIBLE OR LIABLE FOR THE LOSS OF DATA ON ANY COMPUTER OR INFORMATION STORAGE DEVICE.

(c) 2022 EngiLab PC

## 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.

(c) 2022 EngiLab PC

## 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

(c) 2022 EngiLab PC

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.

(c) 2022 EngiLab PC




www.engilab.com