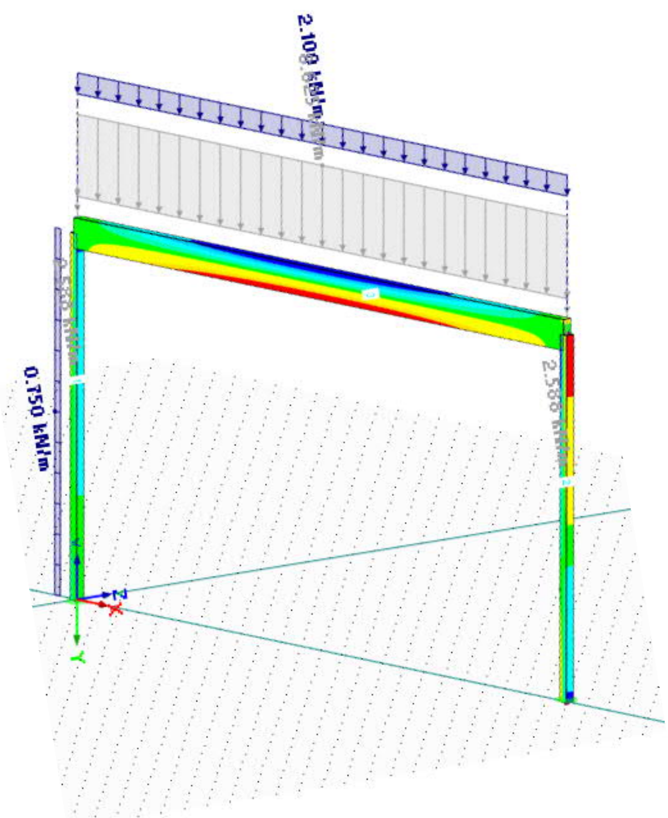
Aalto University
School of Engineering
Department of Civil and Structural Engineering
Risto Syrjä

20210212

RFEM

Directions for Computer Exercise



Content

1 Introduction

1.1 General

These directions will guide you through the basics of using the RFEM program with the following simple examples:

- Bar – Drawn rod as spring model (2D)
- Simple supported beam – Statically determined bent beam (3D)
- Frame – Statically indeterminate (column-)beam structure (3D)
- Truss – Use of bar elements in space structure (3D)
- Arc – Statically determined arc (3D)
- Beam on an elastic foundation – Application of the theory of a semi-infinite beam (3D)

It is a good idea for a beginner to start with the first two examples, which follow a step-by-step guide. An example of the bar is a quick introduction to the use of the program. The simply supported beam includes an introduction to the basic functions of the program and the use of the space coordinate system. The things advised in these examples apply in later examples.

Most of the these FE-models could be solved in 2D coordinate system. Due to the Finnish coordinate system practice and character rules, the 3D coordinate system has been used, which increases the workload somewhat, but makes it easier to interpret the results.

Data transfer (exports and imports) is discussed in the last chapter.

1.2 Program Version

Version RFEM 5.24 is used in these directions (Figure 1).

Do not use the trial version to do the assignment! A model made with it can only be opened on the same computer where the model was made. Use student license or school license.

Figure 1. RFEM.

1.3 Study Material

1.3.1 Home Pages

- Dlubal: https://www.dlubal.com/en
- Free student licence, which is valid for one year: https://www.dlubal.com/en/education/students/free-structural-analysis-software-for-students
- First steps with RFEM (Manuals, tutorials and so on): https://www.dlubal.com/en/products/rfem-fea-software/first-steps-with-rfem
- Importer: Rak Tek Solutions Oy: rakteksolutions.fi/

1.3.2 Lecture Notes

These directions includes references to the corresponding items of the lecture notes, which includes the theory needed for learning.

- Syrjä, Risto: Basics of Utilising Finite Element Method Program. Lecture Notes. Otaniemi 2021. 165 p.

1.3.3 Manuals and Tutorials

- Introductory Example for RFEM 5, September 2020, 54 p. (See Dlubal homepage.)
- Program RFEM 5, User Manual, 2020, 651 p. (See Program Menu → Help → Manual)
- Tutorial for RFEM 5, September 2020, 100 p. (See Dlubal homepage.)

1.4 User Interface

The user interface of the RFEM is show in Figure 2. There is a title bar at the top, a work area in the middle, and a status bar at the bottom. There are several options to input and edit data:

- Menu
- Tools
- Navigator with four sheets
  - Data
  - Display
  - Views
  - Results
- Panel
- Shortcut menu (mouse right click in the Work area)
- Tables (Excel format)
- Keyboard functions

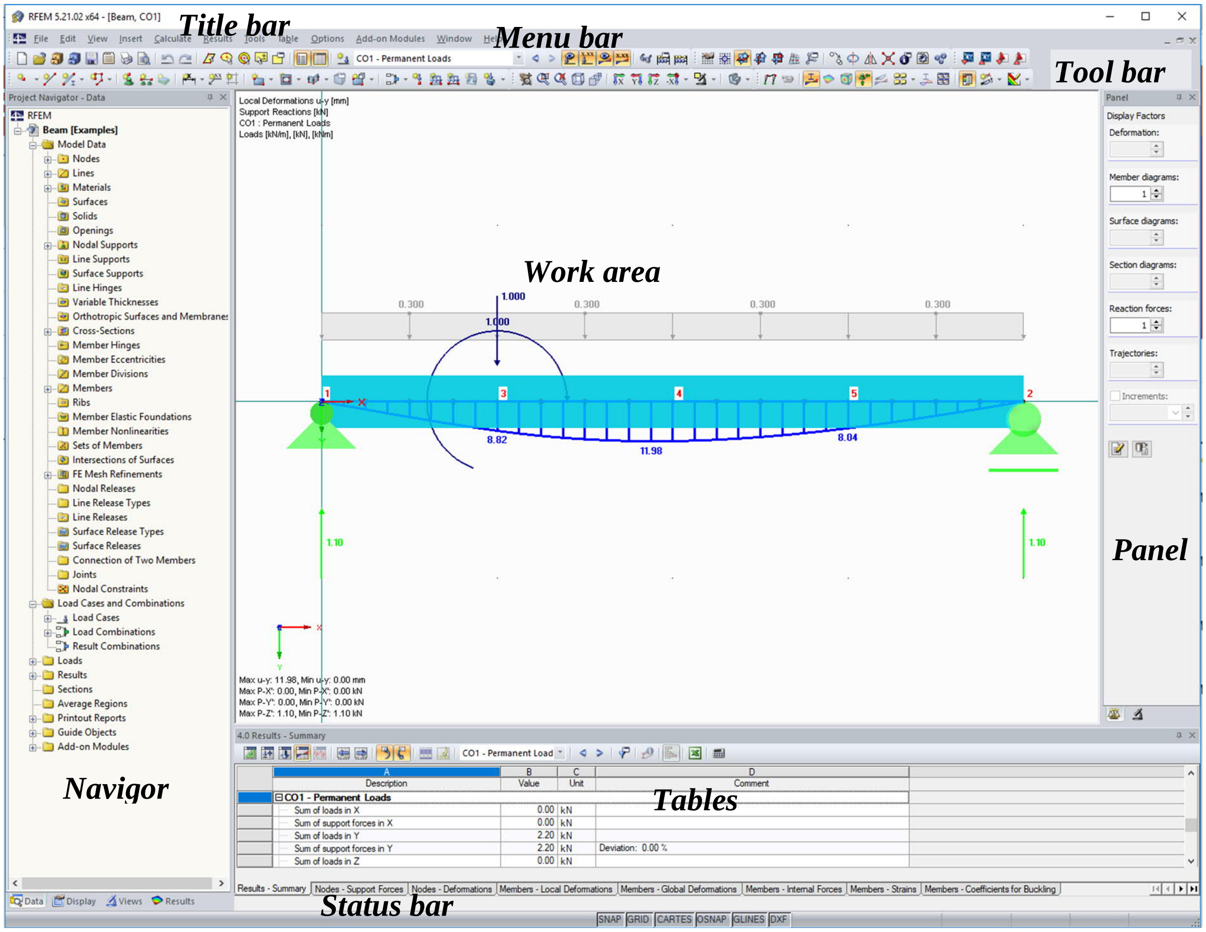


Figure 2. User interface.

Mouse functions are presented in Table 1.

Table 1. Mouse functions.

Left button once	Object selection
Left button + Ctrl	Drag-and-drop
Left button double click	Dialog box
Right button	Shortcut menu
Scrolling by the wheel button	Maximize and minimise the model view in the work area
Pressed wheel button	Move the mode view
Pressed wheel button + Ctrl	Rotate the mode view

The object colours of the work area can be changed from the shortcut menu: Display Properties or Colours in Graphics According to.

The background colour can be changed as follows: Options → Configuration Manager (Figure 3). Double click the left hand side button on the desired option. In this management mode, you can create your own settings (Create New…), save them to a file (Export) and apply them (Import) if necessary. You should use the configuration file, for example, when using the template on different computers. Therefore, the settings are not included in the template file.

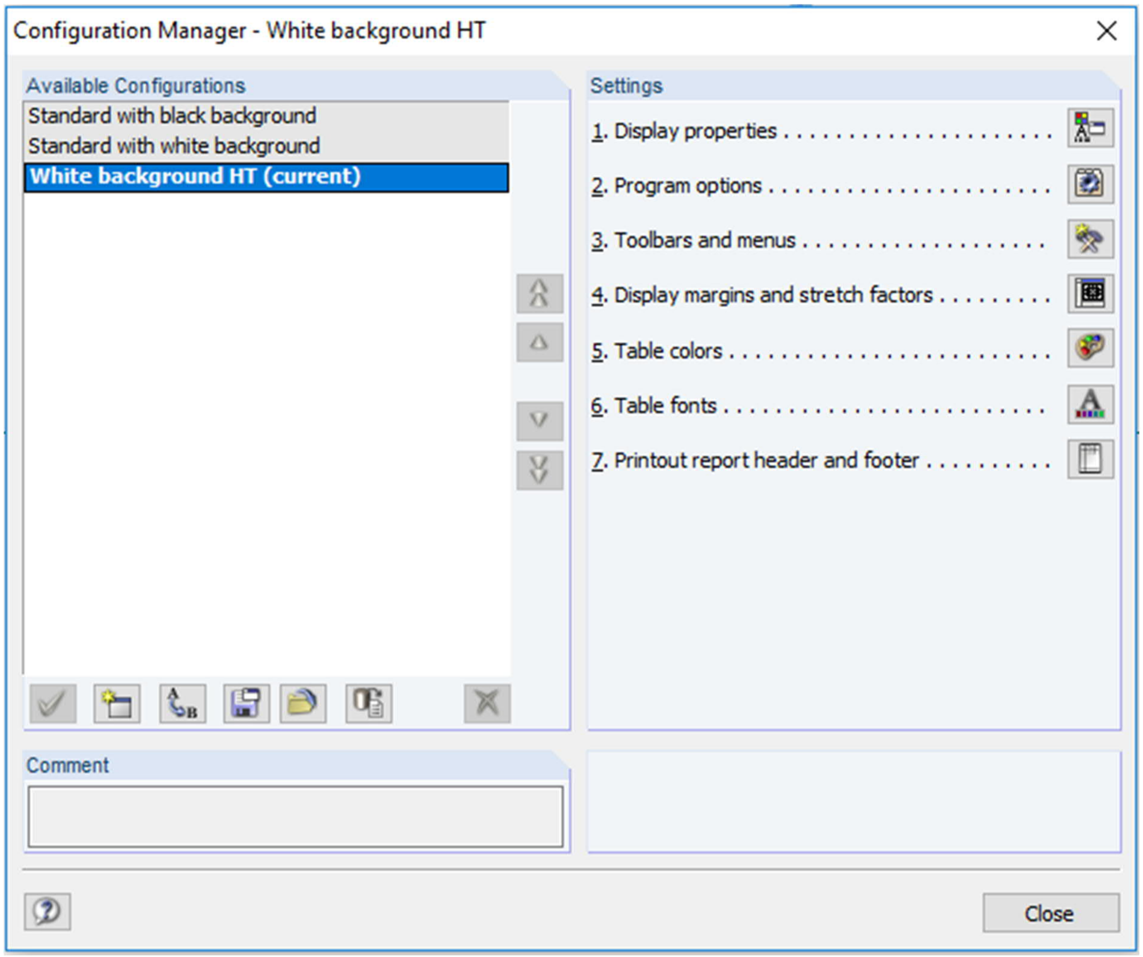


Figure 3. Configuration manager.

2 Bar

2.1 Problem

Figure 4 shows a bar with the left end fixedly supported. The bar is loaded by a point force (*F*).

The task is to determine the spring constant (*k*) of the spring describing the bar by means of the displacement of the free end of the bar.

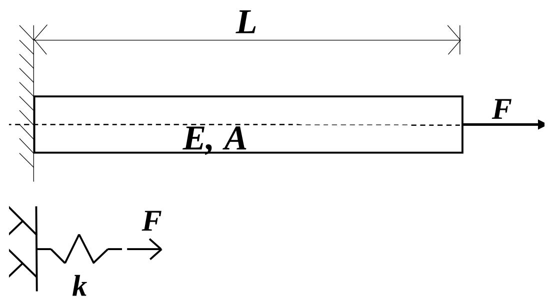


Figure 4. Bar.

The initial values are (*The blue italic part is made by Mathcad program*):

Length

$L := 3m$

Point load

$F := 35kN$

2.2 Preliminary Planning

Figure 5 shows the elevation and cross-section of the bar, sthe dimensions of the structure, the specific loads acting on it, the support conditions and the coordinate system.

Material is isotropic and elastic (linear).

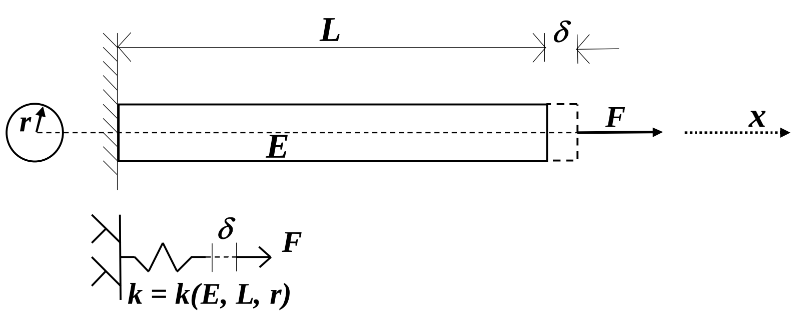


Figure 5. Preliminary planing for the bar.

A circular cross section and the following material properties and dimensions have been selected for the beam (*Mathcad*):

Radius

$$r := 0.0125m$$

Modulus of elasticity

$$E := 210000 \frac{MN}{m^2}$$

Cross-section area

$$A := \pi\, r^2 \qquad\qquad A = 4.909\,cm^2$$

2.3 Starting the Program

Start RFEM program (Figure 6).

Figure 6. RFEM icon.

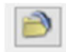
2.4 Starting the Modelling

Start to create a new model: File → New (Ctrl+N). Give a model name: Bar (Figure 7).

Note in which folder the model is saved! See Chapter 2.11 Safety Copying (p. 18). If you want to change the folder, press icon  (Create New Project).

Select the proper coordinate system as shown in Figure 7 (Type of Model; Positive Orientation of Global Z-Axis).

Set the units by pressing  (Units and Decimal Places).

If SI-units are not selected (Figure 8), change the units by pressing icon  (Load Saved Profile) and selecting the metric option (Figure 9).

Update the units, if needed

- modulus of elasticity, MN/m^2 (Figure 8),
- cross-sectional dimension, m (Figure 8),
- load, kN (Figure 10) and
- deflection, mm, with two decimals (Figure 11).

Press OK in the open windows.

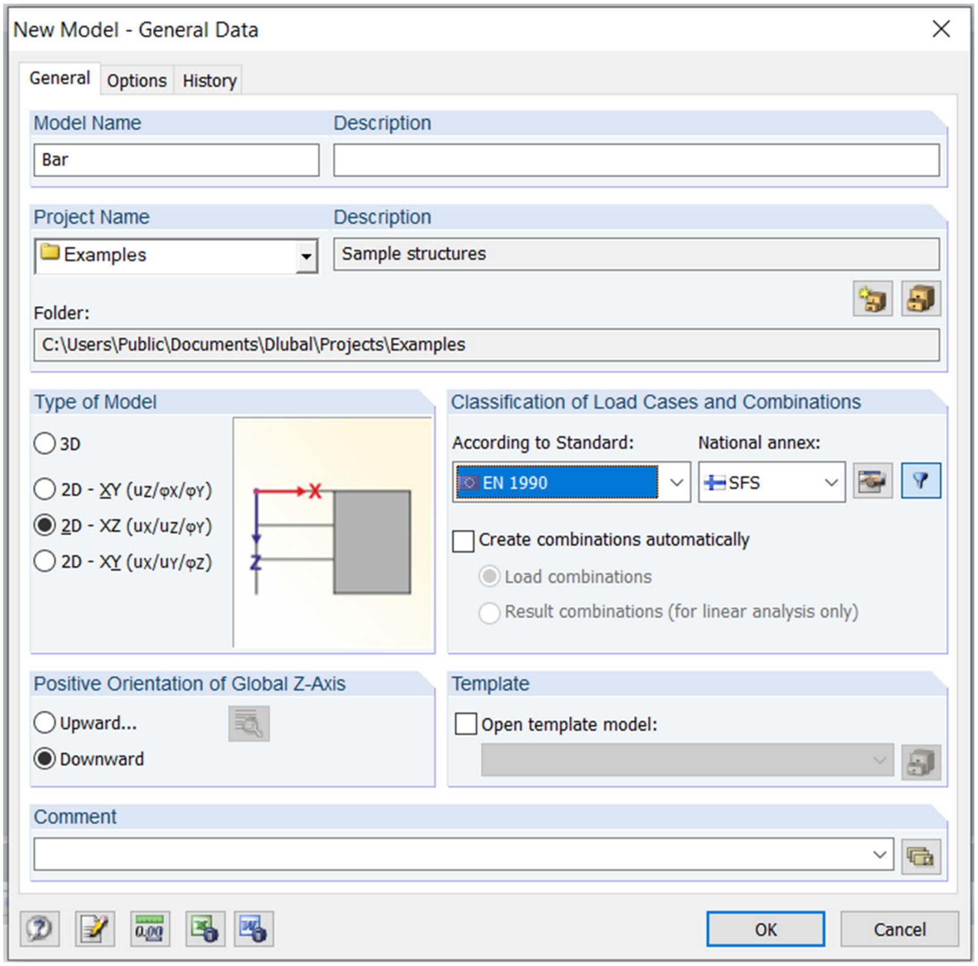


Figure 7. New model.

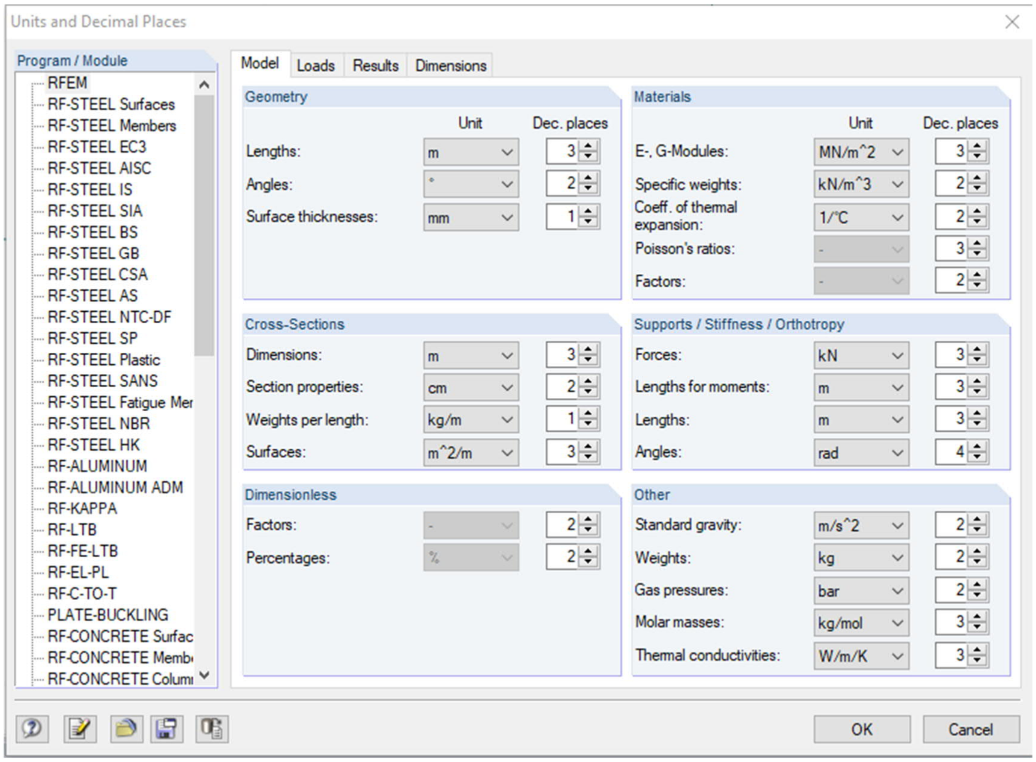


Figure 8. Units and decimal places of the model.

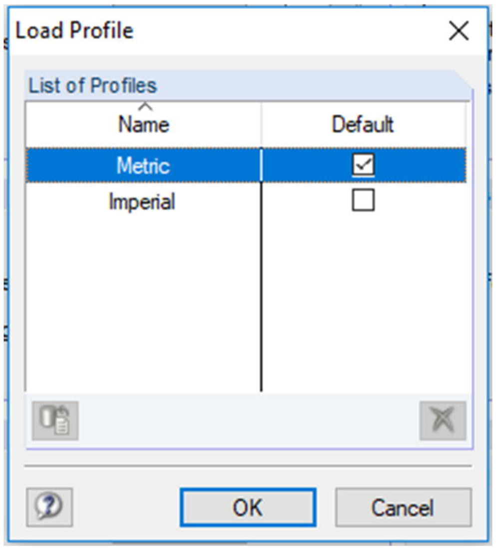


Figure 9. Unit system.

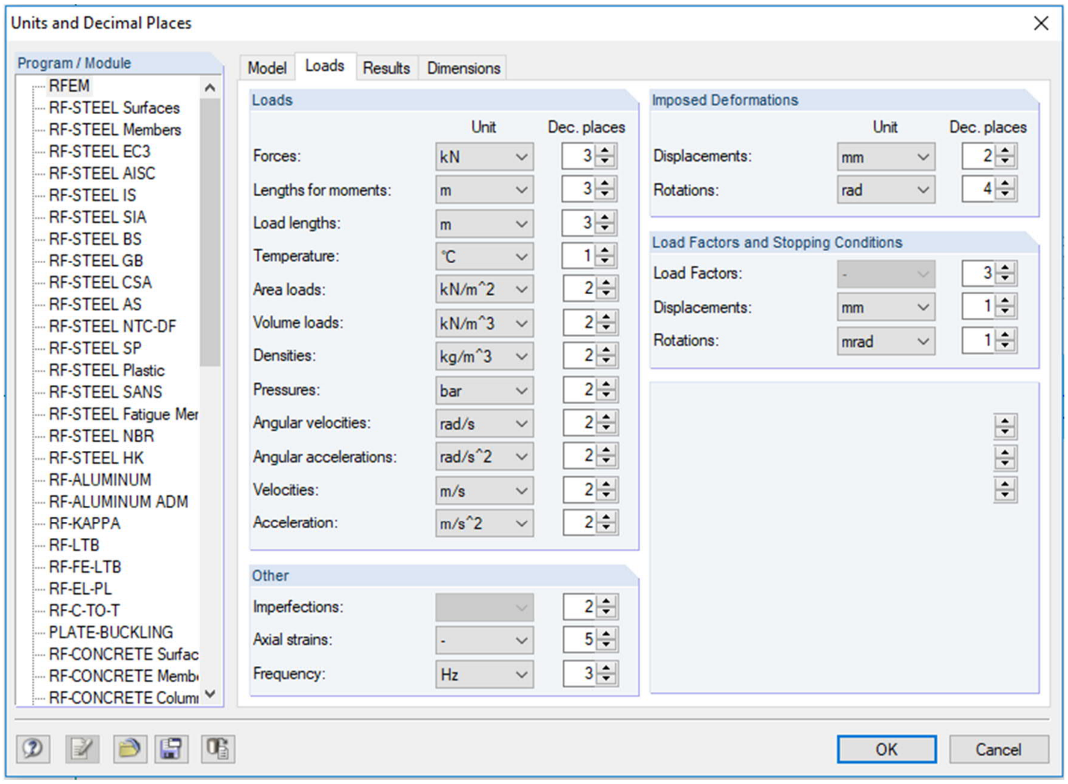


Figure 10. Units and decimal places of the loads.

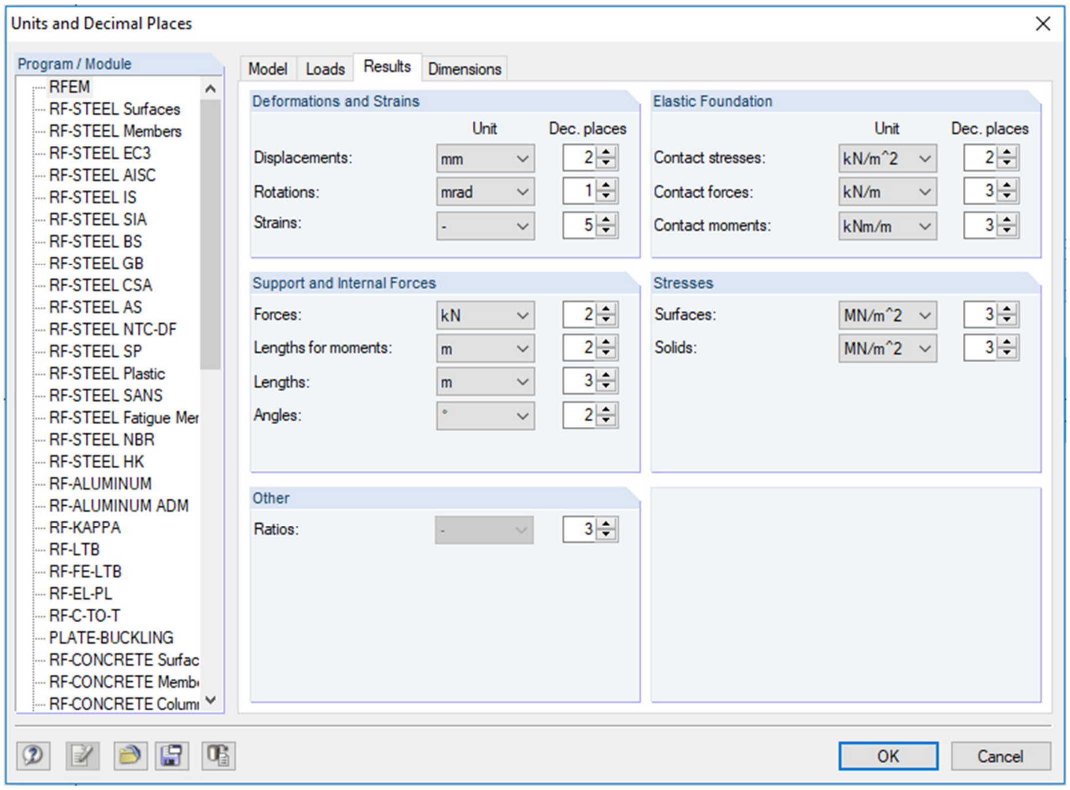


Figure 11. Units and decimal places of the results.

2.5 Structural Member

Create a structural member (bar) by selecting icon  from toolbar or the menu: Insert → Model Data → Members → Graphically → Single.

Select "Beam" as the component type. (Truss type automatically adds joints to the ends of the bar, which would make the structure unstable in this case, so we use a beam.)

Press the icon  (Create New Cross-Section for Member…) to create the cross section of the member. Select the icon  (Import Timber Cross-Sections - Circle). Select as shown in Figure 12. (A steel bar (Round Bar ) would also be available, but due to the surface pattern of the ribbed bar, it is more difficult to calculate its cross-sectional area by hand.)

Change the material to steel S 235. Press the icon  (Edit Material) and check that the value of the modulus of elasticity is correct (Fig. 13). Plress OK in the Material window and in the two Cross-section window.

Check the information in the Member window and press OK (Figure 14).

To set the *L*-length bar to the right of the origin, follow these steps: With the Line window open, draw a line to the right of the origin using the left mouse button (Figure 15). Stop drawing with Esc.

Save: File → Save (Ctrl+s). If you end your session, see Chapter 2.11 Safety Copying (p. 18).

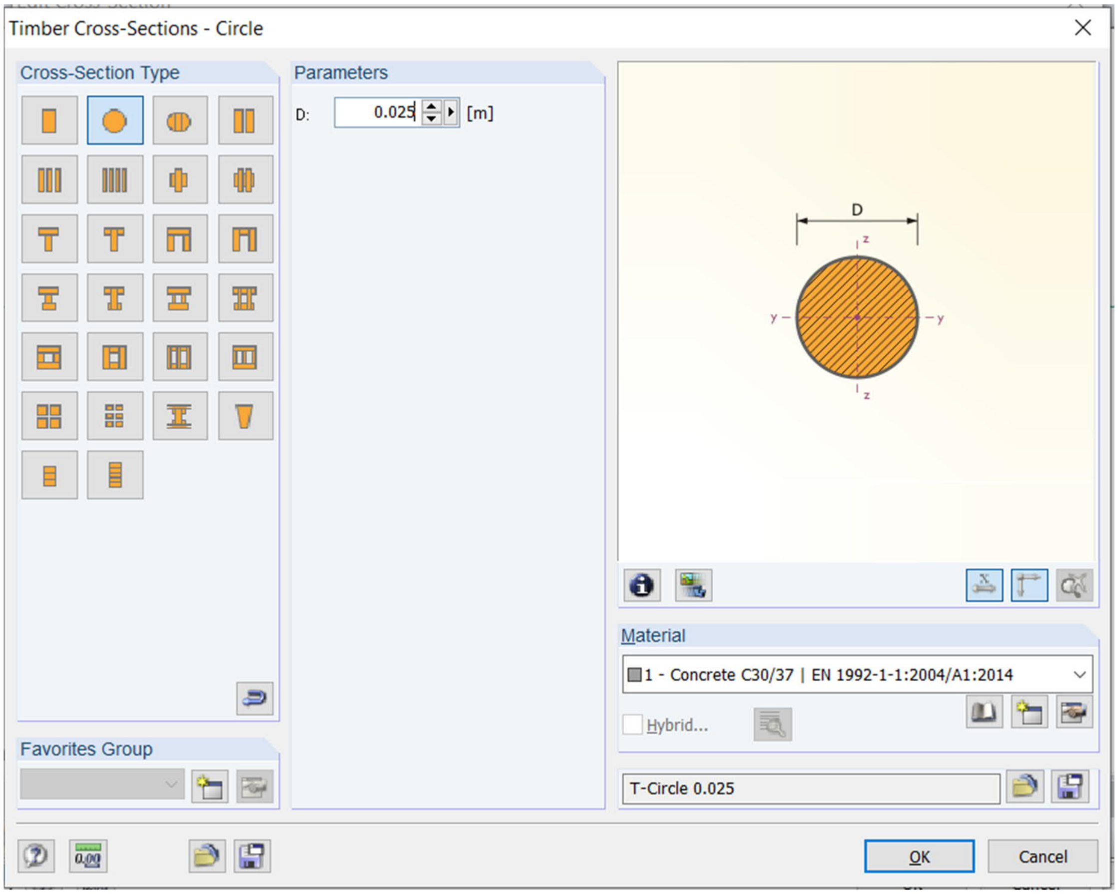


Figure 12. Cross-section of the bar.

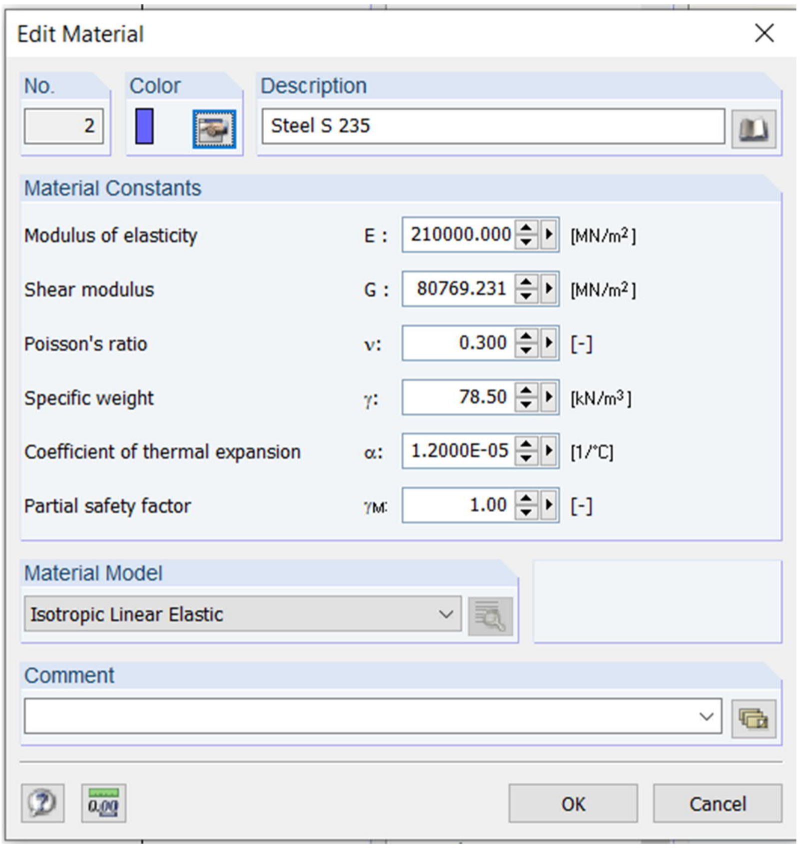


Figure 13. Steel S 235.

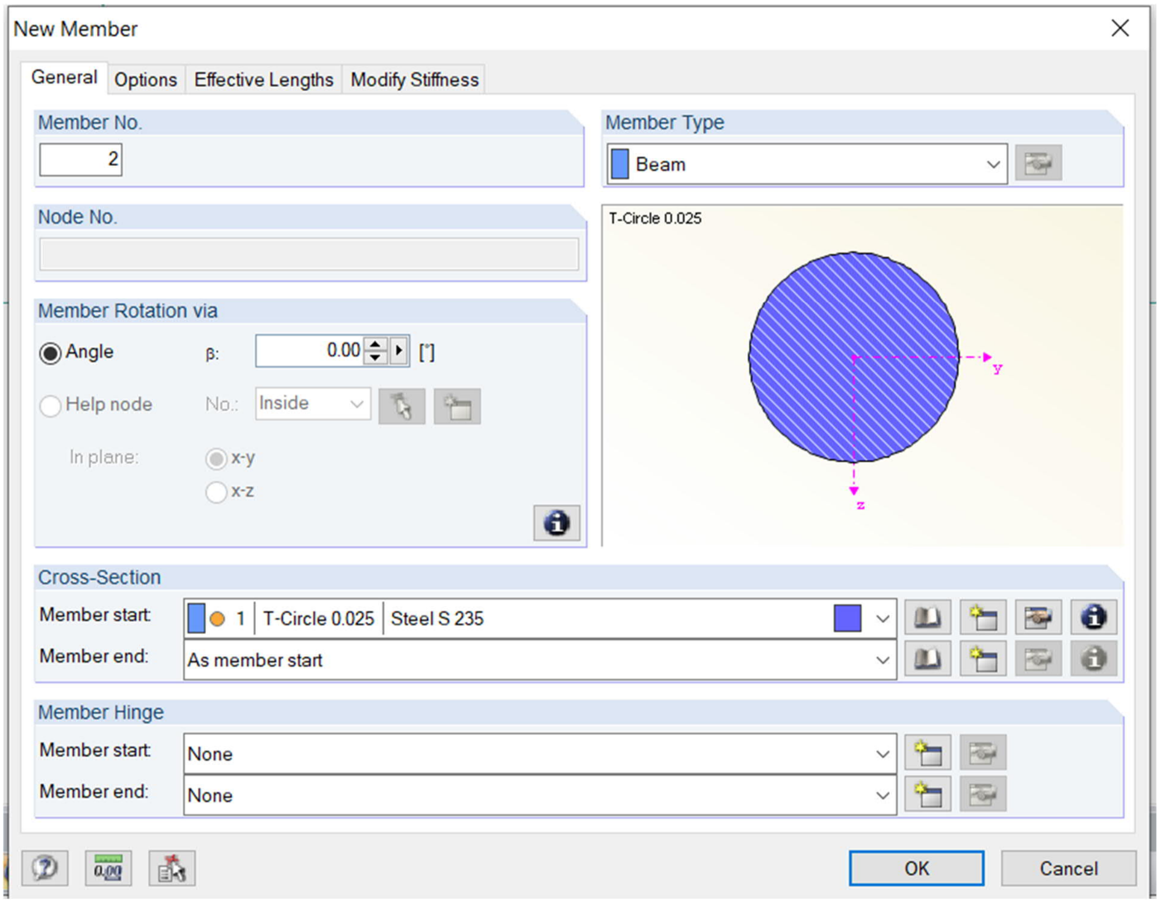


Figure 14. Defining a member.

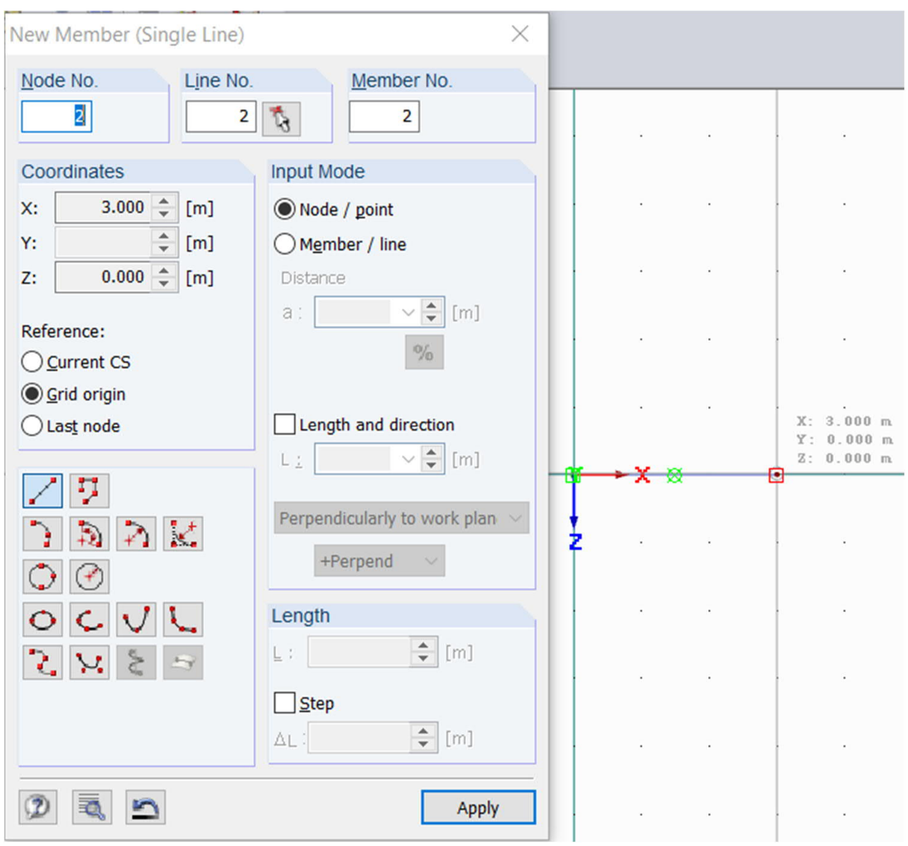


Figure 15. Defining the length of the bar.

2.6 Support

To add fixed support, press the toolbar icon  or select: Insert → Model data → Nodal Supports → Graphically. This will open a window (New Nodal Support). Select Rigid support as shown in Figure 16. Press OK. Use the mouse to select the origin (Figure 17). Press ESC.

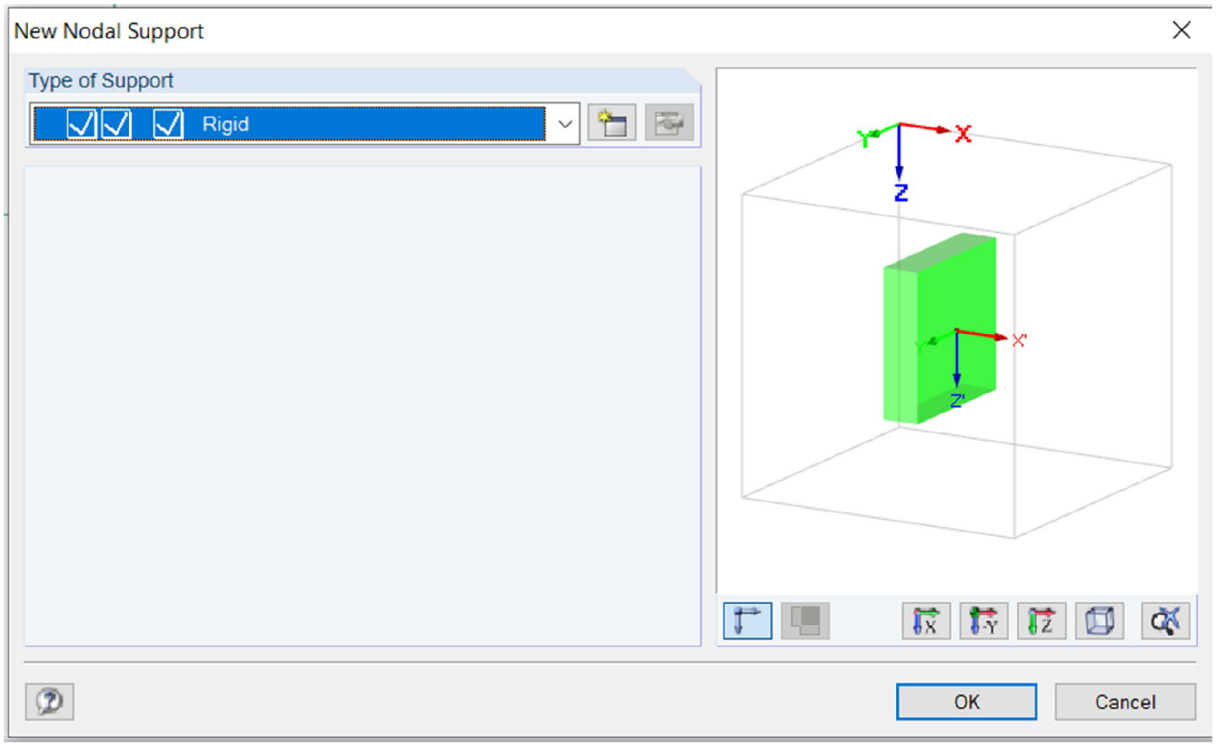


Figure 16. Fixed support.

Figure 17. Supported bar.

Save: File → Save (Ctrl+s). If you end your session, see Chapter 2.11 Safety Copying (p. 18).

2.7 Load

Add Load: Insert → Nodal Load → Graphically. This will open the load case window. Update as shown in Figure 18 and press OK.

In the window that opens, update as shown in Figure 19 and press OK.

Use the mouse to select the right end of the rod (Figure 20). Press ESC.

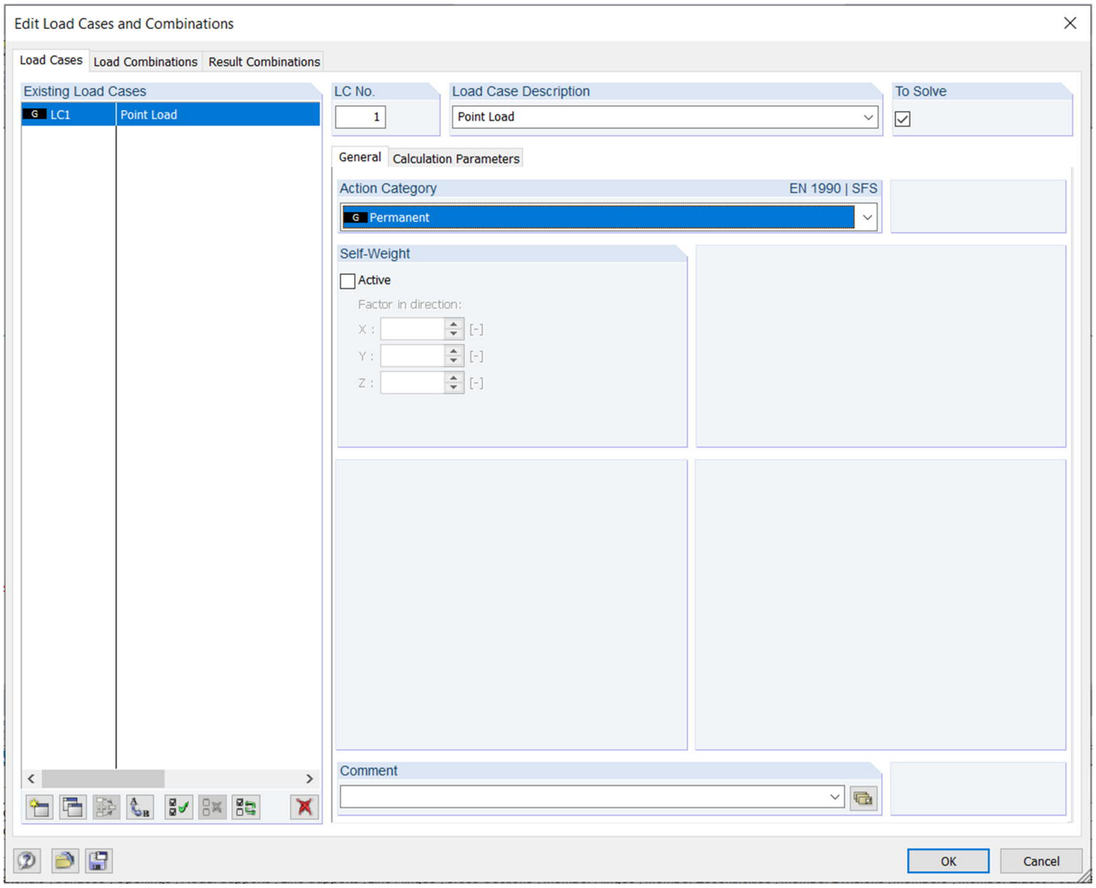


Figure 18. Load case.

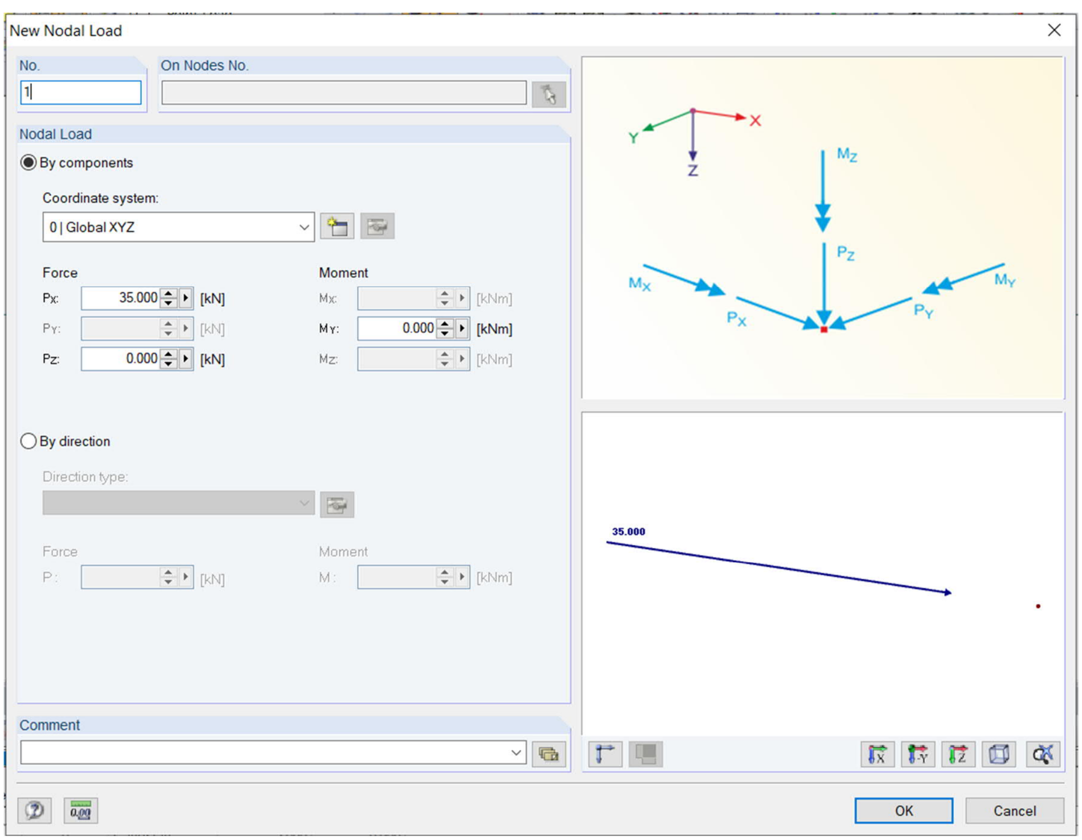


Figure 19. Nodal load.

Figure 20. Loaded bar.

Save: File → Save (Ctrl+s). If you end your session, see Chapter 2.11 Safety Copying (p. 18).

2.8 Analysis

Analyse: Calculate → Calculate All (Figure 21). It takes a short time…

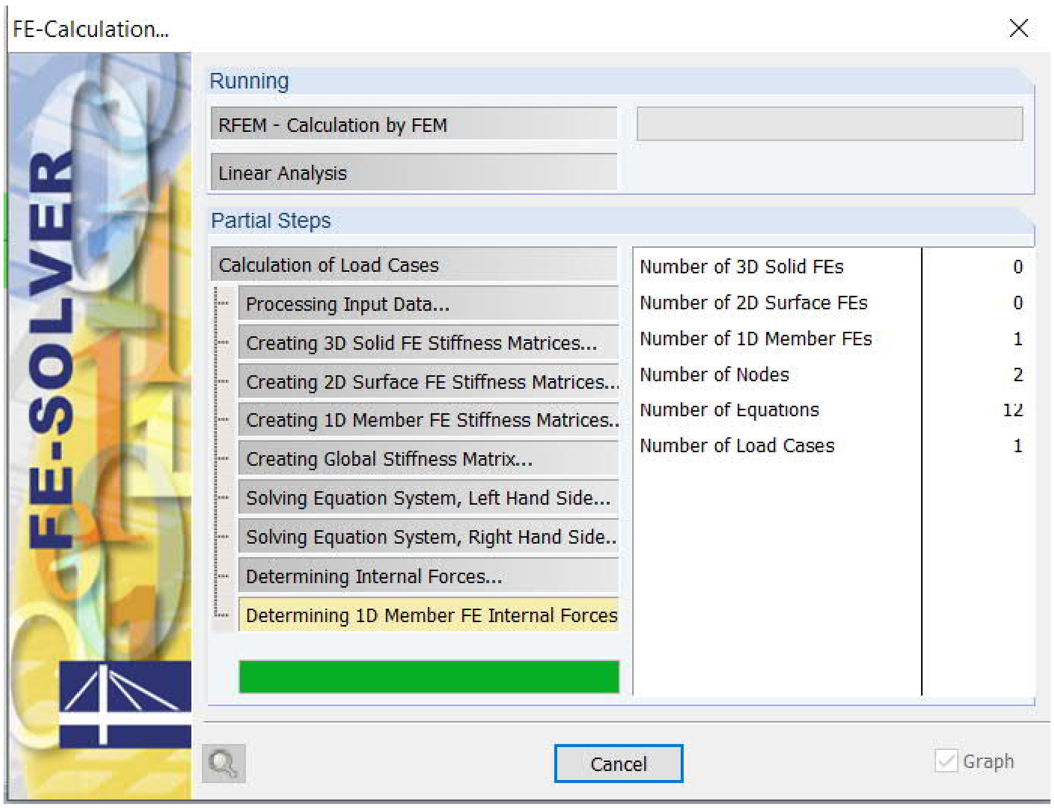


Figure 21. FE-Calculation.

Save: File → Save (Ctrl+s). If you end your session, see Chapter 2.11 Safety Copying (p. 18).

2.9 Results

Select the displacement as the graph: Navigator → Results → Members → Local Deformations → u_x. The graph (Figure 22) shows the displacement of the right end of the bar in the *x*-direction. (Graph settings are discussed in the following examples.)

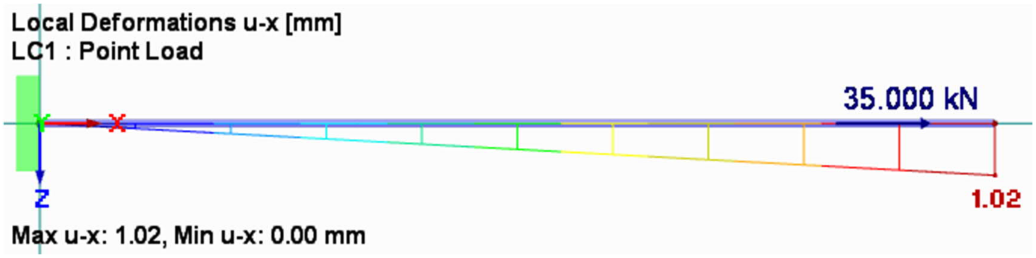


Figure 22. Dispalcement.

The spring equation is

$$F = k\delta \tag{1}$$

where δ is displacement at the free end of the spring, which is the same as the displacement obtained by FEM at the right end of the bar. The spring constant is obtained (*Mathcad*):

Displacement

$$\delta := 0.00102m$$

Spring constant

$$k := \frac{F}{\delta} \qquad k = 34.314\,\frac{MN}{m}$$

2.10 Validity

The spring constant can be calculated directly from the dimensions and modulus of elasticity of the bar (*Mathcad*):

Spring constant

$$k(A,E,L) := \frac{E\,A}{L} \qquad k(A,E,L) = 34.361\,\frac{MN}{m}$$

The result is almost the same as the result obtained with FEM through the displacement. The small difference is due to the fact that the displacement value is rounded.

2.11 Safety Copying

When saving the model (Ctrl+s), the program creates a file Beam.rf5, for example in the folder:

C:\Users\Public\Documents\Dlubal\Projects\Examples\

The file is on the local disc of the computer. After the each session, take a safety copy of the file to some other place too!

- MyCourses → Dashboard → Manage private files
- Email
- USB
- WIN home folder
- and so on...

The model can be copy, for example, to the z-disc and open again by a double click.

Start always a new session by making a safety copy, so that the version history is on the project folder, for example:

- Bar.rf5 (model under editing)

- Bar_v5.rf5
- Bar_v4.rf5
- Bar_v3.rf5
- Bar_v2.rf5
- Bar_v1.rf5 (the oldest safety copy)

3 Simple Supported Beam

3.1 Problem

A simple supported beam, with pin support at the left end and roll at the right one, is shown in Figure 23. The beam is loaded by the self-weight (γ), point load (F) and point moment (M).

The task is to determine the deflection (v), rotation (φ), bending moment (M_y), shear force (V)[1] and axial stress (σ_x) of the beam.

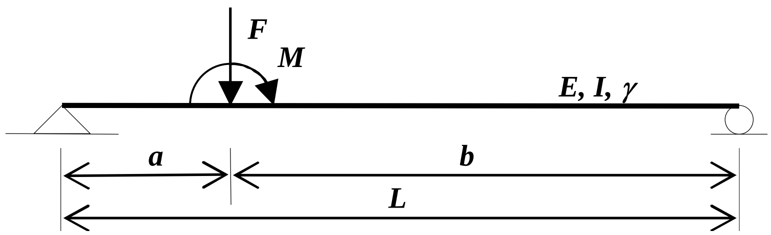


Figure 23. Simple supported beam.

The initial values are (*The blue italic part is made by Mathcad program*):

Lengths

$a := 1m$

$b := 3m$

$L := a + b$ $\qquad\qquad$ $L = 4m$

Unit weight

$$\gamma = 10 \cdot \frac{kN}{m^3}$$

Loads (characteristic value)

$F := 1kN$

$M := 1kN \cdot m$

3.2 Preliminary Planning

See the corresponding chapter in the Lecture Notes: ***Introduction, Types of Structures*** *and* ***Preliminary Planning****.*

The elevation and cross-section with the structural dimensions, loads, supports and the coordinate system are shown in Figure 24. The coordinate system is in accordance with Finnish practice, right hand coordinate system.

Material is isotropic and elastic (linear). Each load is considered as its own load case. The magnitude of all the safety factories are one.

The problem can be solved in the plane case, in the x-y coordinate system. In RFEM, however, this 2D coordinate system in accordance with Finnish practice is not possible, so the 3D coordinate system is used.

[1] The symbol Q is usually used for the shear force. The same notation is selected here as in the RFEM program.

The beam is divided to the sub beams, so that there is a node at the position of the point load. Each sub beams has four beam elements.

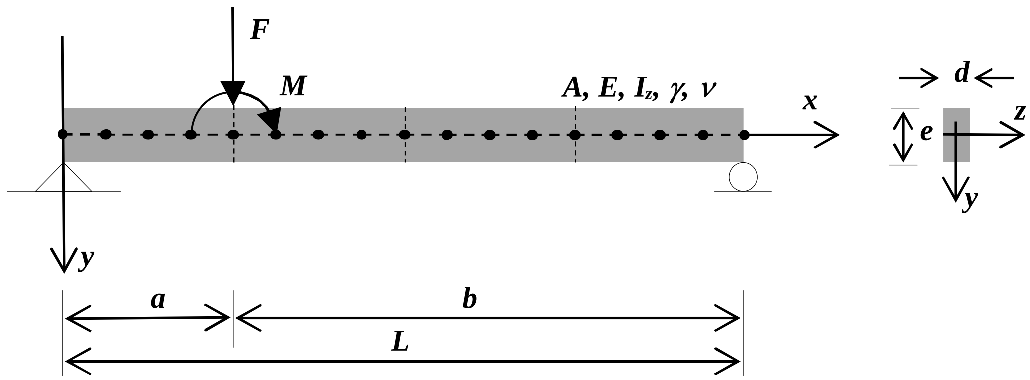


Figure 24. Preliminary planning.

For the beam, the rectangular cross-section shape is chosen. The material and cross-sectional parameters are as follows (*Mathcad*):

Dimensions

$d := 0.1m$

$e := 0.3m$

Cross-section area and moment of inertia

$A := d \cdot e$ $\qquad A = 0.03 \cdot m^2$

$I_z := \frac{d \cdot e^3}{12}$ $\qquad I_z = 2.25 \times 10^{-4} \cdot m^4$

Modulus of elasticity, Poisson's ratio and shear modulus

$E := 1000 \frac{MN}{m^2}$

$\nu := 0$

$G := \frac{E}{2(1+\nu)}$ $\qquad G = 500 \cdot \frac{MN}{m^2}$

3.3 Starting the Modelling

Start to create a new model: File → New (Ctrl+N). Give a model name: Beam (Figure 25).

Select the proper coordinate system as shown in Figure 25 (Type of Model; Positive Orientation of Global Z-Axis). See Chapter 3.5 Coordinate System, p. 23.

Select the proper standards. It doesn’t matter in this example.

Press OK.

These general data can be edit later: Edit → Model Data → General Data.

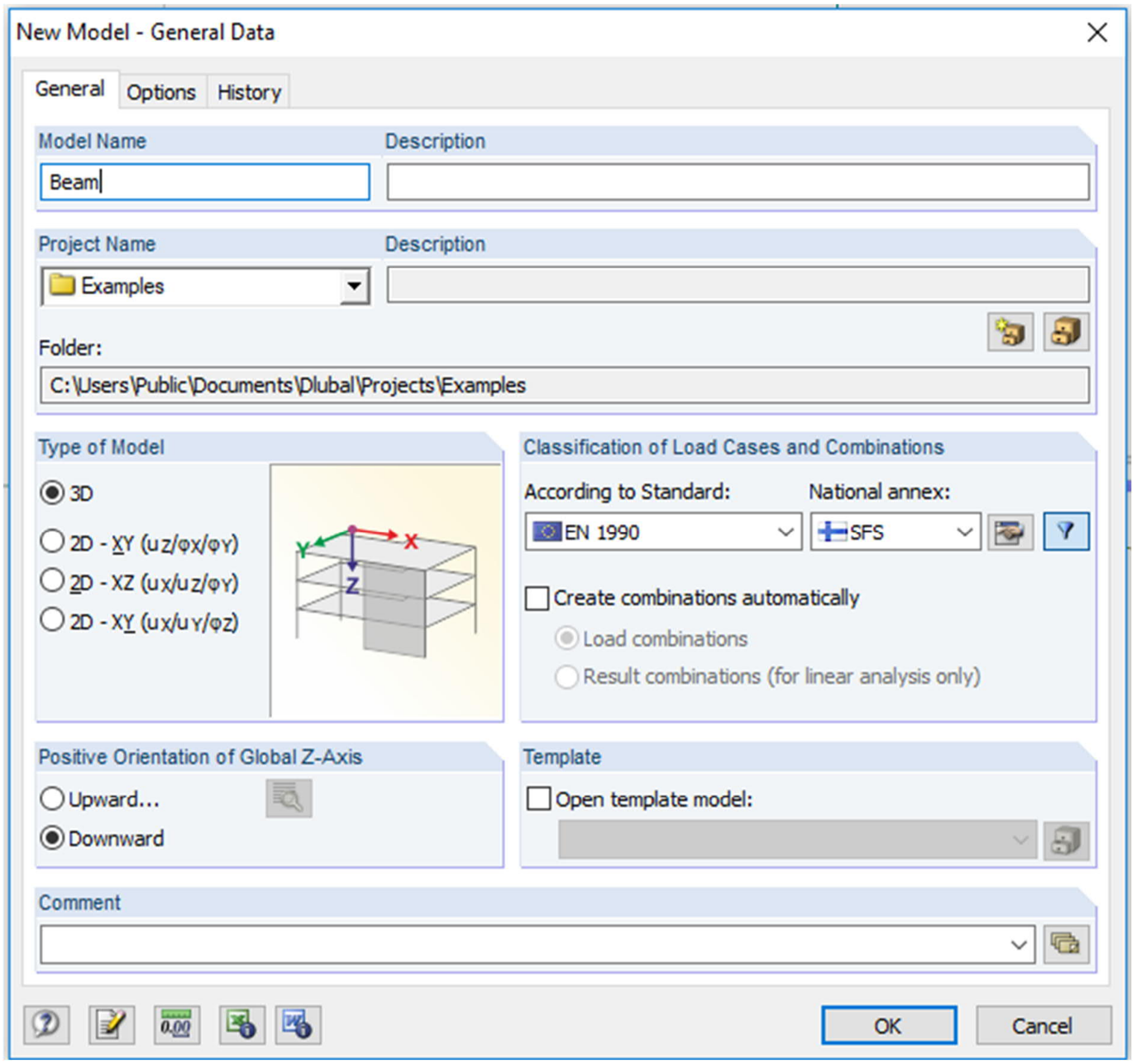


Figure 25. New model.

3.4 Units

See the corresponding chapter in the Lecture Notes: ***Units****.*

Set the units: Edit → Units and Decimal Places.

Update the units, if needed
- modulus of elasticity, MN/m^2 (Figure 8),
- cross-sectional dimension, m (Figure 8),
- load, kN (Figure 10),
- deflection, mm, with two decimals (Figure 11),
- support force, kN (Figure 11) and
- stress, MN/m^2 (Figure 11).

If you frequently use the same units, you may want to save them from icon  (Save as Profile). See Figure 26. The profile can be activated by pressing icon  (Load Saved Profile) icon. See Figure 27.

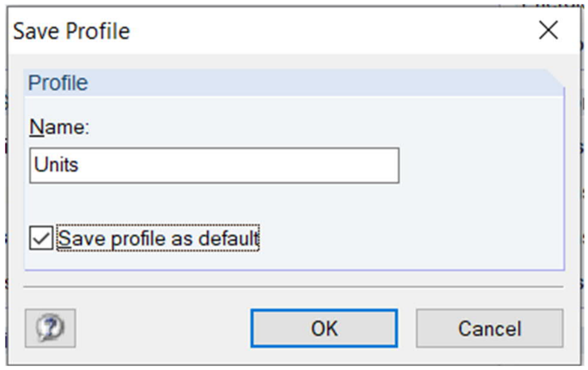


Figure 26. Save unit profile.

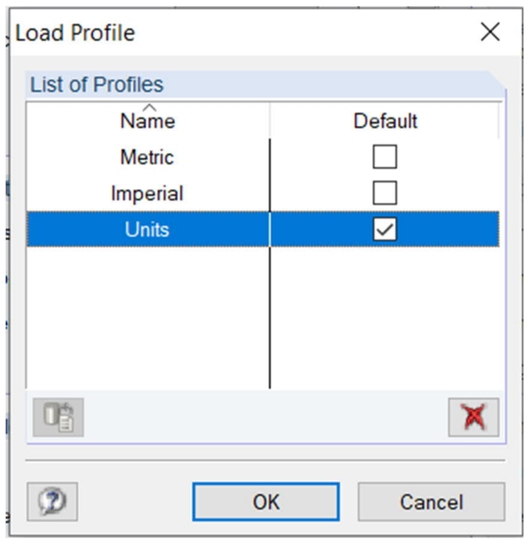


Figure 27. Load unit profile.

Save: File → Save (Ctrl+s). If you end your session, see Chapter 2.11 Safety Copying (p. 18).

3.5 Coordinate System

See the corresponding chapter in the Lecture Notes: ***Coordinate Systems****.*

Right-click in the workspace and select from the context menu , Coordinate System (Figure 28). Press icon  (Create New User Defined Coordinate System).

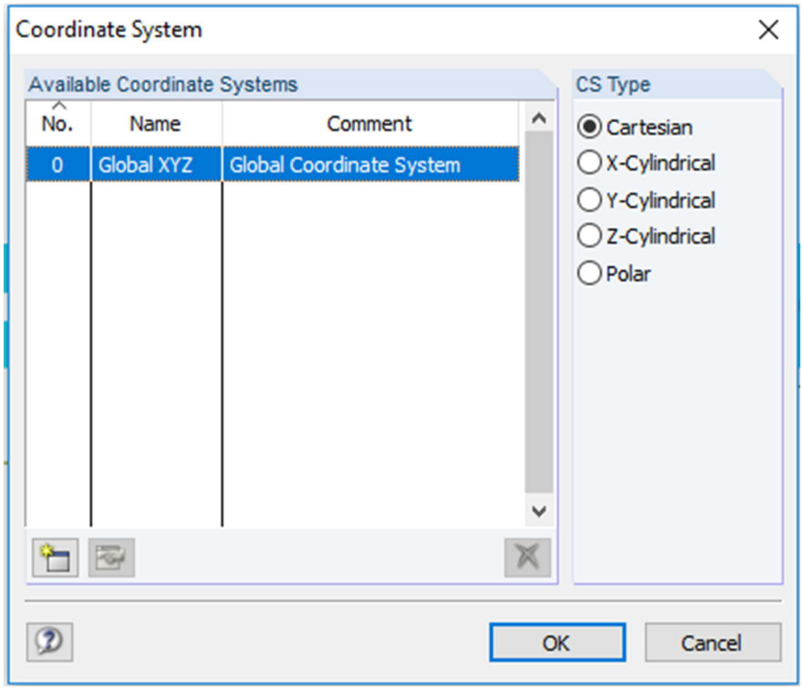


Figure 28. Coordinate system.

Create a local coordinate system relative to the global coordinate system. Update as shown in Figure 29. Press OK in the both windows.

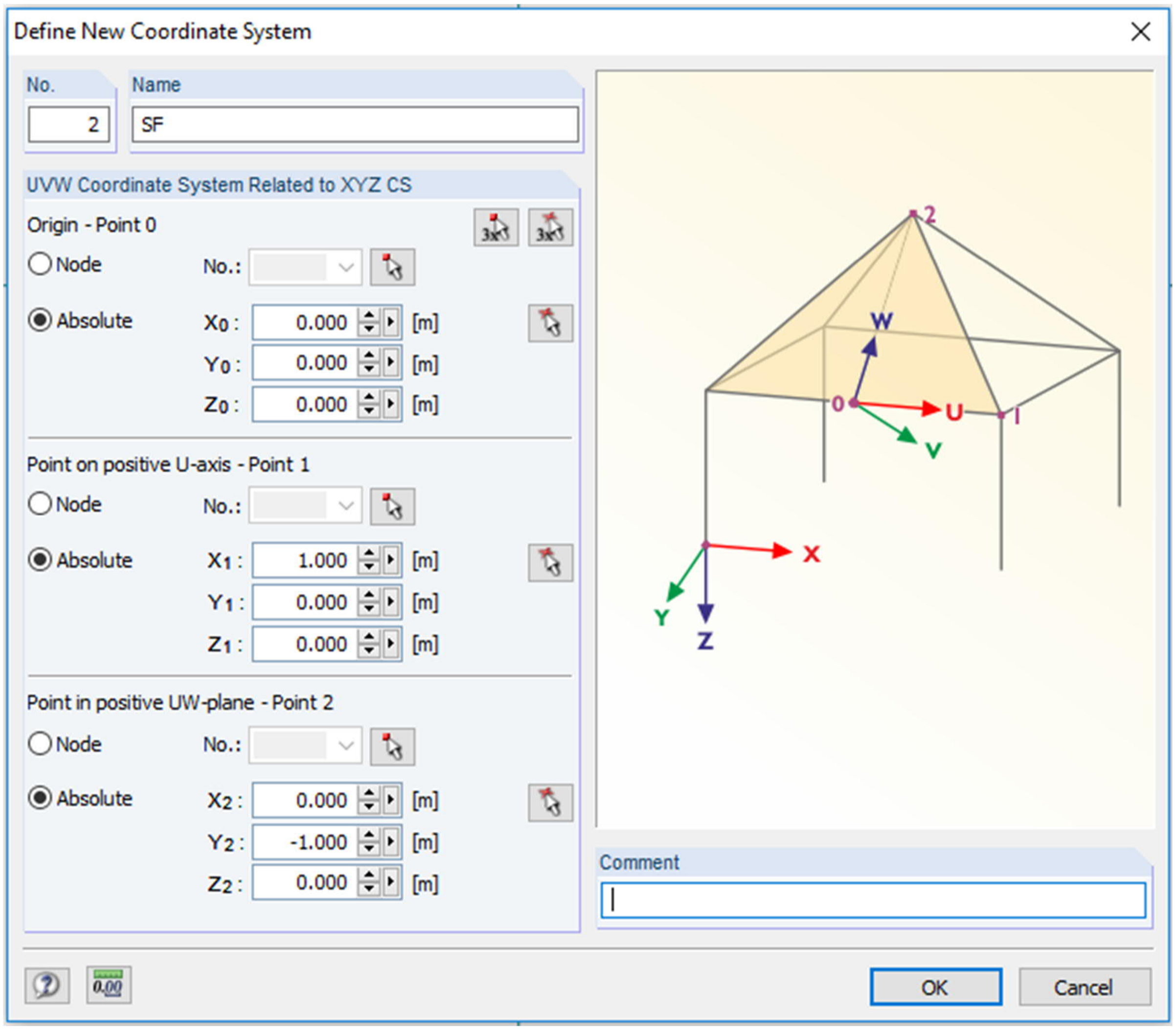


Figure 29. Define new coordinate system.

Right click in work area and select  (Work Plane, Grid/Snap). Give the fitting settings for work plane and grid/snap (Figure 30). Press OK.

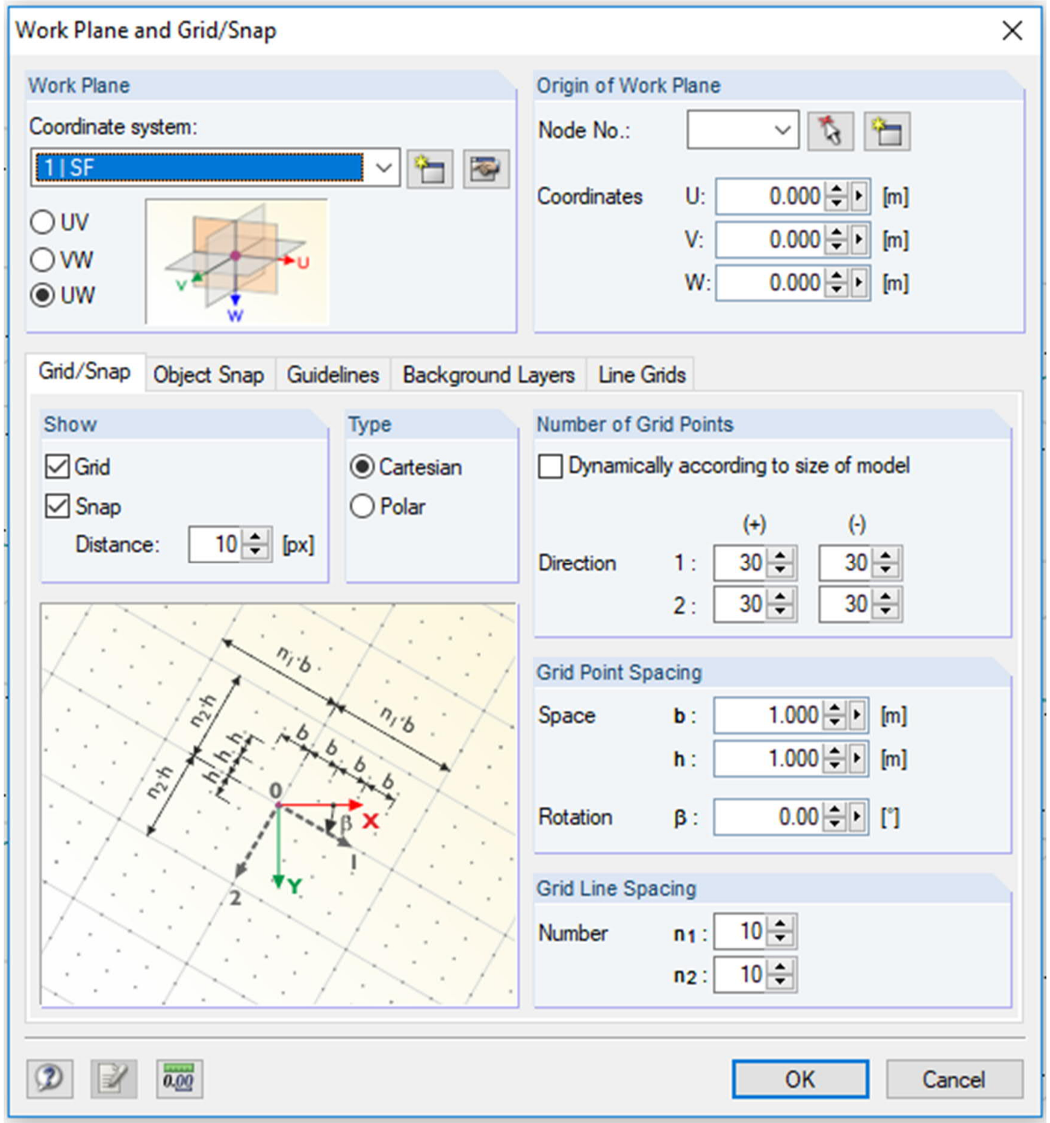
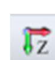


Figure 30. Work plane and grid/snap.

Rotate the view of the work area to the x-y plane: Select icon  or from menu: View -> Select View -> In Z-Direction.

Save: Ctrl+s.

3.6 Material

*See the corresponding chapter in the Lecture Notes: **Materials**.*

Open the material window: Insert → Model Data → Materials → Dialog Box.

Give material data for material ”E = 1000 MN/m^2” as shown in Figure 31 and press OK.

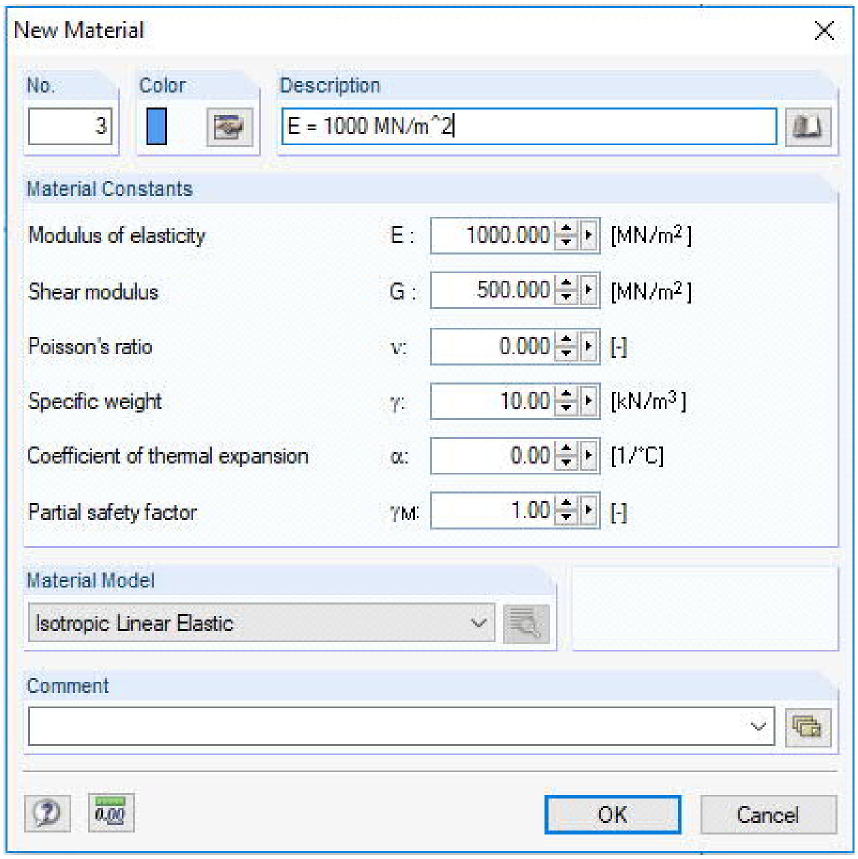


Figure 31. Material.

Save: Ctrl+s.

3.7 Cross-Section

See the corresponding chapter in the Lecture Notes: ***Geometric Properties of the Cross-Section***.

Open cross-section window: Insert → Model Data → Cross Sections → Dialog Box. Press icon  (Rectangle). Give the cross-sectional data, including the material, as shown in Figure 32 and press OK. Note the coordinate system used (see Figure 24). Press OK.

Moment of inertia and cross-section areas are updated as shown in Figure 33. Press OK.

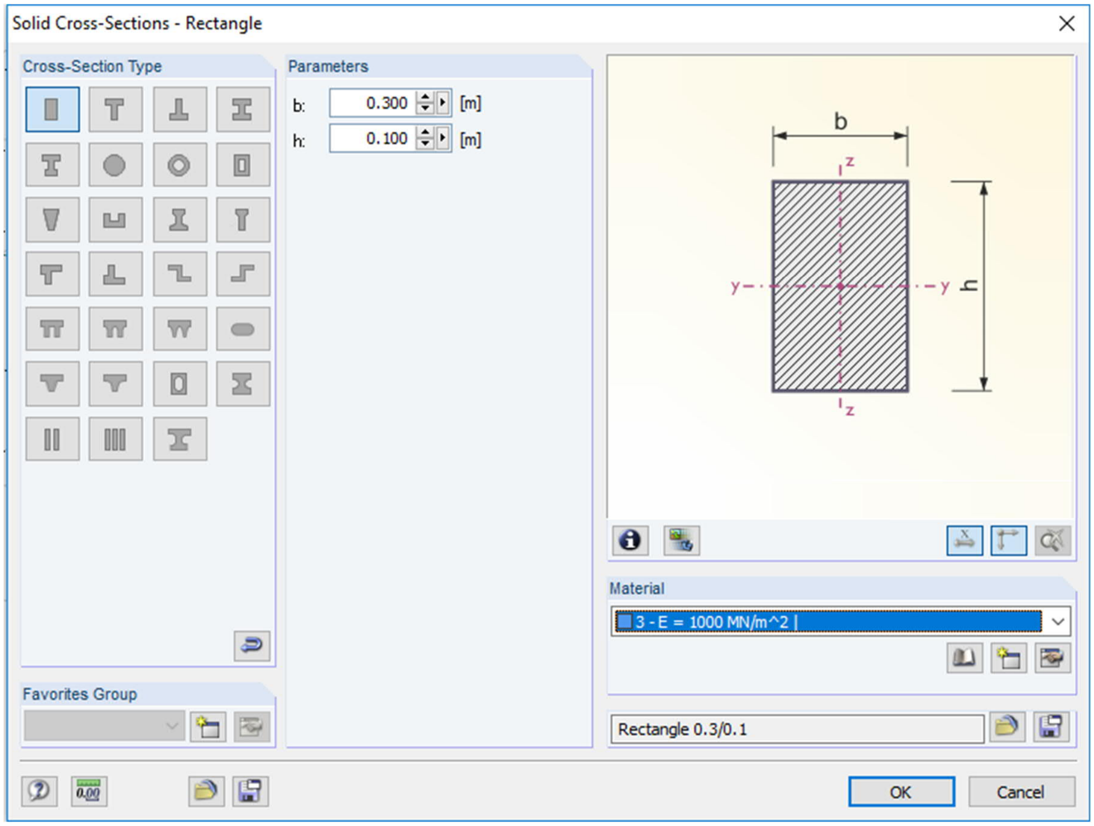


Figure 32. Rectangular cross-section.

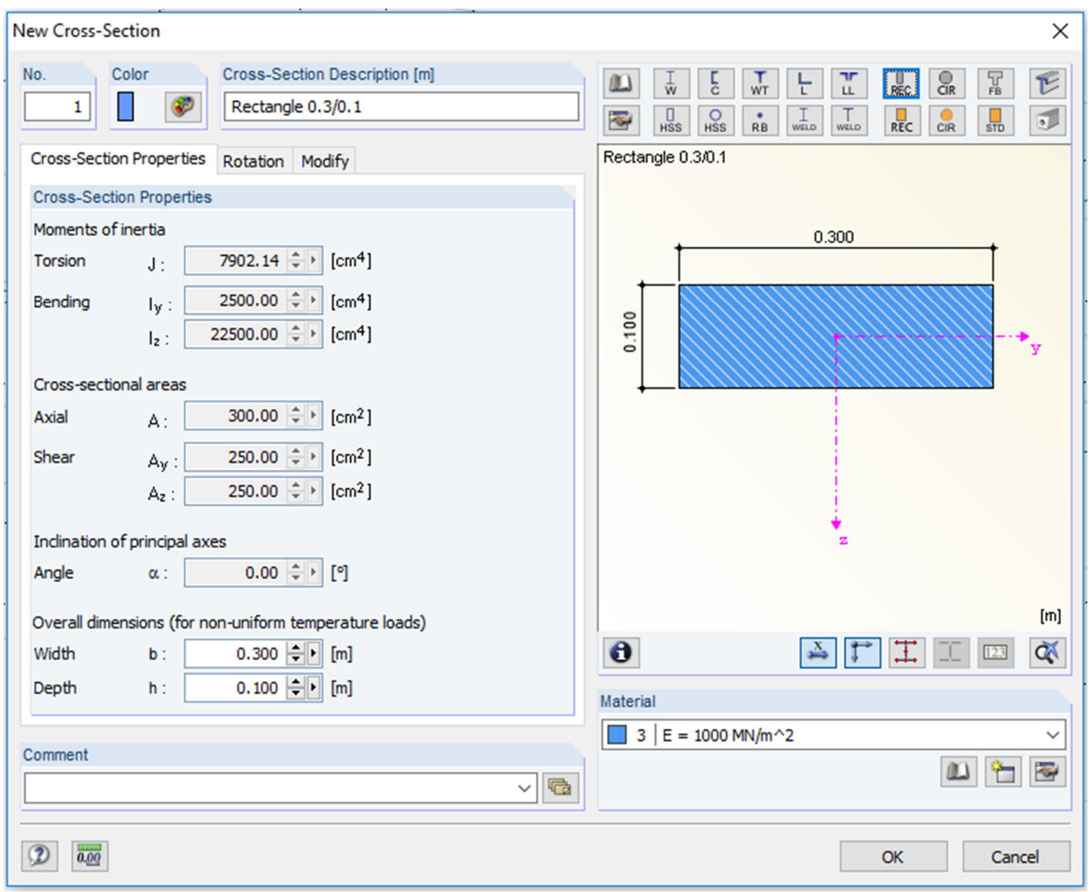


Figure 33. Cross-section.

Save: Ctrl+s.

3.8 Geometry

*See the corresponding chapter in the Lecture Notes: **Geometry**.*

Determine a beam, with length of *L*, from the origin to the right hand side. First, select icon , or from the menu: Insert → Model Data → Members → Graphically → Single. Check the data in the member window and press OK (Figure 34).

When the line window is open, draw a line from the origin to the right hand side by using the left mouse button (Figure 35). End by using Esc-button.

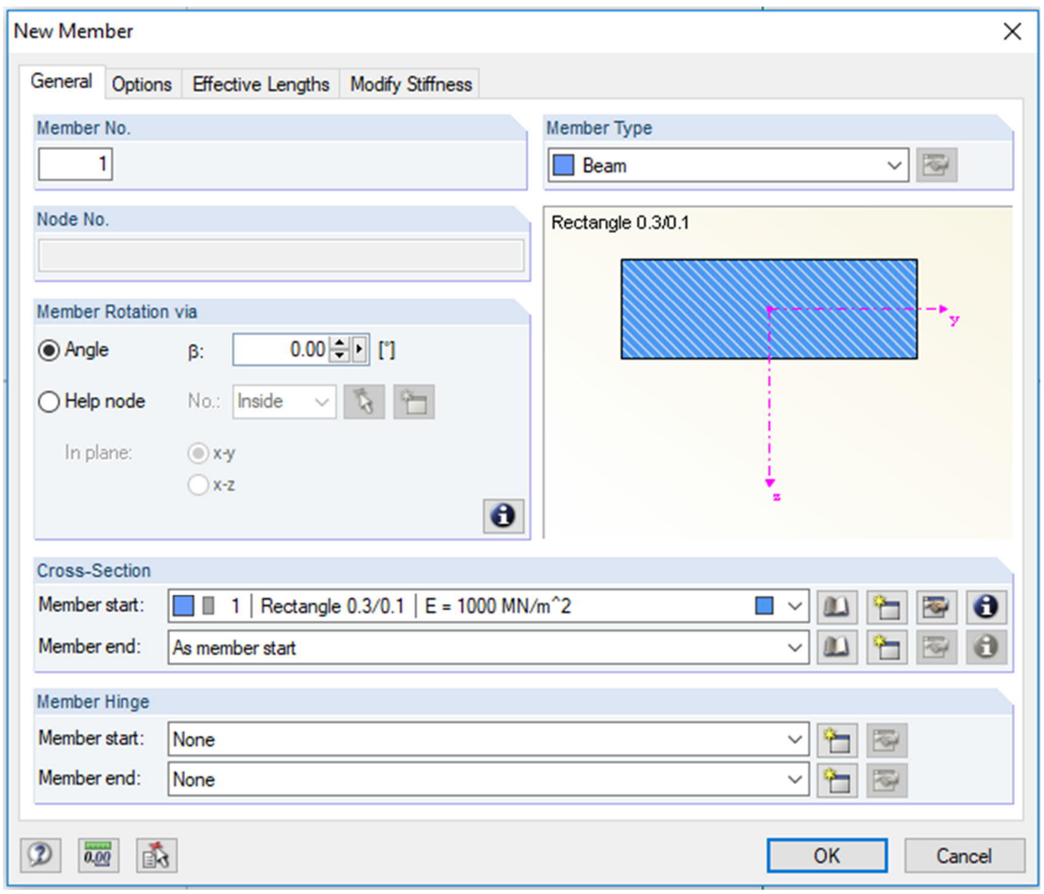


Figure 34. Member.

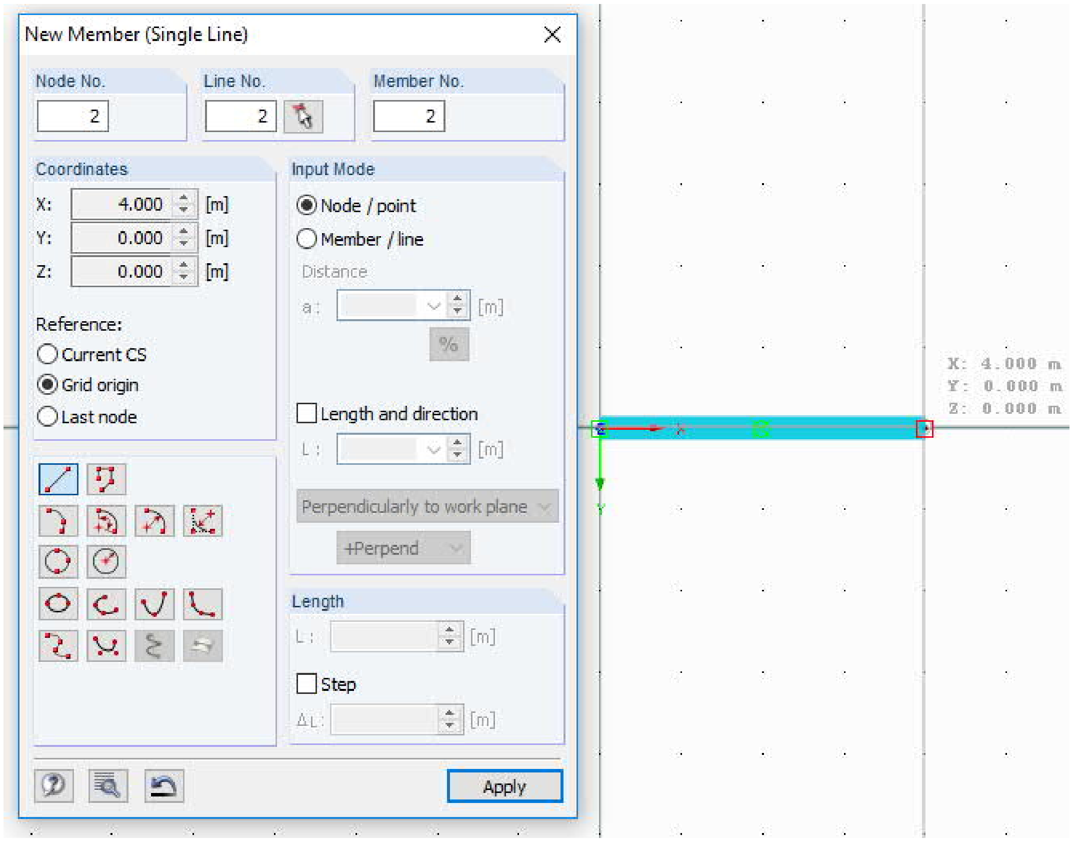


Figure 35. Line.

Point the beam by the right mouse button and select Divide Member → n Intermediate Nodes. Update as shown in Figure 36 and press OK.

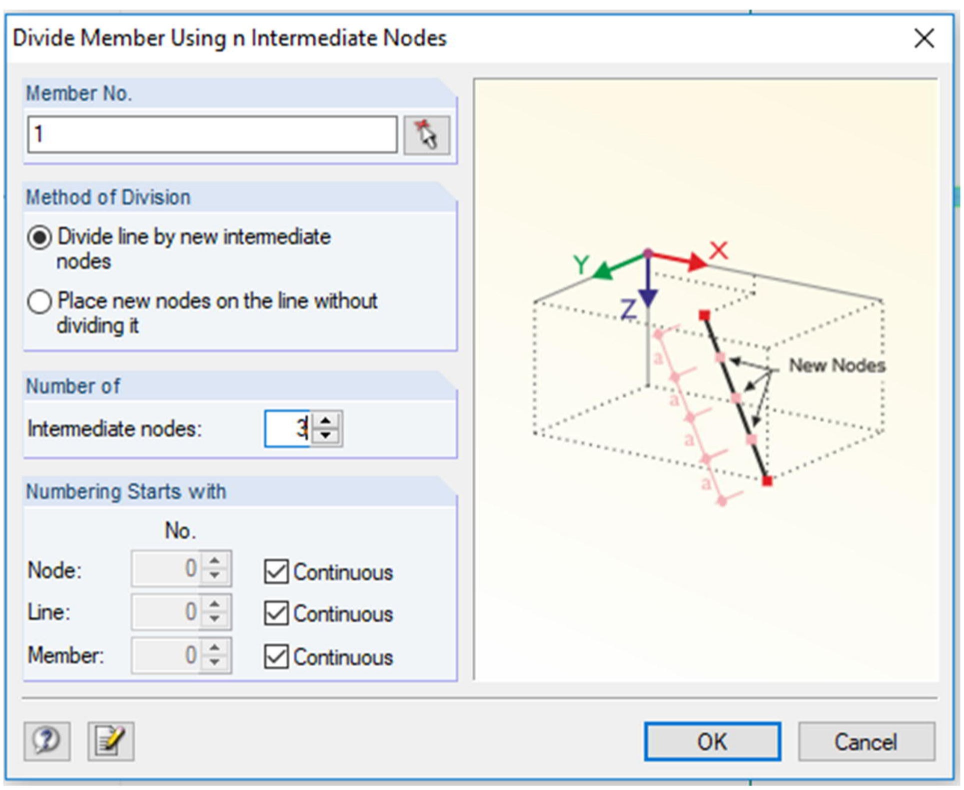


Figure 36. Dividing member.

Select Display-sheet from the Navigator (Tree view) and set the numbering on (Figure 37). Another way, without specified definitions, is to press right mouse button at the work area → Show Numbering.

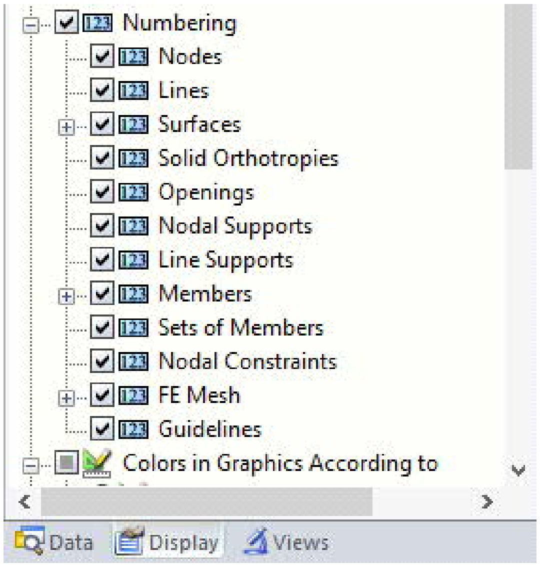


Figure 37. Numbering.

Figure 38 shows that the beam now has four members (M) and that node points 1 and 2 are at the ends of the beam. Another way would be to draw in sequence many consecutive lines; in this case, the node points would be in the numerical order, from left to right. The numbering order does not affect the result of the calculation.

In 2D-model, the point load or point moment can be place only in the node point, that's why proper amount of nodes are created. (In 3D-case, the point load, acting on the surface, can be place in the arbitrary point.)

Figure 38. Numbering of the members (M), the lines and the nodes of the beam.

Save: Ctrl+s.

3.9 Elements and Meshes

See the corresponding chapters in the Lecture Notes: ***Types of Structures*** *and* ***Elements and Meshes****.*

In the member window, the type of the member was defined to be "Beam" (Figure 34). So the beam element is used. RFEM program makes the element dividing (mesh) automatically; the user can change the definition. Change the definition by opening the FE mesh refinement window: Insert → Model Data → FE Mesh Refinements → Dialog Box. Update as shown in Figure 39 and press OK.

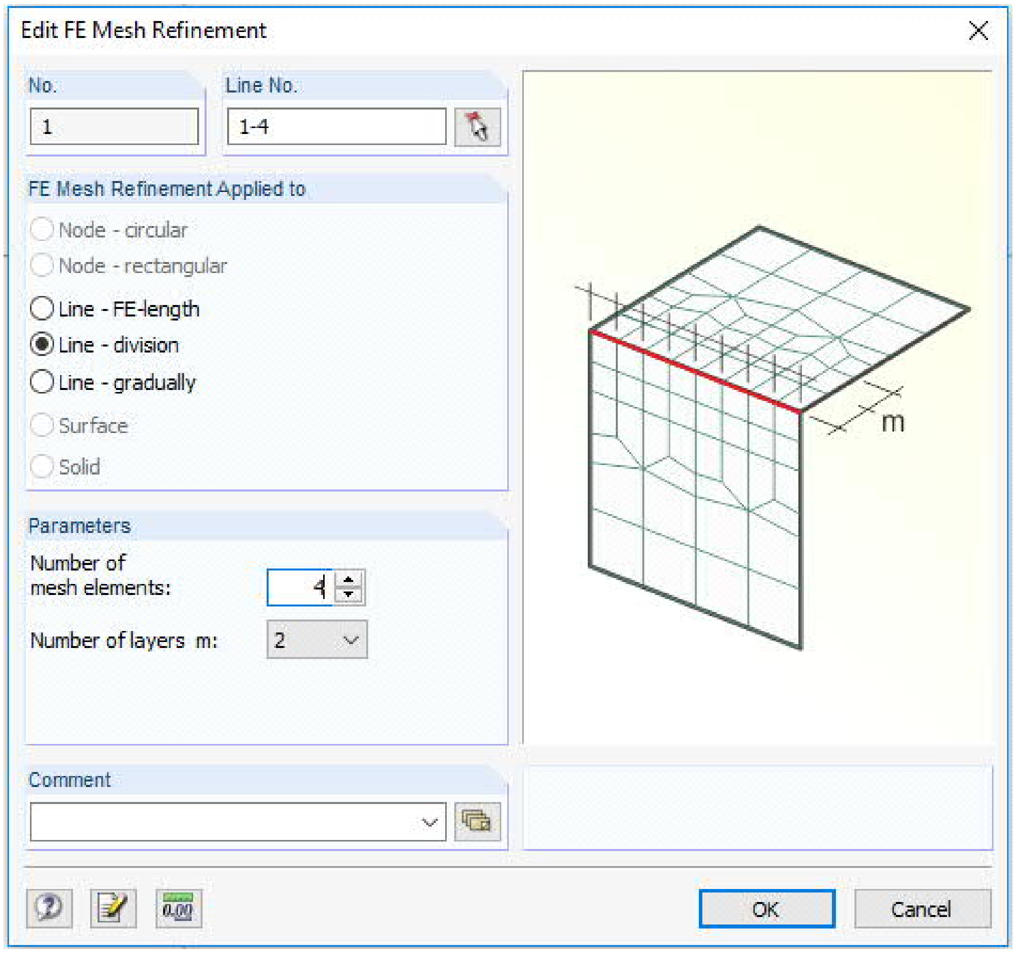


Figure 39. Edit FE mesh refinement.

The visibility of the element dividing (mesh) can be set on or off: Navigator → Display → Model → FE Mesh Refinements (Figure 40). From the same list, the visibility of the members can be, for example, set too.

Figure 40. Element dividing of the beam.

More settings of the element mesh: Calculate → FE Mesh Settings.

Save: Ctrl+s.

3.10 Supports

See the corresponding chapter in the Lecture Notes: ***Supports****.*

Add a pin support (hinged): Insert → Model data → Nodal Supports → Graphically. This will open a new window (New Nodal Support). Since the required 2D case support condition is not already available in the 3D case, click the icon  (New Type of Nodal Support). Update as shown in Figure 41 and press then icon  (Edit User-Define Axis System).

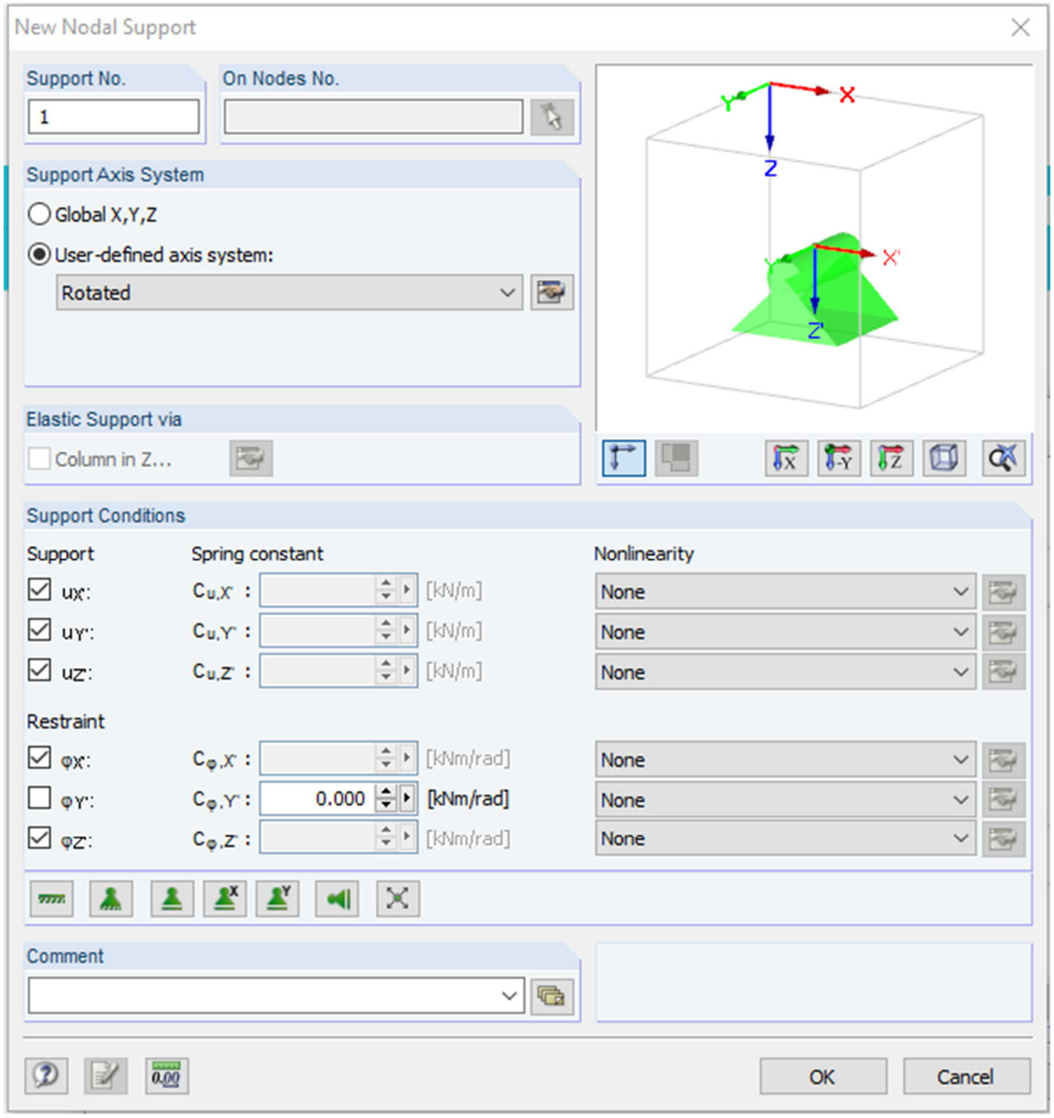


Figure 41. New nodal support.

In the window that opens, update as shown in Figure 42. The direction of the support is set so that it is correctly in the global coordinate system (X, Y, Z); the local, support-specific coordinate system (X ', Y', Z ') is different.

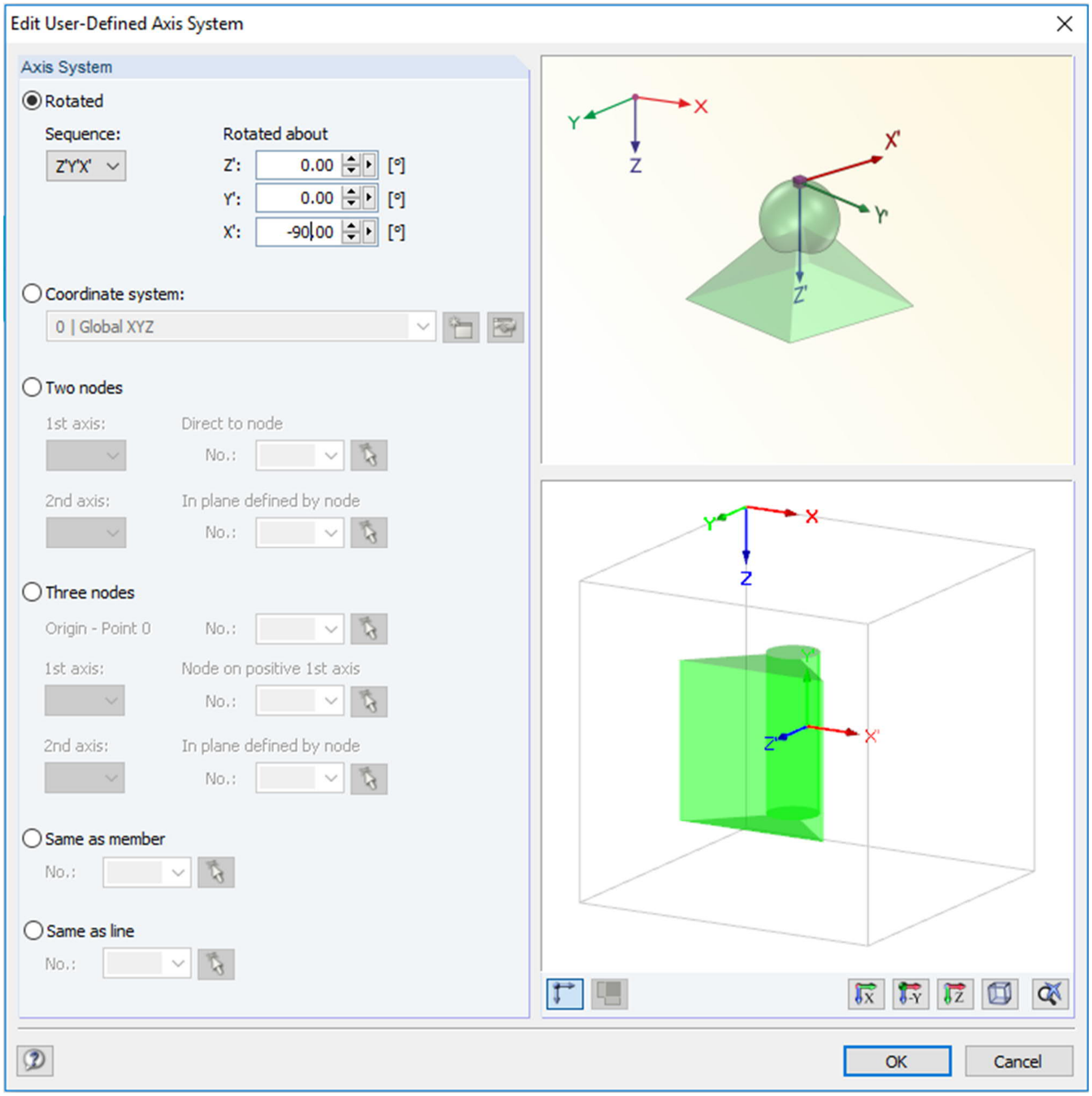


Figure 42. User-defined axis system.

Press OK twice. The pin support is now defined (Figure 43). You can change the view with the icons below the support image. Press OK. Select the node point at the left end of the beam by mouse (Figure 45). Press Esc.

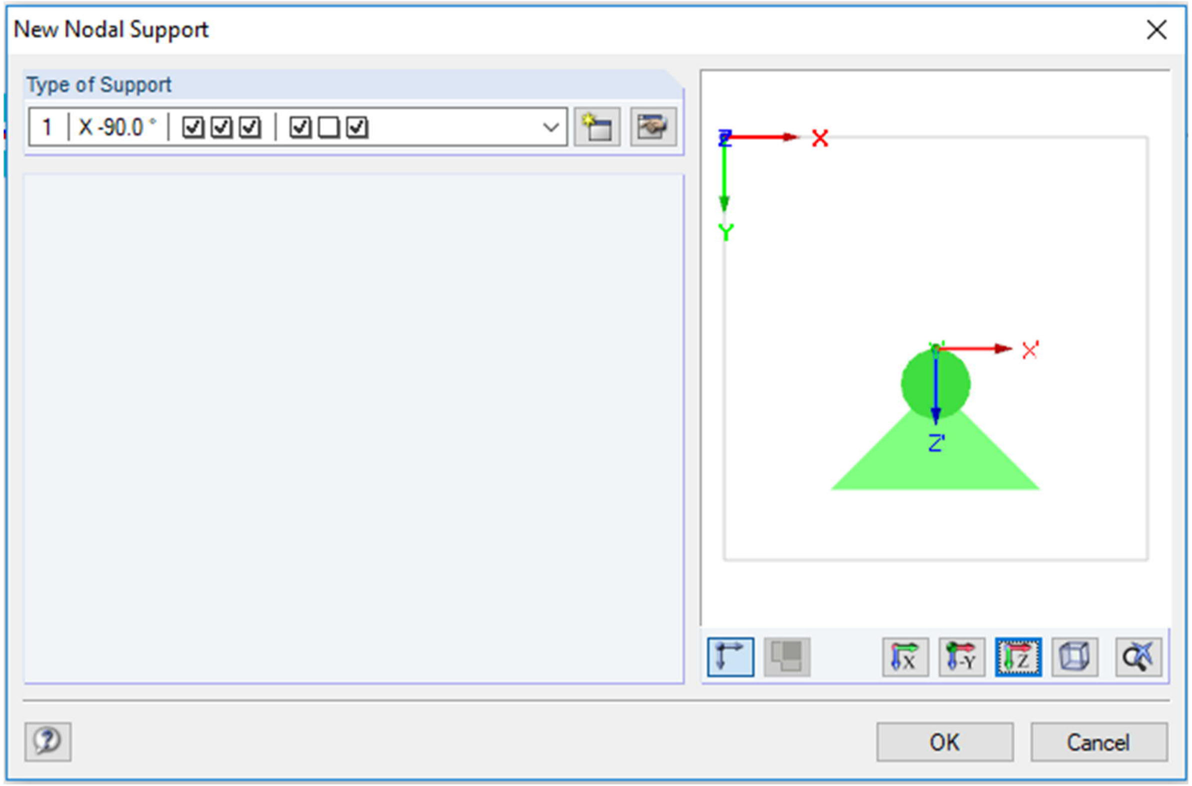


Figure 43. Pin support.

In the similar way, add a roll support (sliding in x). The difference with the previous one is that the transition u_x should be released (Figure 44). Add the roll support at the right end of the beam (Figure 45).

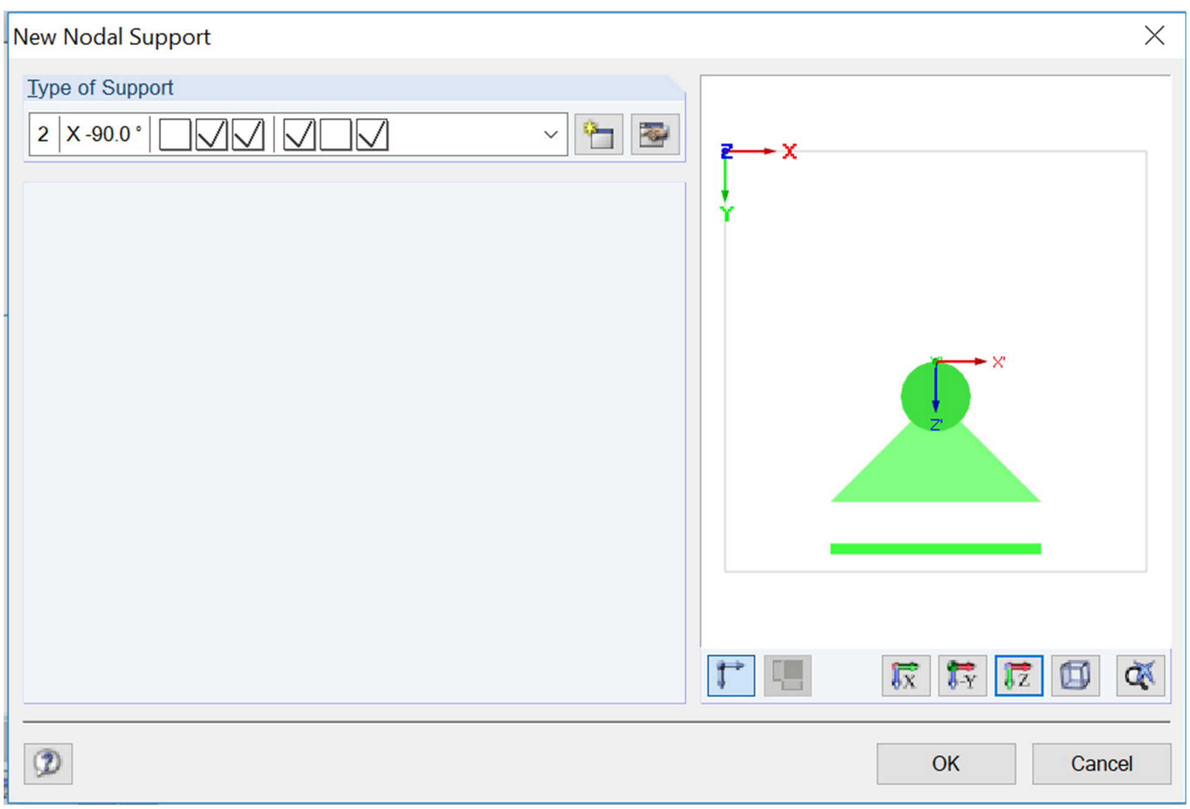


Figure 44. Roll support.

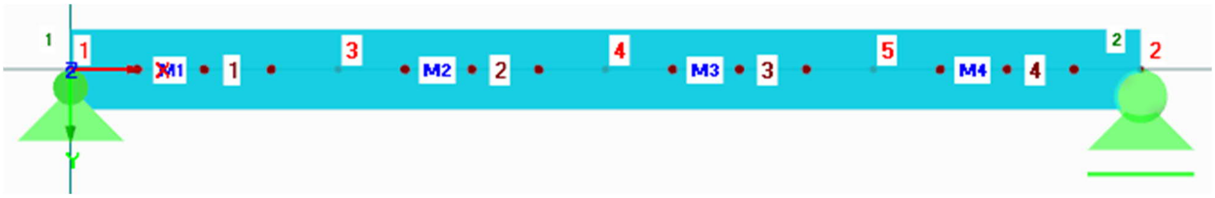


Figure 45. Beam supported at the end points.

Save: Ctrl+s.

3.11 Loading

See the corresponding chapter in the Lecture Notes: ***Loading****.*

Each load (self-weight, point force and point moment) is defined as its own load case. In this way, their effects can be considered separately, which facilitates to check the validity (Chapter 3.15 Validity, p. 53).

3.11.1 Self-Weight

It is good practice to always determine the first load case for self- weight, even if it is not needed to solve the problem. Create the first loading case and add self-weight to load the beam: Insert → Loads → New Load Case. Update the data as shown in Figure 46 by selecting “Self-weight” from the drop-down list and by activating Y-direction. Press OK.

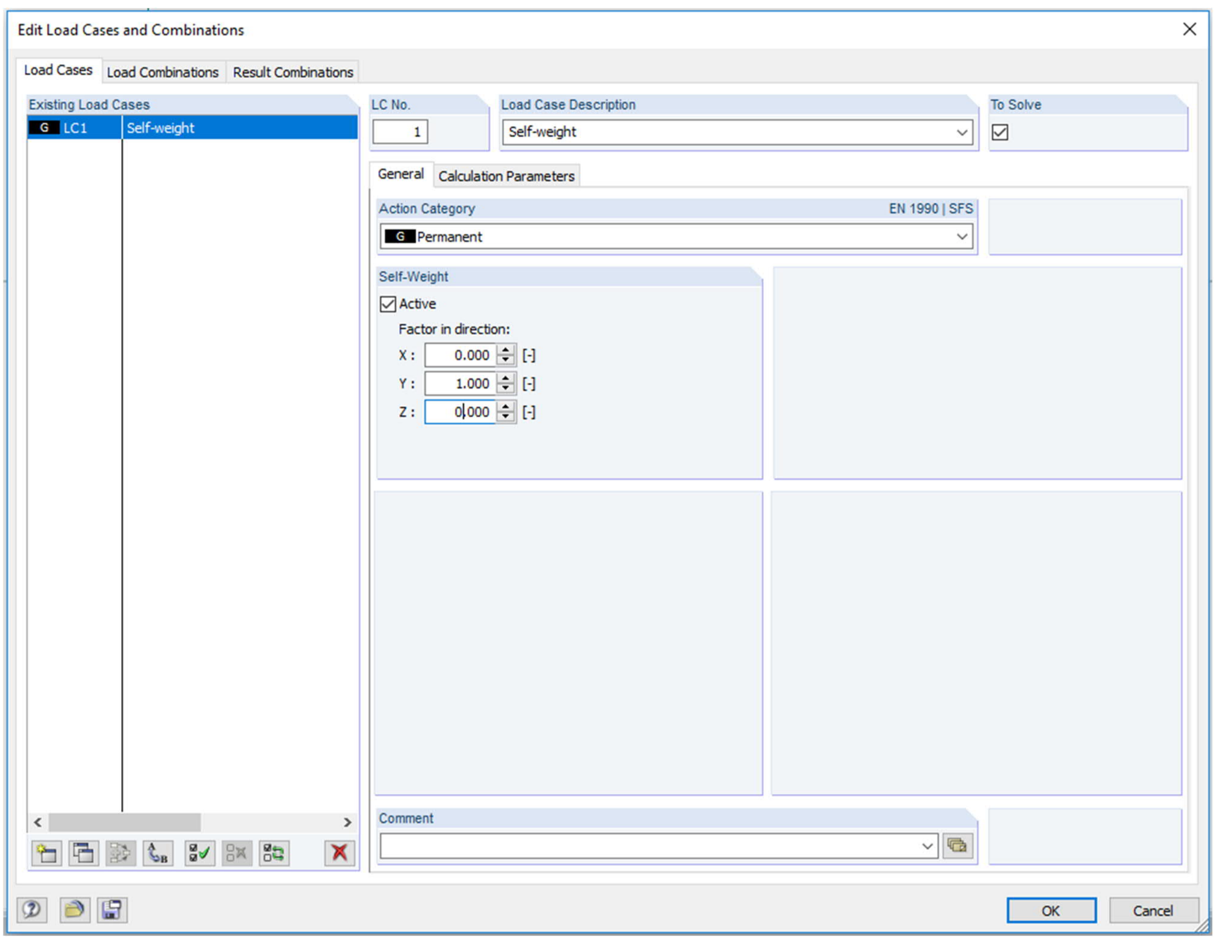


Figure 46. Self-weight load case.

The program calculates the self-weight from the length, cross-section area and unit weight. Set the self-weight visible: Navigator → Display → Loads → Self-weight (Figure 47).

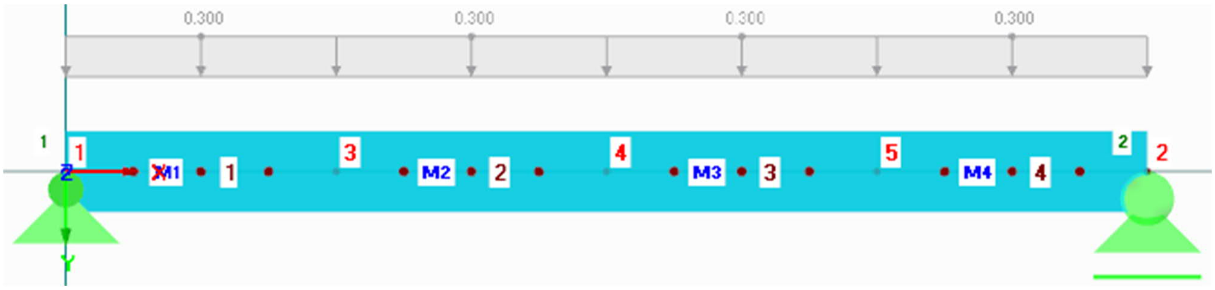


Figure 47. Beam loaded by the self-weight.

Ctrl+s.

3.11.2 Point Force

Add a new loading case: Insert → Loads → New Load Case. Update as shown in Figure 48 and press OK.

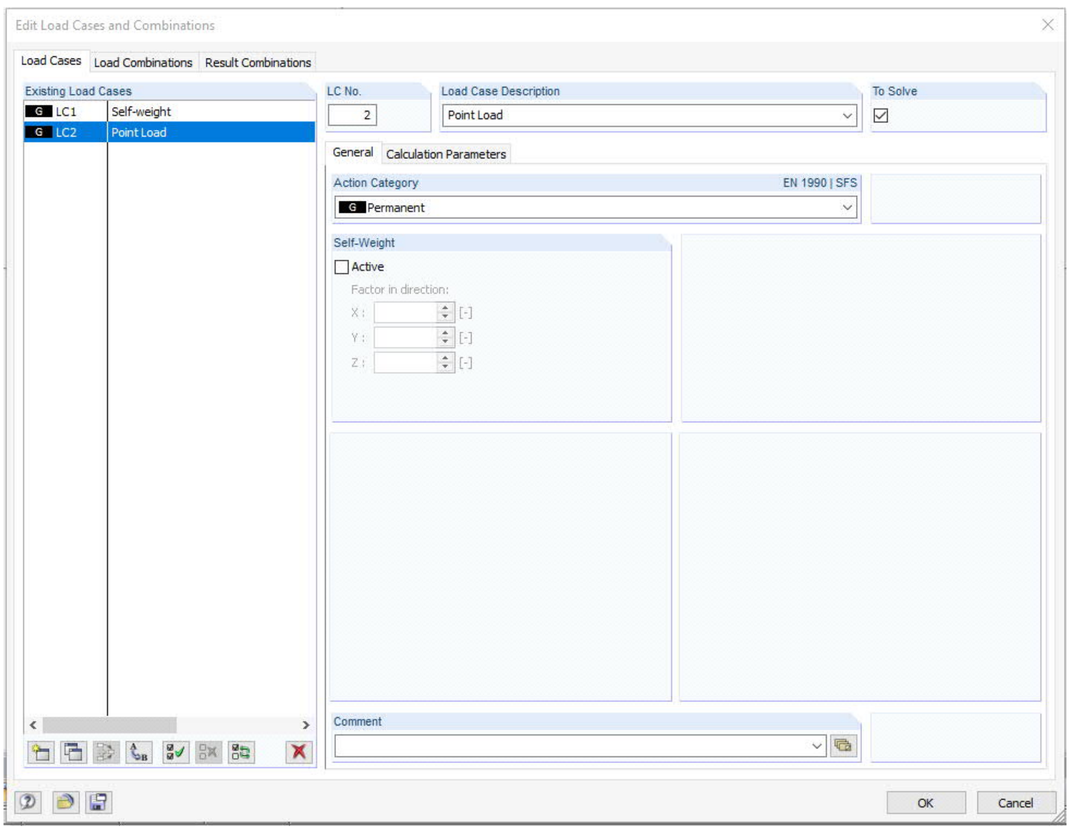


Figure 48. Load case for the point load.

Add point load to load the beam: Insert → Loads → Nodal Loads → Graphically. Update as shown in Figure 49, press OK and apply by mouse the load on node three. Press Esc. The end result is shown in Figure 50. From tree view (Navigator → Data → Loads →…) it can be check that the load is in the right load case.

In 2D case, the point load for a line can be apply only for the node point.

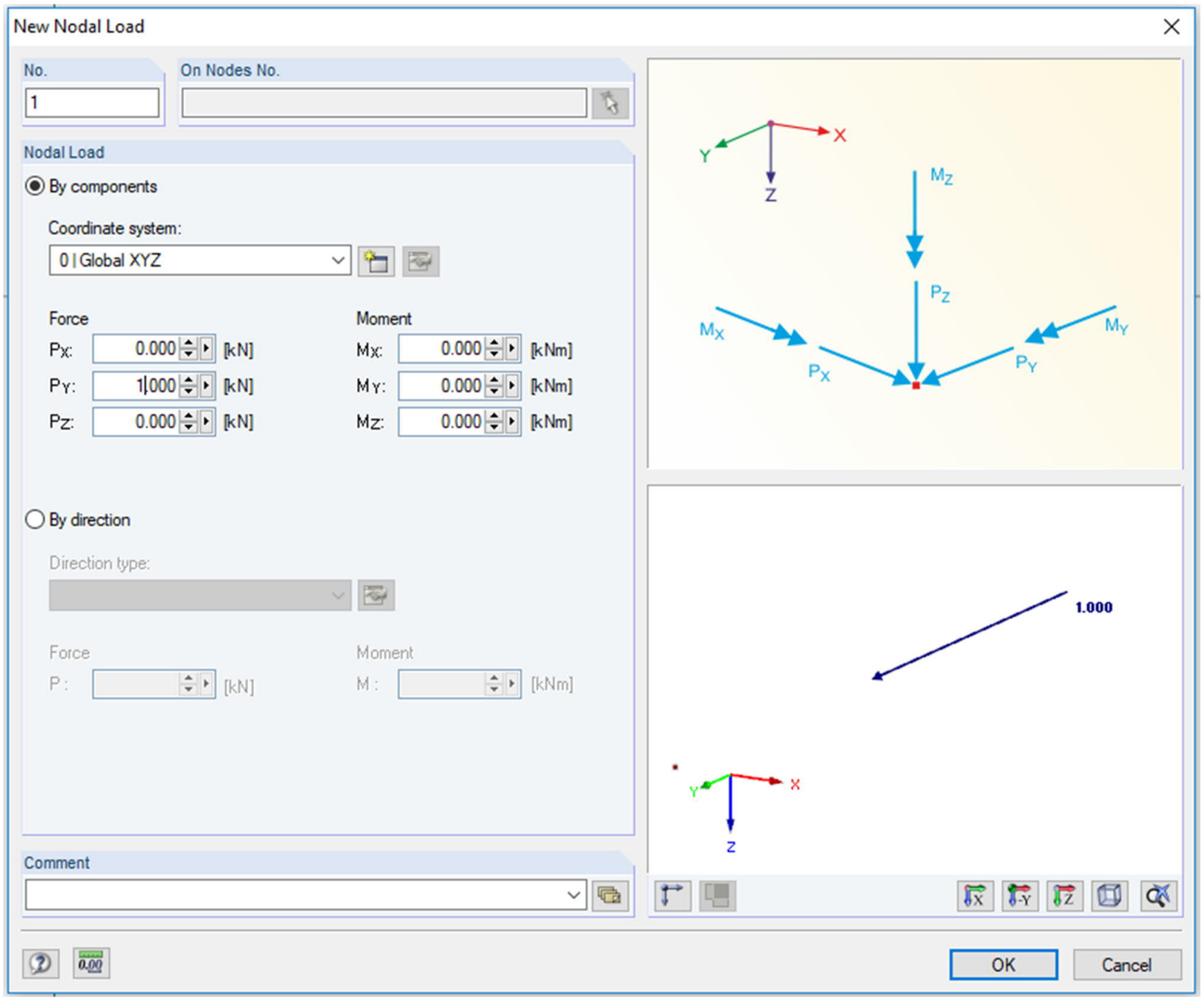


Figure 49. Point force.

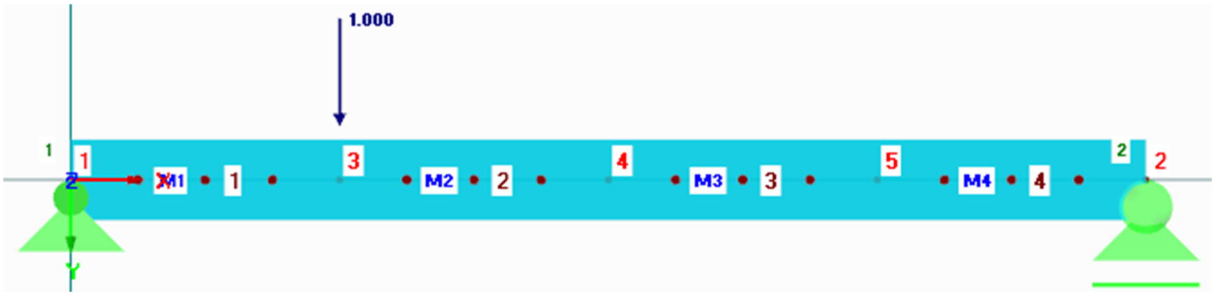


Figure 50. Beam loaded by the point force.

Ctrl+s.

3.11.3 Point Moment

Add a new loading case: Insert → Loads → New Load Case. Update as shown in Figure 51 and press OK.

Add a point moment to load the beam: Insert → Loads → Nodal Loads → Graphically. Update as shown in Figure 52, press OK and apply by mouse the load in node point three. Press Esc. The end result is shown in Figure 53.

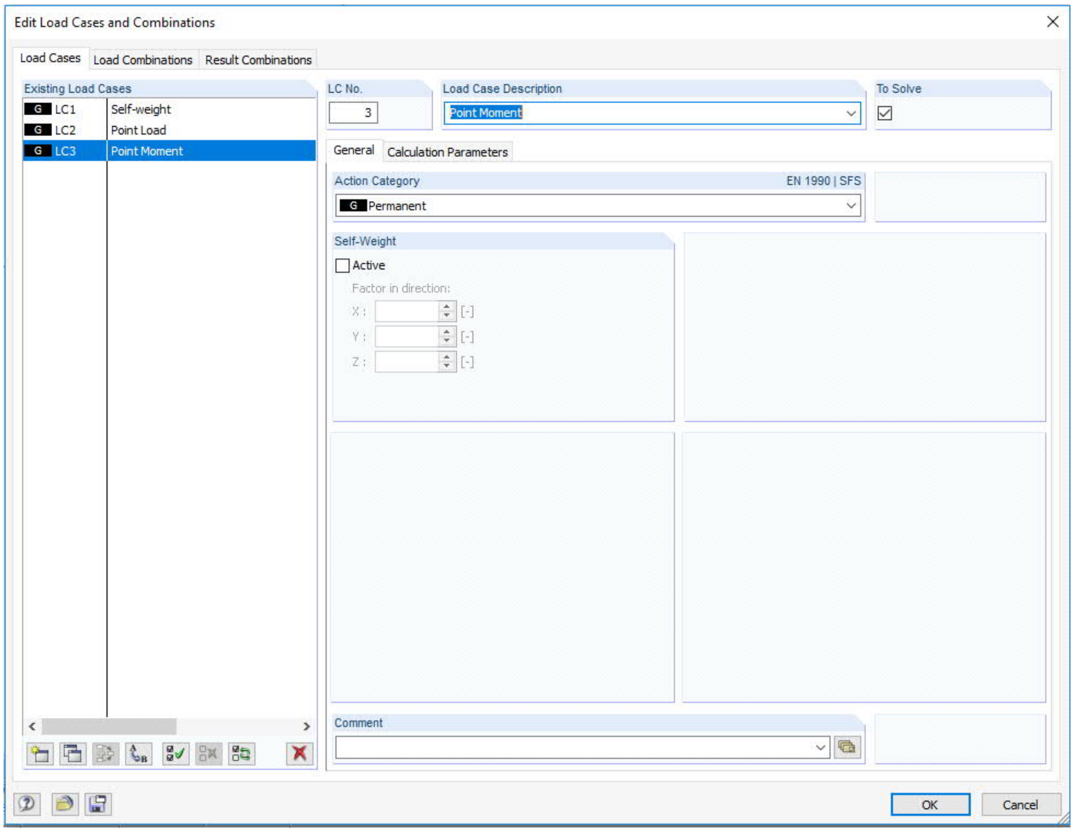


Figure 51. Load case for the point moment.

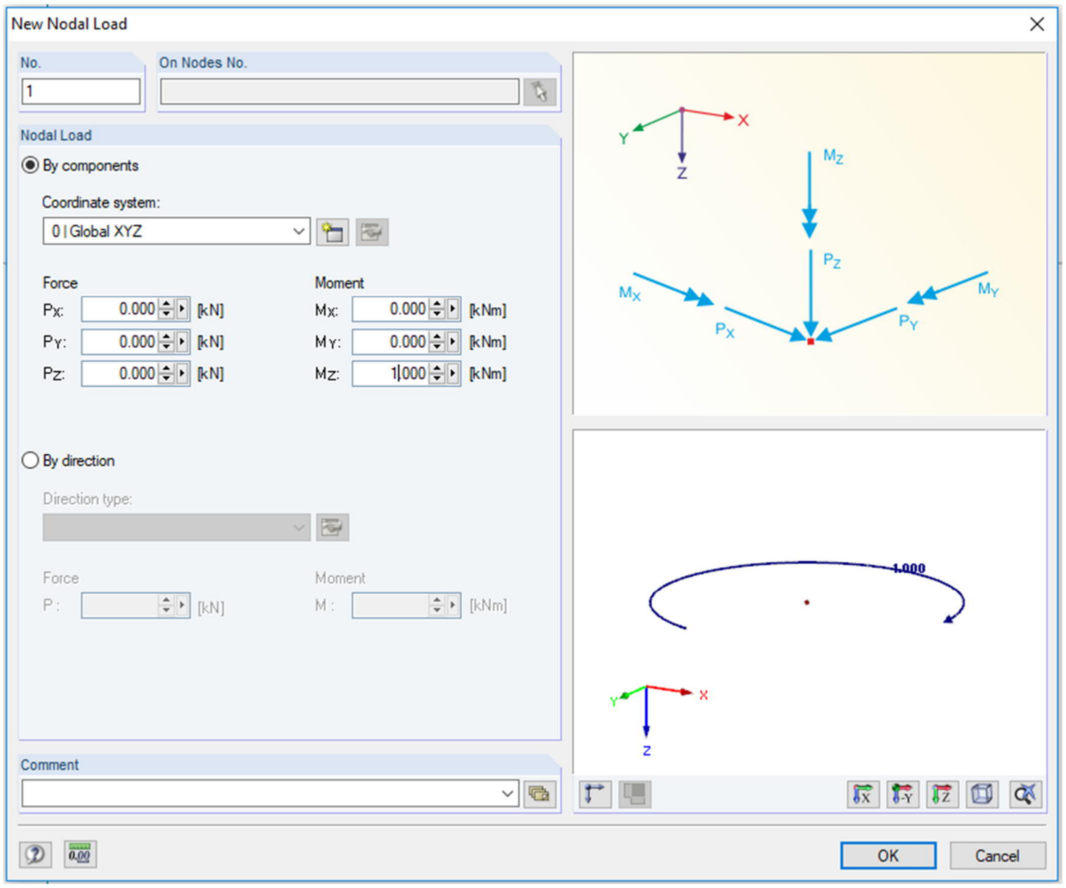


Figure 52. Point moment.

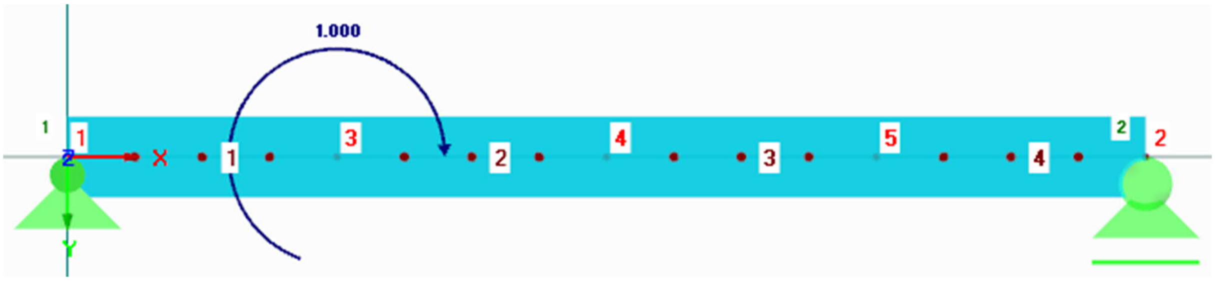


Figure 53. Beam loaded by the point moment.

Ctrl+s.

3.11.4 Load Combination

According to the superposition principle (in a linear system) loads or their responses from different load cases can be added together to obtain a synergy (see Chapter 3.14 Results, p. 43).

Make the load combination, which includes all the loads: Insert → Loads Cases and Combinations → Load Combination. Update as shown in Figure 54. The safety factors can be changed from the drop-down list. Press OK.

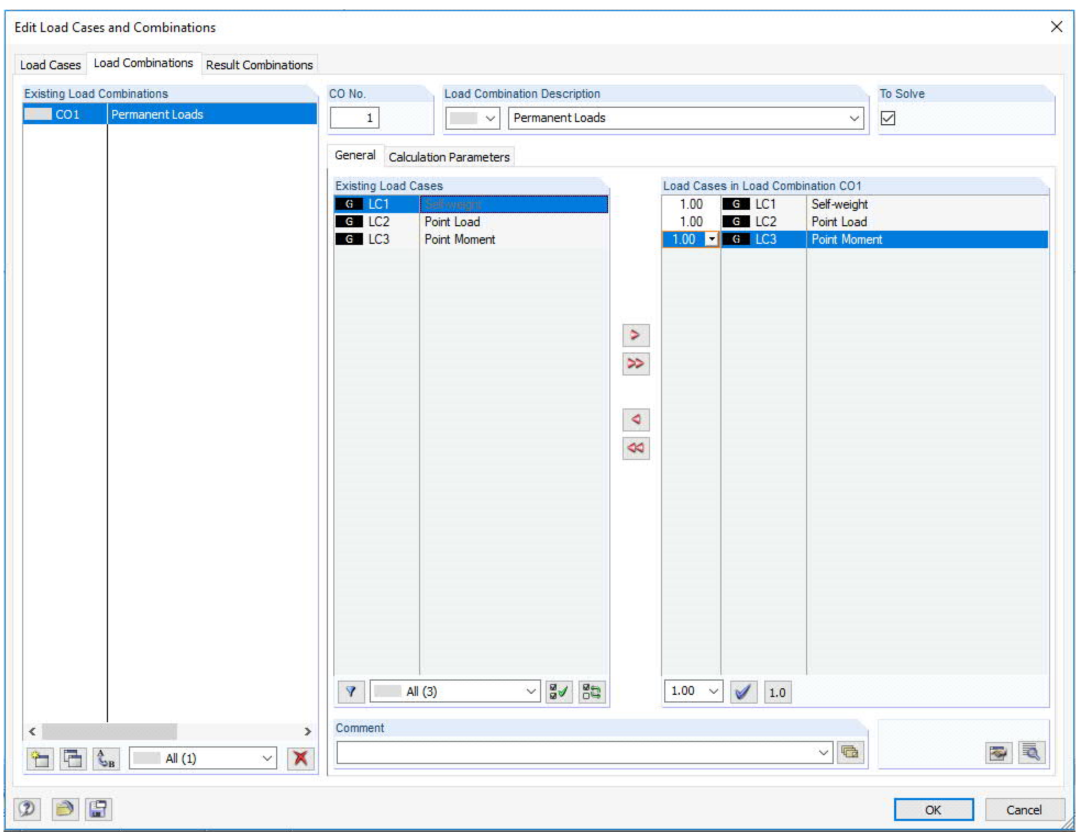


Figure 54. Load combination.

All loads included in the combination are presented in Figure 55.

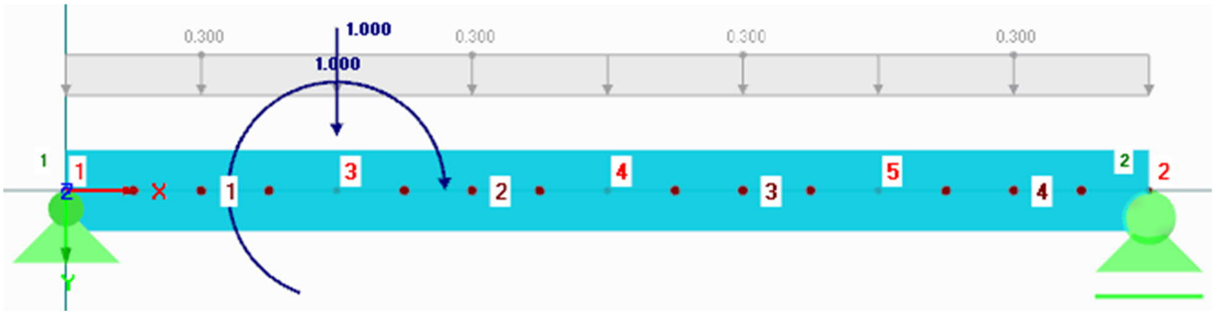


Figure 55. Loaded beam.

Ctrl+s.

3.12 Model Checking

See the corresponding chapter in the Lecture Notes: ***Model Checking****.*

3.12.1 General Checking

Make the general checking: Tools → Plausibility Check. Update as shown in Figure 56, with selected warnings option, and press OK. If there are warnings, find and correct them.

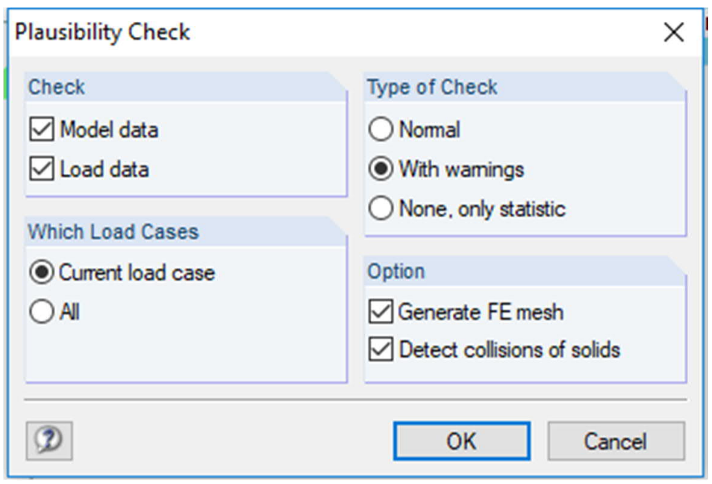


Figure 56. Plausibility check.

The model data can be checked from the pop-up window (Figure 57).

The existence of the essential data is ensured by this tool. However, there can be, for example, wrong numerical input values, which will be checked after the analysis in Chapter 3.15 Validity, p. 53.

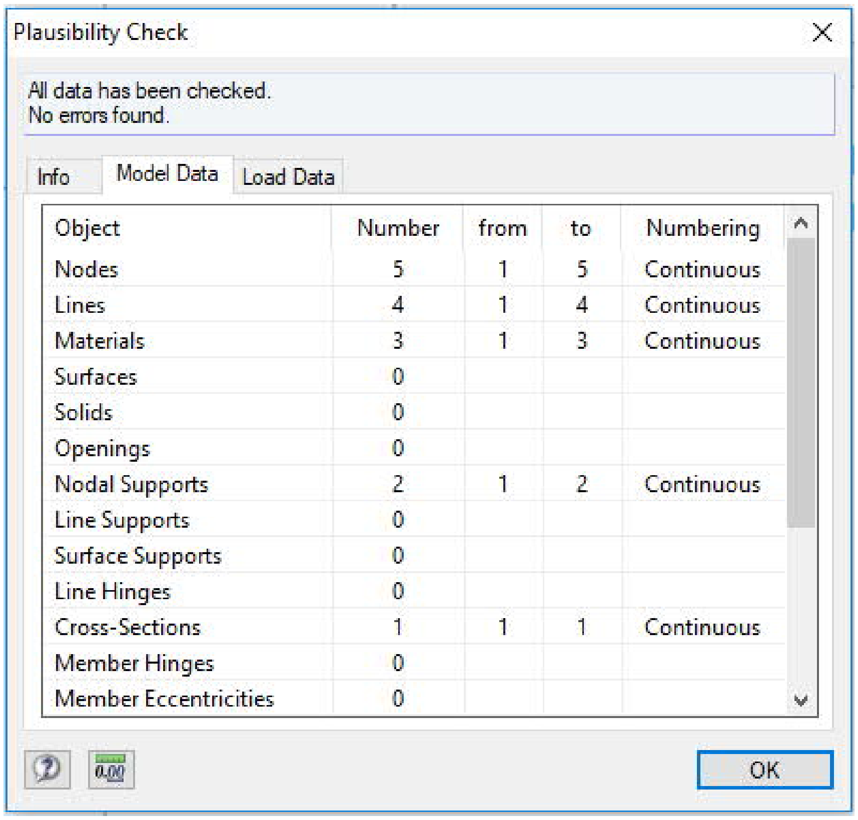


Figure 57. Model data.

3.12.2 Geometric Checking

Check the model for any mismatches: Tools → Model Check →… (Figure 58). The model can have

- identical nodes
- overlapping members
- crossing unconnected members
- overlapping lines
- crossing unconnected lines
- unused zero lines.

The checking result is good, if there are not any extra parts.

Surface check tools are not available for 2D model.

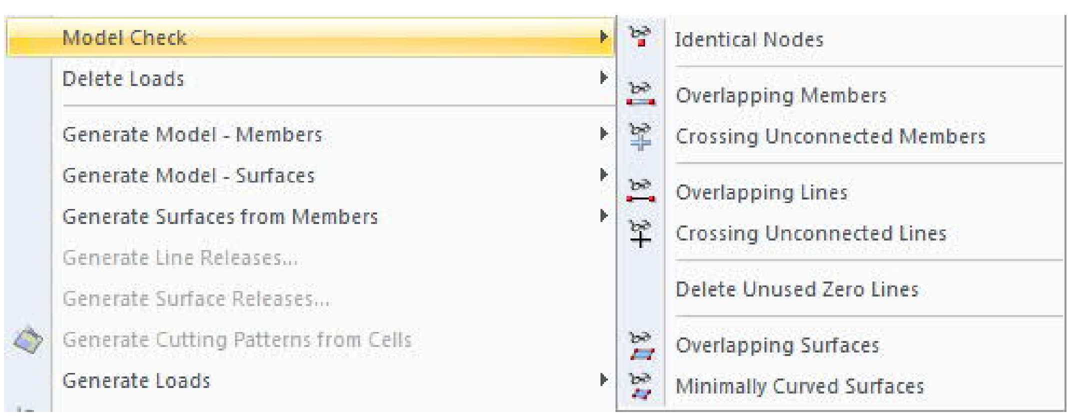


Figure 58. Checking the model geometry.

If the model geometry is made by some other program, there might be some inaccuracy between the parts. In this case, it is good to regenerate the model: Tools → Regenerate Model.

3.12.3 Checking the Loads

Remove any unused loads: Tools → Delete Loads → Not Used Loads.

Ctrl+s.

3.13 Analysis

*See the corresponding chapter in the Lecture Notes: **Analysis**.*

Analyse: Calculate → Calculate All (Figure 59). It takes a short time…

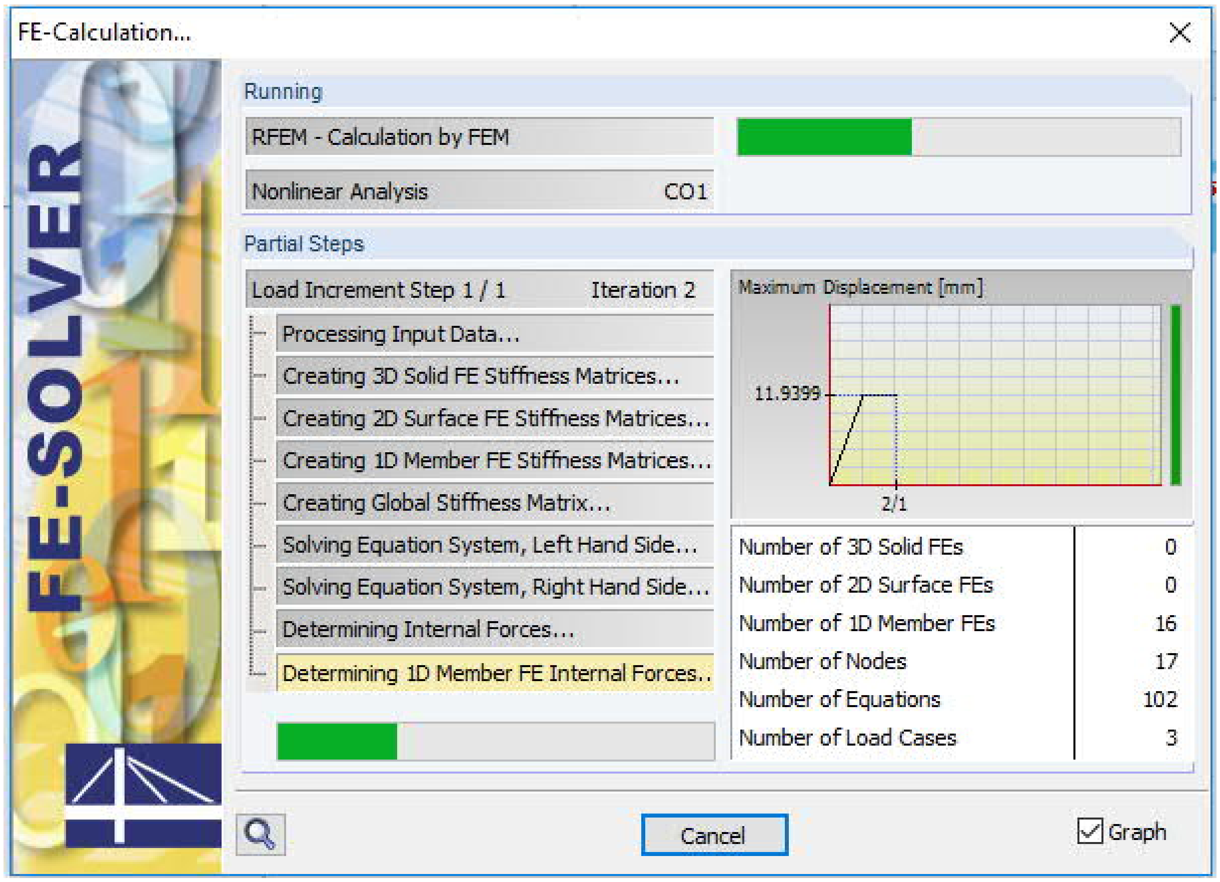


Figure 59. Calculation.

Ctrl+s.

3.14 Results

*See the corresponding chapter in the Lecture Notes: **Results**.*

3.14.1 Deflection

From the toolbar or navigation (Data sheet), select the self-weight as the load case. Select the deflection as the graph: Navigator → Results → Members → Local Deformations → u_y (= v).

The deflection curve is shown in Figure 60. Deflection refers to the displacement in the y-axis direction.

Examine the results separately for each load case and when the loads are combined (Figures 60…63).

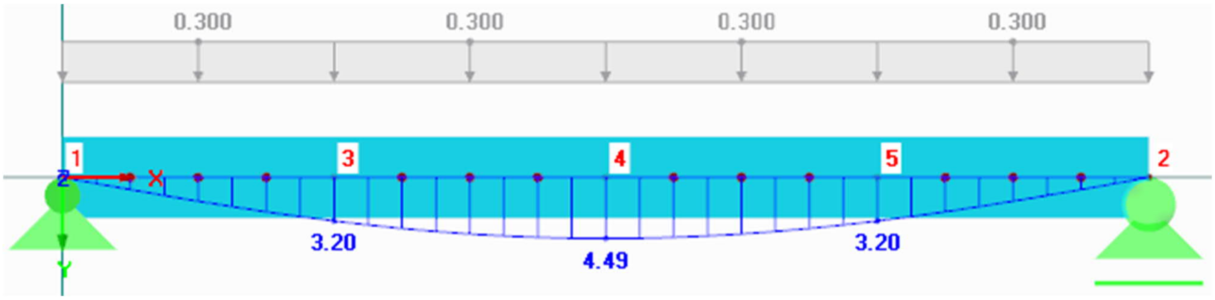


Figure 60. Deflection due to self-weight.

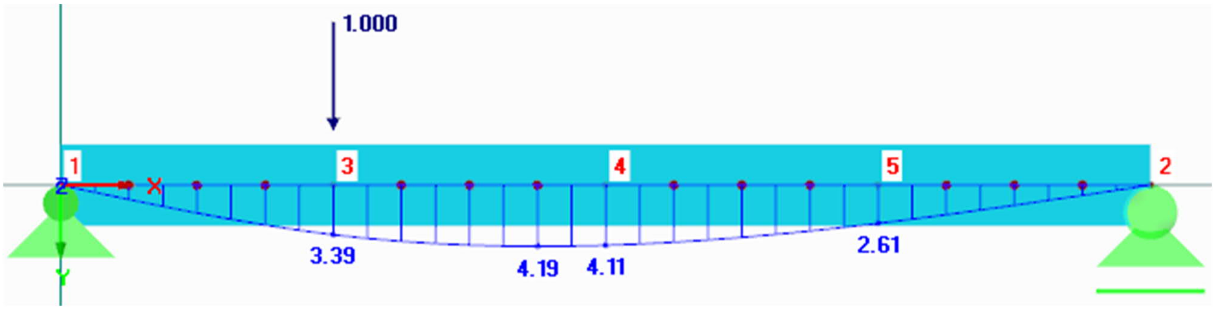


Figure 61. Deflection due to point force.

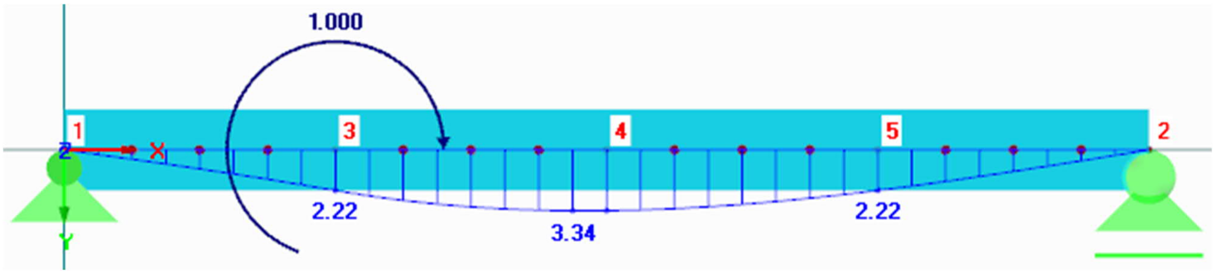


Figure 62. Deflection due to point moment.

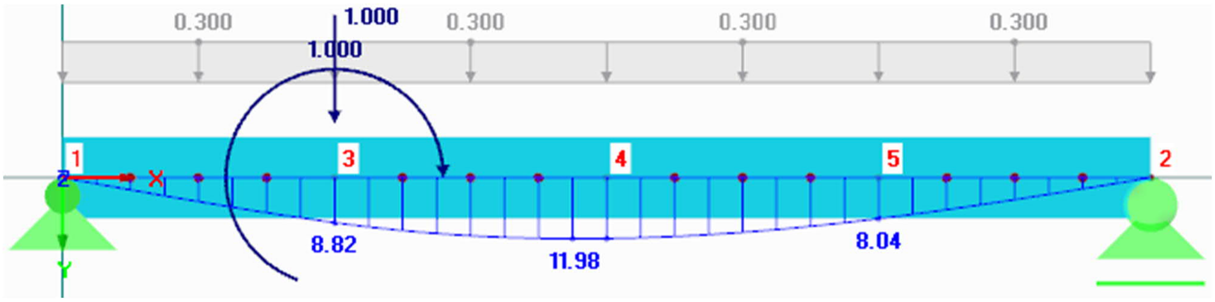


Figure 63. Deflection due to all loads.

The boundary conditions of the deflection fulfils in all load cases: deflection value at the both support is zero.

$$\begin{cases} v(0) = 0 \\ v(L) = 0 \end{cases} \tag{2a, b}$$

3.14.2 Rotation

Select rotation φ_y. Rotation is the first derivate of deflection, angle of heel (rad).

$$\varphi(x) = v'(x) \tag{3}$$

Examine results in all load cases and in the combined one (Figures 64….67).

In the symmetric load case the rotation value is zero in middle of the beam (Figure 64).

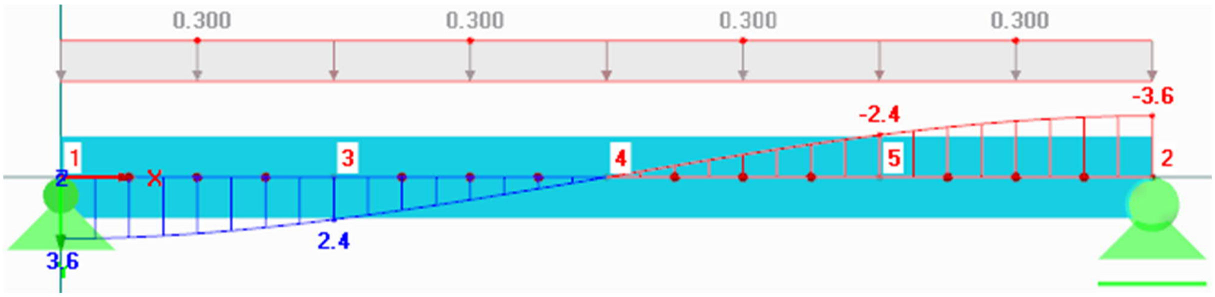


Figure 64. Rotation due to self-weight.

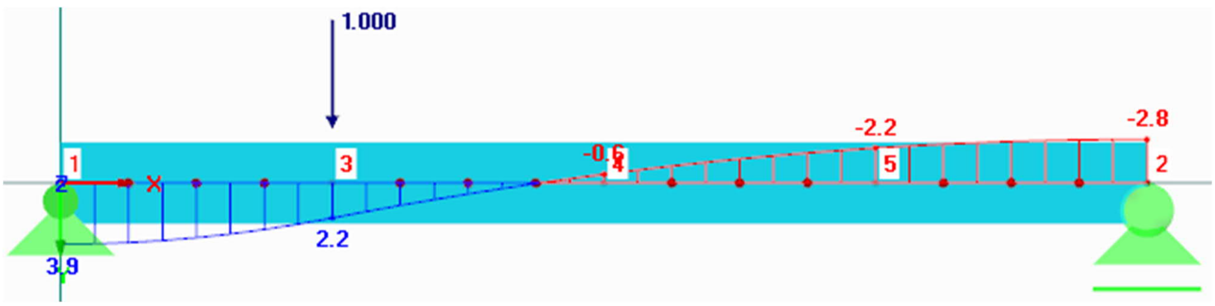


Figure 65. Rotation due to point force.

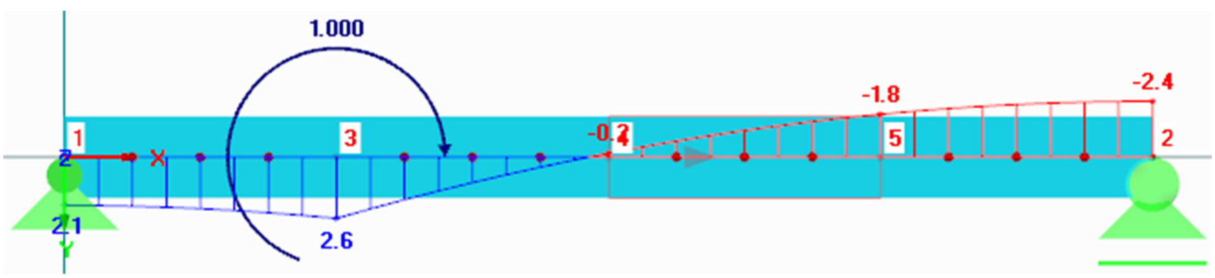


Figure 66. Rotation due to point moment.

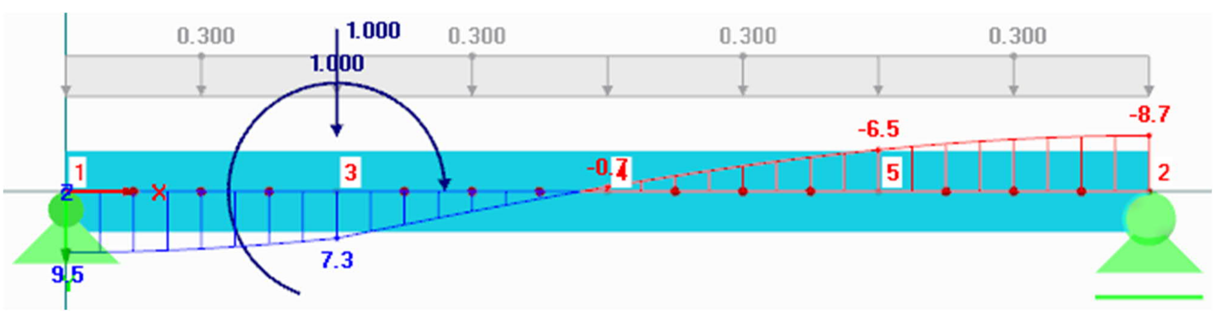


Figure 67. Rotation due to all loads.

3.14.3 Bending Moment

Select bending moment M_z (Figure 68).

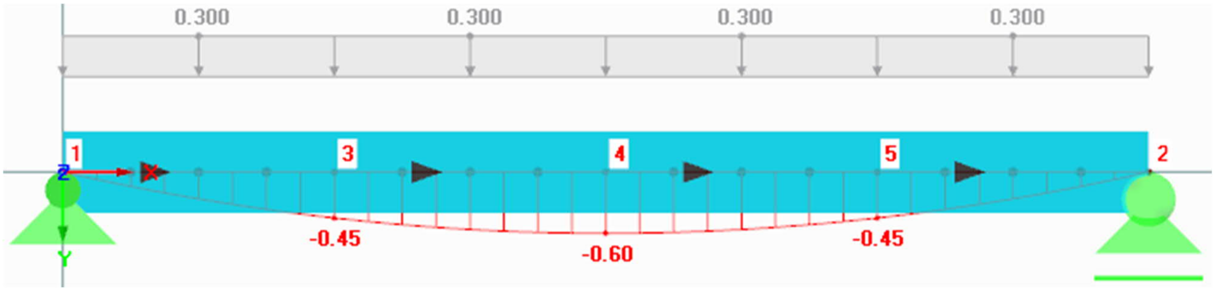


Figure 68. Bending moment due to self-weight.

The bending moment is the opposite of the second derivative of the deflection divided by the bending stiffness (*EI*).

$$M_z(x) = \frac{-v''(x)}{EI_z} \tag{4}$$

where the modulus of elasticity and the moment of inertias are constant, as is the case in this problem.

The sign rule used by the program is different from the one used in Finland. You can get the sign correctly on the graphs by changing the direction of each component. Show component directions: Navigator → Display → Model → Members → Member Orientations (Figure 68).

While holding down the Shift key, left-click on each members (4 pieces). Then select from the right-click menu ”Reverse Member Orientation”. Recalculate: Calculate → Calculate all. Compare Figures 68 and 69.

Using the program's documentation tool makes it easier in practice to use the results provided by the program as is and to comment on different sign rules in the document (Chapter 3.16 Documentation, p. 54).

Examine results in all load cases and in the combined one (Figures 69…72).

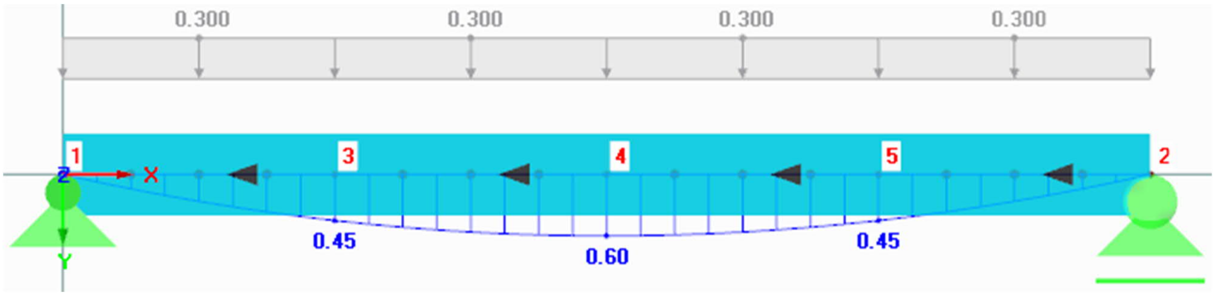


Figure 69. Reversed directions of the members.

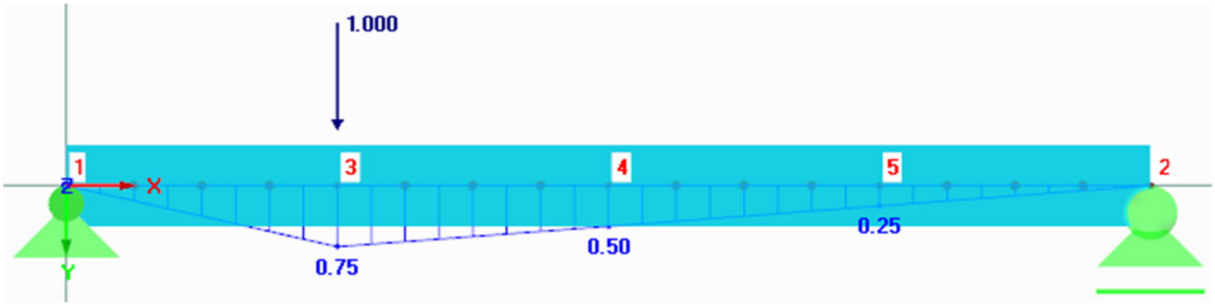


Figure 70. Bending moment due to point load.

In the case of a point moment, the bending moment has two values, the difference of which is equal to the external point moment (Figure 71).

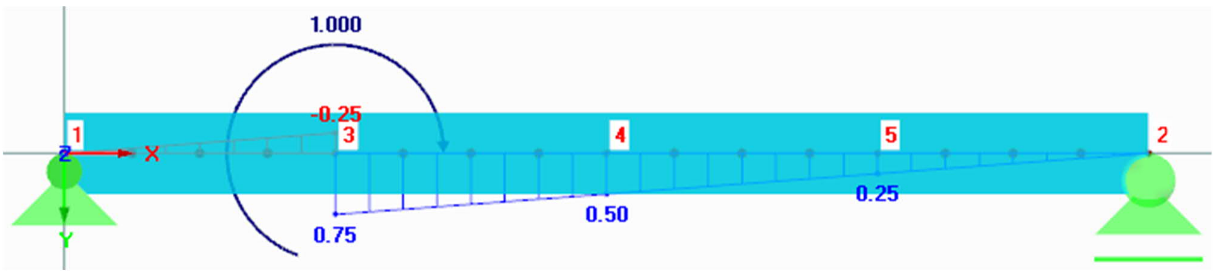


Figure 71. Bending moment due to point moment.

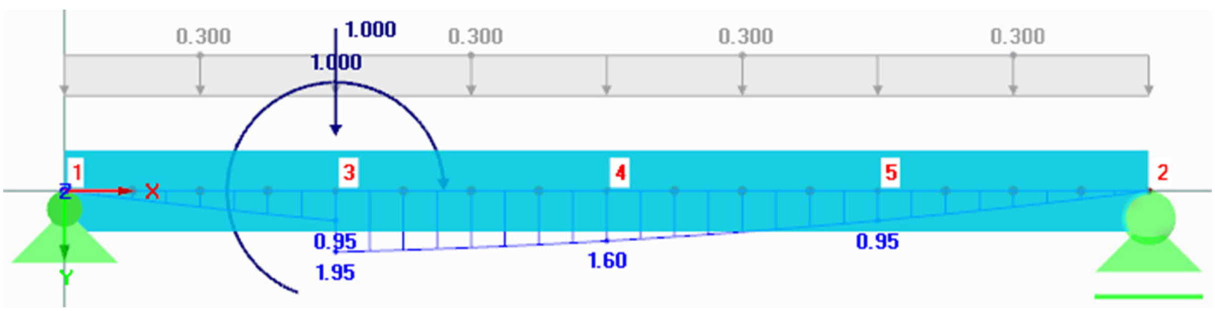


Figure 72. Bending moment due to all loads.

The boundary conditions of the bending moment fulfils in all load cases: bending moment value at the both support is zero.

$$\begin{cases} M(0) = 0 \\ M(L) = 0 \end{cases} \tag{5a, b}$$

$$\Rightarrow \begin{cases} v''(0) = 0 \\ v''(L) = 0 \end{cases} \tag{6a, b}$$

If the external point moment is acting at the support, then the value of the bending moment at this support would be equal to the external point moment.

3.14.4 Shear Force

Select shear force V_y. Shear force is the first derivate of bending moment.

$$V(x) = M_z{}'(x) \tag{7}$$

$$\Rightarrow V(x) = \frac{-v'''(x)}{EI_z} \tag{8}$$

The sum of the forces and support reactions along the *y*-axis is zero.

$$\sum F_y = 0 \tag{9}$$

Support reactions can be set visible from the navigator: Results → Support Reactions → Nodal Supports → Pz. The support reaction (A) on the left support is the same as the value of the shear force at the same point; with the right support, the support reaction (B) is the opposite of the shear force.

$$A = V_z(0) \tag{10}$$
$$B = -V_z(L) \tag{11}$$

Examine results in all load cases and in the combined one (Figures 73…76).

Distributed external load is the opposite of derivative of the shear force (Figure 73).

$$q(x) = -V_z{}'(x) \tag{12}$$
$$\Rightarrow q(x) = \frac{v^{(4)}(x)}{EI_z} \tag{13}$$

In the case of uniformly distributed symmetrical load, the value of the shear force is zero at the middle of the beam.

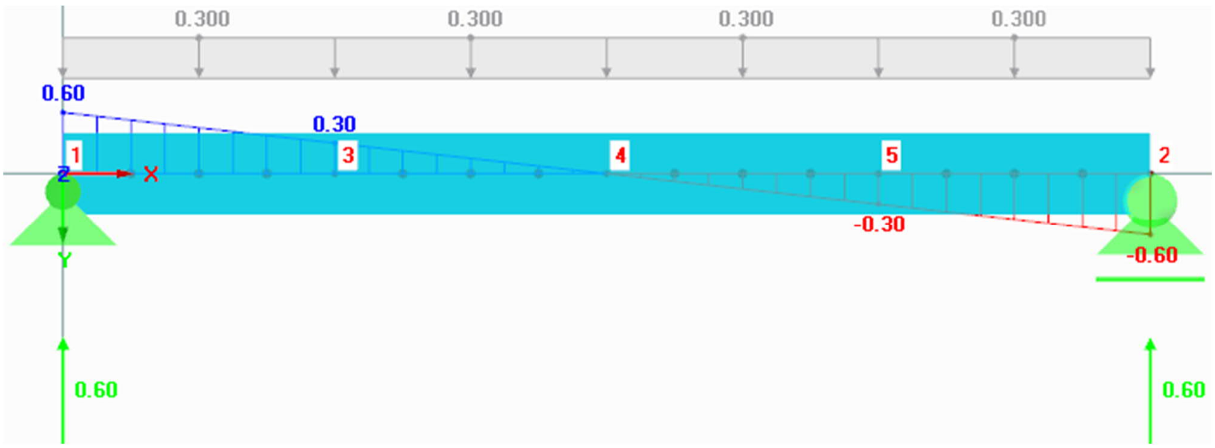


Figure 73. Shear force and reaction forces due to self-weight.

The shear force has two values at the position of the point force, the difference of which is equal to the point force (Figure 74).

Support reactions (*A* and *B*) due to point force can be checked by the lever arm rule:

$$A = \frac{b}{L}F \tag{14}$$

$$B = \frac{a}{L}F \tag{15}$$

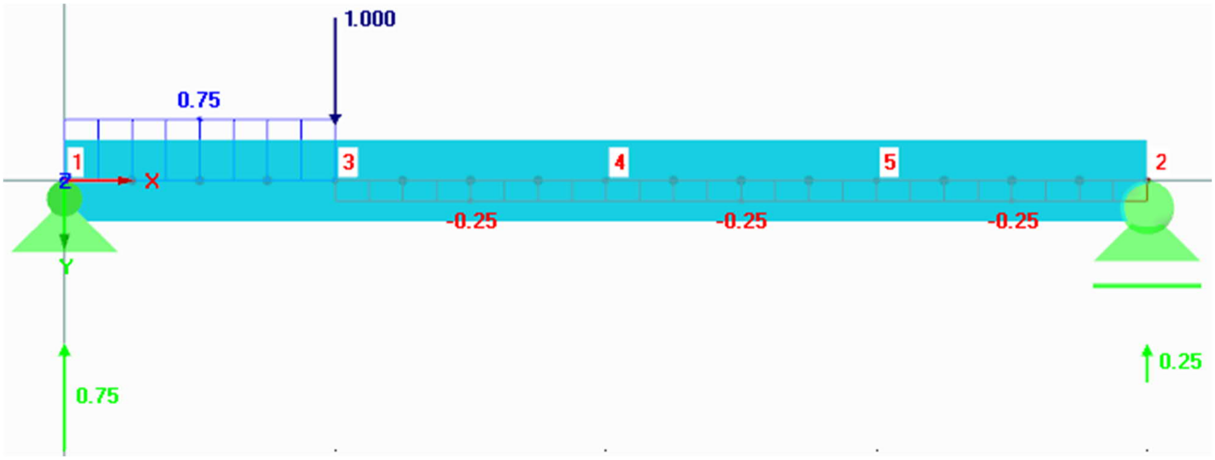


Figure 74. Shear force and reaction forces due to point force.

In the case of point moment, the value of the shear force is constant (Figure 75).

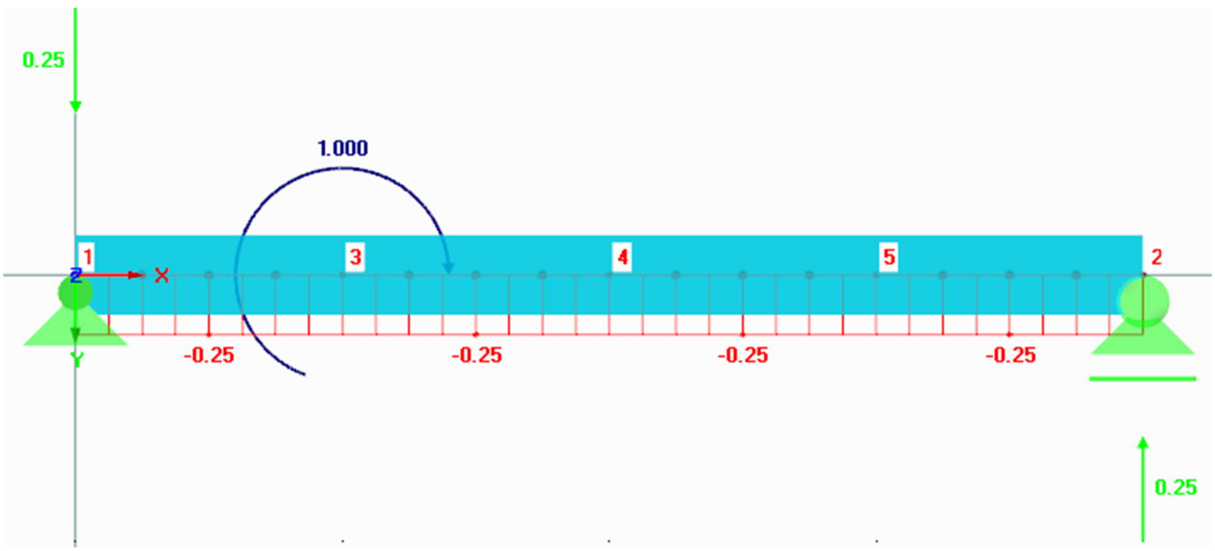


Figure 75. Shear force and reaction forces due to point moment.

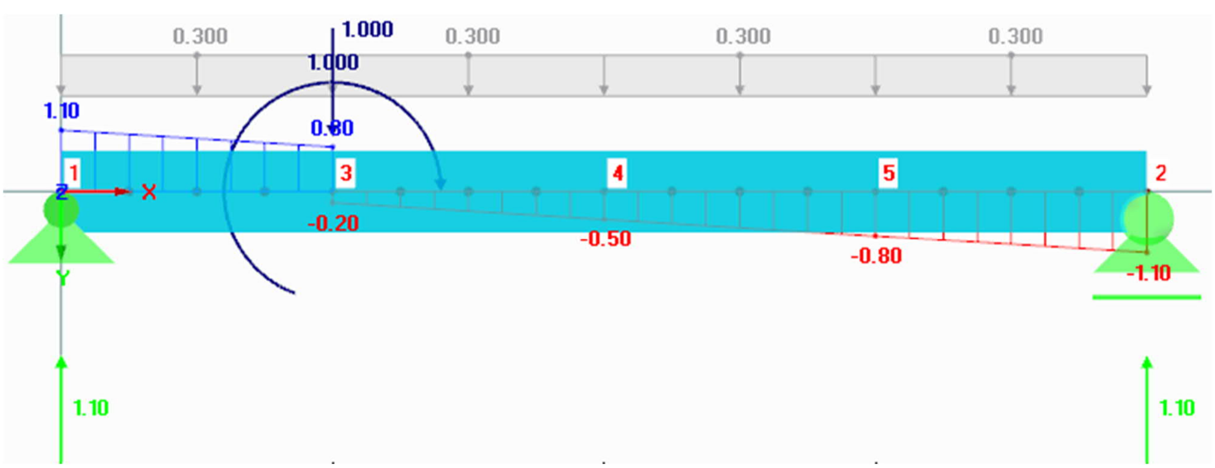


Figure 76. Shear force and reaction forces due to all loads.

3.14.5 Axial Stress

Select axial force σ_x. Axial or normal stress is the ratio of bending moment to moment of inertia multiplied by the *y*-coordinate value. If the beam were affected by its longitudinal normal force *N*(*x*), another term in the normal stress expression would be the ratio of the normal force to the cross-section area.

$$\sigma_x(x,y) = \frac{N_x(x)}{A} + \frac{M_z(x)}{I_z} y \tag{16}$$

The stress value is zero on the neutral axis ($y = 0$) and at the ends of the beam. With a homogeneous rectangular cross-section, the neutral axis coincides with the centre of gravity; normal force acts on this axis. Normal stress affects the longitudinal direction of the beam. A positive value means tension, a negative one compression. The extreme (minimum and maximum) values are at the upper and lower surface at the beam.

$$y_u = -e/2 \tag{17}$$
$$y_l = +e/2 \tag{18}$$

Examine results in all load cases and in the combined one (Figures 77…80).

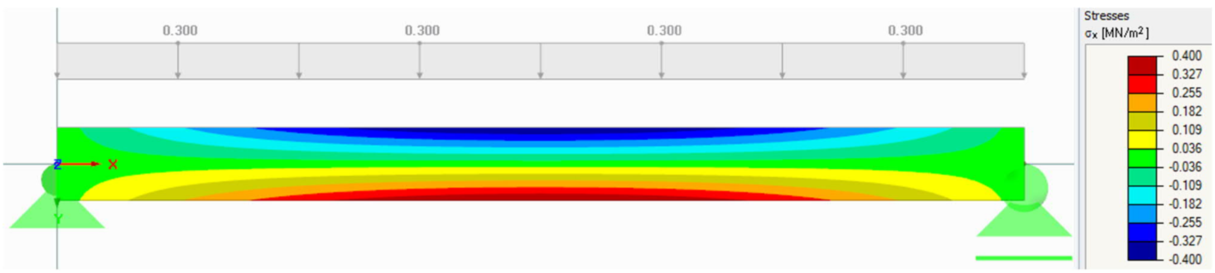


Figure 77. Normal stress due to self-weight.

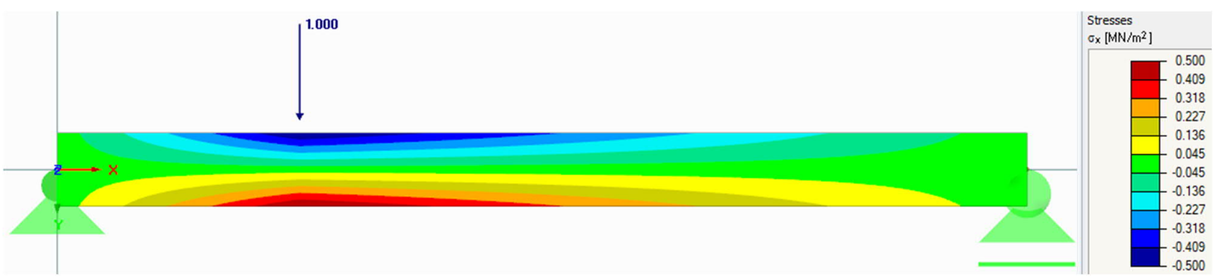


Figure 78. Normal stress due to point force.

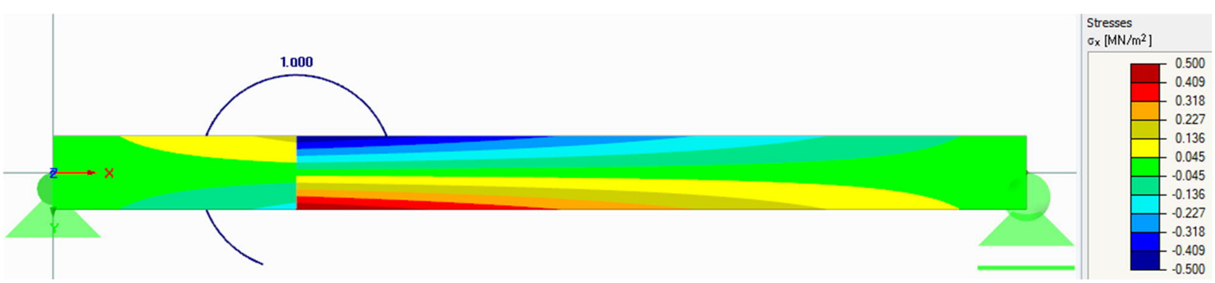


Figure 79. Normal stress due to point moment.

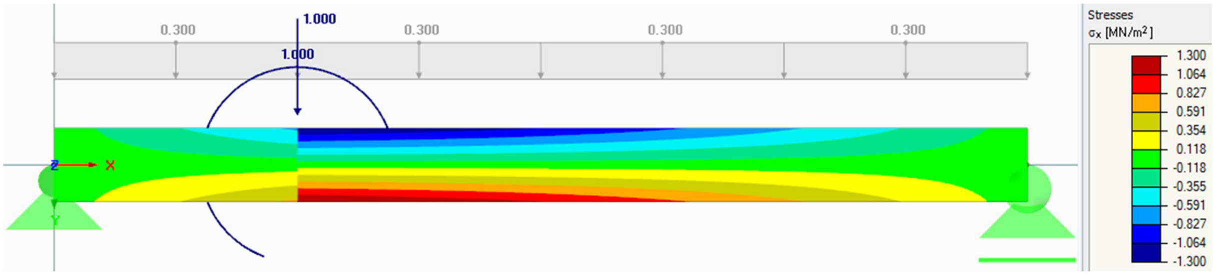


Figure 80. Normal stress due to all load cases.

3.14.6 Result Diagram

You can view the results of different quantities in one load case at the same time as follows: Select all members of the bar as active and then from the menu: Results -> Result Diagrams for Selected Members. In the pop-up Navigator, select u_y, φ_z, M_z, and V_y. Figure 81 shows the graphs in the case of a point load.

The numeric values of the graphs can be exported with an Excel icon (Export to MS Excel) for further processing in a spreadsheet program in *.xlsx or *.csv format.

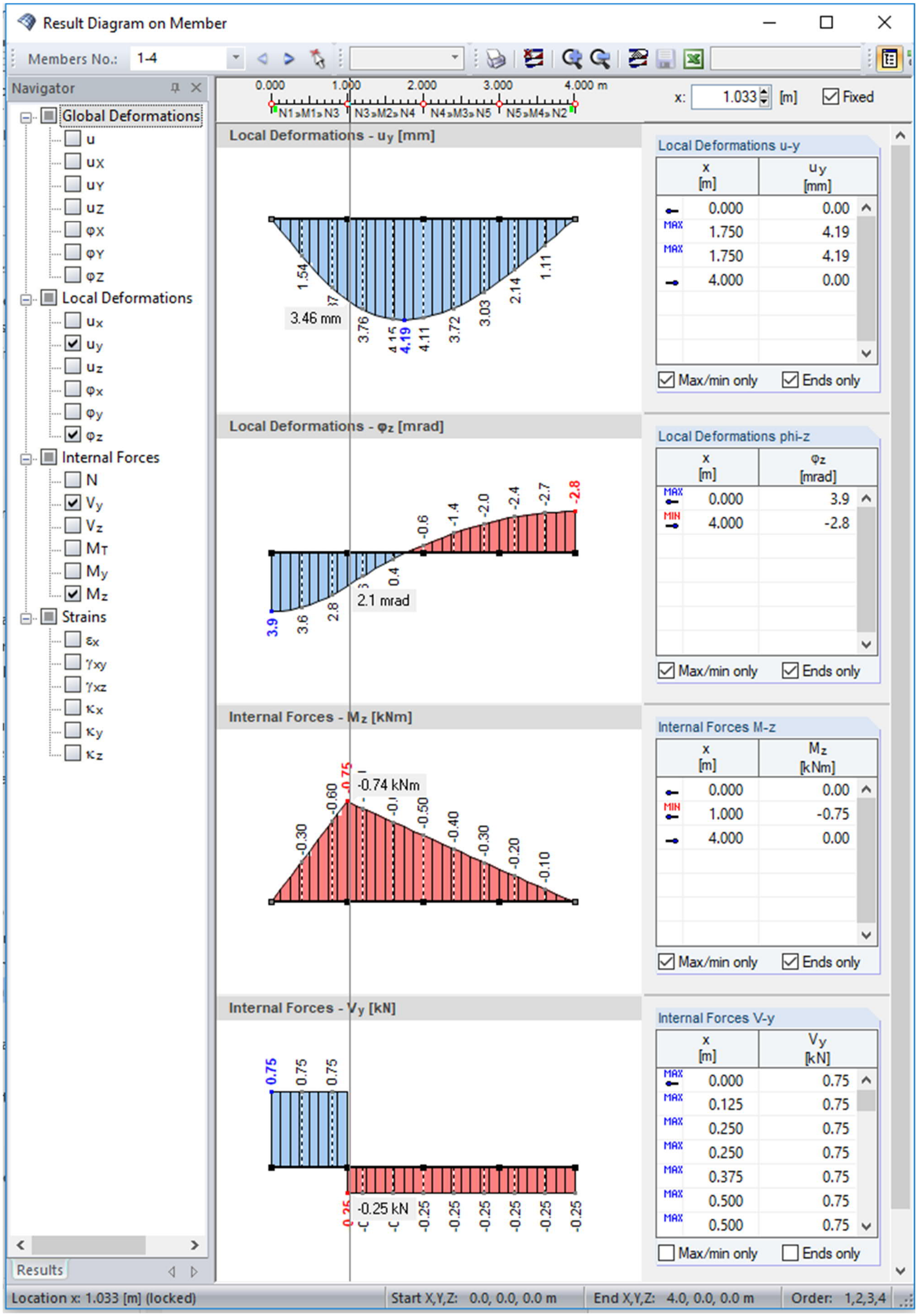


Figure 81. Result diagrams.

3.14.7 Animation

Select the load case, for example point load. Select deflection: Results → Global Deformations → u_y. Select: Results → Animation. End animation by Esc-button.

You can record a video (*.avi) as follows. Adjusts the setting: Options → Program Options → Graphics → OpenGL, remove Hardware acceleration selection. Press OK. Save (Ctrl+s). Close the program and open the model again.

Start the animation. Recording: Tools → Create Video File. Press  (Create Video File) and select the folder. The regarding is started by the red button and ended by the blue one. (Video display requires its own program.)

3.15 Validity

See the corresponding chapter in the Lecture Notes: ***Validity****.*

The validity of the results have always been verified by using some other method.

The use of simplified model and manual calculation method is extremely recommended.

First, the deflections ($v = u_y$) at point $x = a$ due to self-weight q, point load F and point moment M are calculated (*Mathcad*). Then sum of these. Further normal stress (σ_x) due to all load cases is calculated. Because normal stress has two values at point $x = a$, the value is calculated by the small distance δ to the right of this point (compare to Figure 80).

In the vertical direction, the stress is calculated at the lower surface of the beam: $y = e/2$.

Compare with each others the values calculated by Mathcad and FEM. The results should be the same to one decimal place. There is a small difference. The reason is, that in hand calculation Euler-Bernoulli theory is used (flat beam) and in FE analysis, Timoshenko theory is used (high beam with effect of shear force).

Self-weight

$$q := \gamma \cdot A \qquad q = 300 \cdot \frac{N}{m}$$

Deflection due to self-weight

$$v_q(x) := \frac{q \cdot L^4}{24 \cdot E \cdot I} \cdot \left(\frac{x}{L} - 2 \cdot \frac{x^3}{L^3} + \frac{x^4}{L^4} \right) \qquad v_q(a) = 3.17 \cdot mm$$

Deflection due to point load

$$v_F(x) := \frac{F \cdot L^2}{6 \cdot E \cdot I} \cdot \left| \begin{array}{ll} b \cdot \left[\left(1 - \frac{b^2}{L^2} \right) \frac{x}{L} - \frac{x^3}{L^3} \right] & \text{if } x \le a \\ a \cdot \left[-\frac{a^2}{L^2} + \left(2 + \frac{a^2}{L^2} \right) \frac{x}{L} - 3 \cdot \frac{x^2}{L^2} + \frac{x^3}{L^3} \right] & \text{if } x > a \end{array} \right.$$

$$v_F(a) = 3.33 \cdot mm$$

Deflection due to point moment

$$v_M(x) := \frac{M \cdot L^2}{6 \cdot E \cdot I} \cdot \left| \begin{array}{ll} \left(2 - 6 \cdot \frac{a}{L} + 3 \cdot \frac{a^2}{L^2} \right) \cdot \frac{x}{L} + \frac{x^3}{L^3} & \text{if } x \le a \\ -3 + 6 \cdot \frac{b}{L} - 3 \frac{b^2}{L^2} \ldots & \text{if } x > a \\ + \left(5 - 6 \cdot \frac{b}{L} + 3 \cdot \frac{b^2}{L^2} \right) \frac{x}{L} - 3 \cdot \frac{x^2}{L^2} + \frac{x^3}{L^3} & \end{array} \right.$$

$$v_M(a) = 2.22 \cdot mm$$

Deflection due to all loads

$$v_{sum}(x) := v_q(x) + v_F(x) + v_M(x) \qquad v_{sum}(a) = 8.72 \cdot mm$$

Bending moment due to all loads

$$M_{sum}(x) := -E \cdot I \cdot \frac{d^2}{dx^2} v_{sum}(x)$$

Dimension

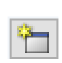
$$\delta := 0.01m$$

Axial stress due to all loads

$$\sigma_{sum}(x, z) := \frac{M_{sum}(x)}{I} z \qquad \sigma_{sum}\left(a + \delta, \frac{h}{2} \right) = 1.3 \cdot \frac{MN}{m^2}$$

3.16 Documentation

See the corresponding chapter in the Lecture Notes: ***Documentation****.*

Make a report: File → Open Printout Report… (Figure 82). Press icon  (New Printout Report).

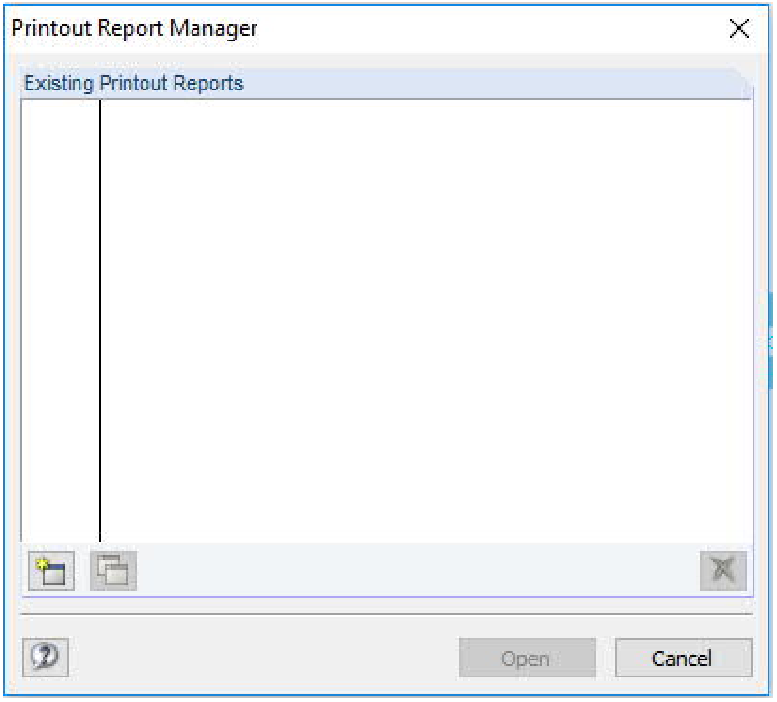


Figure 82. Report manager.

Press OK in the open window (Figure 83). The report manager has its own user interface.

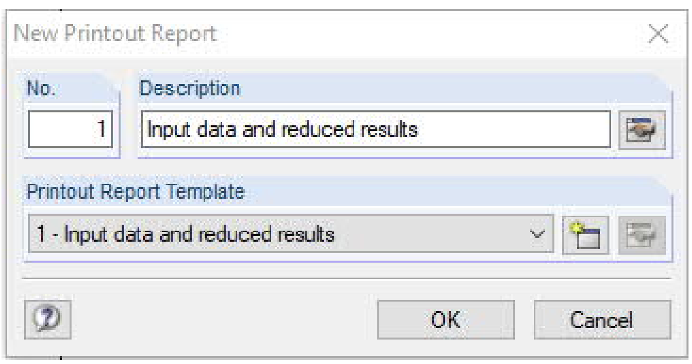


Figure 83. Creating a new report.

Add a cover page for the report: Settings → Cover. Fill the data as shown in Figure 84: project, client and engineer with student index. Press OK.

Save (File → Save) and exit (File → Exit) from the report manager.

Make the member numbering invisible: Right mouse button → Show Numbering. Another way: Navigator → Display → Numbering.

Add figures (distribution curves) to the report: File → Print Graphic. Update as shown in Figure 85 and press icon near the Mass print selection (Mass Print Settings).

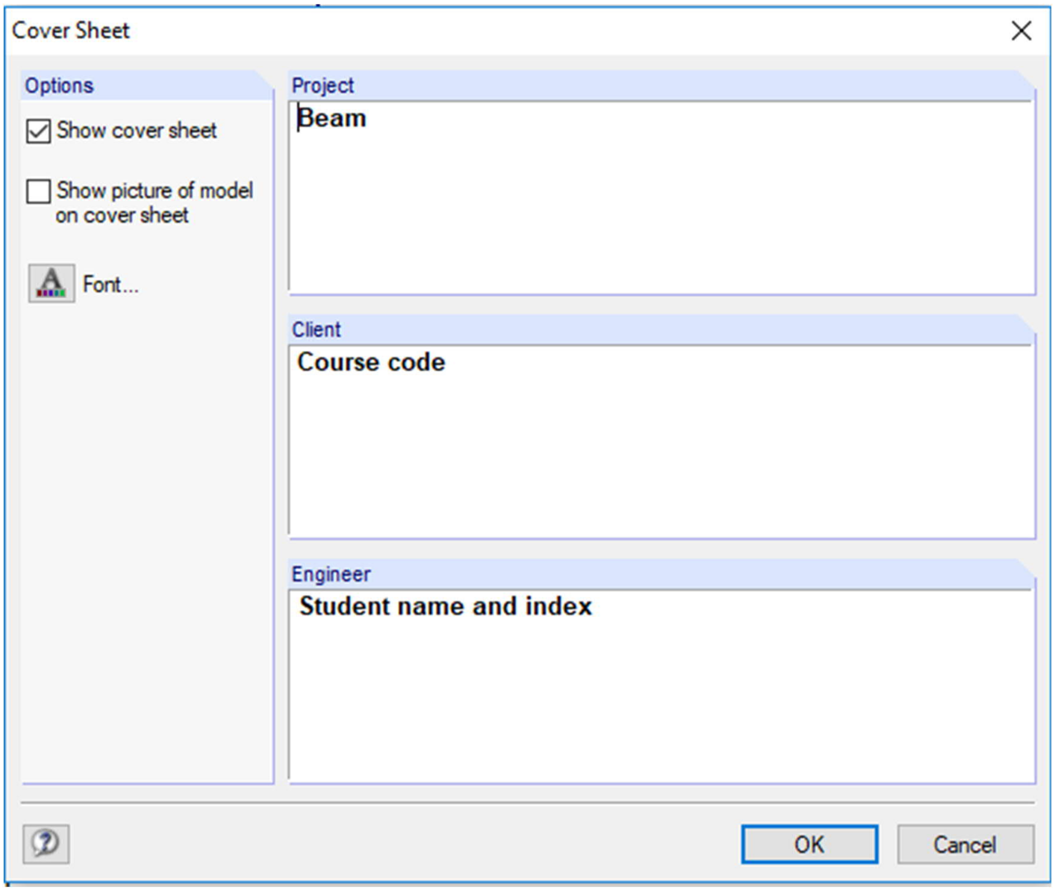


Figure 84. Cover sheet.

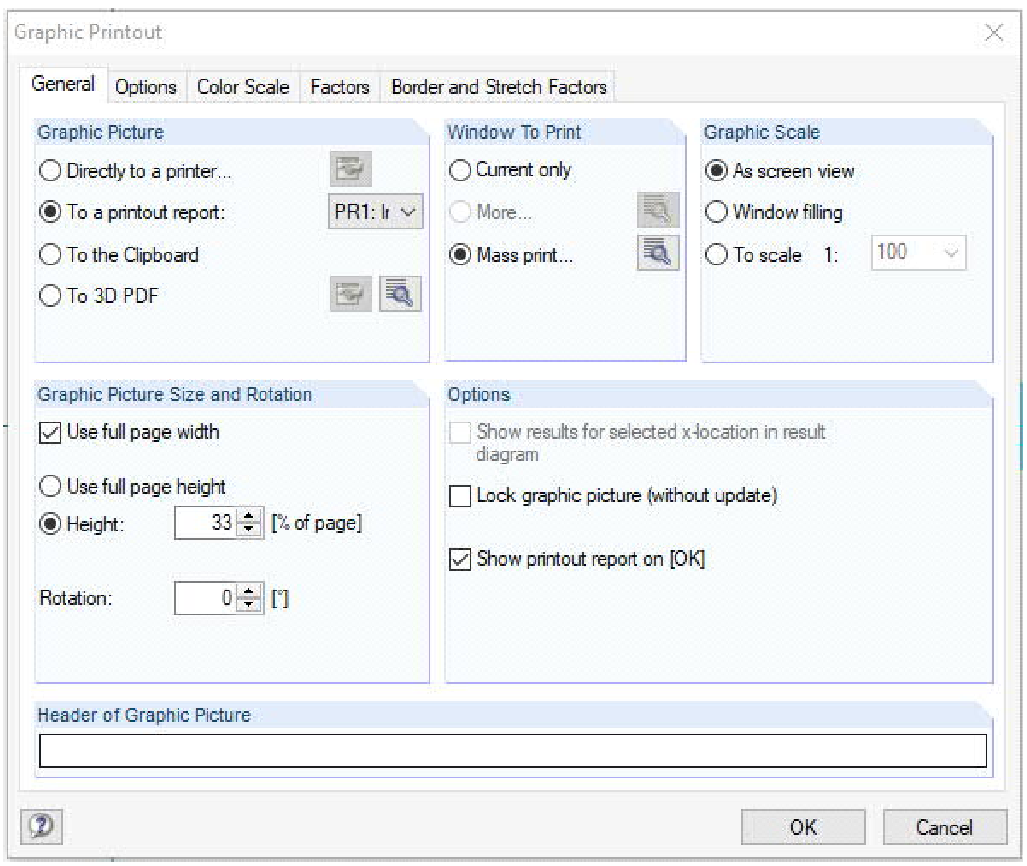


Figure 85. Graphic printout.

Update as shown in Figure 87 and 87 and press OK in the both open windows.

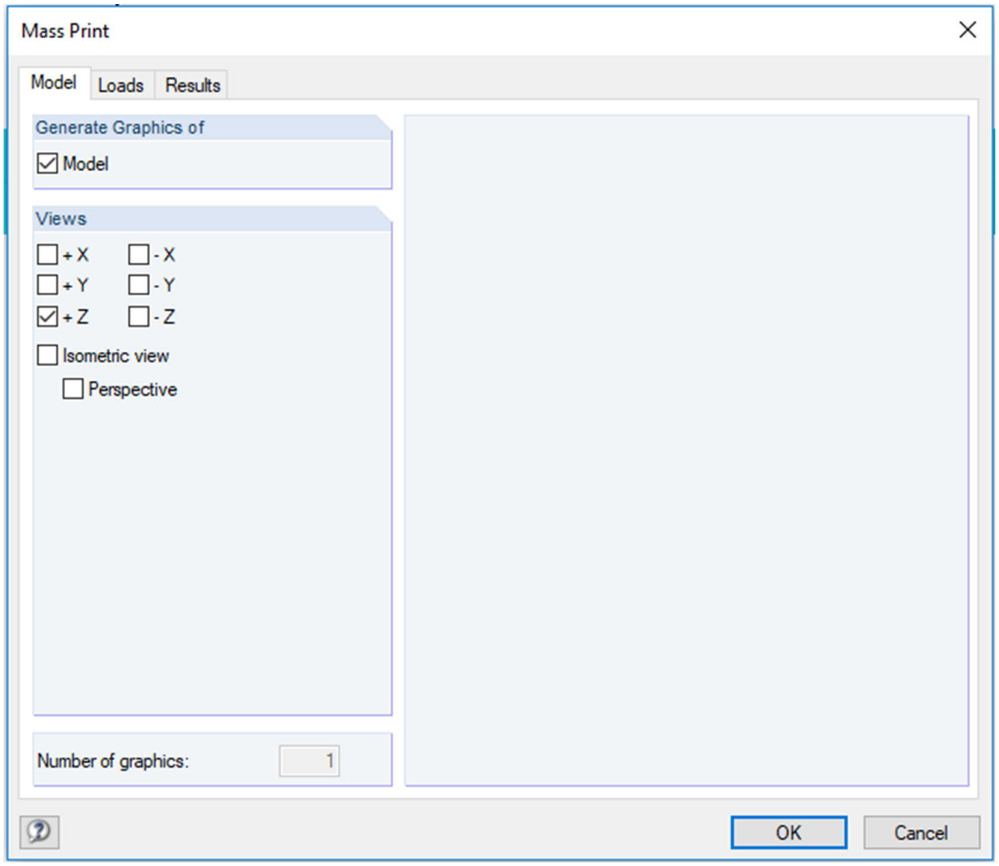


Figure 86. Mass print, model sheet.

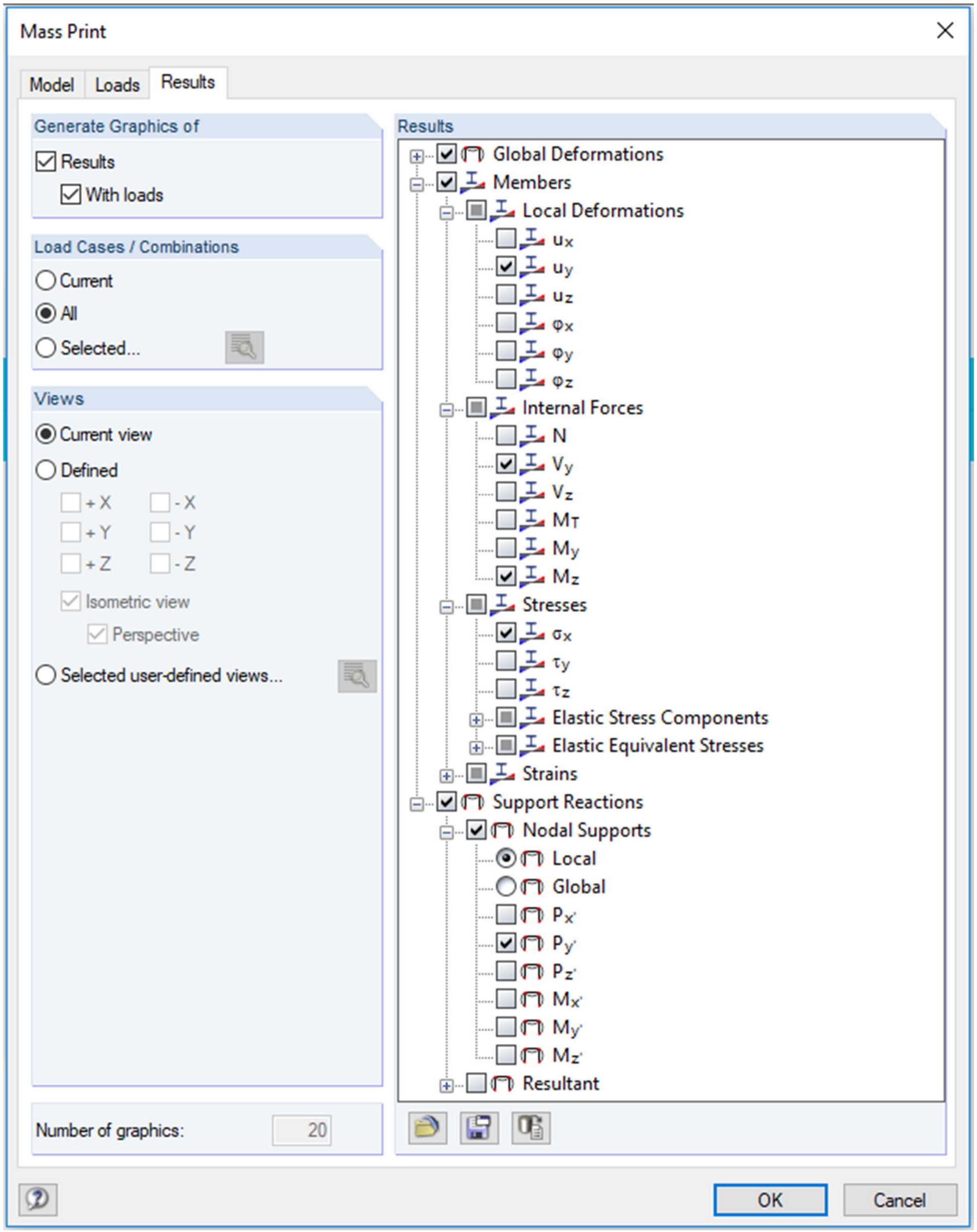


Figure 87. Mass print, results sheet.

Check, that the page size is A4 and that the page is in portrait orientation (File → Printer Setup).

If you want to remove some part of the report, use Delete button in Tree view.

Add a note about a different sing rule in the program: Insert → Text Block. Update as shown in Figure 88 and press OK.

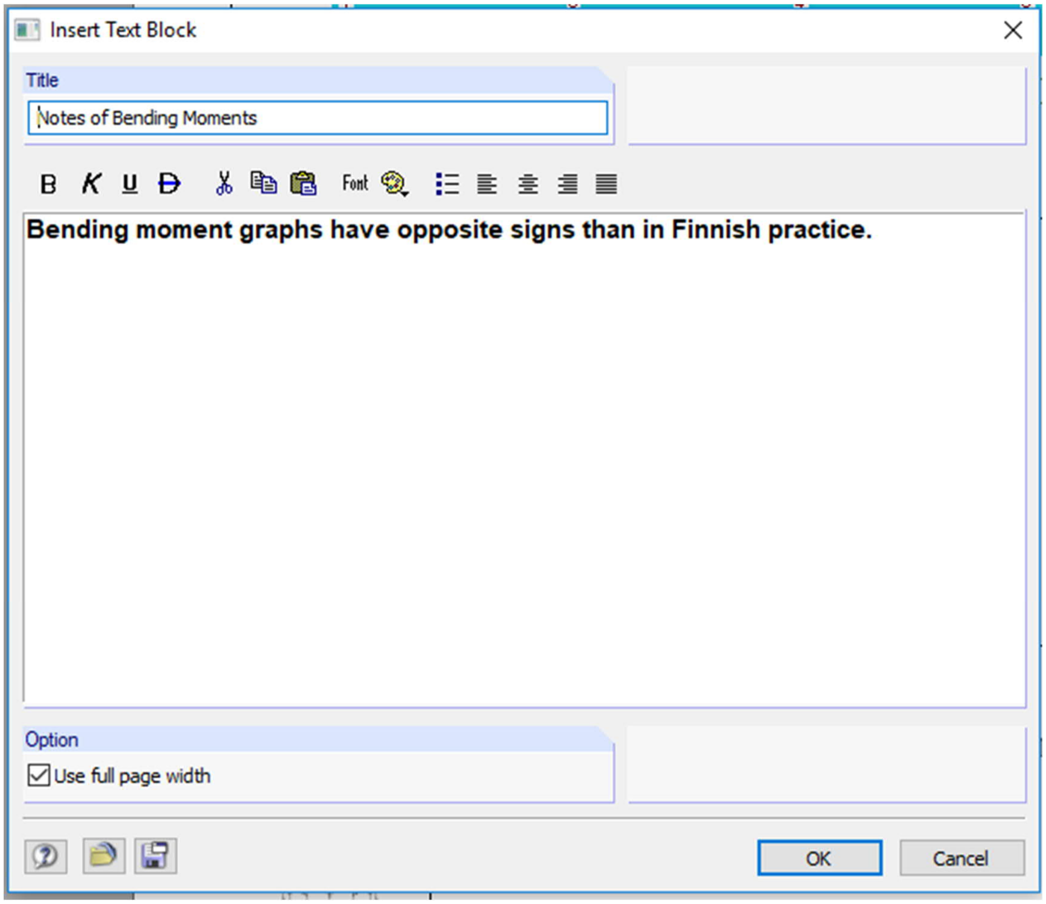


Figure 88. Note text.

Print the report to the pdf-file: File → Export to PDF.

4 Frame

4.1 Problem

Figure 89 shows a rigidly supported frame made of reinforced concrete. The joints between the columns (c) and the beam (b) are rigid. The frame is loaded by its dead weight (g_γ), uniformly distributed wind load (p) and snow load (q).

The task is to determine deflection (u, v), bending moment (M), shear force (V) and normal force (N) distribution curves.

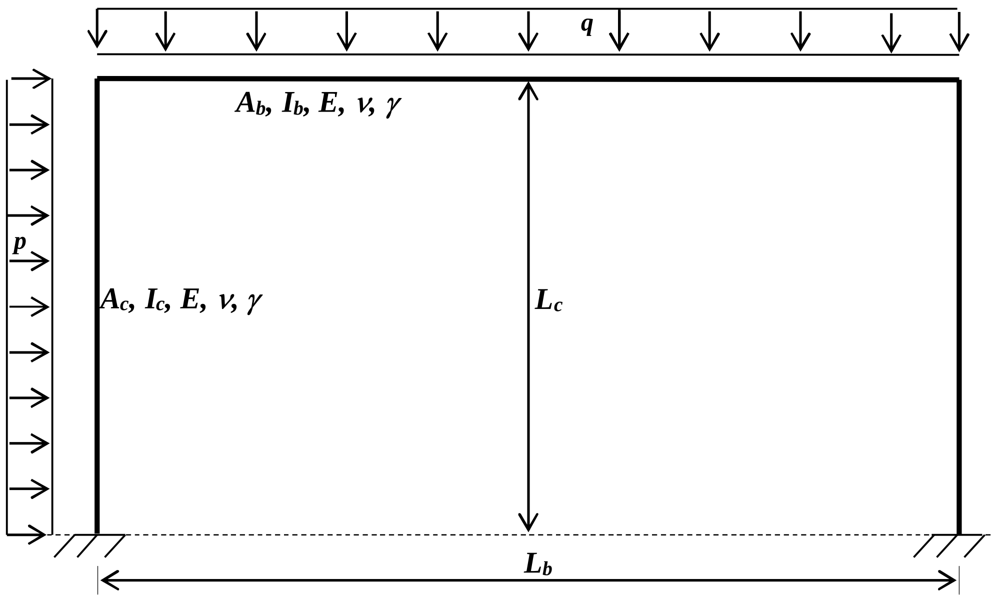


Figure 89. Frame.

Initial values are (*Mathcad*):

Dimensions

$$L_b := 12m$$

$$L_c := 6m$$

Snow and wind load (characteristic value)

$$q := 4\frac{kN}{m}$$

$$p := 1\frac{kN}{m}$$

4.2 Preliminary Plan

Figure 90 shows the elevation of the frame and the cross-sections of the structural parts, showing the dimensions of the structure, the specific loads acting on the frame, the support conditions and the coordinate system.

The coordinate system is in accordance with Finnish practice. The left support is at the origin. The deflection in the x-axis direction is u and in the y-axis direction v.

Material is isotropic and elastic (linear).

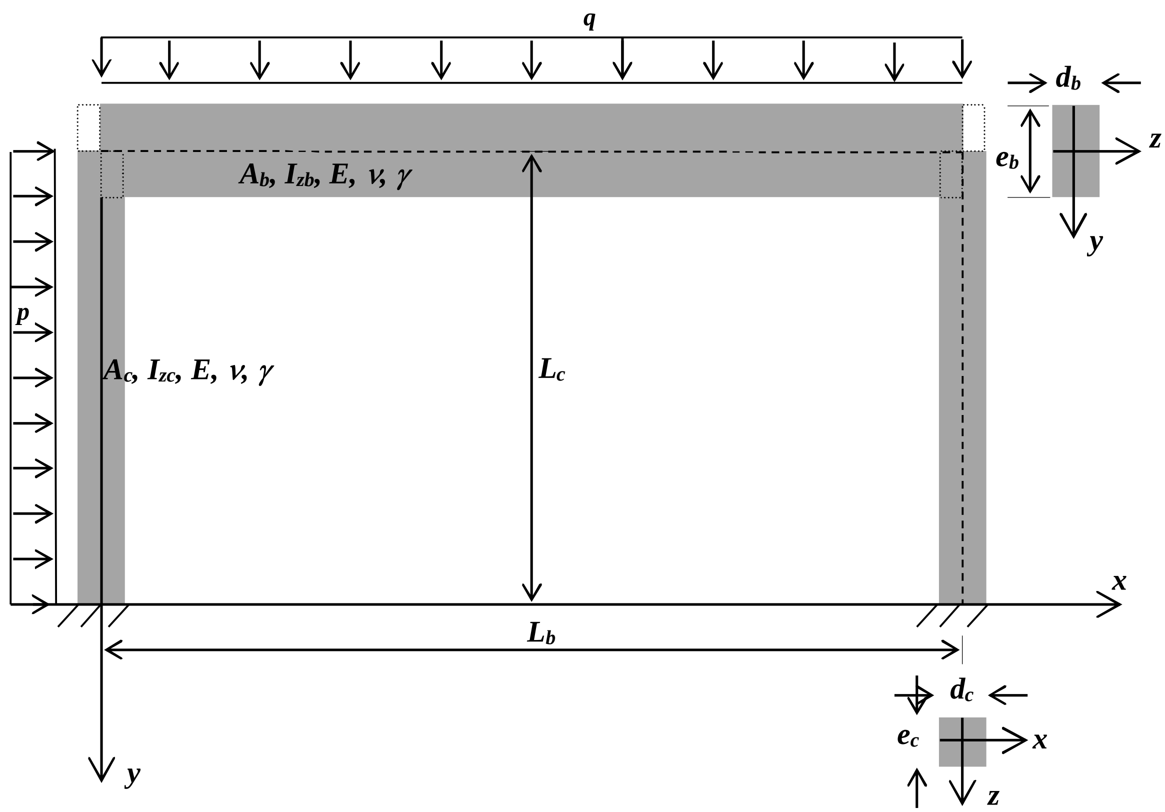


Figure 90. Preliminary plan of the frame.

The wind load is applied to the left column as shown in Figure 90. The length of the action of the load is equal to the length of the centre line of the component; the dashed white area at the junction of the column and the beam is disregarded. For dead weight, the grey area delimited by the dashed line is calculated twice (column + beam).

Each load is considered as its own load case. Depending on the cause of the load, the load cases (LC) are as follows:

- LC1 - Self weight, permanent load
- LC2 - Wind, variable load, line load
- LC3 - Snow, variable load, line load

The Eurocode EN 1990 and the Finnish national annex are used to combine the loads.

The problem can be solved in the plane case, in the x-y coordinate system. In RFEM, however, this 2D coordinate system in accordance with Finnish practice is not possible, so the 3D coordinate system is used.

One meter long elements are used in the beam and column.

Rectangular cross-sections and the following material properties and dimensions have been selected for the beam and columns (*Mathcad*):

Dimensions

$d_b := 0.2m$

$d_c := 0.2m$

$e_b := 0.6m$

$e_c := 0.2m$

Cross-sectional area and moment of inertia

$A_b := d_b \cdot e_b$ $\qquad A_b = 0.12 \cdot m^2$

$A_c := d_c \cdot e_c$ $\qquad A_c = 0.04 \cdot m^2$

$I_b := \dfrac{d_b \cdot e_b{}^3}{12}$ $\qquad I_b = 3.6 \times 10^{-3} \cdot m^4$

$I_c := \dfrac{d_c \cdot e_c{}^3}{12}$ $\qquad I_c = 1.333333 \times 10^{-4} \cdot m^4$

Modulus of elasticity, Poisson's ratio and shear modulus

$E := 36000 \dfrac{MN}{m^2}$

$\nu := 0.2$

$G := \dfrac{E}{2(1+\nu)}$ $\qquad G = 1.5 \times 10^4 \cdot \dfrac{MN}{m^2}$

Unit weight

$\gamma = 25 \cdot \dfrac{kN}{m^3}$

4.3 Modelling

Give a name for the model, for example: B3_Frame.

Set active 3D-coordinate system, standard EN 1990 and national annex SFS as well as automatic load combining (Figure 91).

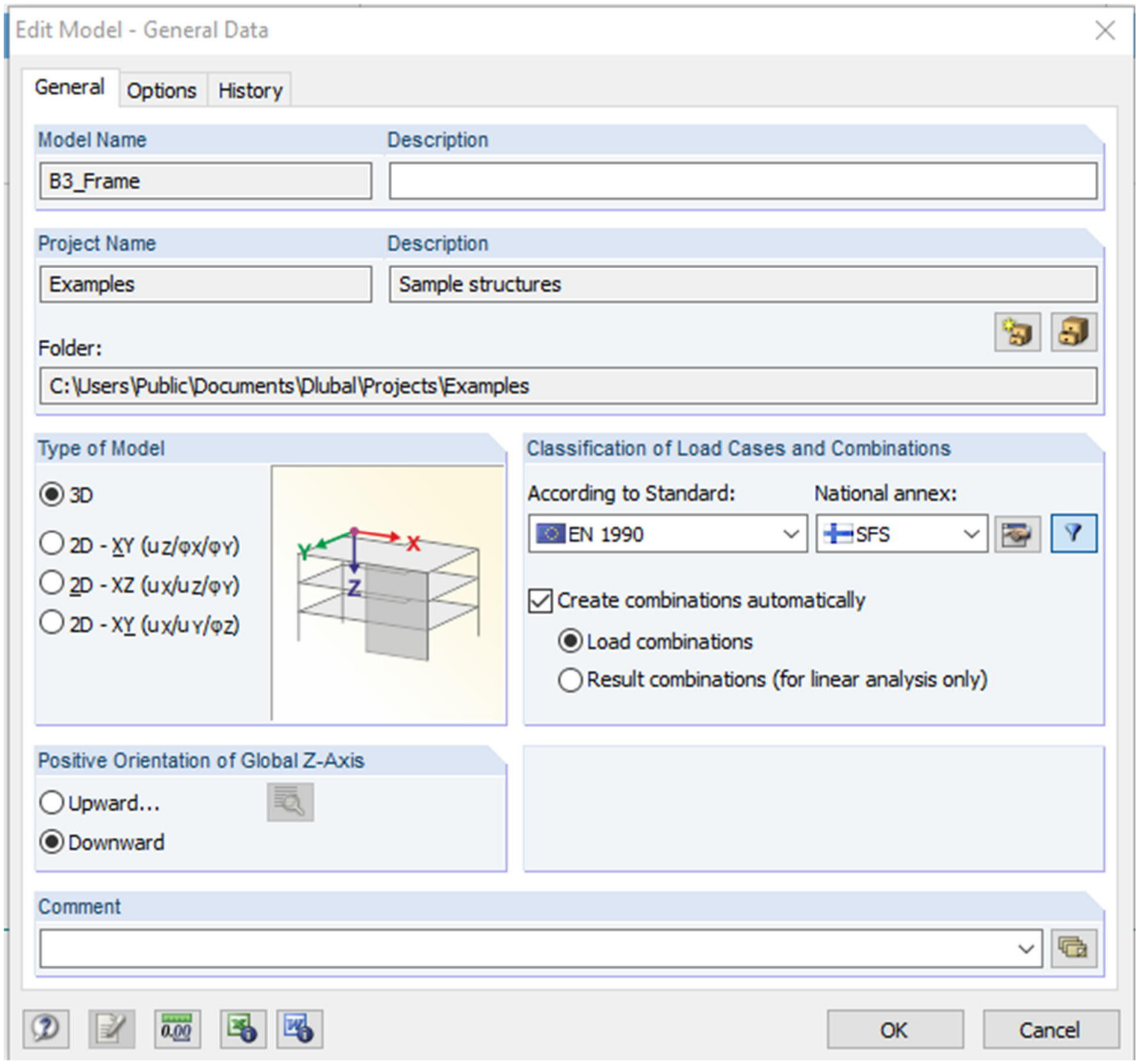


Figure 91. General data.

Create a finite element model, review it, and analyze by applying the instructions previously provided in this guide. Some additional advice is provided in this chapter.

4.3.1 Joints

A fixed joint is automatically placed between the components meeting at the same point (Figure 92).

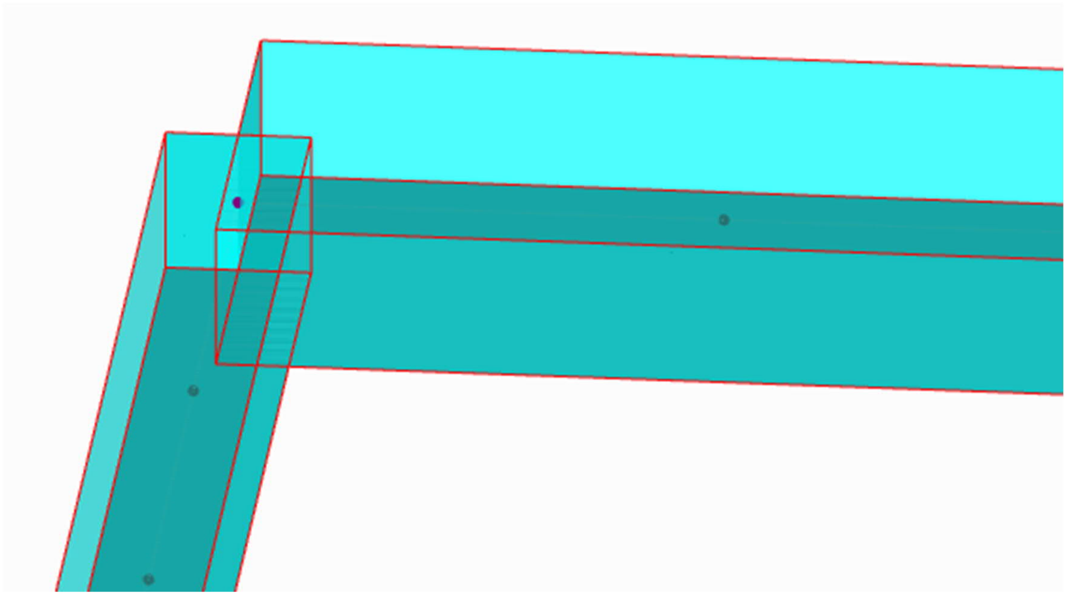


Figure 92. Joint of the column and beam.

4.3.2 Supports

With rigid support, all displacements and rotations are prevented (Figure 93). At the beginning and end of the column, the support should be rotated in different directions so that they appear to be at the same level (compare to Figure 96).

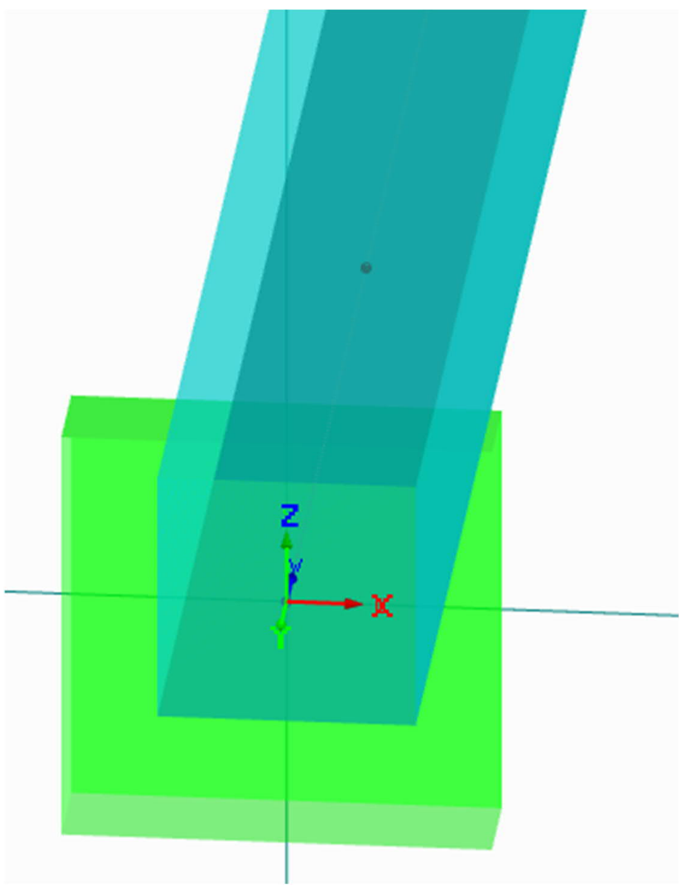


Figure 93. Support for the column.

4.3.3 Loads and Load Combination

Create the load cases:

- LC1 - Self-weight, permanent load
- LC2 - Wind, transient, uniform, member load,)
- LC3 - Snow, transient, uniform, member load, $s_k < 2{,}75$ kN/m^2).

Use Member load for a uniformly distributed load; it affects on members. Line loads are forces and moments that act on lines; it can only be used when the line belongs to a surface.

Use drop-down lists ”Load Case Description” and ”Action Category” when creating the load cases (Figure 94).

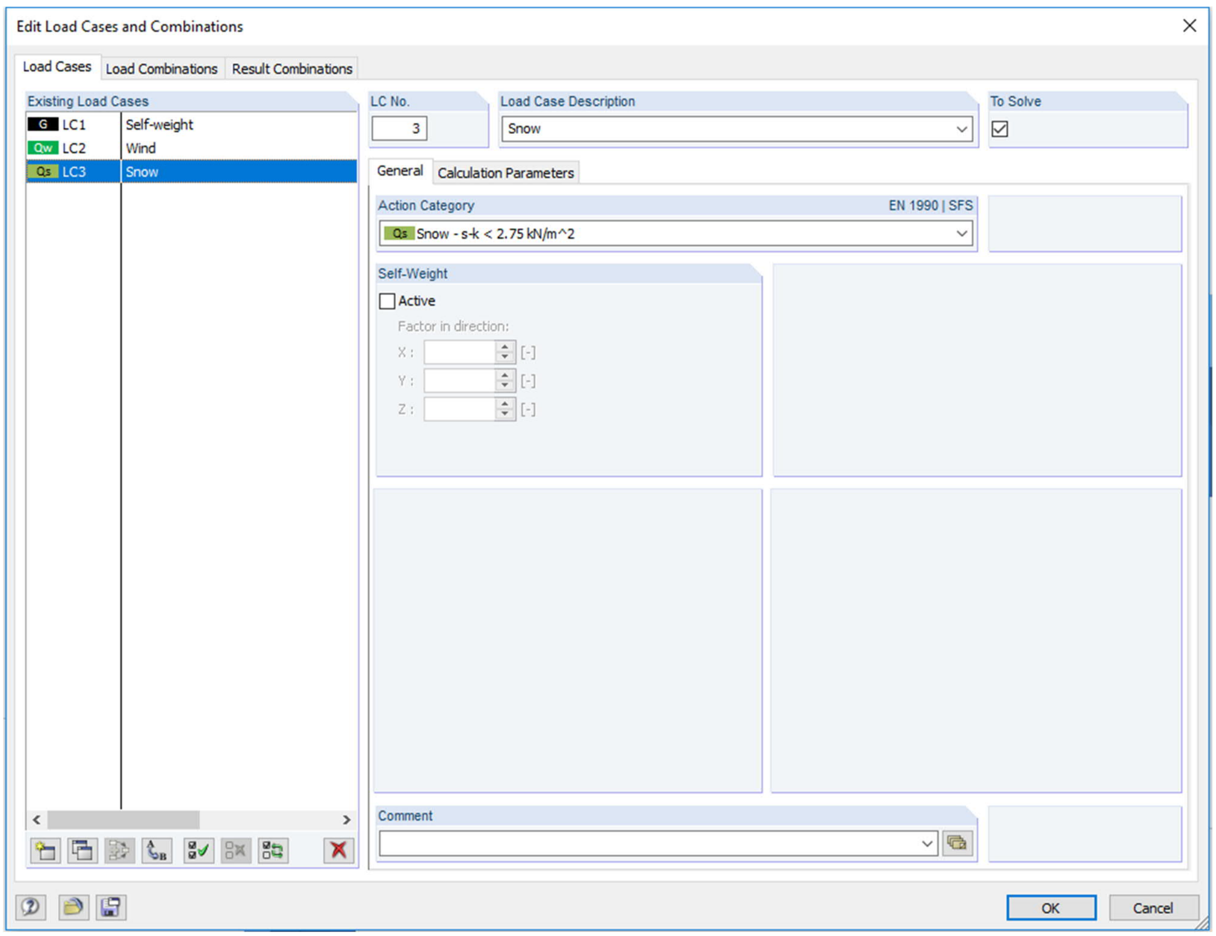


Figure 94. Load cases.

Combine the loads: Insert → Load Cases and Combinations → Load Combinations. The program automatically creates the combination according to the given standards. Do not add other combinations!

Figure 95 shows each load case on its own line (CO – Combination). Compare with the corresponding example in the lectures.

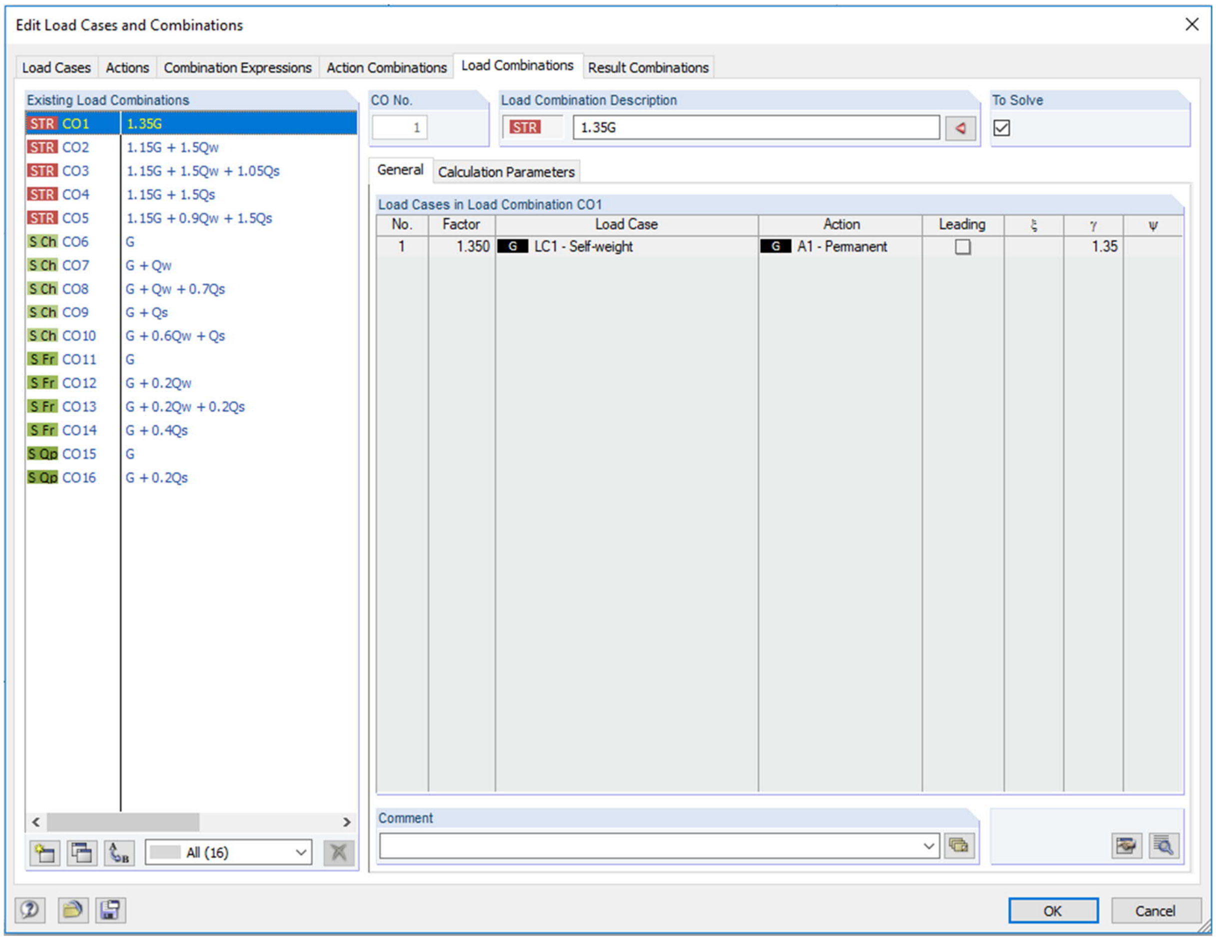


Figure 95. Load combination.

Figure 96 shows the modelled frame in one particular load case.

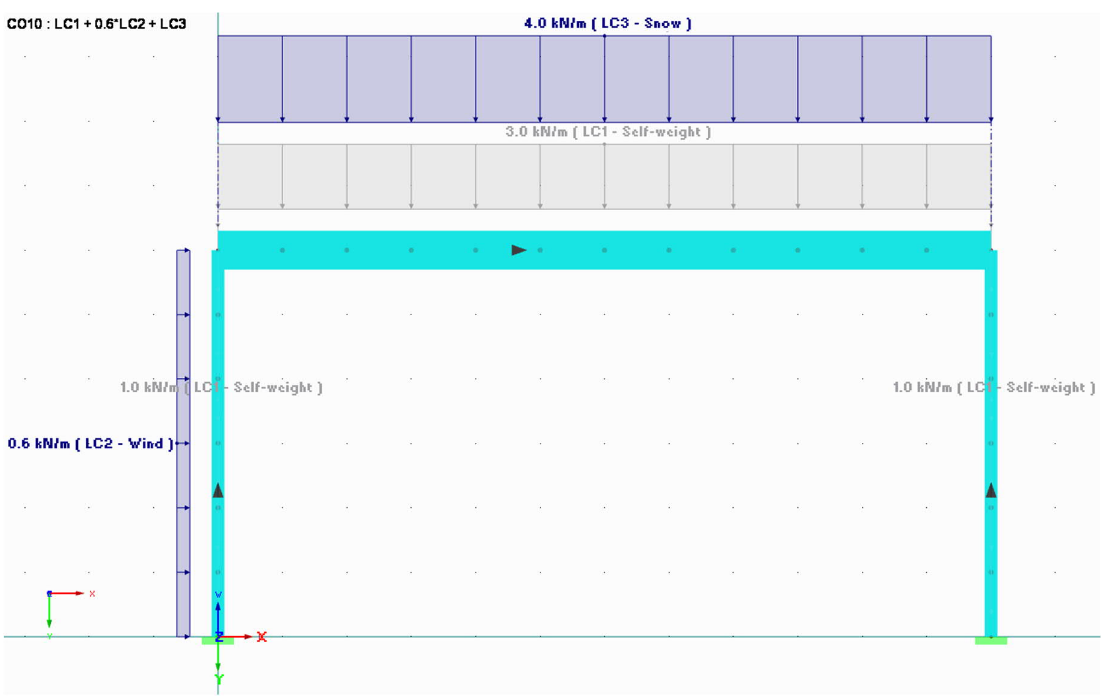


Figure 96. FE-model of the frame.

4.4 Results

In the case of snow load, consider the following: deflection (Figure 97), bending moment (Figure 98), shear force (Figure 99), and normal force (Figure 100).

Positive normal force means tension, negative compression.

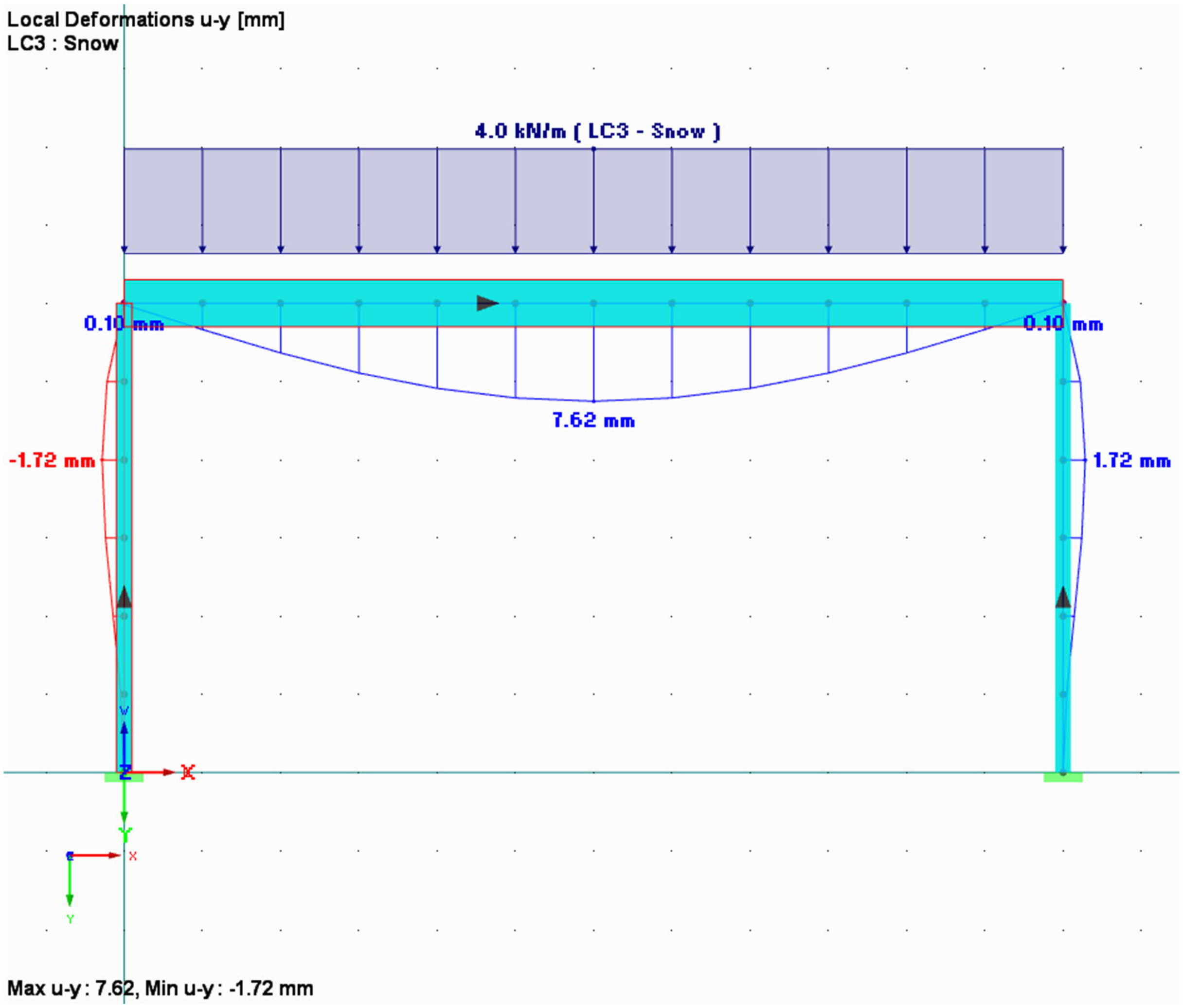


Figure 97. Deflection due to snow load.

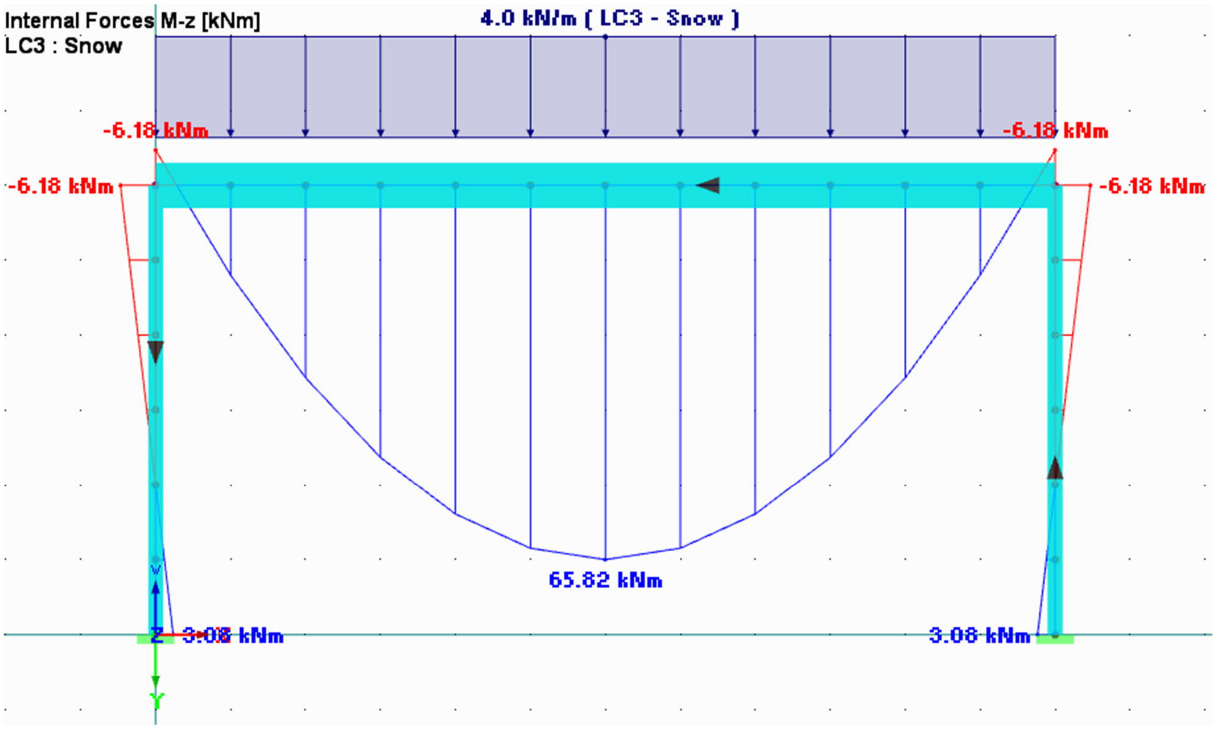


Figure 98. Bending moment diagram of snow load.

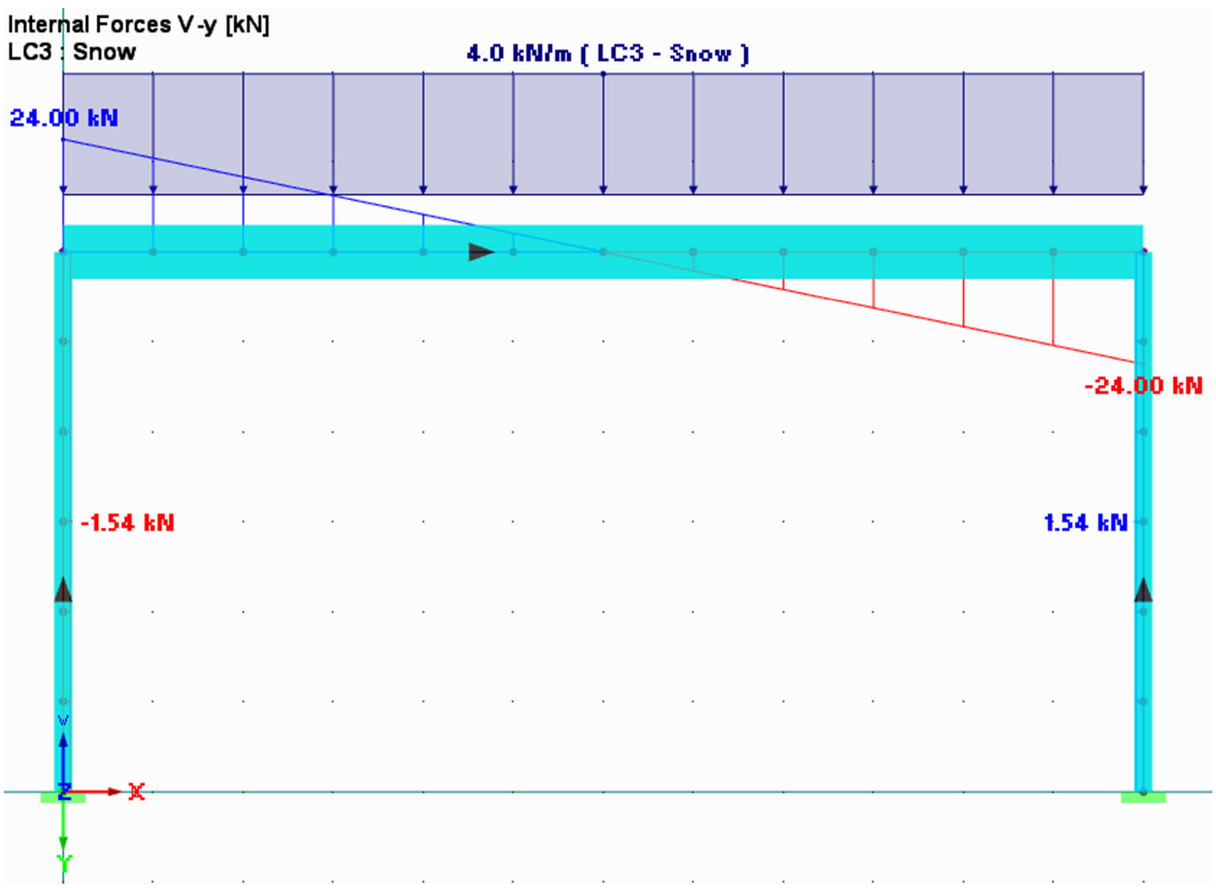


Figure 99. Shear force diagram of snow load.

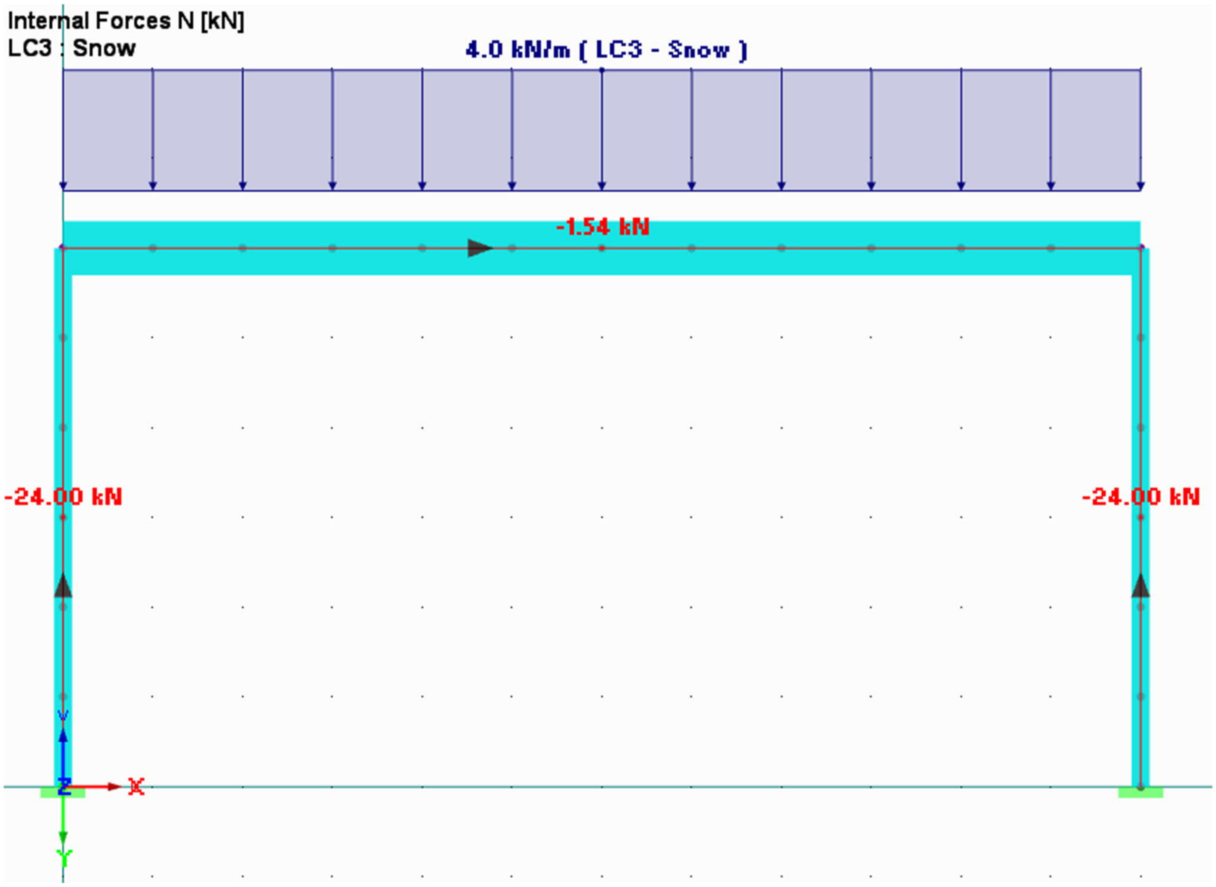


Figure 100. Normal force diagram of snow load.

4.5 Validity

The bending moment (M), shear force (V) and normal force (N) from the snow load q **alone** will be calculated. The load is then symmetrical and the shear force at the centre of the beam is zero (compare to beam example).

A statically determined structure is done by cutting the structure in the middle of the beam. Unknown forces at this point are bending moment (X_1) and normal force (X_2). The unknown forces is solved by the unit force method, ignoring the effect of normal and shear forces when calculating the displacement variables (δ_{ij}). In the unit force method, the unknown generalized force quantity is initially set to one (Figure 101).

Once the unknown forces have been resolved, the bending moment, shear force and normal force are calculated. Finally, the deflection (v) of the centre of the beam is calculated using the expression for the bending moment of the beam.

Compare the numerical values obtained with Mathcad and FEM in case of snow load! The results should be the same to one decimal place. The slight difference is due to the fact that classical beam theory (Euler & Bernoulli, low beams) has been used in the manual calculation, while the beam element also takes into account the effect of shear force (Timoshenko, high beams).

Similarly, in the case of other individual loads, consider the correctness of the **shape** of each graph! (For numerical values, the correctness has not been checked by hand, except in the case of snow load. For self-weight, the results would be obtained in the same way as for snow load. The shear force caused by wind load in the middle of the beam is not zero; there are three unknown forces.)

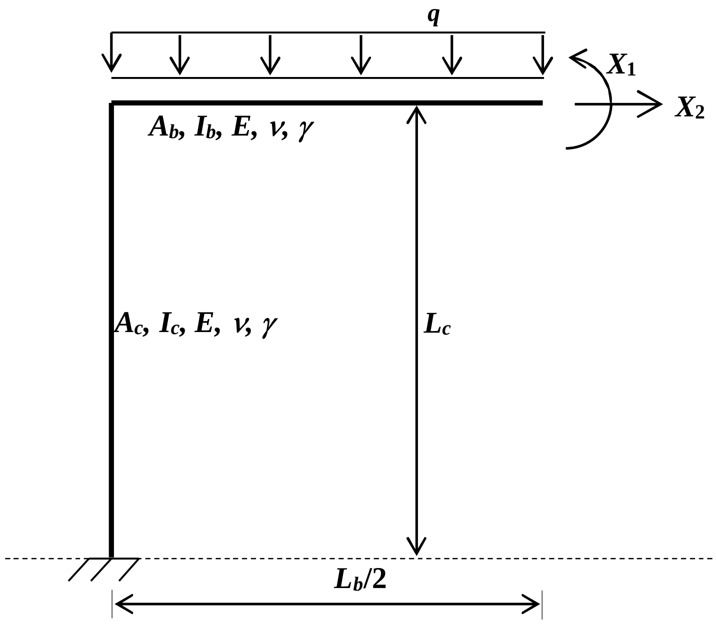


Figure 101. Unknown generalized forces.

Bending moments of a statically determined structure due to external load in the beam and column

$$M_{0b}(x) := \frac{-q}{2} \cdot \left(\frac{L_b}{2} - x \right)^2$$

$$M_{0c} := M_{0b}(0)$$

Bending moments due to unit moment

$$M_{1b} := 1$$

$$M_{1c} := 1$$

Bending moments due to unit force

$$M_{2b} := 0$$

$$M_{2c}(y) := y - L_c$$

Load vector elements

$$\delta_{10} := \int_0^{\frac{L_b}{2}} \frac{M_{1b} \cdot M_{0b}(x)}{E \cdot I_b} \, dx + \int_0^{L_c} \frac{M_{1c} \cdot M_{0c}}{E \cdot I_c} \, dx \qquad \delta_{10} = -0.091111$$

$$\delta_{20} := \int_0^{\frac{L_b}{2}} \frac{M_{2b} \cdot M_{0b}(x)}{E \cdot I_b} \, dx + \int_0^{L_c} \frac{M_{2c}(y) \cdot M_{0c}}{E \cdot I_c} \, dy \qquad \delta_{20} = 0.27\,m$$

The elements of the coefficient matrix

$$\delta_{11} := \int_0^{\frac{L_b}{2}} \frac{M_{1b}^2}{E \cdot I_b}\,dx + \int_0^{L_c} \frac{M_{1c}^2}{E \cdot I_c}\,dy \qquad \delta_{11} = 1.296296 \times 10^{-6} \cdot \frac{1}{N \cdot m}$$

$$\delta_{12} := \int_0^{\frac{L_b}{2}} \frac{M_{1b} \cdot M_{2b}}{E \cdot I_b}\,dx + \int_0^{L_c} \frac{M_{1c} \cdot M_{2c}(y)}{E \cdot I_c}\,dy \qquad \delta_{12} = -3.75 \times 10^{-6}\,\frac{1}{N}$$

$$\delta_{21} := \delta_{12} \qquad \delta_{21} = -3.75 \times 10^{-6}\,\frac{1}{N}$$

$$\delta_{22} := \int_0^{\frac{L_b}{2}} \frac{M_{2b}^2}{E \cdot I_b}\,dx + \int_0^{L_c} \frac{\left(M_{2c}(y)\right)^2}{E \cdot I_c}\,dy \qquad \delta_{22} = 1.5 \times 10^{-5} \cdot \frac{m}{N}$$

Generalized forces in the middle of of the beam

Given

$$X_1 \cdot \delta_{11} + X_2 \cdot \delta_{12} + \delta_{10} = 0$$

$$X_1 \cdot \delta_{21} + X_2 \cdot \delta_{22} + \delta_{20} = 0$$

$$\mathrm{Find}\left(X_1, X_2\right) \rightarrow \begin{pmatrix} 0.065806451612903225809 MN \cdot m \\ -0.0015483870967741935 48 MN \end{pmatrix}$$

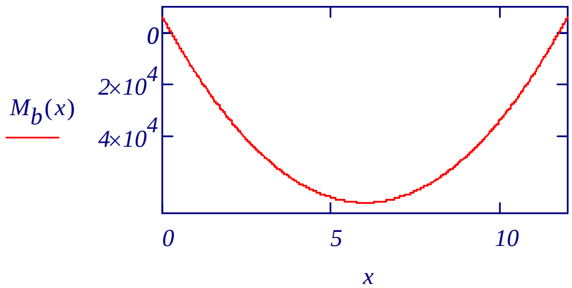
Insert the solved force quantities as values of the variables

$$X_1 := 0.065806451612903225809 MN \cdot m$$

$$X_2 := -0.001548387096774193548 MN$$

Bending moment in the beam and left column

$$M_b(x) := M_{0b}(x) + X_1 \cdot M_{1b} + X_2 \cdot M_{2b}$$

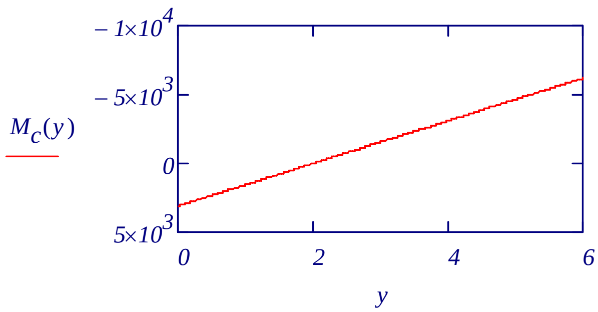


$M_b(0) = -6.193548 \cdot kN \cdot m$

$M_b\left(\dfrac{L_b}{2}\right) = 65.806452 \cdot kN \cdot m$

$M_b\left(L_b\right) = -6.193548 \cdot kN \cdot m$

$$M_c(y) := M_{0c} + X_1 \cdot M_{1c} + X_2 \cdot M_{2c}(y)$$

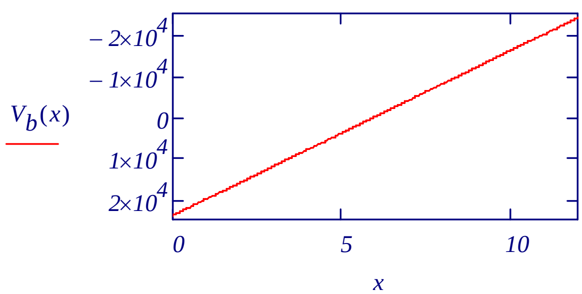


$M_c(0) = 3.096774 \cdot kN \cdot m$

$M_c\left(L_c\right) = -6.193548 \cdot kN \cdot m$

Shear force in the beam and left column

$$V_b(x) := \frac{d}{dx} M_b(x)$$

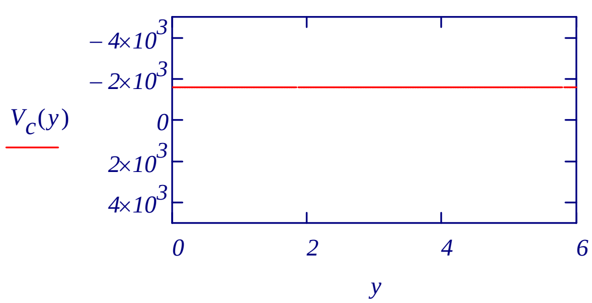


$V_b(0) = 24 \cdot kN$

$V_b\left(\dfrac{L_b}{2}\right) = 0 \cdot kN$

$V_b\left(L_b\right) = -24 \cdot kN$

$$V_c(y) := \frac{d}{dy} M_c(y)$$



$V_c(0) = -1.548387 \cdot kN$

Normal forces in the beam and left column

$N_b := X_2$ $\qquad N_b = -1.548387 \cdot kN$

$N_c := \frac{-q \cdot L_b}{2}$ $\qquad N_c = -2.4 \times 10^4 N$

Bending moment in the beam

$$M_b(x) := \frac{-q}{2} \cdot \left(\frac{L_b}{2} - x\right)^2 + X_1$$

Rotation of the beam

$$\varphi_b(C_1, x) := \int \frac{-M_b(x)}{E \cdot I_b} \, dx + C_1$$

Deflection of the beam

$$v_b(C_1, C_2, x) := \int \varphi_b(C_1, x) \, dx + C_2$$

Boundary conditions at the beam ends

Given

$v_b(C_1, C_2, 0) = 0$

$v_b(C_1, C_2, L_b) = 0$

Solving the integral constants

$$C := Find(C_1, C_2) \rightarrow \begin{pmatrix} 0.0019354838709677419356 \\ 0 \end{pmatrix}$$

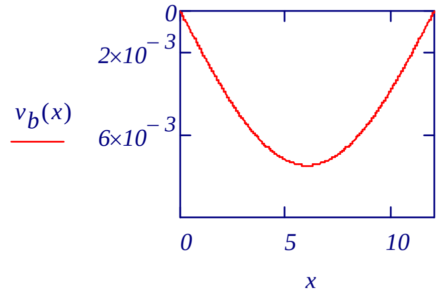
Insert the solved constants as values of the variables

$C_1 := C_0$ $\qquad C_1 = 1.935484 \times 10^{-3}$

$C_2 := C_1$ $\qquad C_2 = 0$

Deflection

$$v_b(x) := C_1 \cdot x + \frac{x^2 \cdot \left(L_b{}^2 \cdot q - 8 \cdot X_1\right)}{16 \cdot E \cdot I_b} + \frac{q \cdot x^4}{24 \cdot E \cdot I_b} - \frac{L_b \cdot q \cdot x^3}{12 \cdot E \cdot I_b}$$



$v_b(0) = 0m$

$v_b\left(\frac{L_b}{2}\right) = 7.473118 \times 10^{-3} m$

$v_b(L_b) = 0m$

4.6 Documentation

Make a report. Add the following to the report that the program has generated automatically:

- Cover page stating the following: name of the assignment, course, author's name and index.
- Deflection graph u_y of wind load
- Normal force graph N of self-weight.
- Deflection curve u (Global deformation) for the ultimate limit load case CO4: 1.15 * LC1 + 1.5 * LC3.
- Shear force graph V_y in the case of ultimate limit load case CO4.
- Bending moment graph M_z for the ultimate limit state load case CO4.

Do not add result pictures other than those mentioned above!

5 Truss

5.1 Problem

Fig. 102 shows a space truss formed by a cantilever, which is supported at three points. The truss bars are made of steel; the cross section and material are the same for all bars. The length of the eight bars is *a*. The cantilever truss is loaded at its end by a point load *P*.

The task is to determine the normal forces (N_i) and the deflection *v* at the point to which the point load is applied. The effect of self weight is not taken into account.

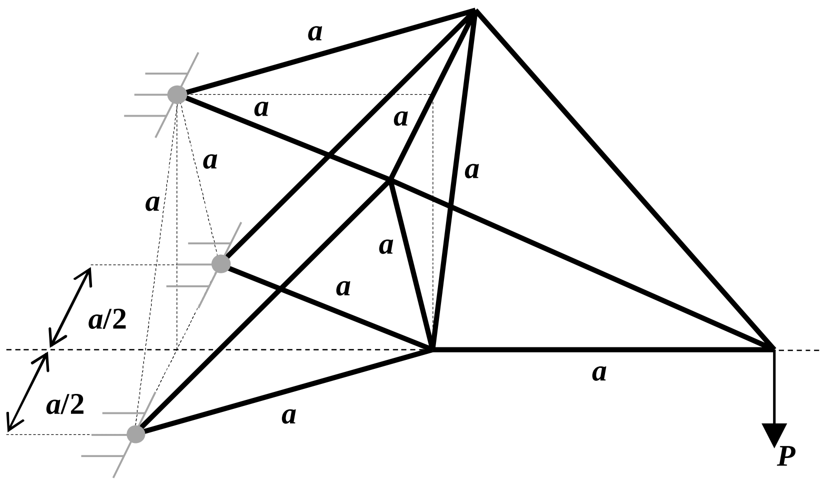


Figure 102. Truss.

Initial values are (*Mathcad*):

Dimension

$a := 2m$

Point load (characteristic value)

$P := 10kN$

5.2 Preliminary Planing

The truss consists of bars. A bar is a structure in which only axial force (or torque) parallel to the bar can occur. In the truss, the ends of the bars are assumed to be connected by a hinges to each other and to the supports.

Several triangles have been used in the truss of the problem. The triangles stiffens the structure so that it does not create a mechanism. If the structure is a mechanism, the group of equations describing the element model cannot be solved.

The coordinate system is determined and the corner points are marded with the letters A… G (Fig. 103). It is observed that the supports forms an equilateral triangle ABC; each side dimension is *a*. The truss is symmetric with respect to the *x*-*y* plane, which information is utilized in the check calculation.

The material is isotropic and ideally elastic (linearly elastic). Each load is considered as its own load case. The self weight is included in the calculation, although it is not needed to solve the problem. All safety factors are ones.

For the bars, Ruukki's steel pipe profile CHS 101.6x5, which is made of steel S 235, is selected from the cross-sectional library of the program (*Mathcad*):

Cross-section dimensions

$$D := 0.1016m$$

$$t := 0.005m$$

Modulus of elasticity

$$E := 210000\frac{MN}{m^2}$$

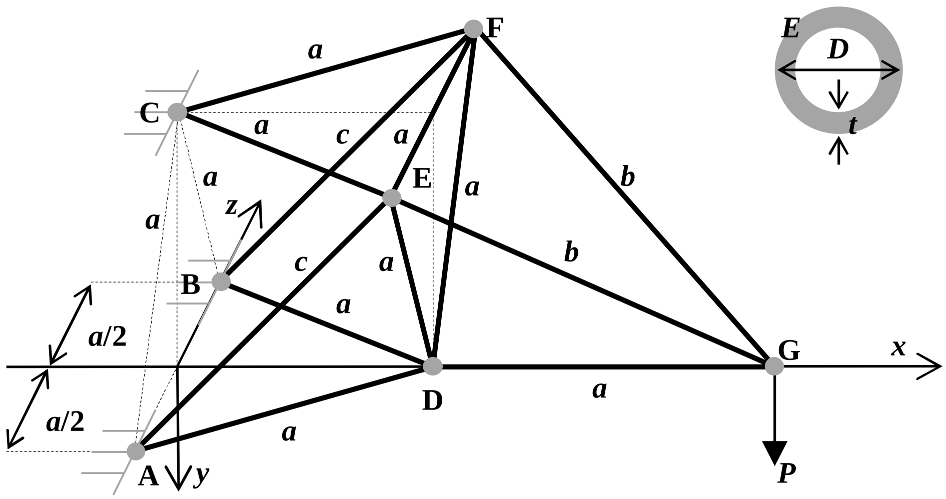


Figure 103. Preliminary plan of the truss.

Calculating gives the cross-section area and lengths of the rods (*Mathcad*):

Cross-section area of the bar

$$A := \pi\cdot\left[\left(\frac{D}{2}\right)^2 - \left(\frac{D-2\cdot t}{2}\right)^2\right] \qquad A = 1.517389\times 10^{-3}\cdot m^2$$

Dimensions

$$b := \sqrt{2}\cdot a \qquad b = 2.828427m$$

$$c := \sqrt{\left(\sqrt{3}\right)^2 + \left(\sqrt{3}\right)^2 + \left(\frac{1}{2}\right)^2}\cdot\frac{a}{2} \qquad c = 2.5m$$

Length and th number of the bars (note symmetry)

$$L_{AD} := a \qquad n_{AD} := 2 \qquad L_{AD} = 2m$$

$$L_{AE} := c \qquad n_{AE} := 2 \qquad L_{AE} = 2.5m$$

$$L_{CE} := a \qquad n_{CE} := 2 \qquad L_{CE} = 2m$$

$$L_{DE} := a \qquad n_{DE} := 2 \qquad L_{DE} = 2m$$

$$L_{DG} := a \qquad n_{DG} := 1 \qquad L_{DG} = 2m$$

$$L_{EG} := b \qquad n_{EG} := 2 \qquad L_{EG} = 2.828427m$$

$$L_{EF} := a \qquad n_{EF} := 1 \qquad L_{EF} = 2m$$

5.3 Modelling

Name the model, for example: Truss.

Select 3D coordinate system.

Create a finite element model, review it, and analyze by applying the instructions previously provided in this guide. Some additional advice is provided in this chapter.

5.3.1 Bar

Select Truss (only N) as the member type. In this case, only normal force acts on the bar and there are hinges at the ends of the bar.

5.3.2 Truss

First, use the bars to define the auxiliary triangle ABC connecting the support points. The bars will be removed later. To help with the definition, select Y-Z as the work surface: Tools → Work Plane…

Use the “Length and direction” feature in the New Member window to define the bars of same lengths. See Figure 104. When you move the cursor to the work area, you will see the direction of the angle specified in the window.

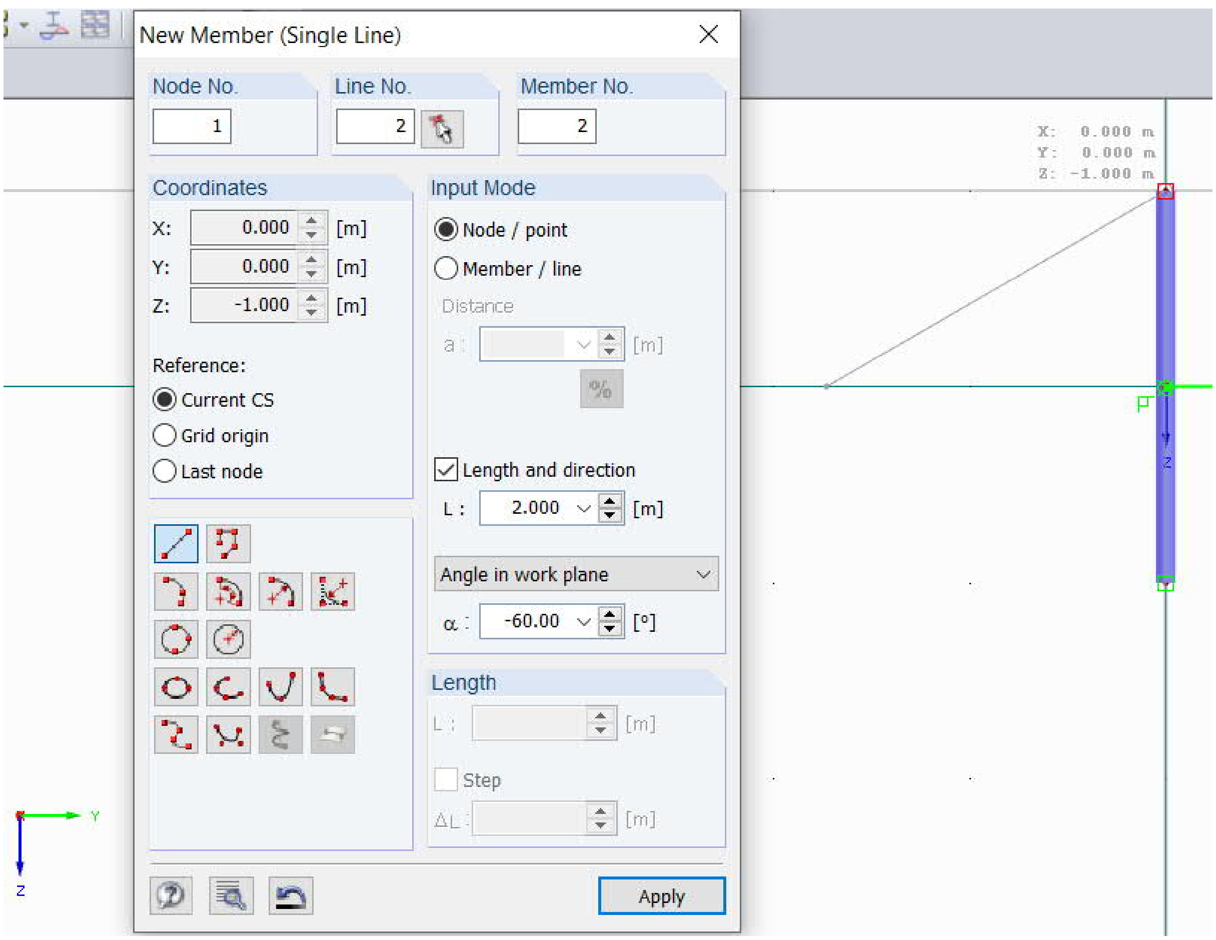


Figure 104. New member.

Using the previous instruction, determine the triangle ABD, then DEF, and then CEF. Add rods AE and BF.

You can check the length of the bar by hovering the cursor over the bar. Another way is to double-click the bar and check the length on the “Effective Length” tab of the Edit Member window.

Remove the auxiliary triangle ABC bars. Add supports and self weight (LC1) to the truss. Perform the calculation. It is a good idea to do the calculation to check the correctness of the sub-model.

When the model is OK, complete the missing rods and point load (LC2).

5.3.3 Loading

Create load cases:

- LC1 - Self-weight, permanent
- LC2 - Point load, permanent

5.4 Results

The bar forces and support reactions of the truss are presented in Figure 105.

Positive bar force means tension and negative one compression. If the bar force is zero, then the numerical value is not marked. Such zero bars in the case of the load in question are DE and DF. The presence of zero bars depends on the load case. If the point load is changed to be parallel to the z-axis, then DG and EF are zero bars.

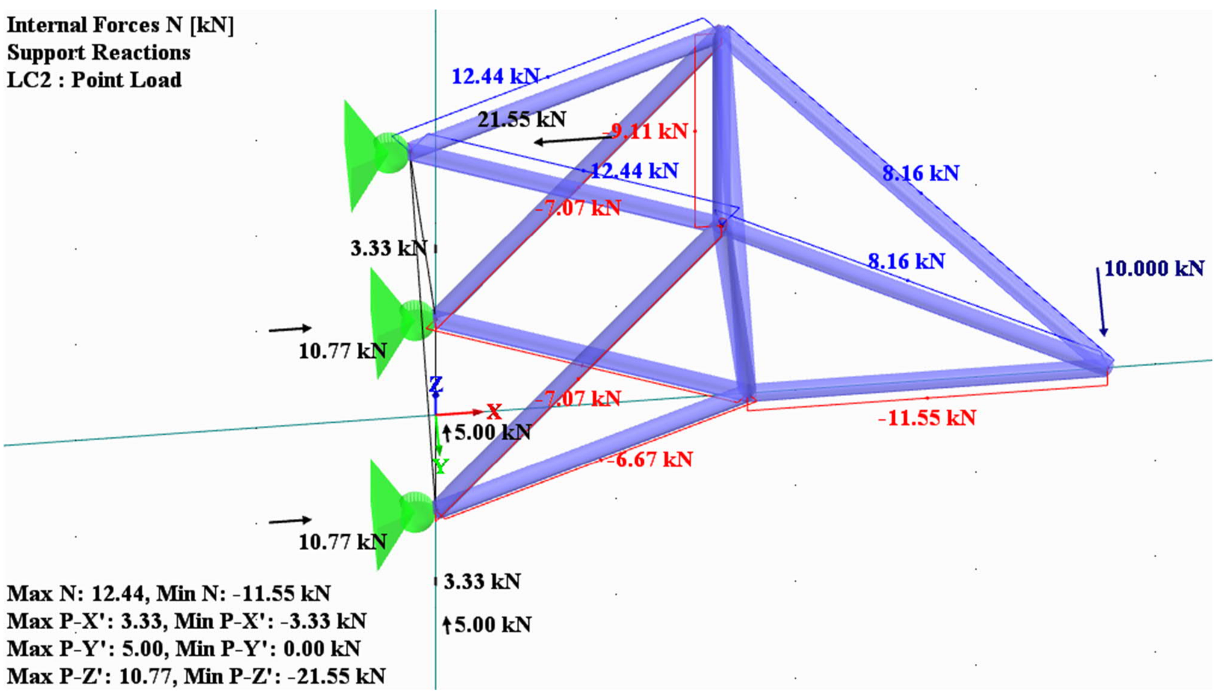


Figure 105. Bar forces and support reactions of the truss.

The deflection curve is presented in Figure 106.

The G point also moves slightly horizontally (*x*-axis direction) towards the origin. (This was not asked to be specified in the assignment.) The displacement is due to changes in the length of the bars and circular motion about the *y*-axis.

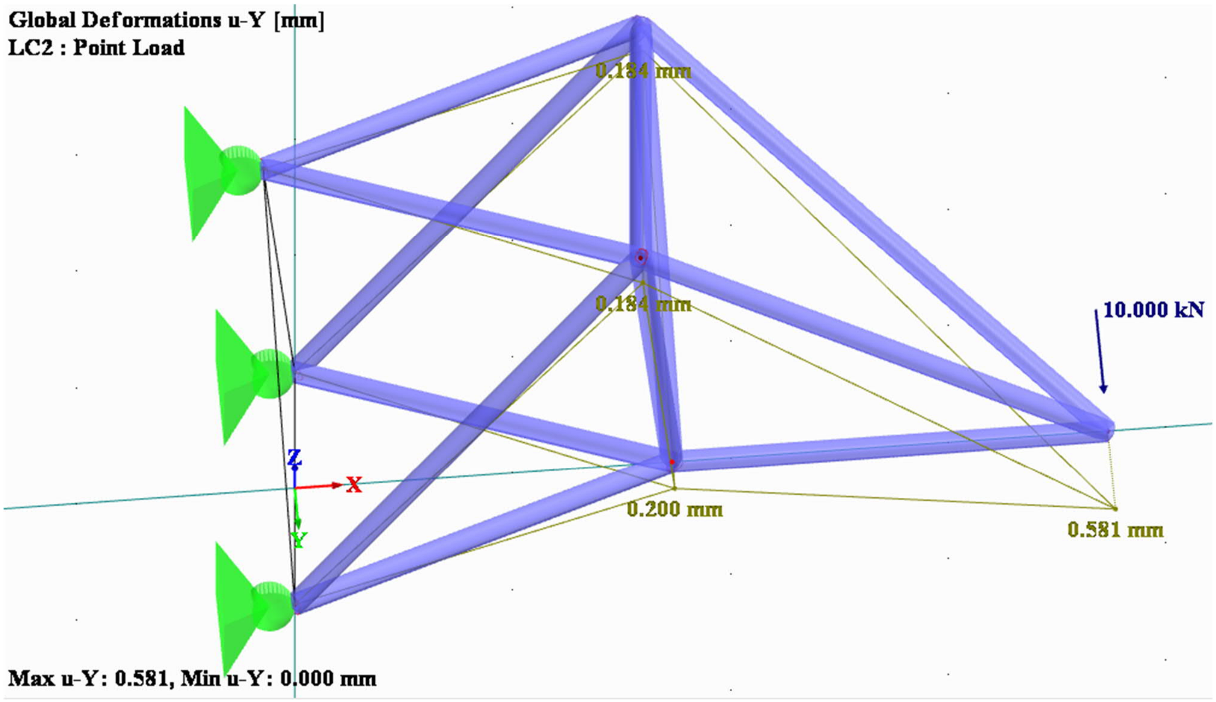


Figure 106. Deflection curve of the truss.

5.5 Validity

The support reactions are calculated from the moment equilibrium condition with respect to the origin and from the equilibrium conditions of the horizontal and vertical forces. (A support reaction along the z-axis is not required to solve the problem.)

Support reactions

$$\sum_n M_{z\_O} = 0$$

$$C_x := \frac{-\left(1 + \frac{\sqrt{3}}{2}\right)}{\frac{\sqrt{3}}{2}} \cdot P \qquad\qquad C_x = -21.547005 \cdot kN$$

$$\sum_n F_x = 0$$

$$A_x := \frac{-C_x}{2} \qquad\qquad A_x = 10.773503 \cdot kN$$

$$\sum_n F_y = 0$$

$$A_y := \frac{P}{2} \qquad\qquad A_y = 5 \cdot kN$$

The bar forces are calculated by using trigonometry from the equilibrium conditions of the forces along the coordinate axes.

Bar forces (note symmetry)

$$N_{DG} := \frac{-2}{\sqrt{3}} \cdot P \qquad N_{DG} = -11.547005 \cdot kN$$

$$N_{EG} := \sqrt{\left(\frac{2}{\sqrt{3}}\right)^2 + 1 + \left(\frac{1}{\sqrt{3}}\right)^2} \cdot \frac{P}{2} \qquad N_{EG} = 8.164966 \cdot kN$$

$$N_{DE} := 0 \qquad N_{DE} = 0$$

$$N_{AD} := \frac{2}{\sqrt{3}} \cdot \frac{N_{DG}}{2} \qquad N_{AD} = -6.666667 \cdot kN$$

$$N_{CE} := \frac{-2}{\sqrt{3}} \cdot \frac{C_x}{2} \qquad N_{CE} = 12.440169 \cdot kN$$

$$N_{EF} := \frac{a}{2 \cdot b} \cdot N_{EG} + \frac{1}{2} N_{CE} \qquad N_{EF} = 9.106836 \cdot kN$$

$$N_{AEx} := A_x + \frac{N_{DG}}{2} \qquad N_{AEx} = 5 \cdot kN$$

$$N_{AE} := \sqrt{N_{AEx}^{\ 2} + A_y^{\ 2}} \qquad N_{AE} = 7.071068 \cdot kN$$

The end deflection of the cantilever is calculated using the unit force method.

The number of the bars (symmetry), the lenghts of the bars and bar forces due to load P and 1

$$n := \begin{pmatrix} n_{AD} \\ n_{AE} \\ n_{CE} \\ n_{DE} \\ n_{DG} \\ n_{EG} \\ n_{EF} \end{pmatrix} \qquad L := \begin{pmatrix} L_{AD} \\ L_{AE} \\ L_{CE} \\ L_{DE} \\ L_{DG} \\ L_{EG} \\ L_{EF} \end{pmatrix} \qquad N := \begin{pmatrix} N_{AD} \\ N_{AE} \\ N_{CE} \\ N_{DE} \\ N_{DG} \\ N_{EG} \\ N_{EF} \end{pmatrix} \qquad N_1 := \frac{N}{P}$$

Sum of the products

$$\sum \overrightarrow{\left(n \cdot L \cdot N \cdot N_1\right)} = 1.856468 \times 10^5 \, J$$

Deflection at point G

$$v_G := \frac{\sum \overrightarrow{\left(n \cdot L \cdot N \cdot N_1\right)}}{E \cdot A} \qquad v_G = 0.582601 \cdot mm$$

Compare the numerical values obtained with Mathcad and FEM in the case of a point load! The results should be the same to a few decimal places.

6 Arch

6.1 Problem

Figure 107 shows an arch with the left end supported by a fixed hinge and the right by a roller support. The arch is made of reinforced concrete. The curve is loaded by an uniformly distributed traffic load (q) as shown.

The task is to determine the displacement (u, w) and rotation (φ) of the arc, as well as the distribution of bending moment (M), shear force (V), normal force (N) and normal stress (σ). The effect of self weight is not taken into account.

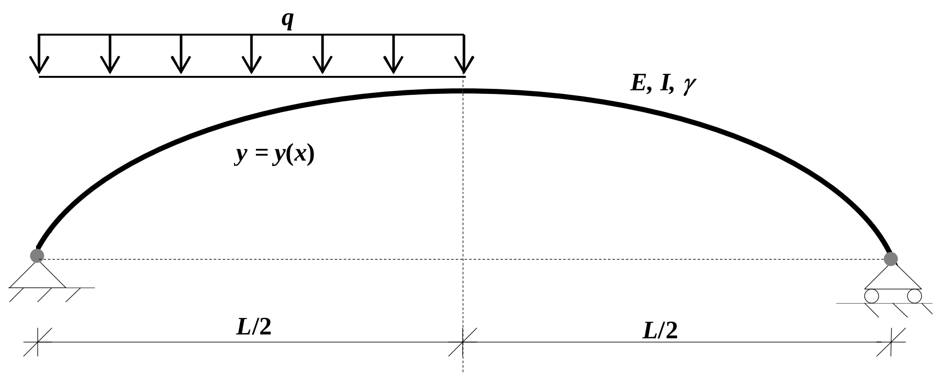


Figure 107. Arch.

Initial values are (*Mathcad*):

Length

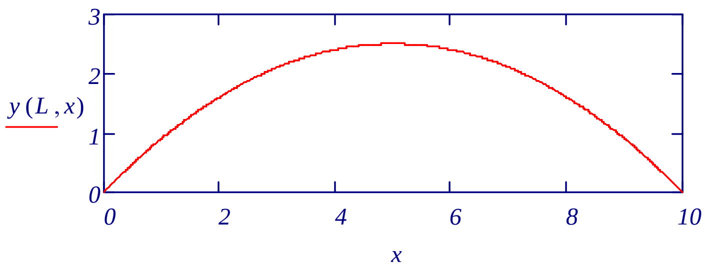
$$L := 10m$$

Load

$$q := 1\frac{kN}{m}$$

Shape of the arch

$$y(L,x) := x\left(1-\frac{x}{L}\right)$$



Apex

$$y\left(L,\frac{L}{2}\right) = 2.5m$$

6.2 Preliminary Planing

Figure 108 shows the elevation and cross-section of the arch, showing the dimensions of the structure, the specific load acting on the arch, the support conditions and the coordinate system.

The origin is positioned on the left support.

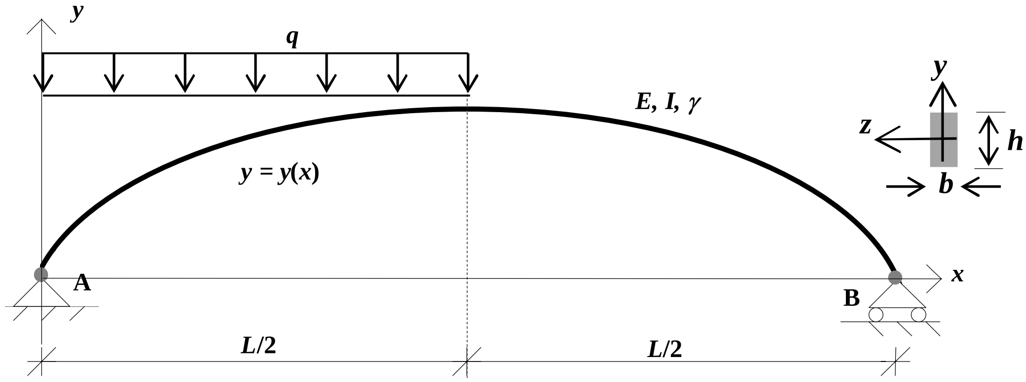


Figure 108. Preliminary planing of the arch.

The material is isotropic and ideally elastic (linearly elastic). Each load is considered as its own load case. The self weight is included in the calculation, although it is not needed to solve the problem. All safety factors are ones.

A rectangular cross section and the following material properties and dimensions have been selected for the arch (*Mathcad*):

Width and height of the cross-section

$b := 0.3m$

$h := 0.6m$

Modulus of elasticity

$$E := 36000 \frac{MN}{m^2}$$

Poisson's ratio

$\nu := 0.2$

Unit weight

$$\gamma := 24.525 \frac{kN}{m^3}$$

6.3 Modelling

Enter a name for the model, for example: Arch.

Select 3D coordinate system.

Create a finite element model, review it, and analyze it using the instructions previously provided in this guide. Some additional advice is provided in this chapter..

6.3.1 Arch

First, create a material and cross section of the member. Then create the shape of the arc using a line: Insert → Model Data → Lines → Parabola → Graphically. Define the arc by entering three points: left support (0, 0), arc peak (L/2, z(L/2)) and right support (L, 0).

Double-click the arch you created. On the Member tab of the Edit window, select Available and apply the cross-section and material you created (Figures 109 and 110). This connects the profile of the member to an arc line.

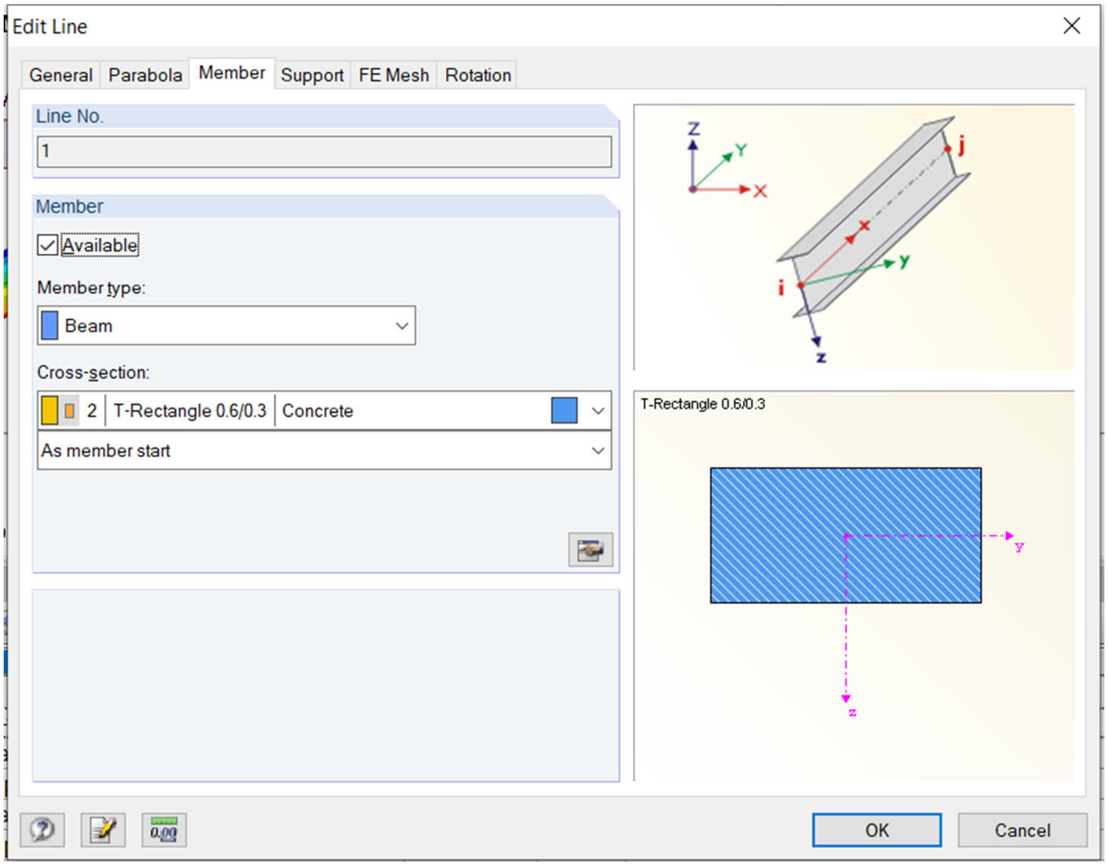


Figure 109. Edit line.

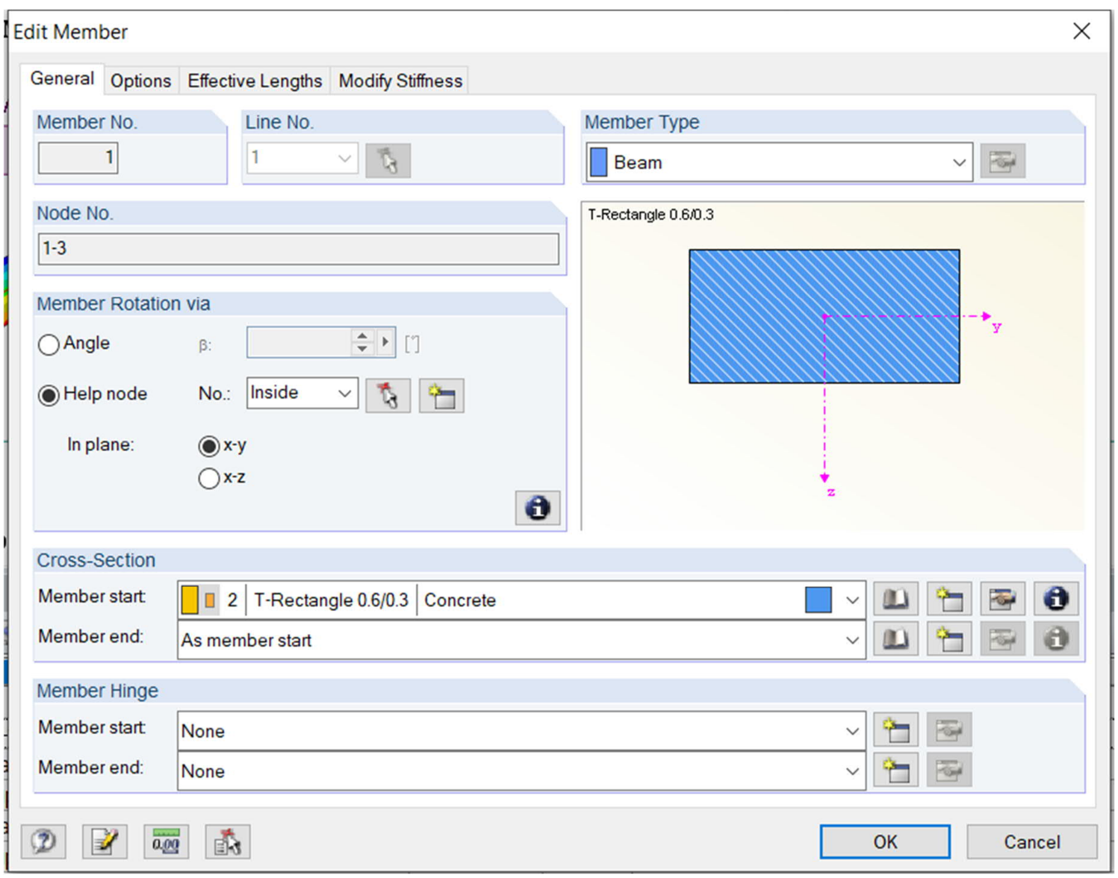


Figure 110. Edit member.

6.3.2 Loading

Create load cases:

- LC1 - Self-weight, permanent
- LC2 - Traffic-load, transient, member load, uniform

For a uniformly distributed load, use the Member load option as shown in Figure 111.

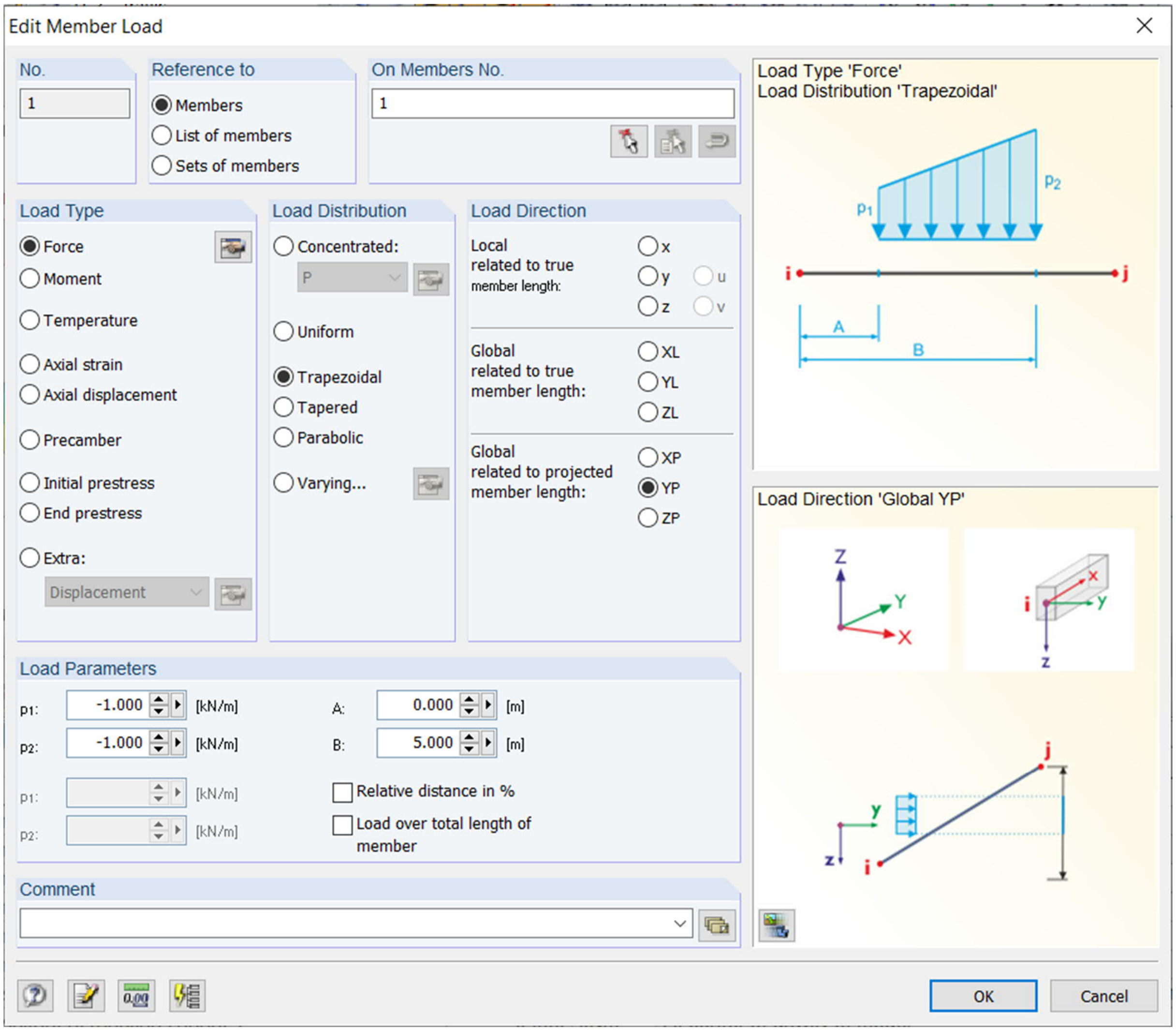


Figure 111. Edit member load.

6.4 Results

The shear force distibution and support reactions of the arc are shown in Fig. 112, the bending moment distribution in Fig. 113, the normal force distribution in Fig. 114, normal stress distribution in Fig. 115, the deflection in Fig. 116 and the rotatio in Fig. 117.

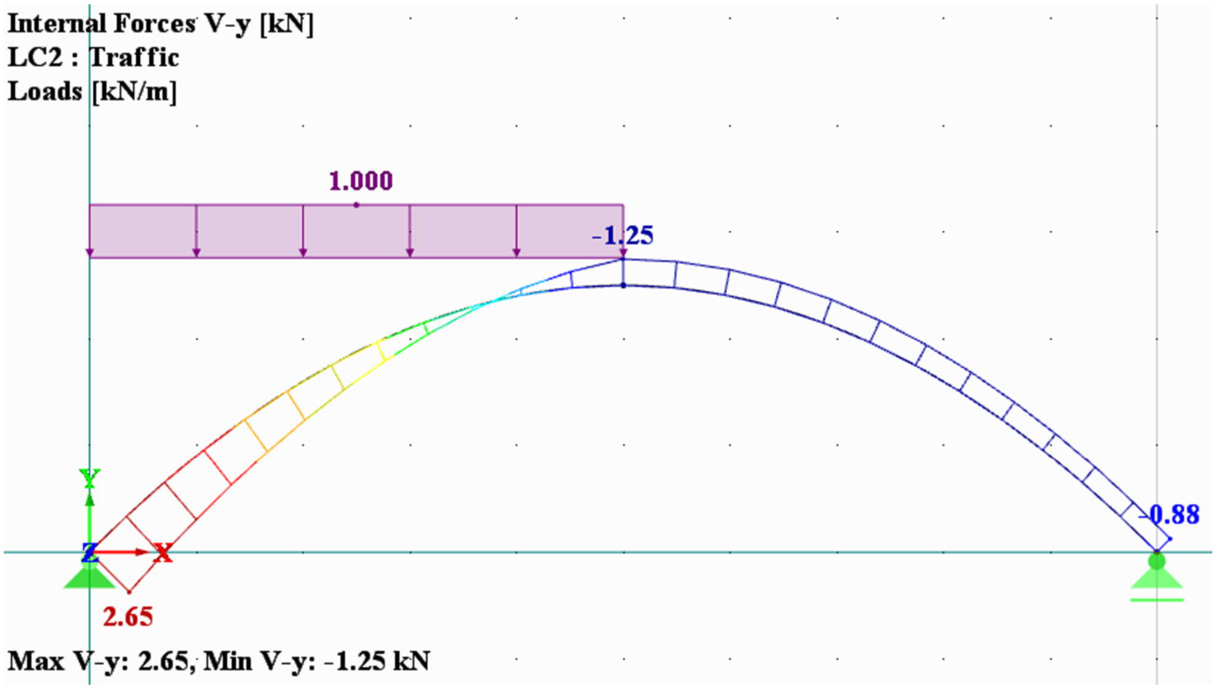


Figure 112. Shear force distibution and support reactions of the arc.

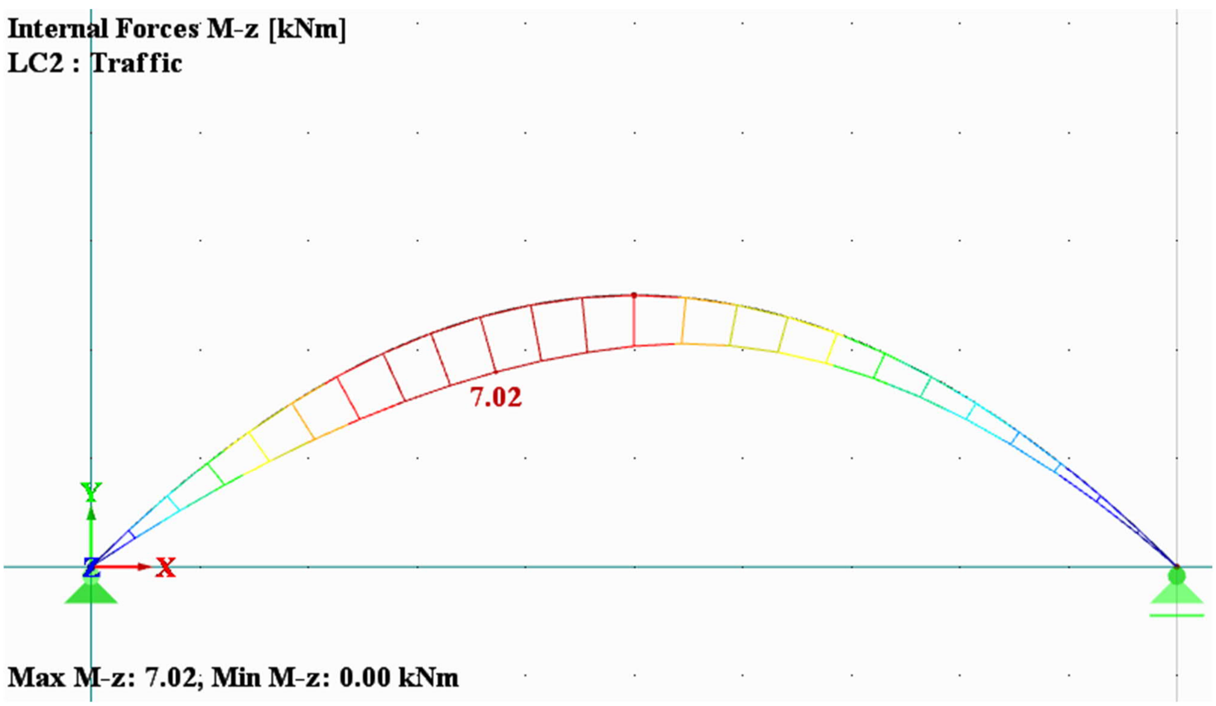


Figure 113. Bending moment distibution of the arc.

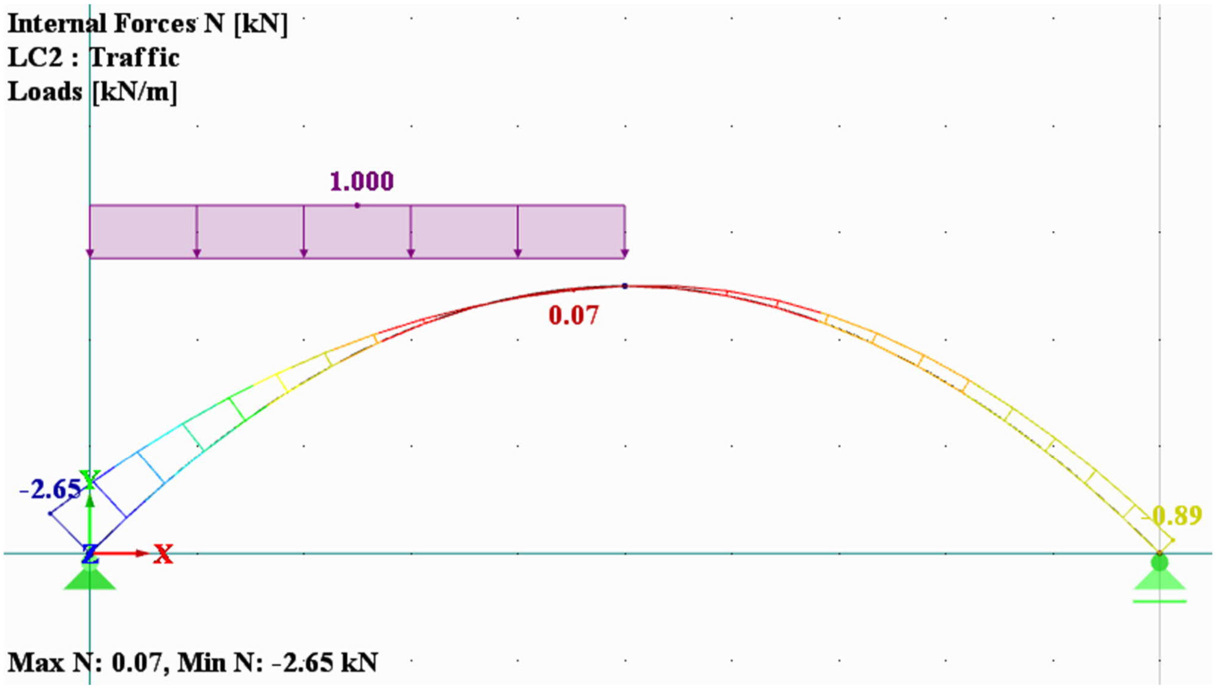


Figure 114. Normal force distribution of the arch.

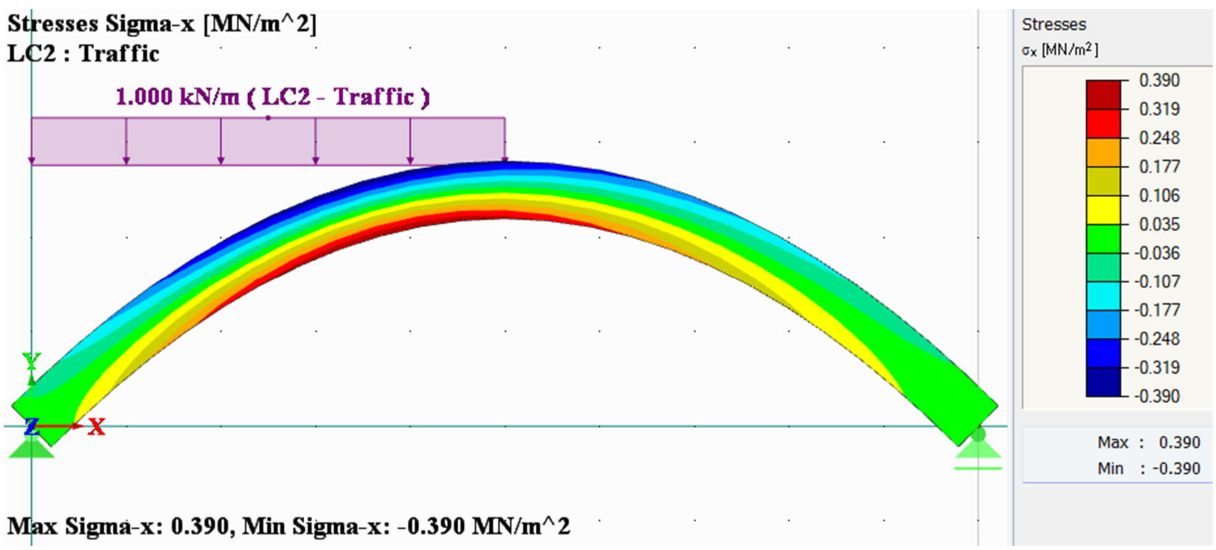


Figure 115. Normal stress distribution of the arch.

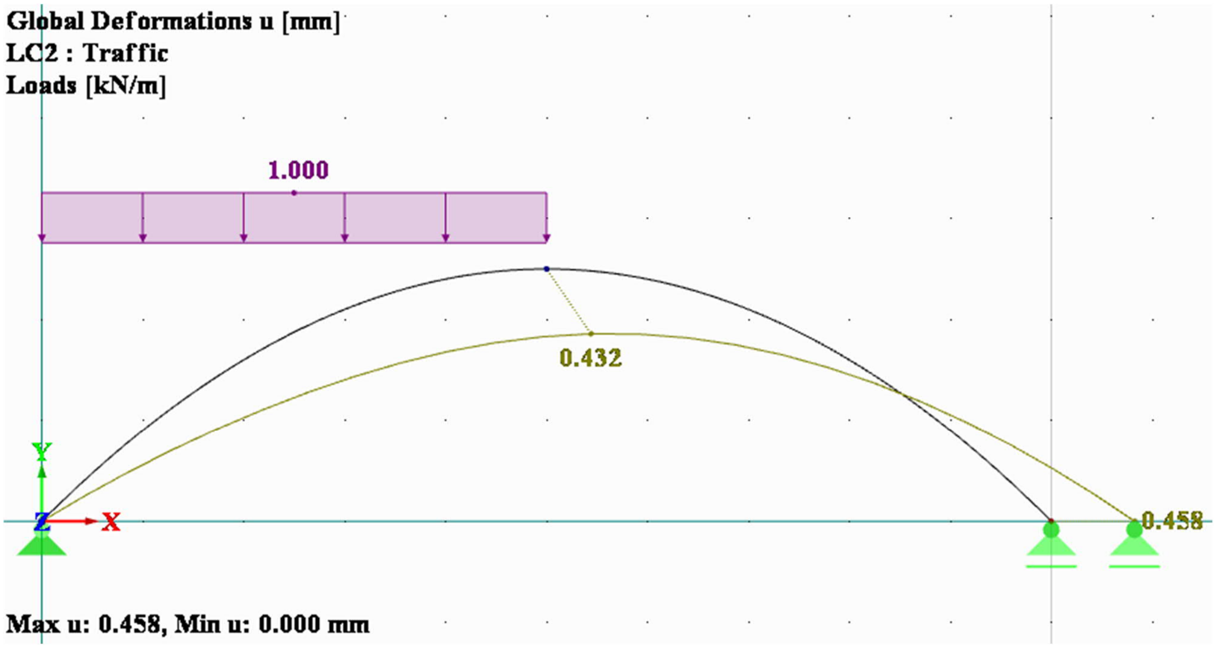


Figure 116. Deflection of the arch.

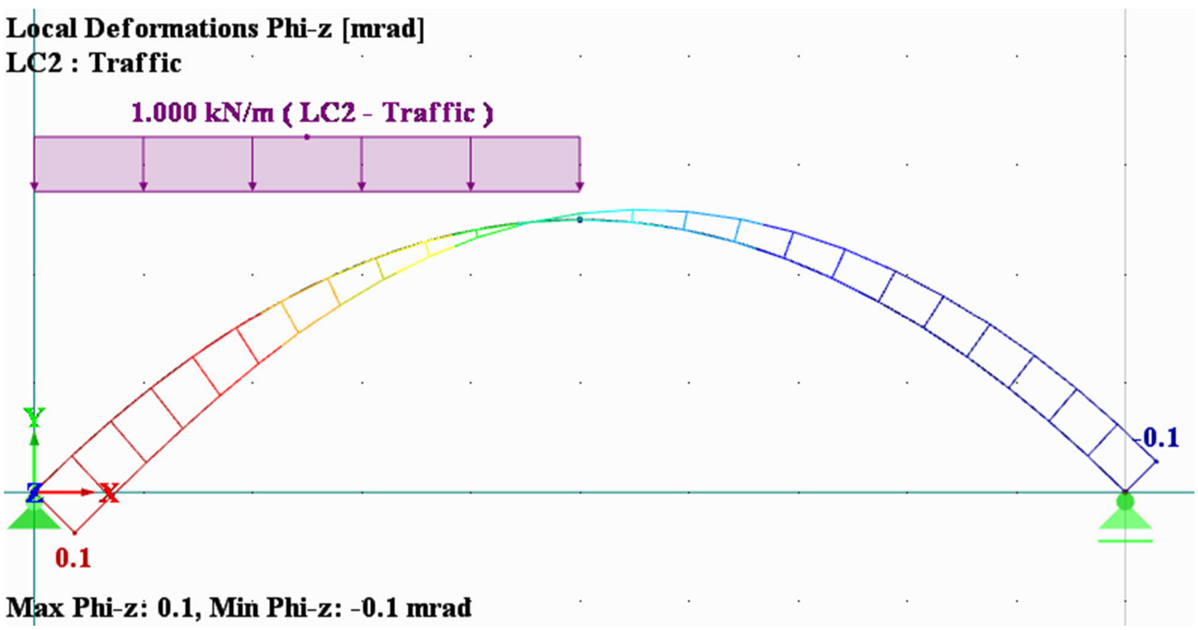


Figure 117. Rotation of the arch.

6.5 Validity

First, the cross-sectional quantities, the derivative of the arc equation, and the support reactions are calculated.

Cross-section area

$A := b\,h$ $\quad\quad\quad\quad$ $A = 0.18m^2$

Moment of inertia

$I := \frac{b\,h^3}{12}$ $\quad\quad\quad\quad$ $I = 0.0054m^4$

Derivative of the arch shape

$$D_y(x) := \frac{d}{dx}y(L,x)$$

Support reactions (by lever arm rule)

$T_A := \frac{q\,L}{2}\,\frac{3}{4}$ $\quad\quad\quad\quad$ $T_A = 3750\,N$

$T_B := \frac{q\,L}{2}\,\frac{1}{4}$ $\quad\quad\quad\quad$ $T_B = 1250\,N$

$H_A := 0$ $\quad\quad\quad\quad$ $H_A = 0$

Horizontal force

$$H(x) := 0$$

The normal and shear forces of the arc are calculated in the tangential direction and in the direction perpendicular to it. See Figure 118.

Normal foce parallel to the arc tangent

$$N_A(x) := H(x)\frac{1}{\sqrt{1+D_y(x)^2}} - V_{Ay}(x)\frac{D_y(x)}{\sqrt{1+D_y(x)^2}}$$

Shear force perpendicular to tangent

$$V_A(x) := H(x)\frac{D_y(x)}{\sqrt{1+D_y(x)^2}} + V_{Ay}(x)\frac{1}{\sqrt{1+D_y(x)^2}}$$

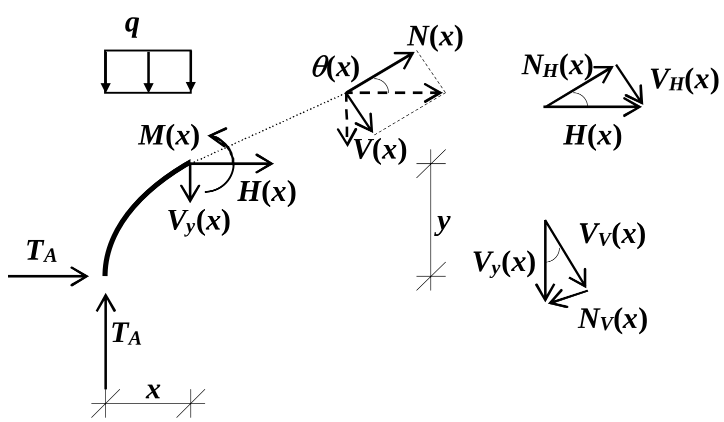


Figure 118. Defining the internal forces of the arch.

By considering a very short part of the arc (Fig. 119), the trigonometric functions expressed by the angle θ can be expressed as a function of *x*.

Trigonometric functions expressed by the angle

$$sin(\theta) = \frac{tan(\theta)}{\sqrt{1 + (tan(\theta))^2}}$$

$$cos(\theta) = \frac{1}{\sqrt{1 + (tan(\theta))^2}}$$

Trigonometric functions expressed by x

$$sin(\theta(x)) = \frac{D_y(x)}{\sqrt{1 + D_y(x)^2}}$$

$$cos(\theta(x)) = \frac{1}{\sqrt{1 + D_y(x)^2}}$$

Normal force parallel to tangent

$$N(x, \theta(x)) = H(x)\,cos(\theta(x)) - V(x)\,sin(\theta(x))$$

Shear force perpendicular to tangent

$$Q(x, \theta(x)) = H(x)\,sin(\theta(x)) + V(x)\,cos(\theta(x))$$

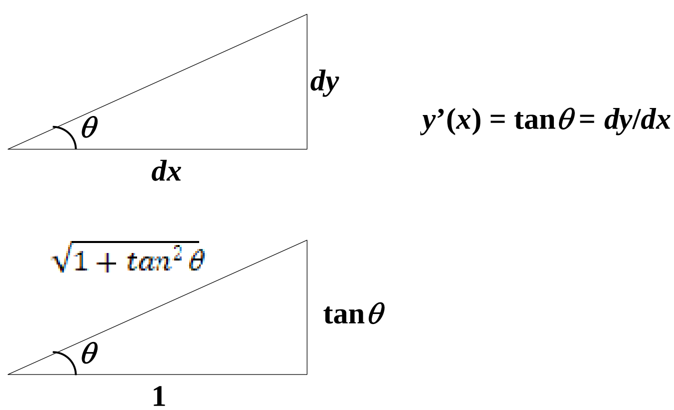


Figure 119. Derivation of the arc function.

Due to the asymmetric load, the expressions of bending moment and shear and normal force are determined piecewise.

Left hand side of the arch: $0 \le x \le \frac{L}{2}$

Bending moment

$$M_A(x) := T_A x - q\frac{x^2}{2}$$

Vertical shear force

$$V_{Ay}(x) := \frac{d}{dx}M_A(x)$$

Normal foce parallel to the arc tangent

$$N_A(x) := H(x)\frac{1}{\sqrt{1 + D_y(x)^2}} - V_{Ay}(x)\frac{D_y(x)}{\sqrt{1 + D_y(x)^2}}$$

Shear force perpendicular to tangent

$$V_A(x) := H(x)\frac{D_y(x)}{\sqrt{1 + D_y(x)^2}} + V_{Ay}(x)\frac{1}{\sqrt{1 + D_y(x)^2}}$$

Right side of the arch: $\frac{L}{2} \le x \le L$

Bending moment

$$M_B(x) := T_B(L - x)$$

Vertical shear force

$$V_{By}(x) := \frac{d}{dx}M_B(x)$$

Normal foce parallel to the arc tangent

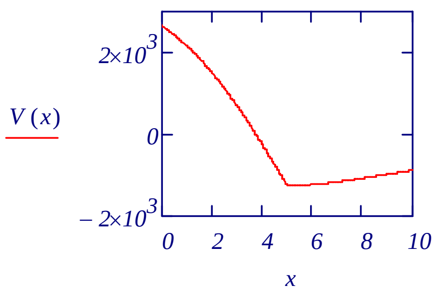
$$N_B(x) := H(x)\frac{1}{\sqrt{1 + D_y(x)^2}} - V_{By}(x)\frac{D_y(x)}{\sqrt{1 + D_y(x)^2}}$$

Shear force perpendicular to tangent

$$V_B(x) := H(x)\frac{D_y(x)}{\sqrt{1 + D_y(x)^2}} + V_{By}(x)\frac{1}{\sqrt{1 + D_y(x)^2}}$$

Shear force perpendicular to tangent

$$V(x) := \begin{vmatrix} V_A(x) & \text{if } x \le \dfrac{L}{2} \\ V_B(x) & \text{otherwise} \end{vmatrix}$$



$V(0) = 2.652kN$

$V_A\left(\dfrac{L}{2}\right) = -1.25\,kN$

$V_B\left(\dfrac{L}{2}\right) = -1.25\,kN$

$V(L) = -0.884kN$

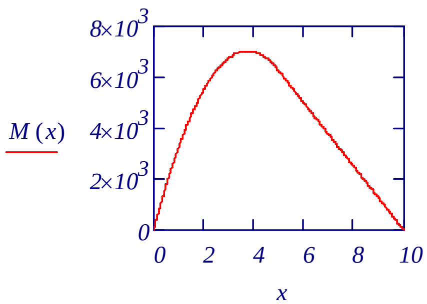
Given

$V_A(x) = 0$

$x_V := Find(x) \rightarrow \dfrac{15m}{4}$

Bending moment

$$M(x) := \begin{vmatrix} M_A(x) & \text{if } x \le \dfrac{L}{2} \\ M_B(x) & \text{otherwise} \end{vmatrix}$$



$M(0) = 0\,kN\,m$

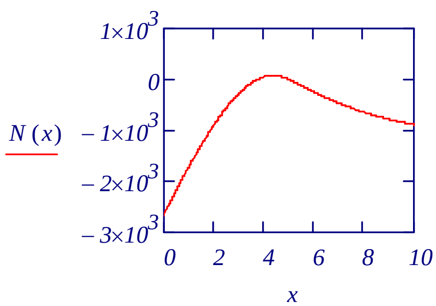
$M(x_V) = 7.031\,kN\,m$

$M\left(\dfrac{L}{2}\right) = 6.25\,kN\,m$

$M(L) = 0\,kN\,m$

Normal force parallel to the arc tangent

$$N(x) := \begin{vmatrix} N_A(x) & \text{if } x \le \dfrac{L}{2} \\ N_B(x) & \text{otherwise} \end{vmatrix}$$



$N(0) = -2.652kN$

$N\left(\dfrac{L}{2}\right) = 0\,kN$

$N(L) = -0.884kN$

The horizontal displacement of the right end is checked by the unit force method.

Normal stress (the proportion of normal force is small, so the stress at the point where the bending moment is high is calculated)

$$\sigma(x,y) := \frac{N(x)}{A} + \frac{M(x)}{I}y$$

$$\sigma_{max} := \sigma\left(x_V, \frac{h}{2}\right) \qquad \sigma_{max} = 0.391\frac{MN}{m^2}$$

Normal force due horizontal unit load acting at the right end of the arch

$$H_1 := -1$$

Bending moment

$$M_1(L,x) := y(L,x)$$

Normal force parallel to the arc tangent

$$N_1(x) := H_1 \frac{1}{\sqrt{1 + D_y(x)^2}}$$

Shear force perpendicular to tangent

$$V_1(x) := H_1 \frac{D_y(x)}{\sqrt{1 + D_y(x)^2}}$$

Horizontal displacement of the right end of the arc

$$u_B := \int_0^{\frac{L}{2}} \frac{M_A(x) M_1(L,x)}{E\,I}\,dx + \int_{\frac{L}{2}}^{L} \frac{M_B(x) M_1(L,x)}{E\,I}\,dx \ ... \\ + \int_0^{\frac{L}{2}} \frac{N_A(x) N_1(x)}{E\,A}\,dx + \int_{\frac{L}{2}}^{L} \frac{N_B(x) N_1(x)}{E\,A}\,dx$$

$$u_B = 0.43\,mm$$

Compare the numerical values obtained with Mathcad and FEM in the case of a point load! The results should be close to each other.

7 Beam on an Elastic Foundation

7.1 Problem

Figure 120 shows a semi-infinite beam on an elastic foundation (elastic constant *k*). The beam is made of wood. At the end of the beam, the point force F_0 and the bending moment M_0 act.

The task is to determine the deflection (*w*) and rotation (φ) of the beam as well as the distribution of bending moment (*M*) and shear force (*V*). The effect of self weight is not taken into account.

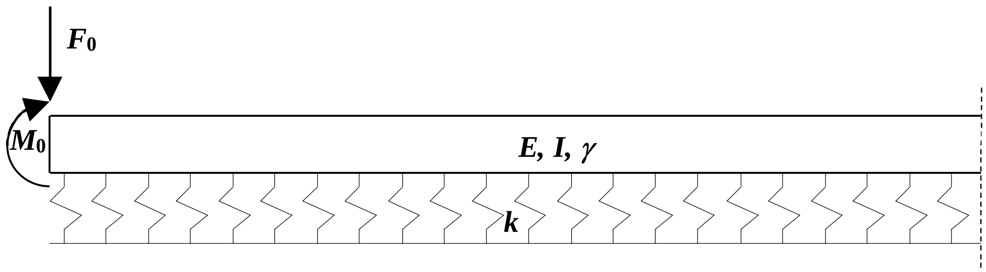


Figure 120. Sem-infinite beam on an elastic foundation.

Initial valeus are (*Mathcad*):

Loads

$F_0 := 7kN$

$M_0 := 10kN\ m$

7.2 Preliminary Planing

A coordinate system for the beam and a sufficiently long finite length measure *L* (Fig. 121) are selected. A roller support is placed at the right end of the beam to allow vertical displacement and rotation about the z-axis (other directions are blocked). Without the support condition, the FEM model cannot be calculated.

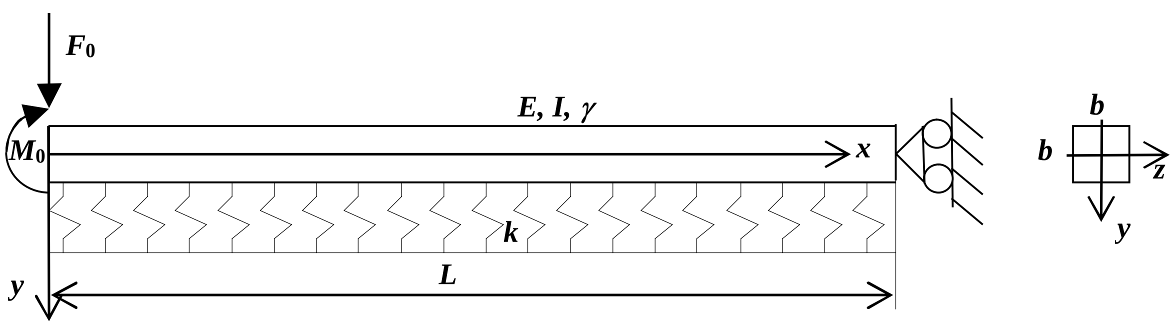


Figure 121. Beam on an elastic foundation.

The material of the beam is isotropic and ideally elastic (linearly elastic). Each load is considered as its own load case. The self weight is included in the calculation, although it is not needed to solve the problem. All safety factors are ones.

A square cross-section and the following material properties and dimensions have been selected for the beam (*Mathcad*):

Length

$L := 10m$

Cross-section dimension

$b := 0.2m$

Modulus of elasticity

$E := 11000\frac{MN}{m^2}$

Poisson's ratio

$\nu := 0.33$

Unit weight

$\gamma := 5.9\frac{kN}{m^3}$

Foundation coefficient

$\kappa := 5000$

Moment of inertia

$I := \frac{b^4}{12}$ $\qquad I =$

Elastic constant

$k := \kappa\frac{E\,I}{L^4}$ $\qquad k =$

7.3 Modelling

Name the model, for example: BEF (Beam on an Elastic Foundation).

Select 3D coordinate system. The y-axis direction is down.

Create a finite element model, review it, and analyze it using the instructions previously provided in this guide. Some additional advice is provided in this chapter.

7.3.1 Elastic Foundation

An elastic support is defined as a property of a structural member. On the Options tab of the New Member (or Edit Member) window, under the text "Member Elastic Foundation" click icon  (Create New Member Elastic Foundation…). See Fig. 122. In the window that will open, specify the base number as shown in Fig. 123.

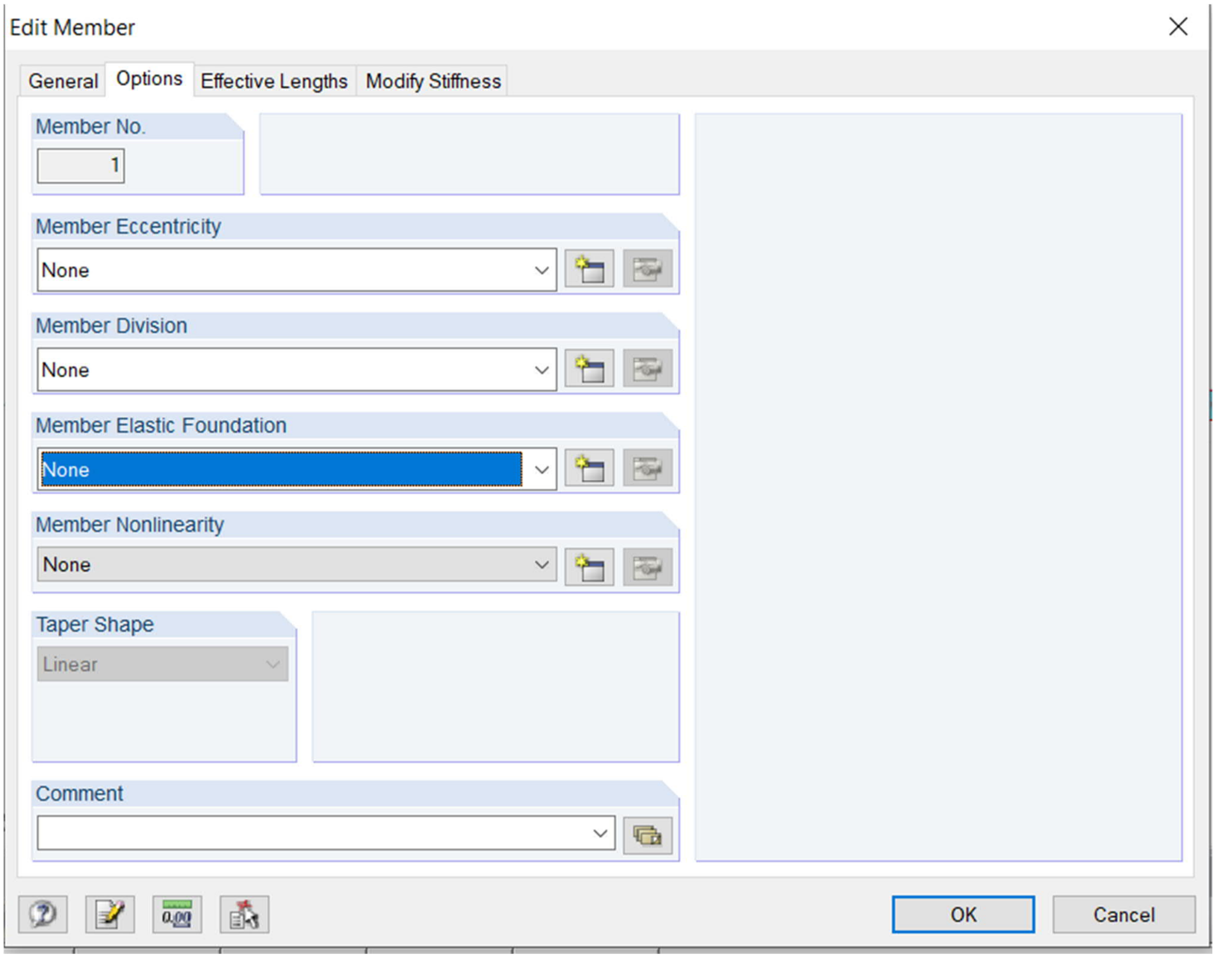


Figure 122. Edit member.

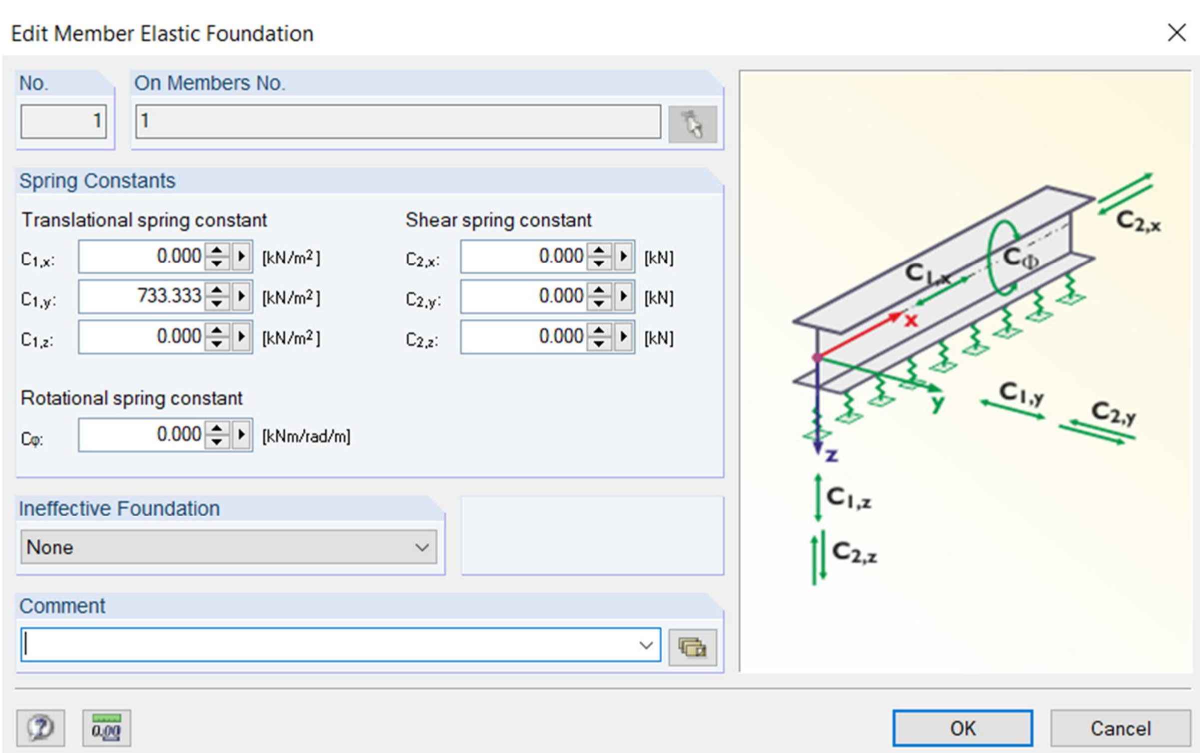


Figure 123. Edit member elastic foundation.

7.3.2 Loading

Create load cases:

- LC1 - Self-weight, permanent
- LC2 - Point load, nodal load, permanent
- LC3 - Point moment, nodal load, permanent

Create load combinations by including load cases LC2 and LC3.

7.4 Results

The shear force distribution of the beam is shown in Fig. 124, the bending moment distribution in Fig. 125, and the normal stress distribution in Fig. 126. In the bending moment distribution view, the line direction is reversed so that the bending moment sign is correct.

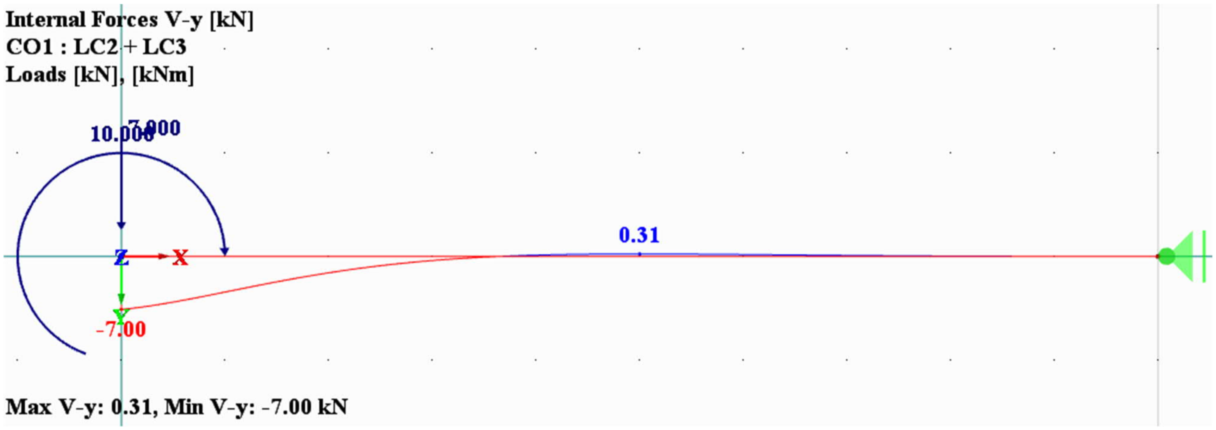


Figure 124. Shear force distribution of the beam..

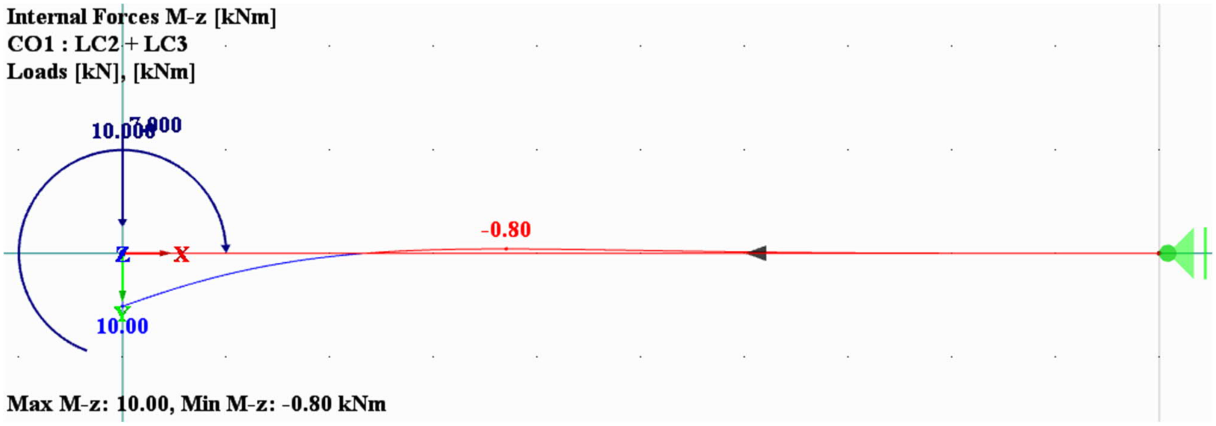


Figure 125. Bending moment distribution of the beam.

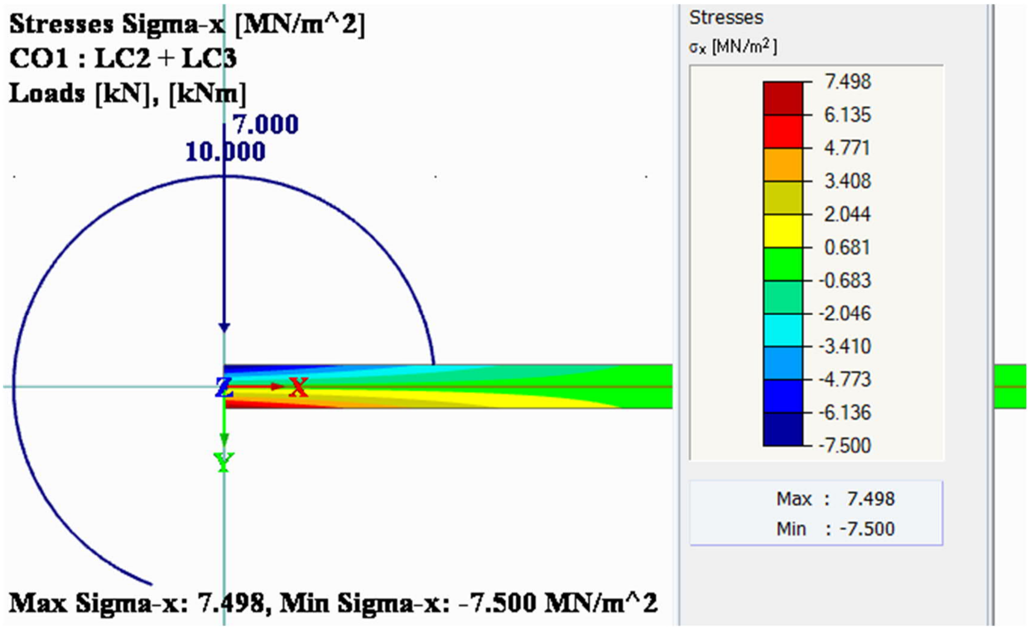


Figure 126. Normal stress distribution of the beam.

Rotation graph is prsentedin Fig. 127 and deflection graph in Fig. 128.

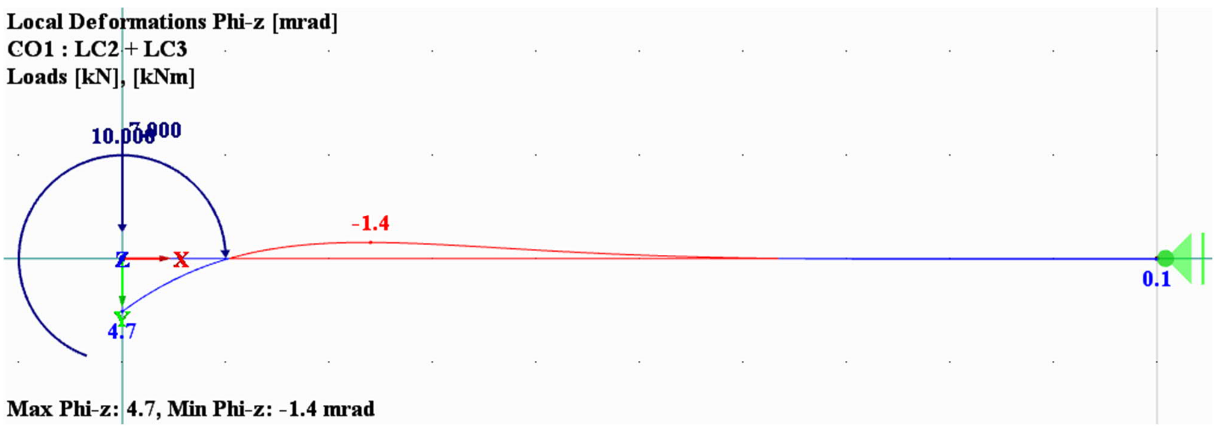


Figure 127. Rotation of the beam.

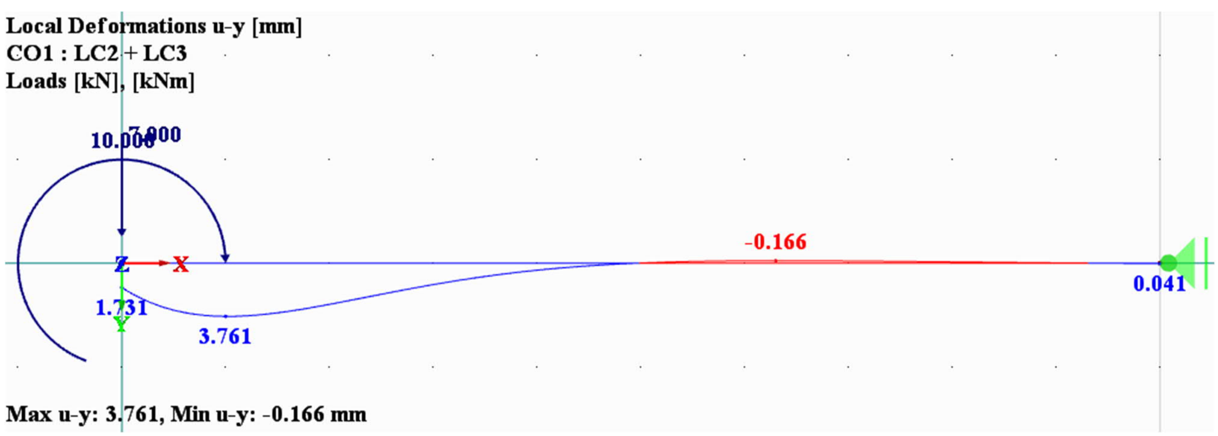


Figure 128. Deflection of the beam.

7.5 Validity

The expressions are written separately for point force and bending moment and finally combined according to the superposition principle (*Mathcad*):

Differential equation

$$\frac{d^4}{dx^4}v(x) + 4\beta^4 v(x) = \frac{q}{E\,I}$$

Solution is

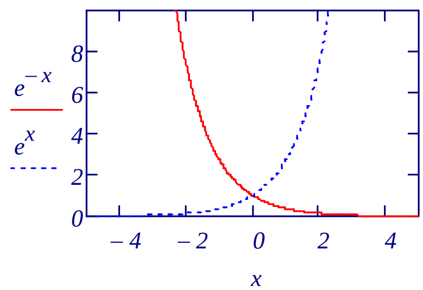
$$v(x) = e^{-\beta x}(A\cos(\beta x) + B\sin(\beta x)) + e^{\beta x}(C\cos(\beta x) + D\sin(\beta x)) + v_o(x)$$

When there is no external force, the particular solution vanish

$$v_o(x) := 0$$

Shape of two basic fuctions with

$$\beta = 1$$



When

$$x \geq 0$$

deflection have to be zero when x approaches infinity. Thus

$$C := 0$$

$$D := 0$$

Boundary conditions

$$M(0) = M_o$$

$$Q(0) = Q_o$$

From these, the integration constants A and B can be solved.

Deflection

$$v_F(F_0, E, I, \beta, x) := \frac{F_0}{2E\,I\,\beta^3} e^{-\beta x} \cos(\beta x)$$

$$v_M(M_0, E, I, \beta, x) := \frac{M_0}{2E\,I\,\beta^2} e^{-\beta x} (\sin(\beta x) - \cos(\beta x))$$

$$v(F_0, M_0, E, I, \beta, x) := v_F(F_0, E, I, \beta, x) + v_M(M_0, E, I, \beta, x)$$

Rotation

$$\varphi_F(F_0, E, I, \beta, x) := \frac{d}{dx} v_F(F_0, E, I, \beta, x)$$

$$\varphi_F(F_0, E, I, \beta, x) \text{ simplify } \rightarrow -\frac{F_0 e^{-\beta x} (\cos(\beta x) + \sin(\beta x))}{2E\,I\,\beta^2}$$

$$\varphi_M(M_0, E, I, \beta, x) := \frac{d}{dx} v_M(M_0, E, I, \beta, x)$$

$$\varphi_M(M_0, E, I, \beta, x) \text{ simplify } \rightarrow \frac{M_0 e^{-\beta x} \cos(\beta x)}{E\,I\,\beta}$$

$$\varphi(F_0, M_0, E, I, \beta, x) := \varphi_F(F_0, E, I, \beta, x) + \varphi_M(M_0, E, I, \beta, x)$$

Bending moment

$$M_F(F_0, E, I, \beta, x) := -E\,I\left(\frac{d}{dx} \varphi_F(F_0, E, I, \beta, x)\right)$$

$$M_F(F_0, E, I, \beta, x) \text{ simplify } \rightarrow -\frac{F_0 e^{-\beta x} \sin(\beta x)}{\beta}$$

$$M_M(M_0, E, I, \beta, x) := -E\,I\left(\frac{d}{dx} \varphi_M(M_0, E, I, \beta, x)\right)$$

$$M_M(M_0, E, I, \beta, x) \text{ simplify } \rightarrow M_0 e^{-\beta x} (\cos(\beta x) + \sin(\beta x))$$

$$M(F_0, M_0, E, I, \beta, x) := M_F(F_0, E, I, \beta, x) + M_M(M_0, E, I, \beta, x)$$

Shear force

$$Q_F(F_0, E, I, \beta, x) := \frac{d}{dx} M_F(F_0, E, I, \beta, x)$$

$$Q_F(F_0, E, I, \beta, x) \text{ simplify } \rightarrow -F_0 e^{-\beta x} (\cos(\beta x) - \sin(\beta x))$$

$$Q_M(M_0, E, I, \beta, x) := \frac{d}{dx} M_M(M_0, E, I, \beta, x)$$

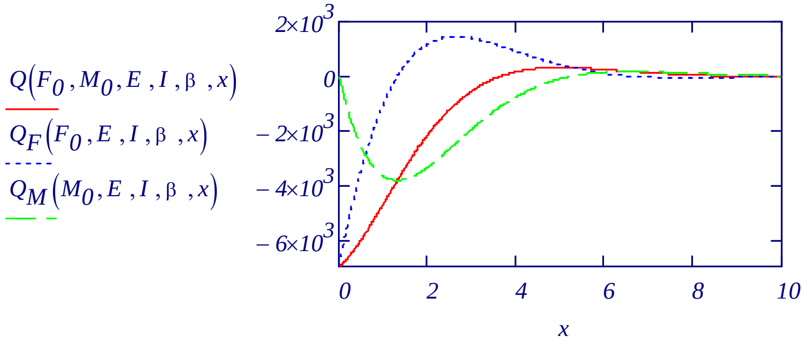
$$Q_M(M_0, E, I, \beta, x) \text{ simplify } \rightarrow -2M_0 \beta\, e^{-\beta x} \sin(\beta x)$$

$$Q(F_0, M_0, E, I, \beta, x) := Q_F(F_0, E, I, \beta, x) + Q_M(M_0, E, I, \beta, x)$$

Parameter

$$\beta := \sqrt[4]{\frac{k}{4E\,I}} \qquad\qquad \beta = 0.595\frac{1}{m}$$

Shear force

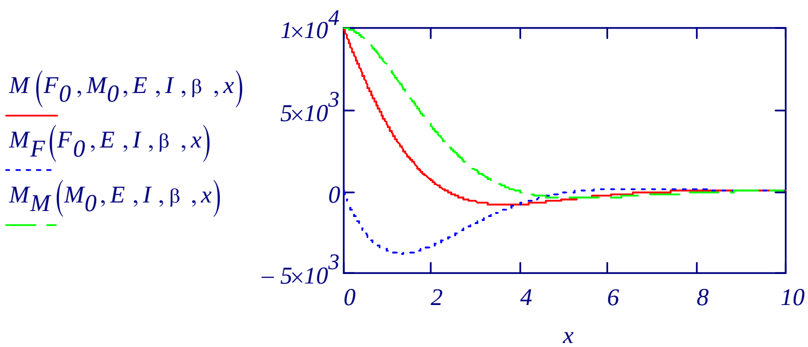


$$Q(F_0, M_0, E, I, \beta, 0) = -7\,kN$$

$$x_Q := 3.667m$$

$$Q(F_0, M_0, E, I, \beta, x_Q) = -0\,kN$$

Bending moment



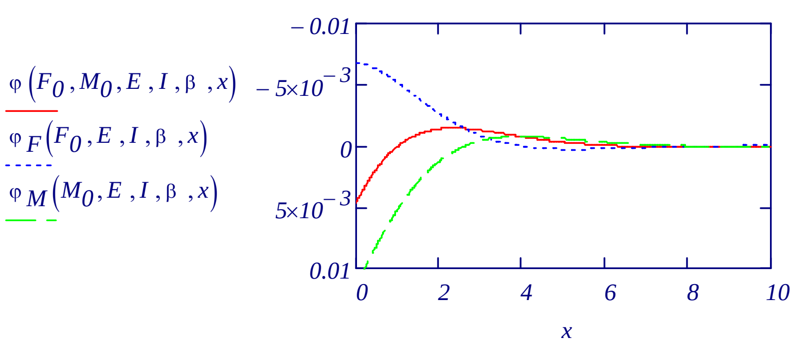
$$M(F_0, M_0, E, I, \beta, 0) = 10kN\ m$$

$$M_{min} := M(F_0, M_0, E, I, \beta, x_Q) \qquad\qquad M_{min} = -0.811\,kN\ m$$

$$x_M := 2.347m$$

$$M(F_0, M_0, E, I, \beta, x_M) = -0\,kN\ m$$

Rotation



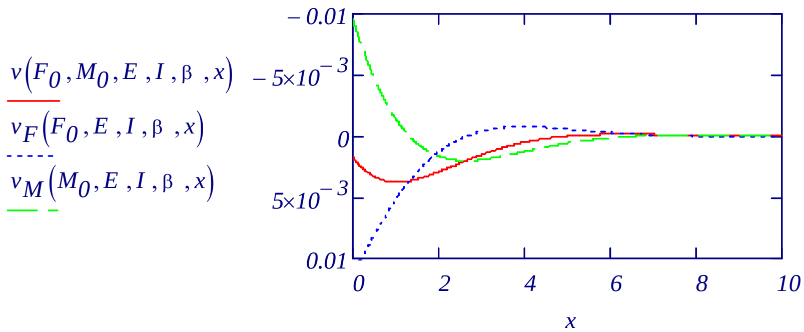
$$\varphi\left(F_0, M_0, E, I, \beta, 0\right) = 0.004717$$

$$\varphi_{min} := \varphi\left(F_0, M_0, E, I, \beta, x_M\right) \qquad \varphi_{min} = -0.001$$

$$x_\varphi := 1.026m$$

$$\varphi\left(F_0, M_0, E, I, \beta, x_\varphi\right) = -0$$

Deflection



$$v\left(F_0, M_0, E, I, \beta, 0\right) = 0.001709m$$

$$v_{max} := v\left(F_0, M_0, E, I, \beta, x_\varphi\right) \qquad v_{max} = 0.003763\,m$$

Maximum normal stress

$$\sigma(x, y) := \frac{M\left(F_0, M_0, E, I, \beta, x\right)}{I} y$$

$$\sigma_{max} := \sigma\left(0, \frac{b}{2}\right) \qquad \sigma_{max} = 7.5 \frac{MN}{m^2}$$

Compare the numerical values obtained with Mathcad and FEM in the case of a point load! The results should be close to each other.

8 Exporting and Importing

Here is shown the export of the RFEM model to an Excel file and the import back to RFEM. Using other file formats works similarly.

8.1 Exporting

Open and leave an empty Excel file open (Figure 129).

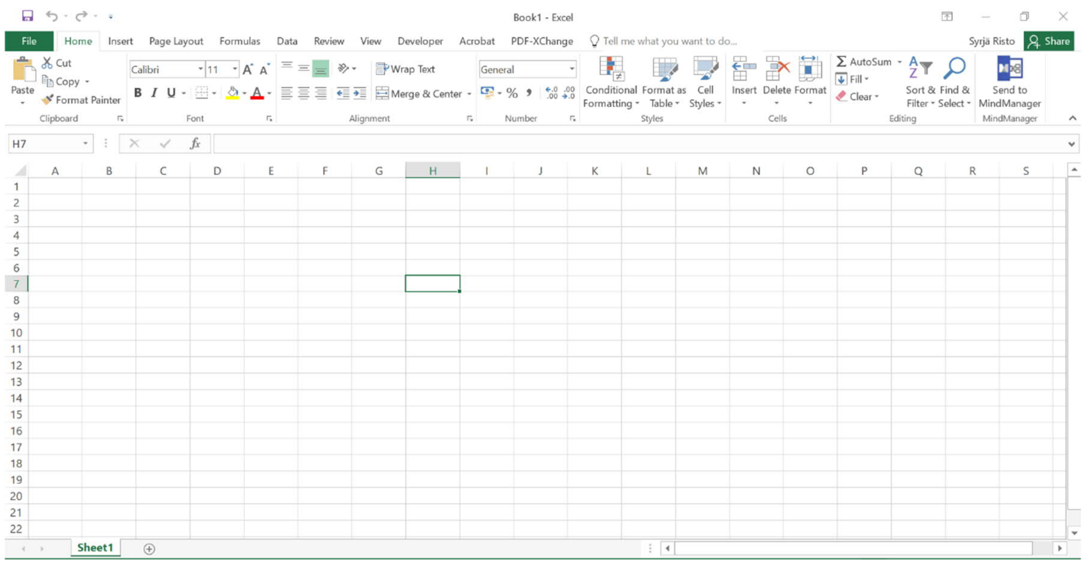


Figure 129. Excel-file.

In RFEM: File -> Export. Update as shown in Figure 130.

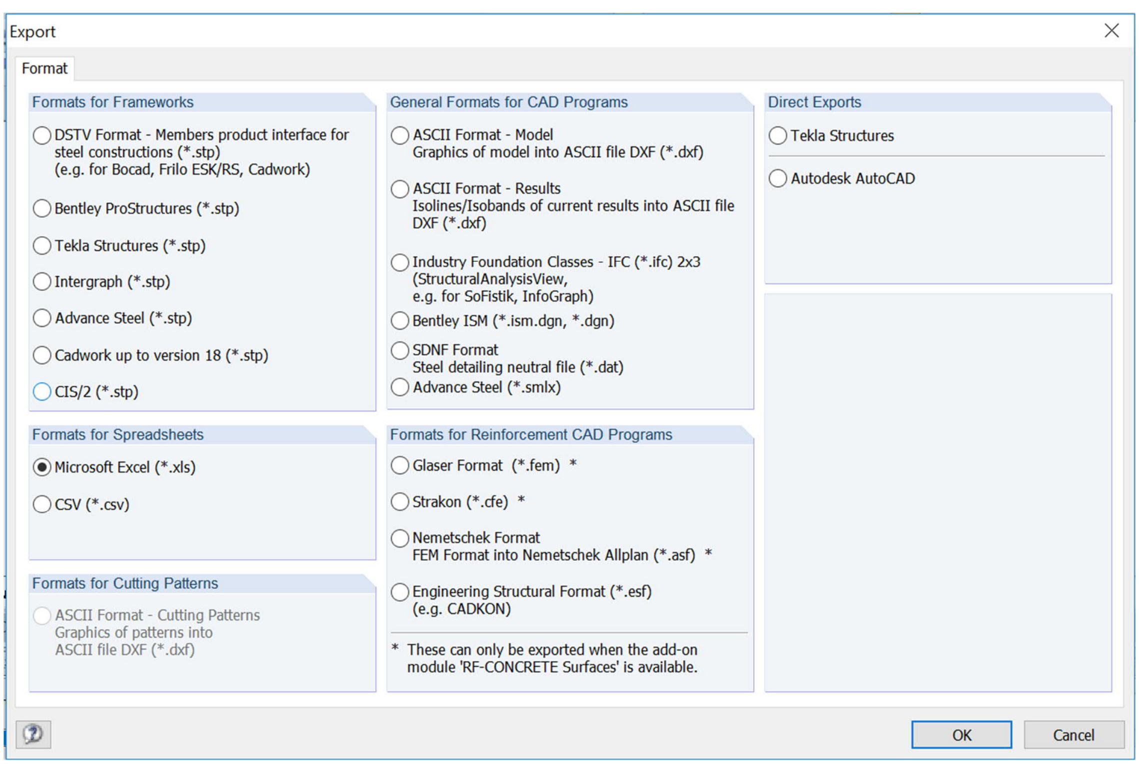


Figure 130. Export.

Press OK. Update as shown in Figure 131.

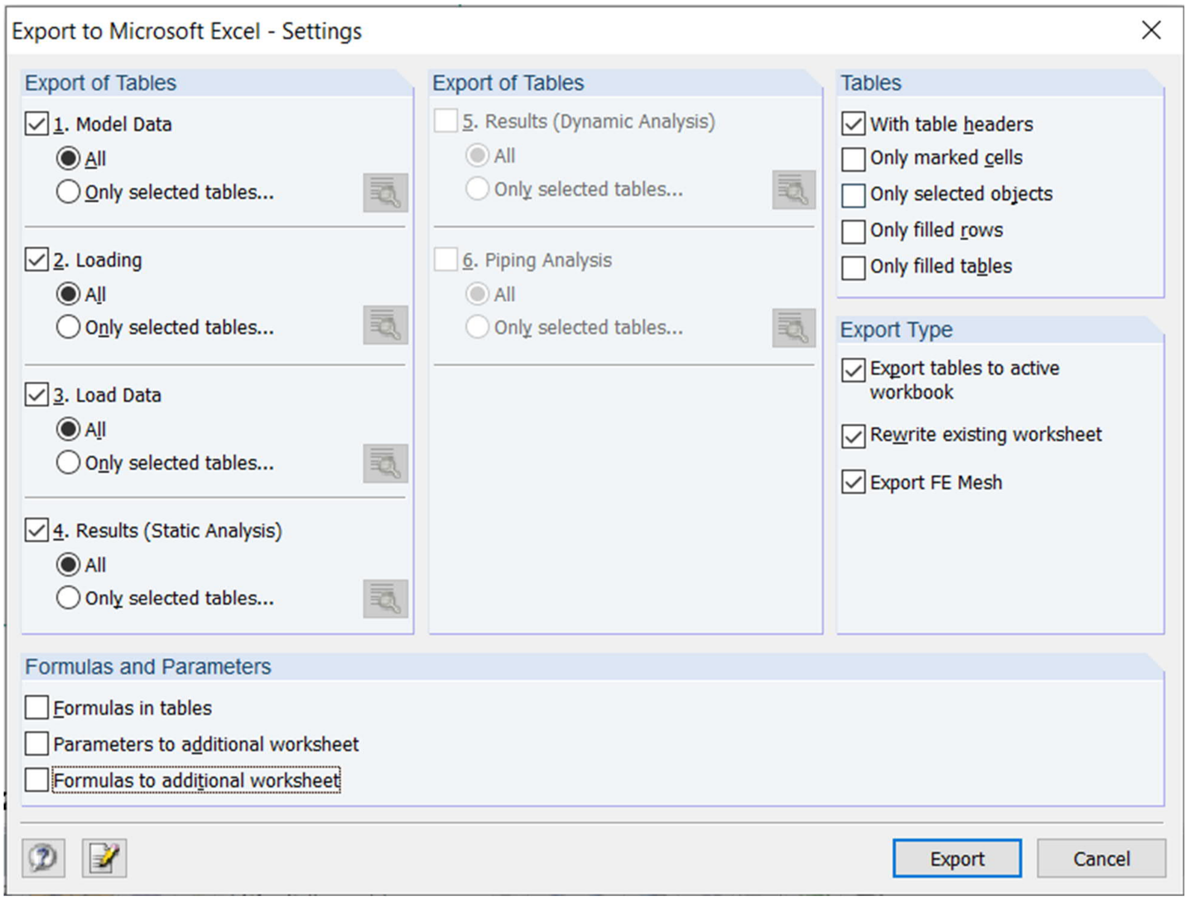


Figure 131. Export to MS Excel.

Press Export. It will take some time (Figure 132)…

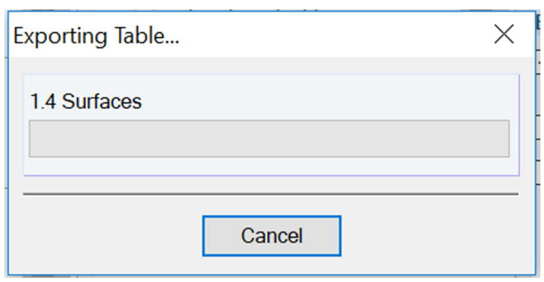


Figure 132. Exporting table.

The data comes in different sheets (Figure 133). Save the Excel file with a suitable name, for example: B3_Frame.xlsx.

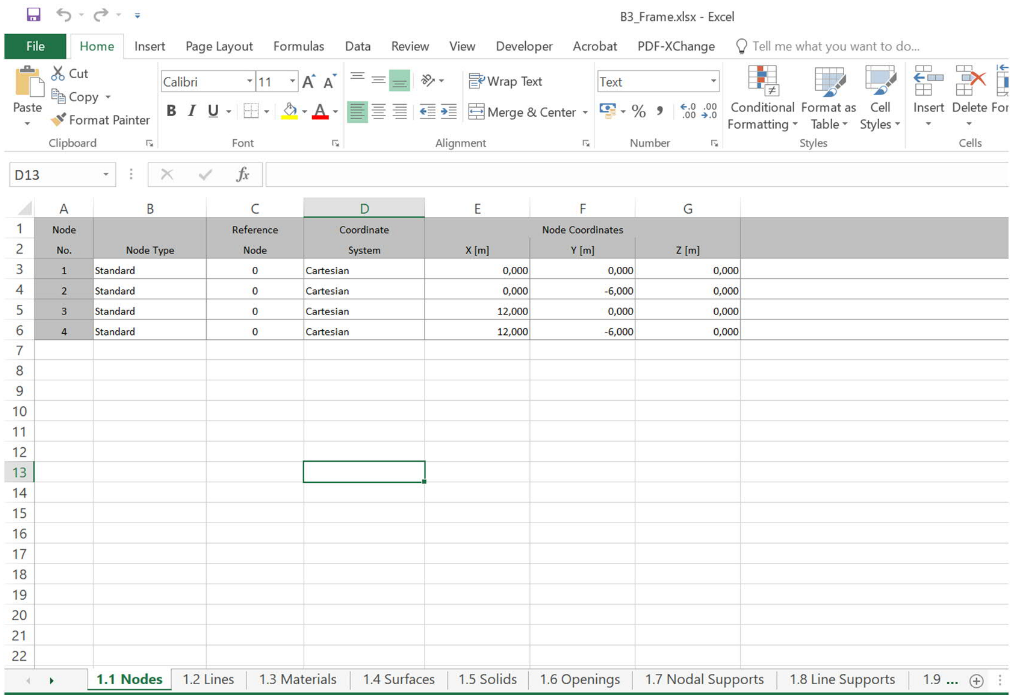


Figure 133. Exporting file.

8.2 Importing

Open and leave the Excel file to be imported open. Create a new RFEM model with the appropriate initial settings. Import the file: File -> Import. Update as shown in Figure 134.

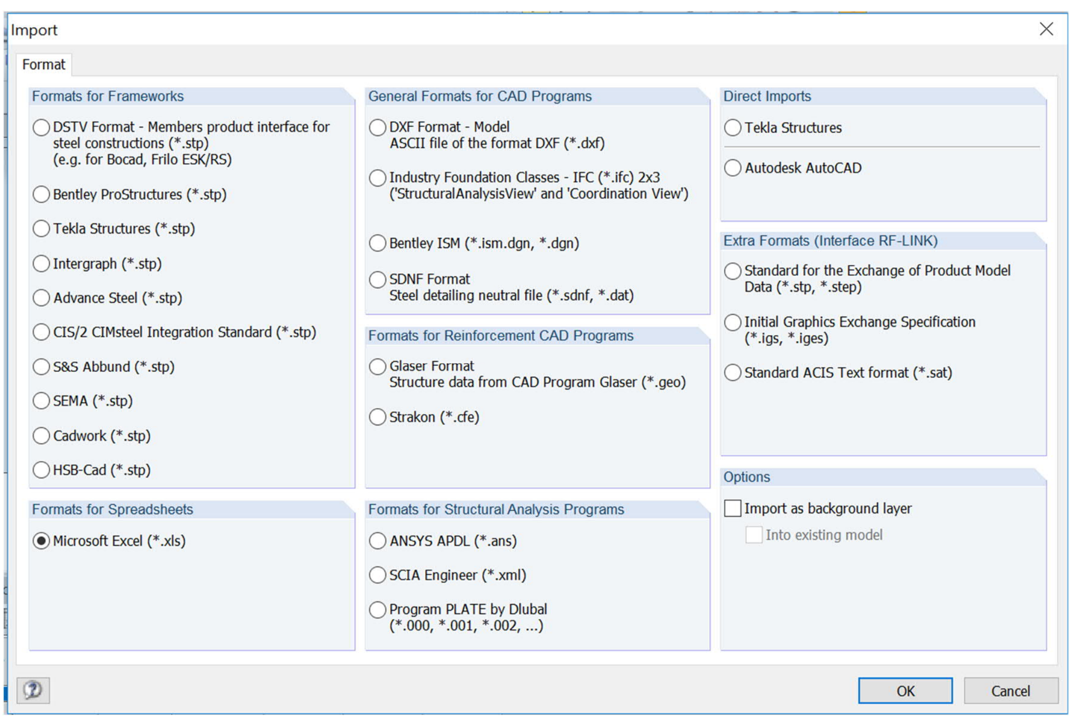


Figure 134. Importing.

Press OK. Update as shown in Figure 135. All sheets can be selected with the icon  (Select All Worksheets).

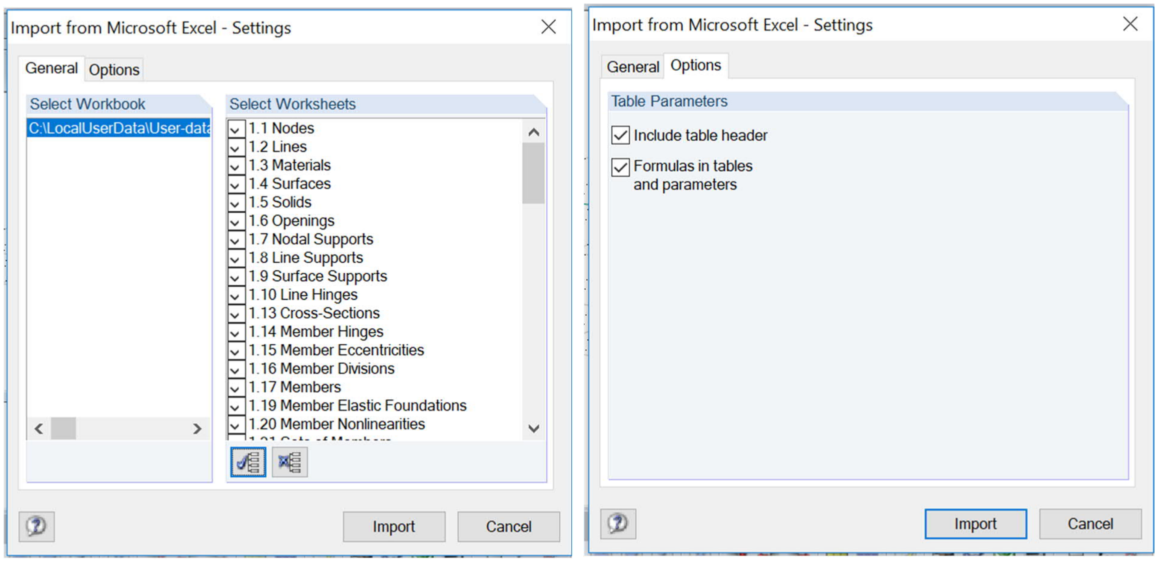


Figure 135. Import from MS Excel.

Press Import. It will take some time (Figure 136)…

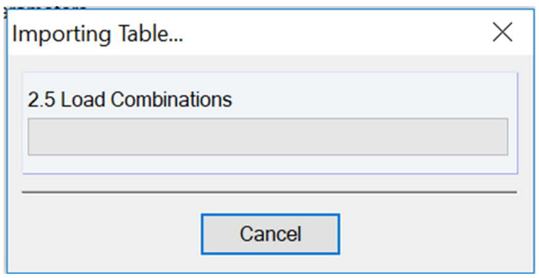


Figure 136. Importing table.

Save the model and re-analyze it. Not all information may be transferred correctly. Repair if necessary.