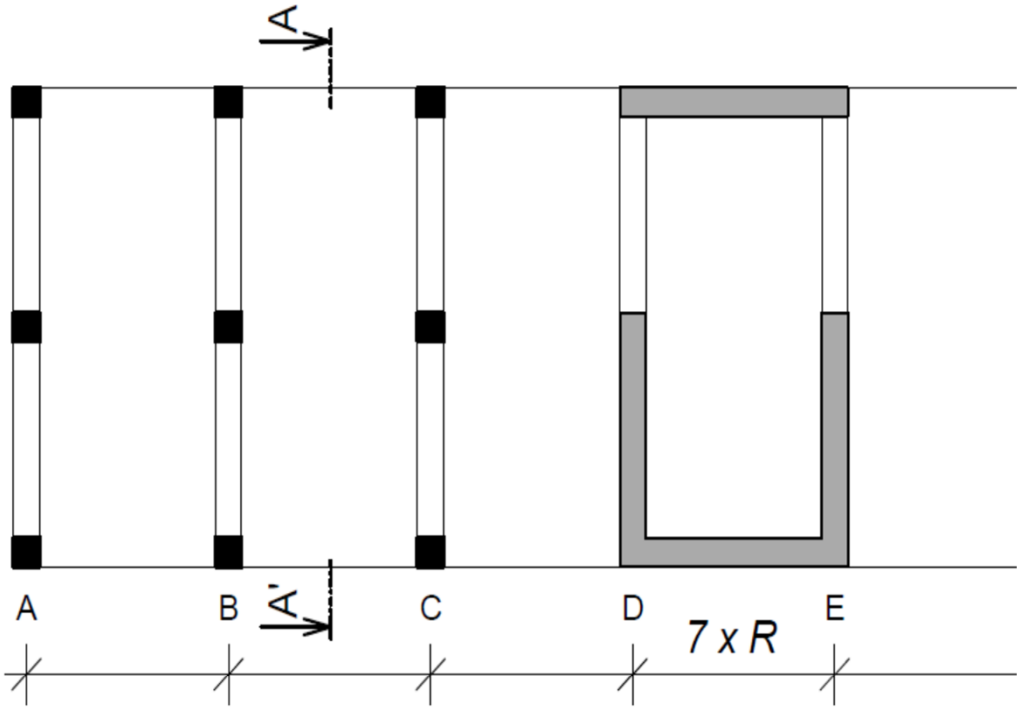
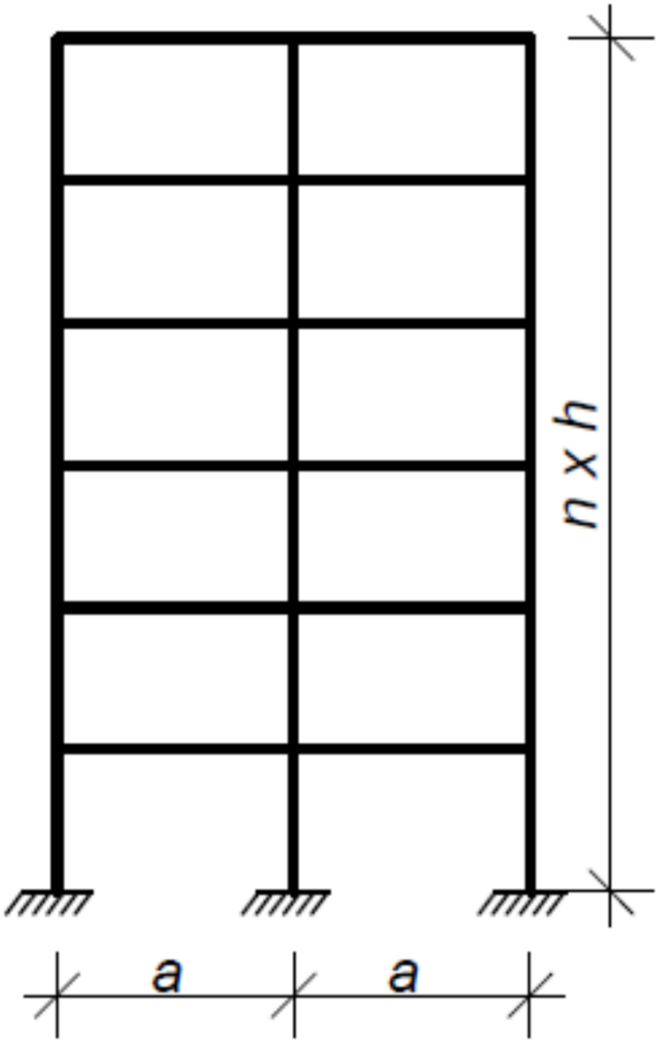
1st task:
Frame Structure



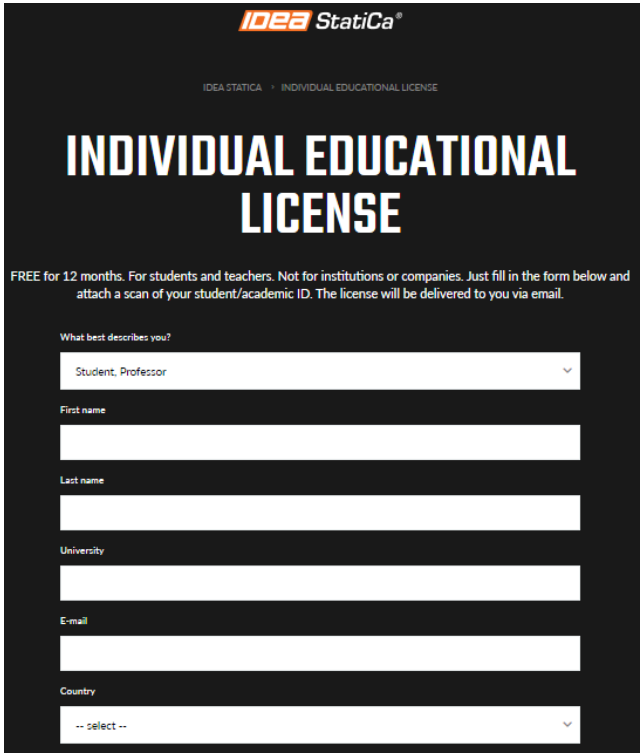


Detailed calculation of 2D frame

- Calculation of loads
- Modelling 2D frame in Finite Element Method (FEM) software
- Calculation of bending moments, shear and normal forces in the **bottom floor** of the frame
- Calculation using **Idea Statica** will be described, but you can use any software you want (Scia, Dlubal, RIBTec…) – calculation steps will be the same
- You can also use **manual calculation** (slope deflection method or force method) that you know from structural mechanics subjects

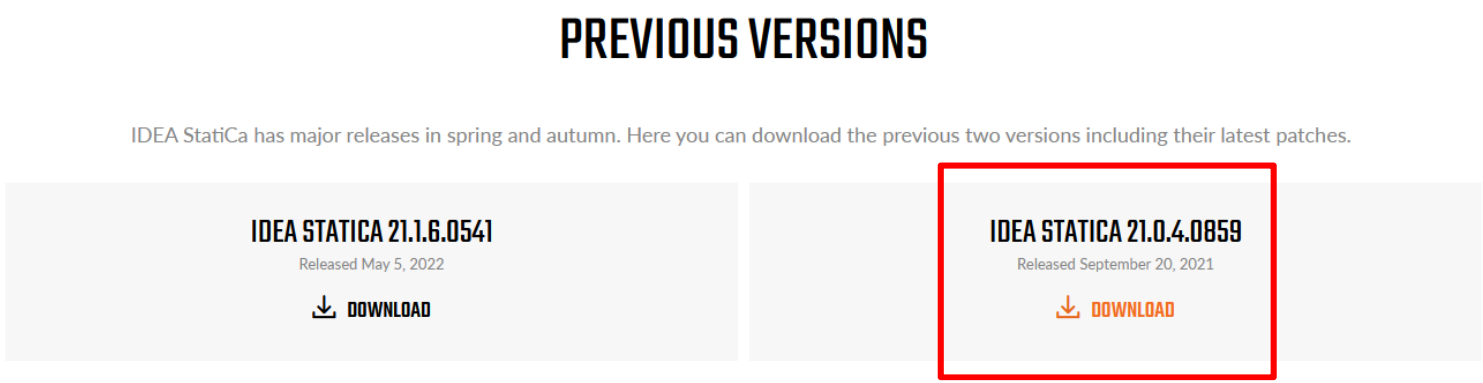
1. Installation of the program

- Go to https://www.ideastatica.com/educational-license/
- Fill in the information. Use your institutional (NOT personal) e-mail
- Wait until you receive an activation code (by e-mail, may take up to 3 days)



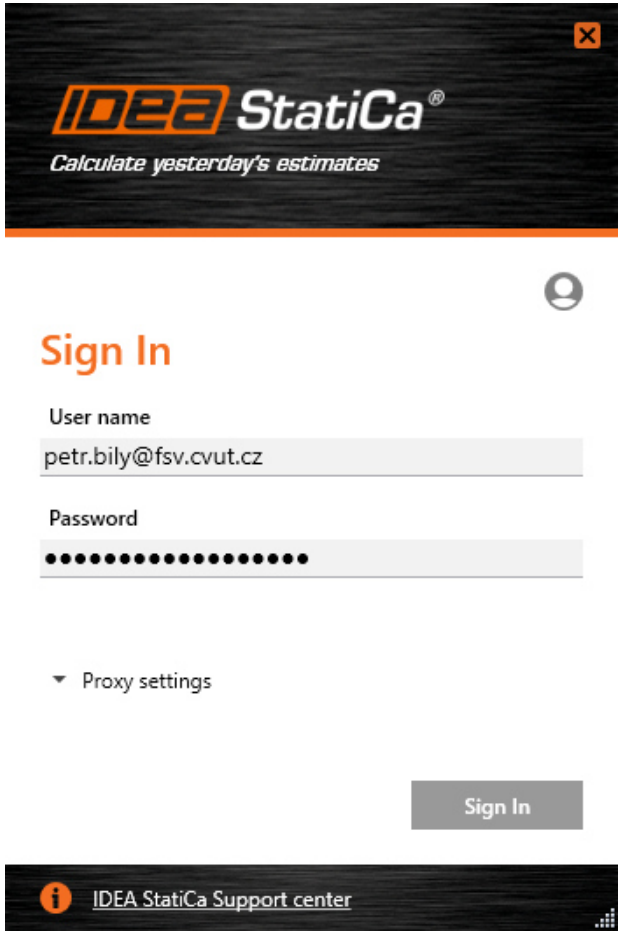
1. Installation of the program

- Go to https://www.ideastatica.com/product-downloads
- Download IdeaStatica version 21.0
- Do **NOT** download the version 22 or 21.1 – it does not containt the module for calculation of frames that we need
- Install IdeaStatica 21.0 to your computer



1. Installation of the program

- Run the program
- Fill in the login details from activation e-mail

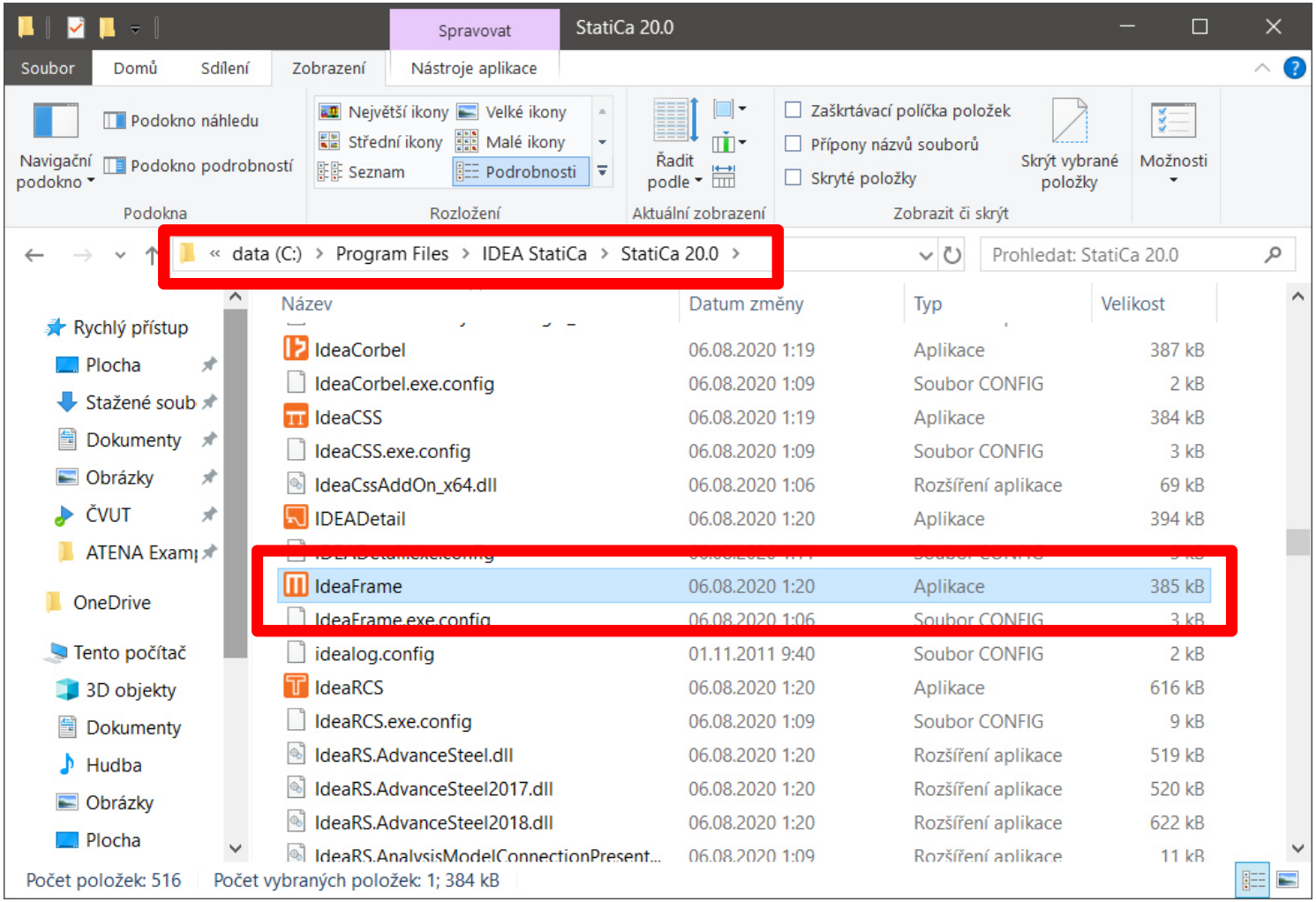


2. Opening the program

- Frame 2D module is **NOT** accessible from main program window

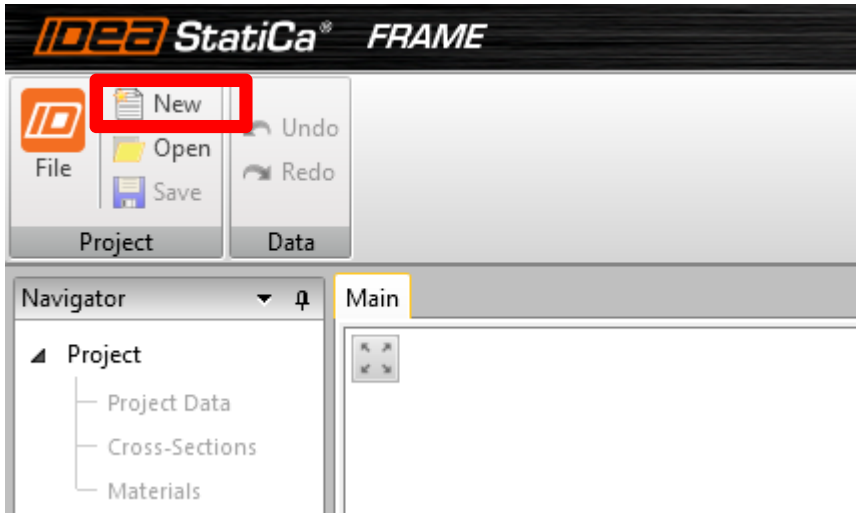
2. Opening the program

- You have to locate IdeaStatica folder in your computer and run the IdeaFrame.exe file directly



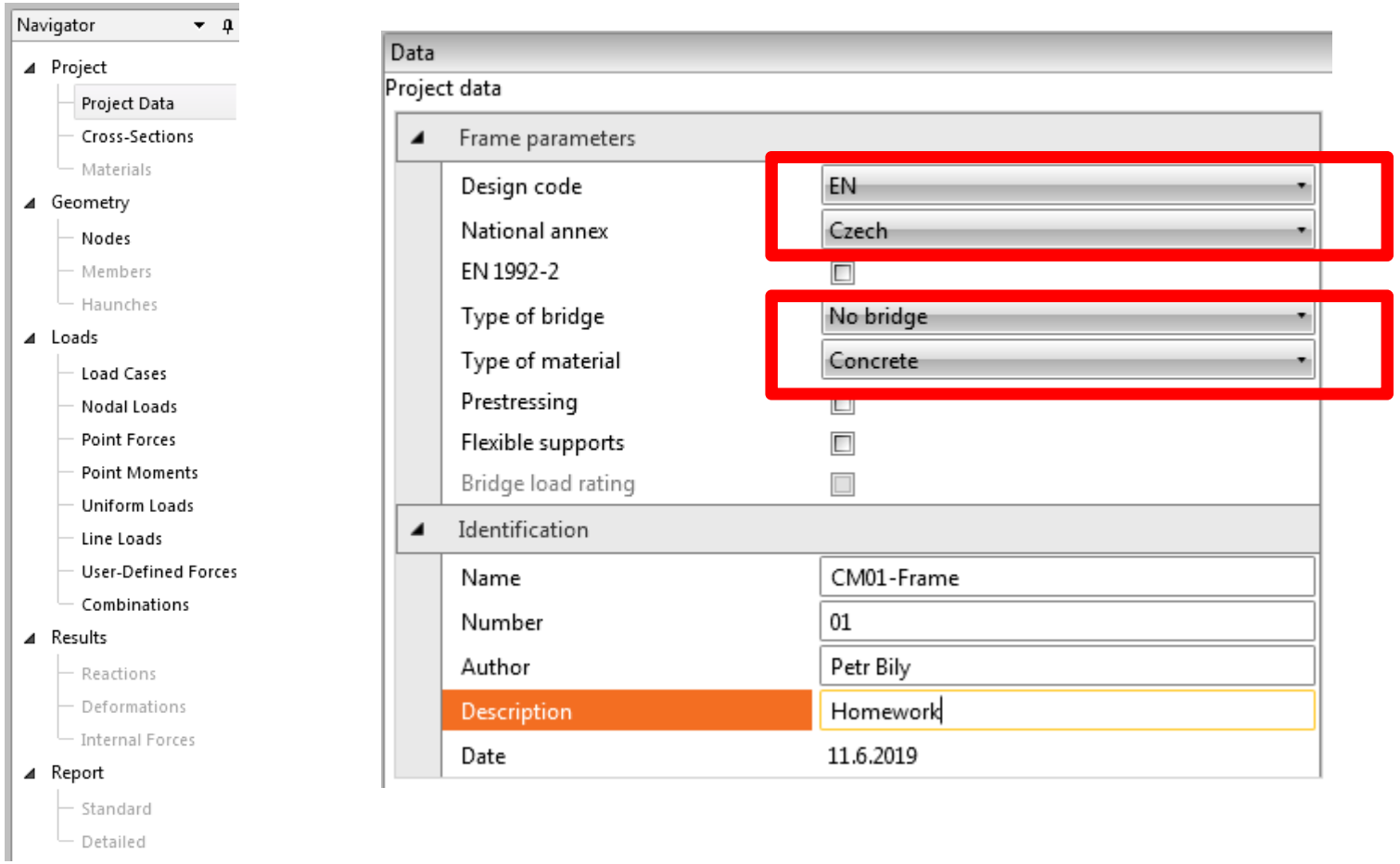
2. Opening the program

- Create new project



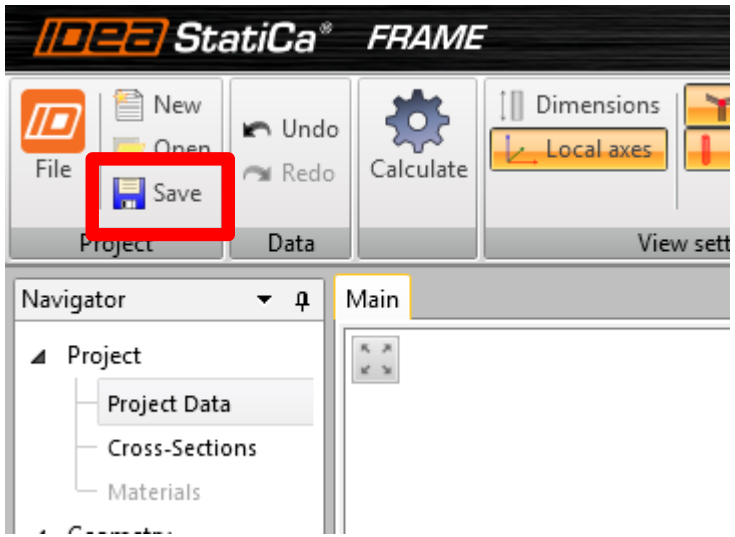
3. Project data

- Use Navigator menu to enter the structure
- Change Project data as follows:



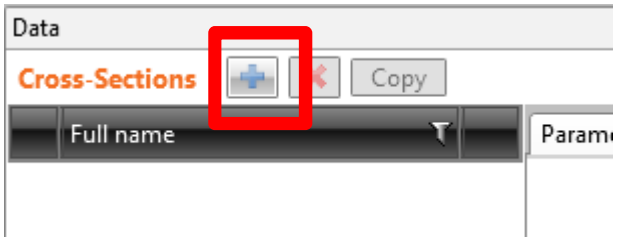
3. Project data

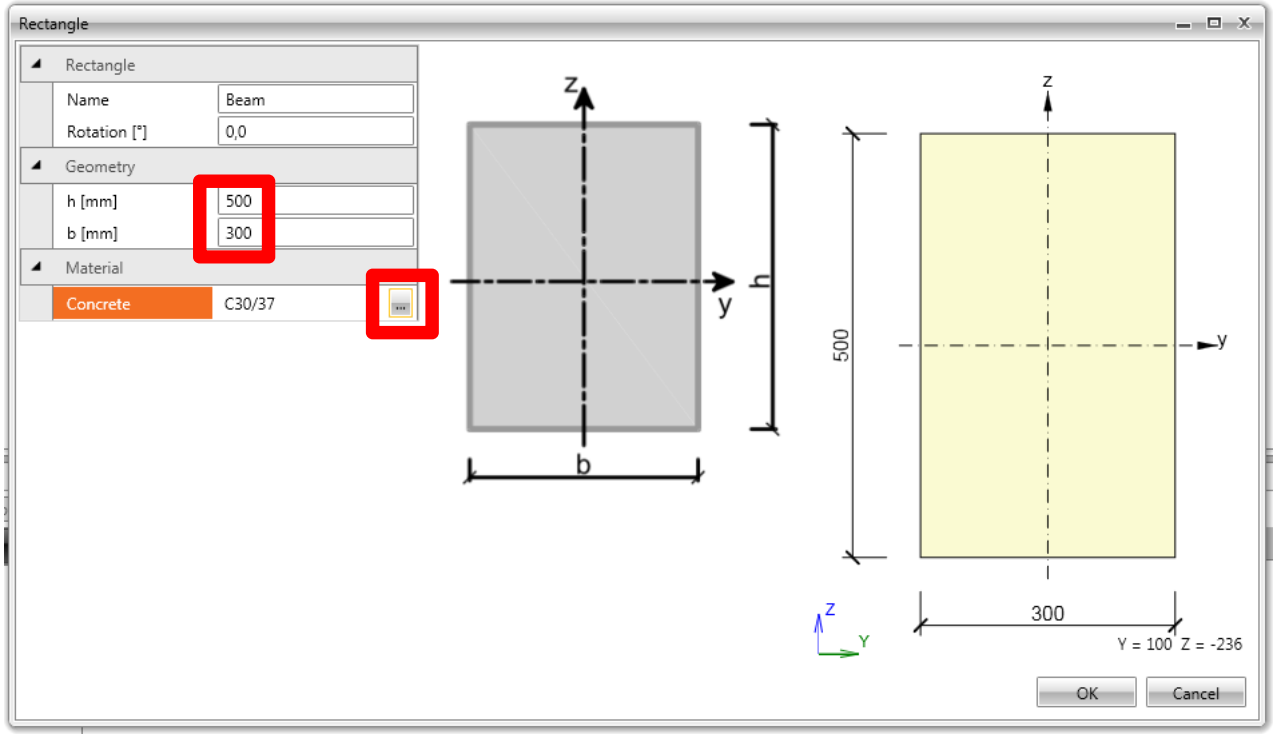
- Save and name your project
- **Save your project regularly to avoid losing unsaved data!!!**



4. Cross-sections

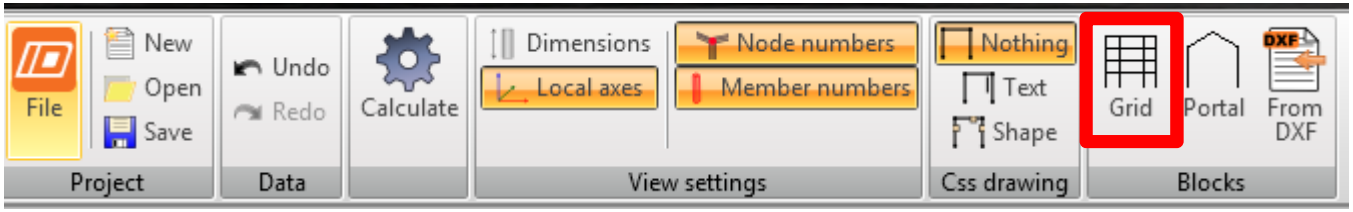
- Enter cross-sections of **<u>your</u>** beam and column (name, dimensions, material)

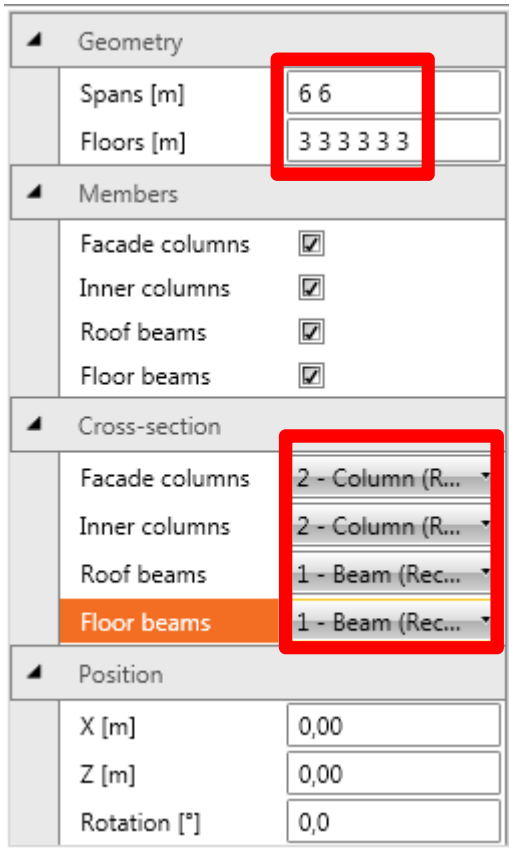




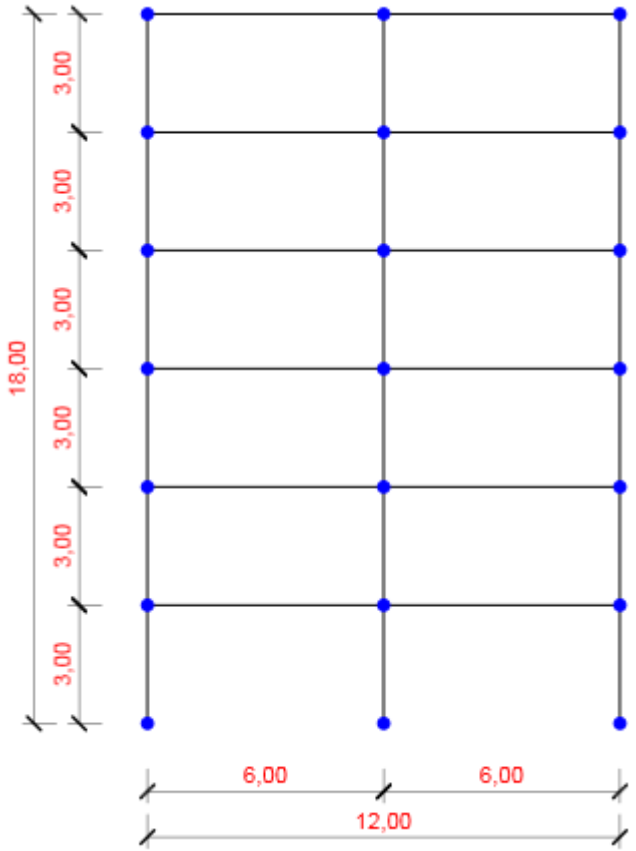
5. Creating the frame

- Go to Geometry > Nodes, use Grid to create the frame



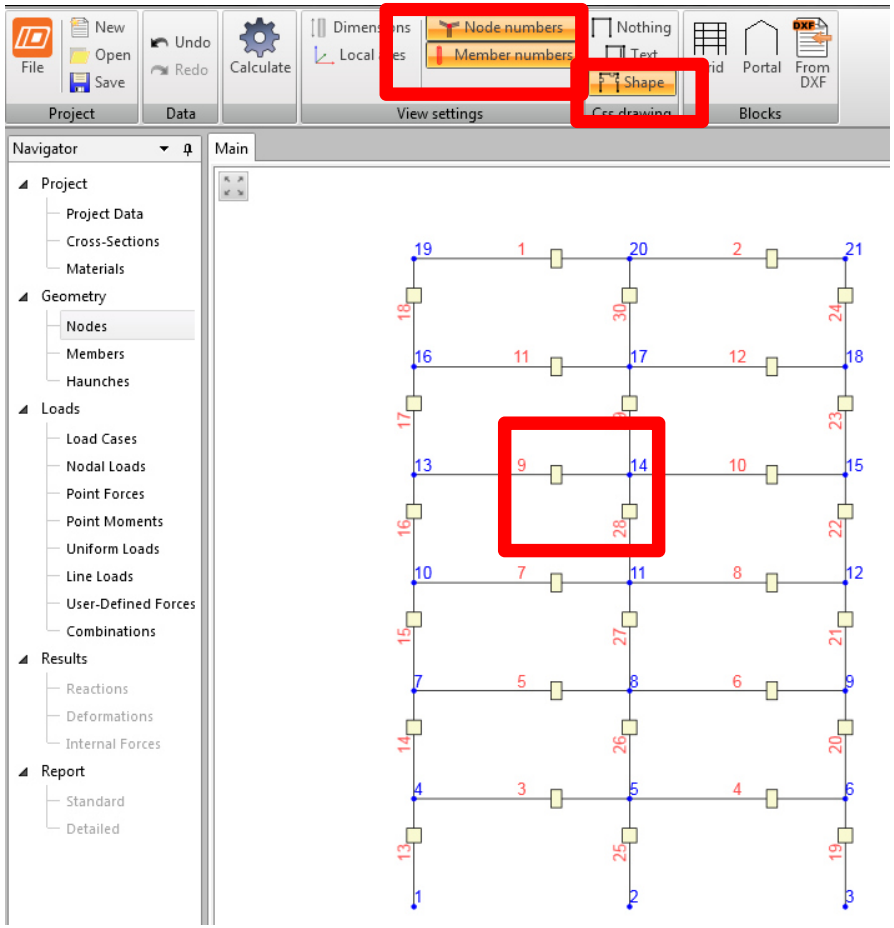


- Set dimensions and number of floors (divide the numbers simply by spacebar)
- Select cross-sections – different for beam/column



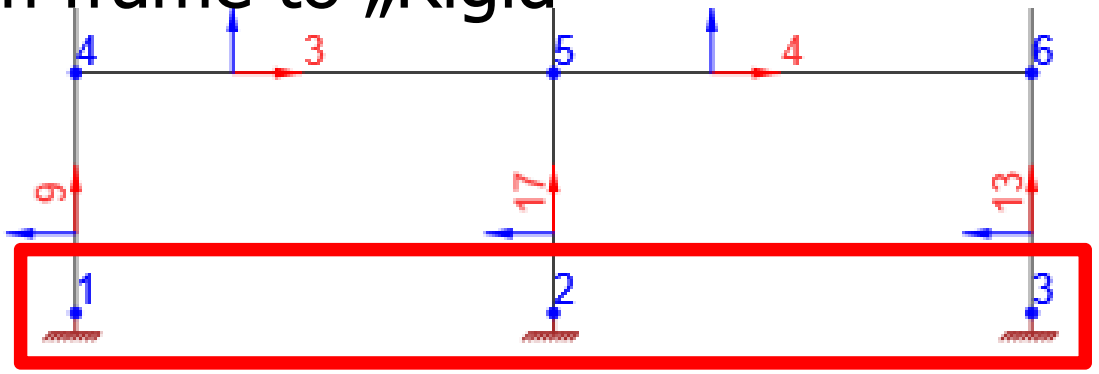
5. Creating the frame

- Display shapes to check whether you entered the cross sections correctly
- Display node and member numbers



5. Creating the frame

- Enter **supports**
- Go to Geometry > Nodes and set all degrees of freedom of nodes in the feet of the columns in bottom frame to „Rigid“



Data

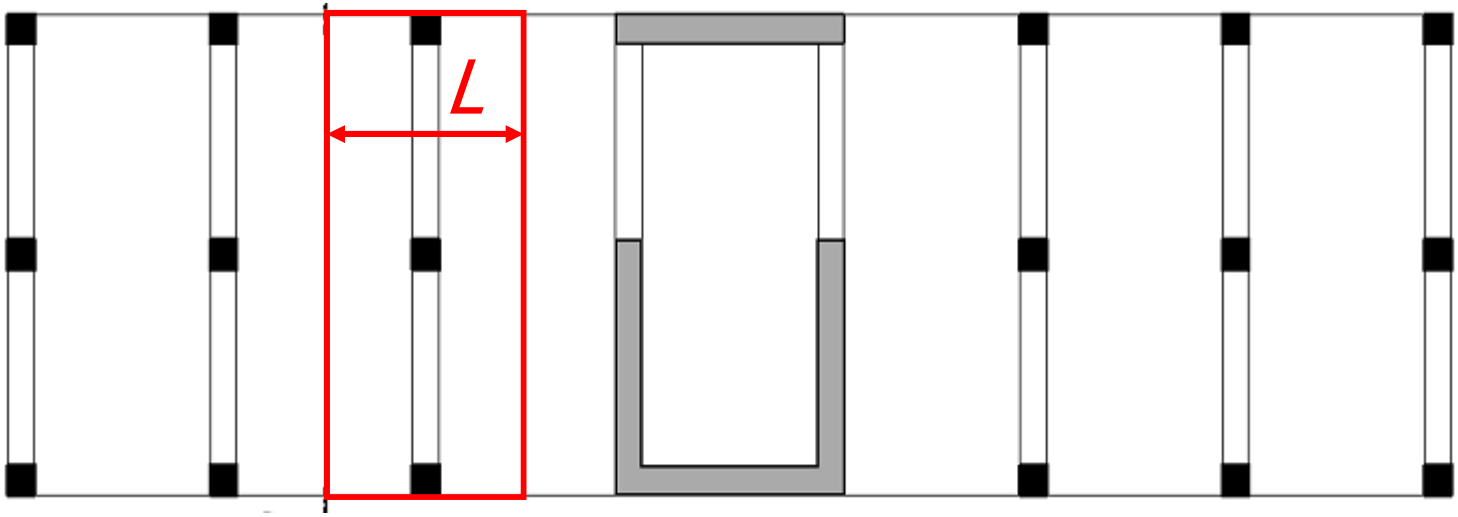
Nodes | Cleanup

Node	X [m]	Z [m]	Nodal Support X	Z	Ry
1	0,00	0,00	☑	☑	☑
2	6,00	0,00	☑	☑	☑
3	12,00	0,00	☑	☑	☑
4	0,00	3,00	☐	☐	☐

6. Loads

Self-weight of the slab +
Other permanent load

- **Manually** calculate 4 values of linear loads per 1 meter of the beam (in kN/m):
  - Characteristic permanent load in typical floor $g_{k,t}$
  - Characteristic permanent load on the roof $g_{k,r}$
  - Characteristic variable load in typical floor $q_{k,t}$
  - Characteristic variable load on the roof $q_{k,r}$
- Linear load [kN/m] = area load [kN/m^2] * L [m]

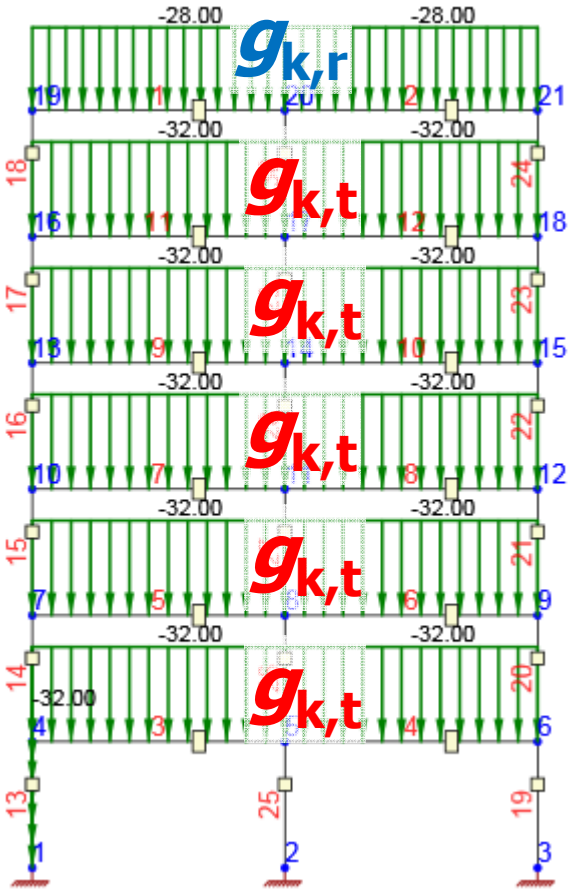


7. Load cases

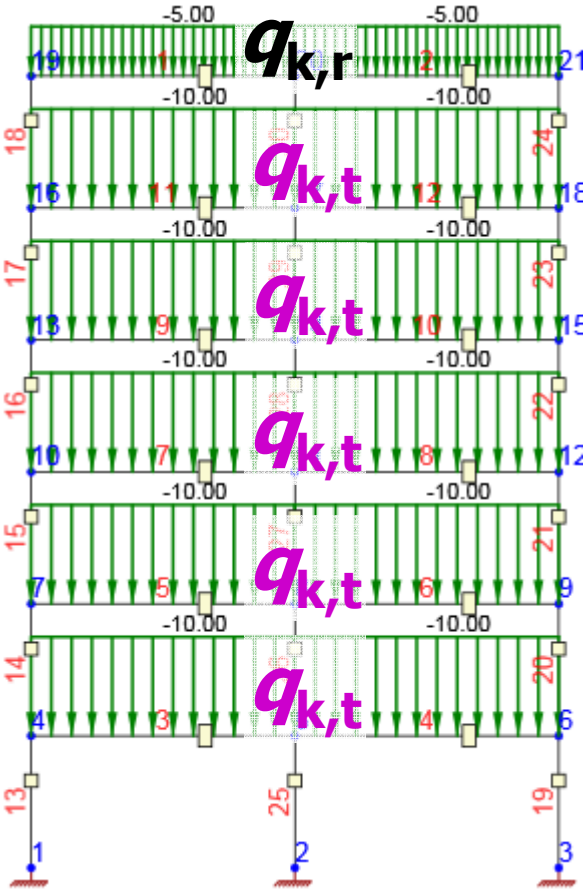
- You should consider 4 load cases:

SW: Self-weight of the frame – calculated automatically by IdeaStatica

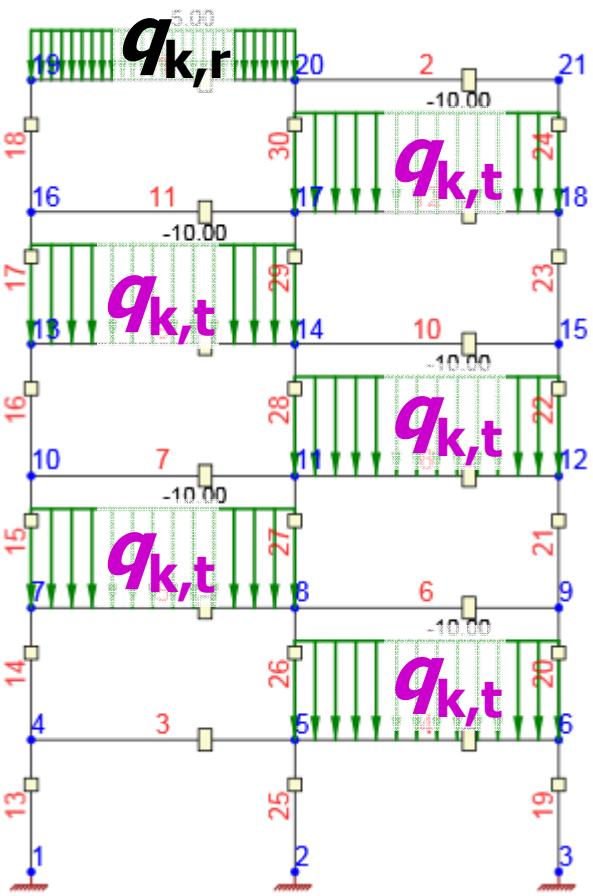
LC1: Full permanent load



LC2: Full variable load

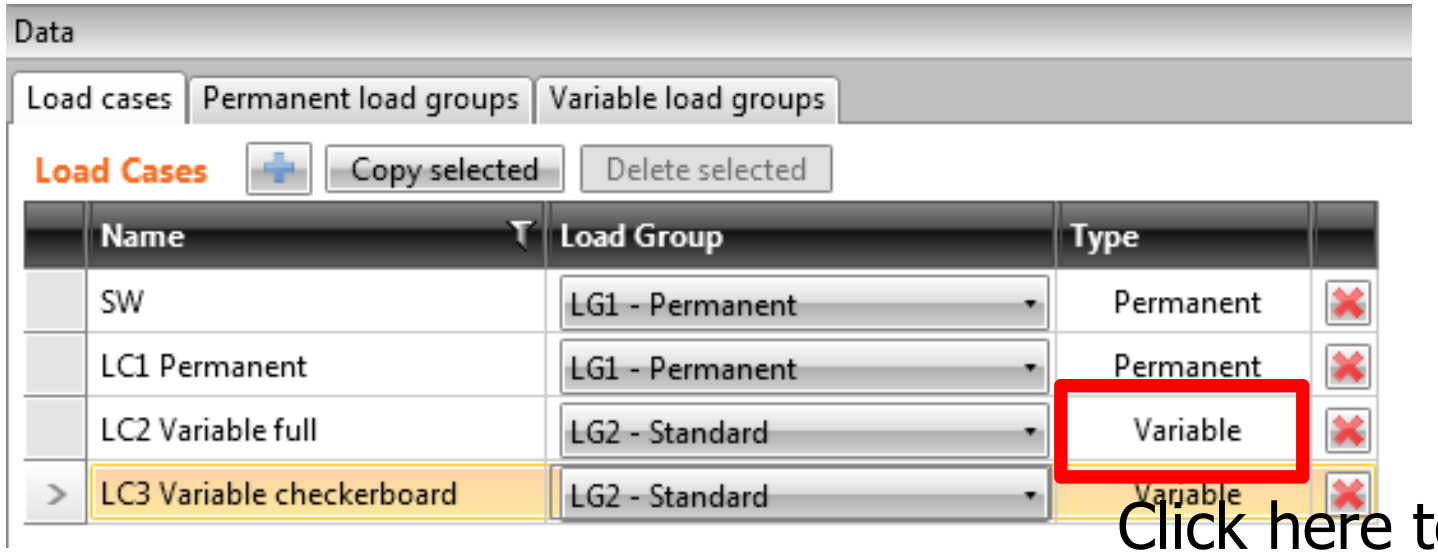


LC3: Checkerboard variable load



7. Load cases

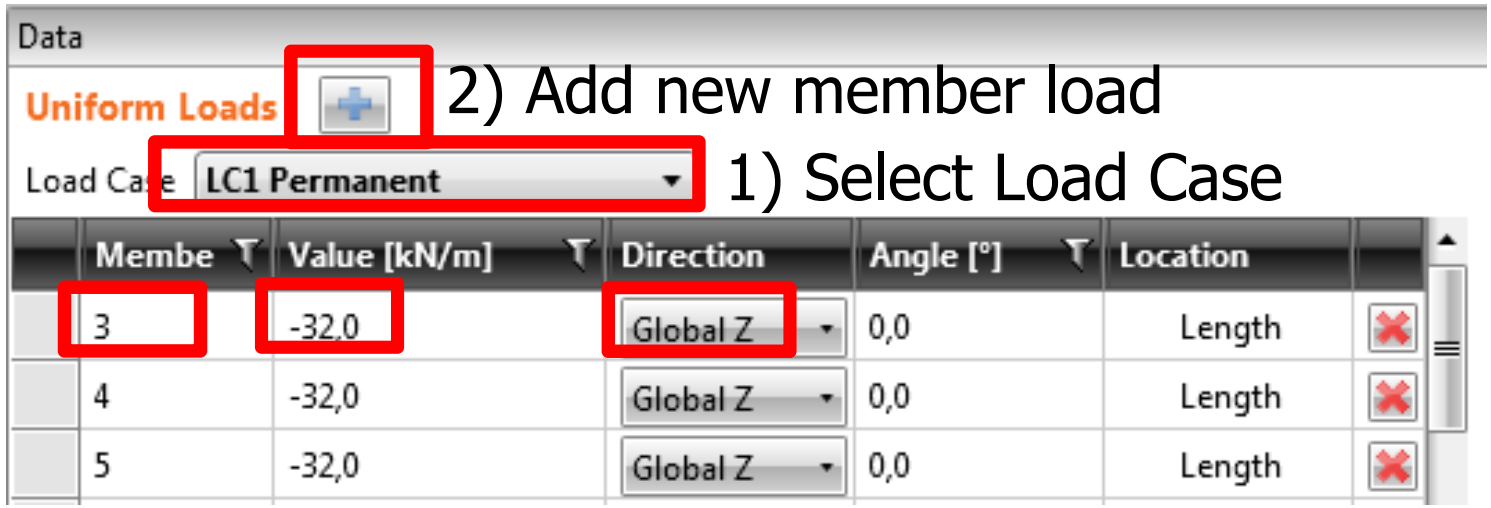
- Create the load cases, adjust Load Group and Type as follows:



- SW = self-weight of the modeled structure (frame) – automatically generated

6. Loads

- Go to „Uniform Loads“, select LC1 and enter the loads. Repeat for LC2 and LC3.



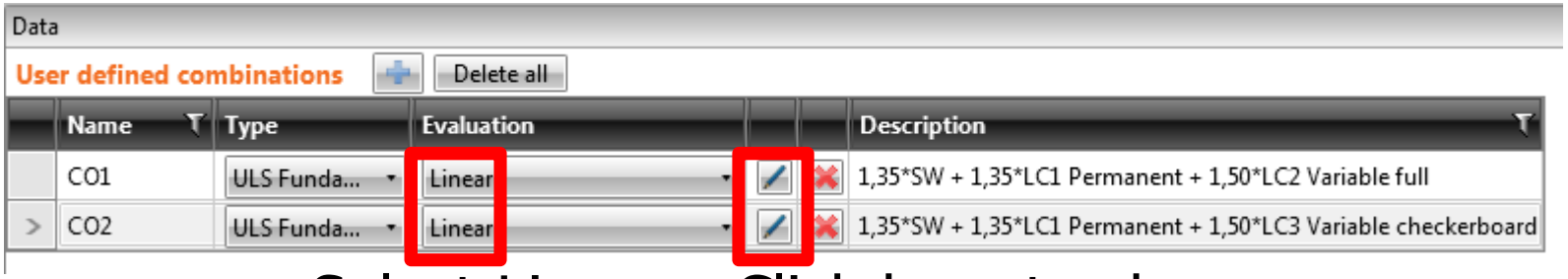
3) Enter member number

4) Enter load value (with „minus“ !)

5) Select vertical (Z) direction

8. Load combinations

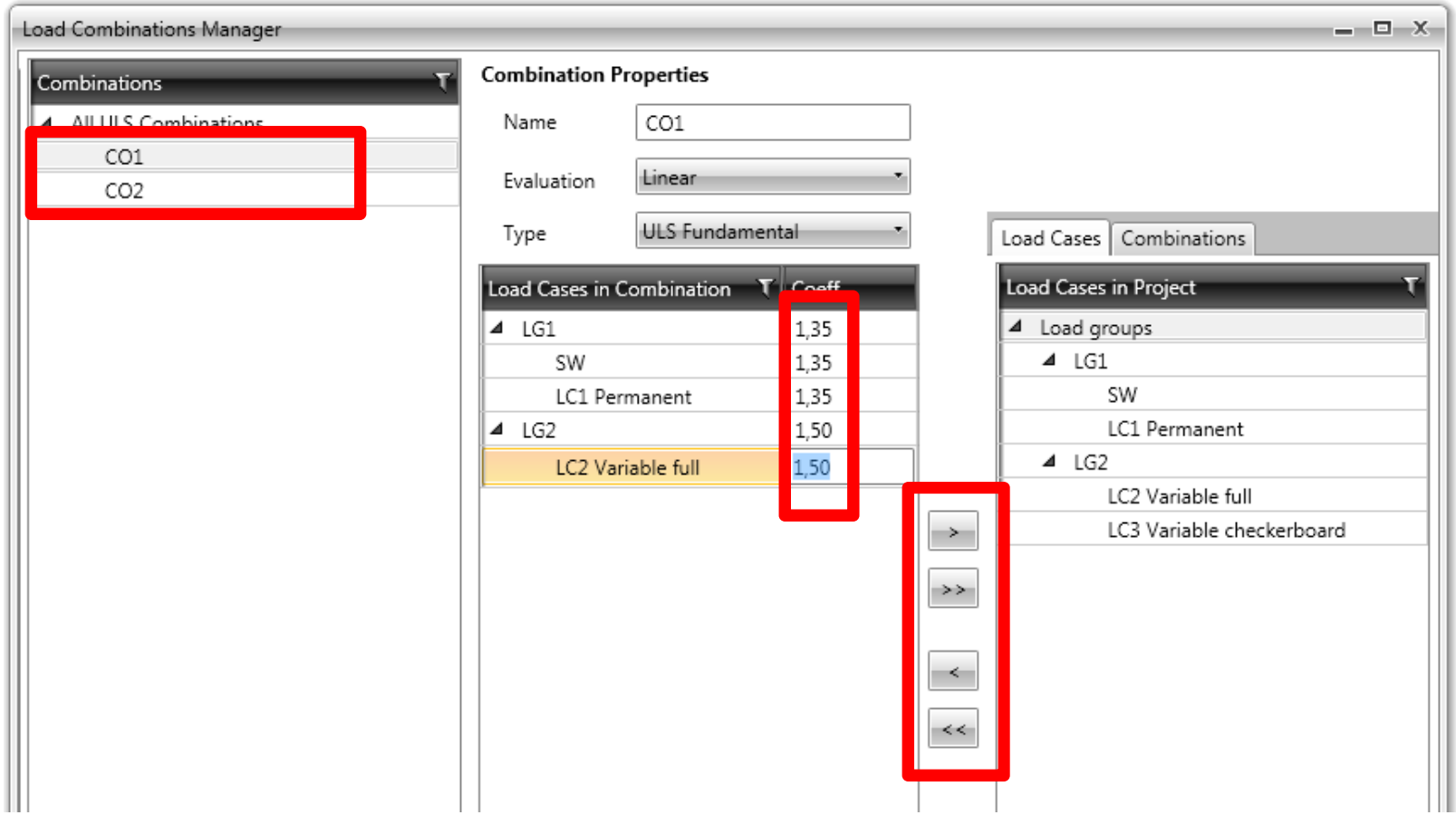
- Create 2 load combinations:
  - CO1 Full = SW+LC1+LC2
  - CO2 Checkerboard = SW+LC1+LC3



Select Linear Click here to change

- Why more combinations?
  - CO1 Full = Extreme N and support M
  - CO2 Checkerboard = Extreme V and midspan M

8. Load combinations



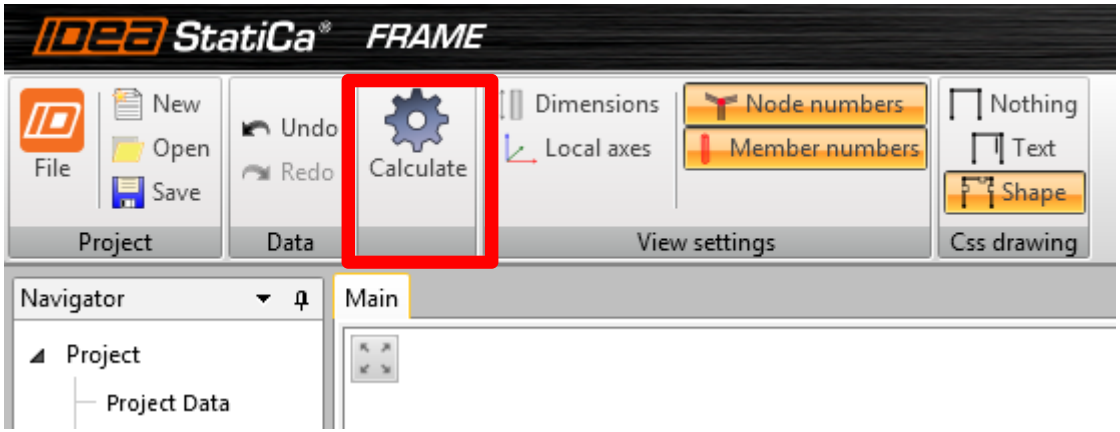
1) Select the combination

3) Adjust the coefficients (1.35 for permanent loads, 1.5 for variable)

2) Add/remove load cases to the combination

9. Results

- Calculate results



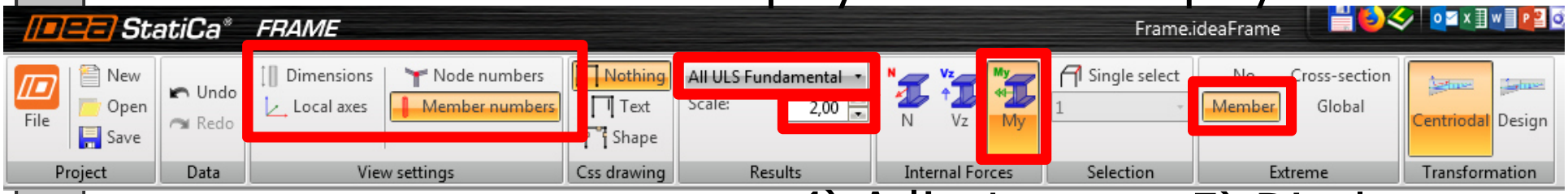
9. Results

- Go to Results > Internal Forces; check int. forces for CO1, CO2 and „envelope“ (All ULS)

1) Turn off all labels **except** Member numbers

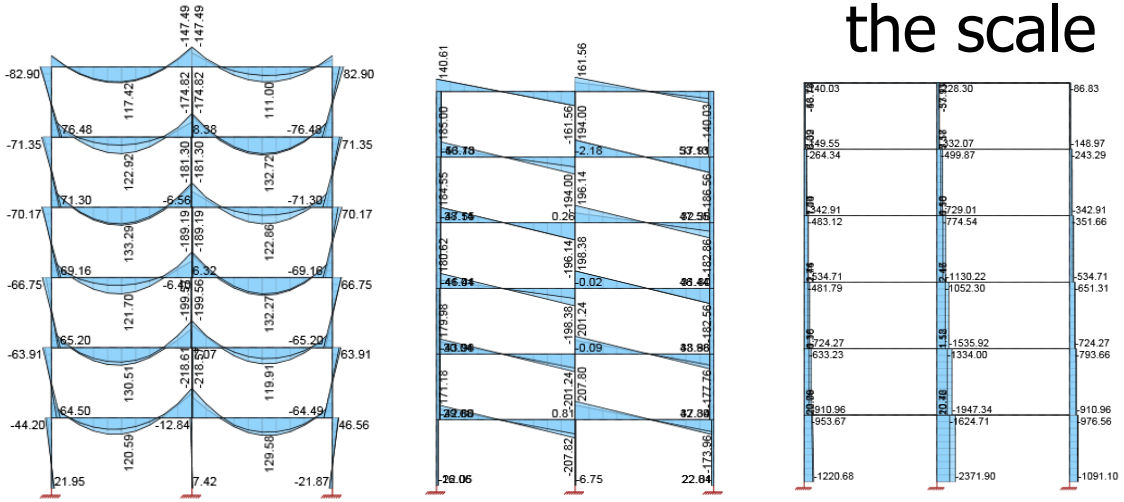
2) Select load combination to display

3) Select internal force to display



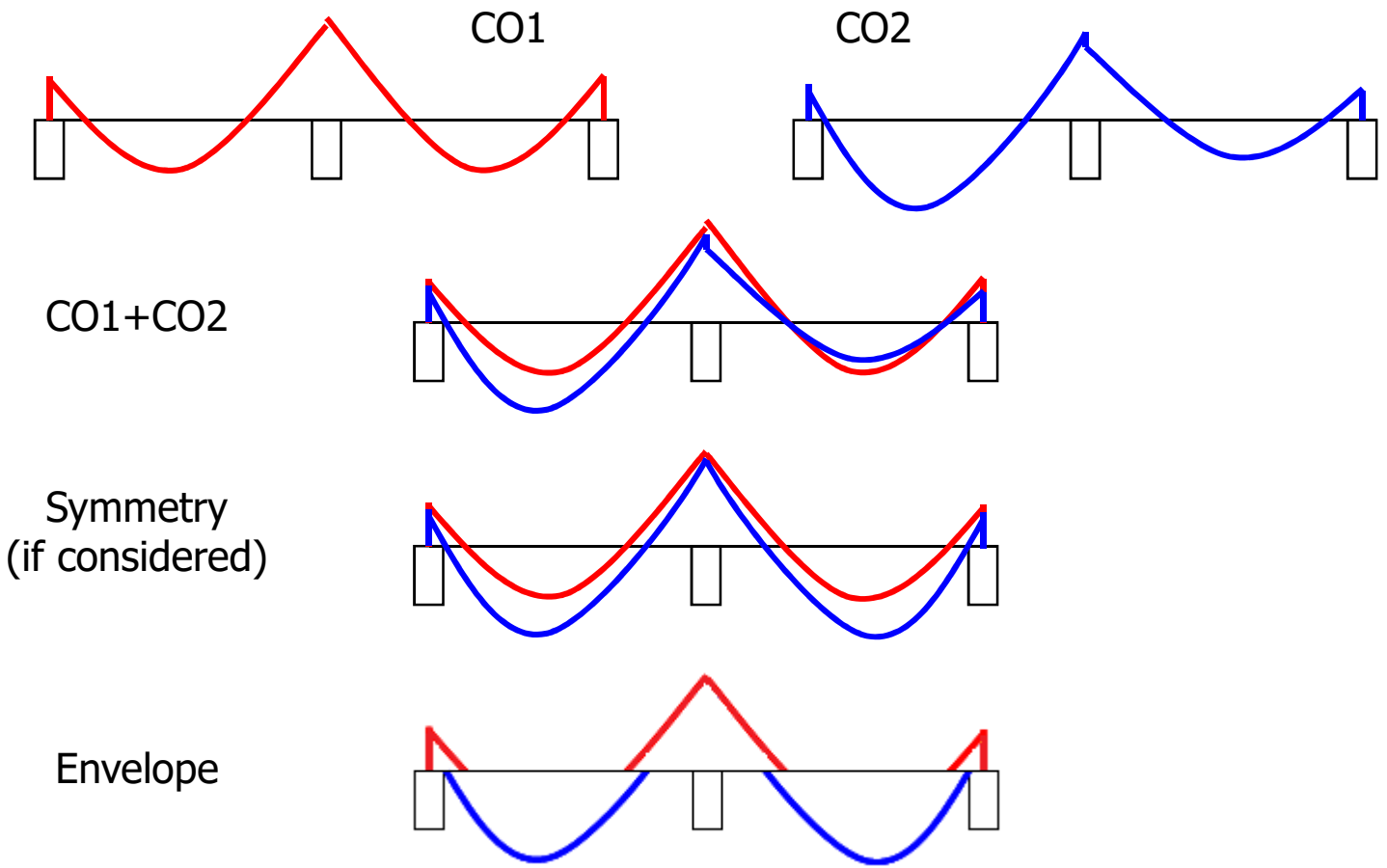
4) Adjust the scale

5) Display extreme values on members



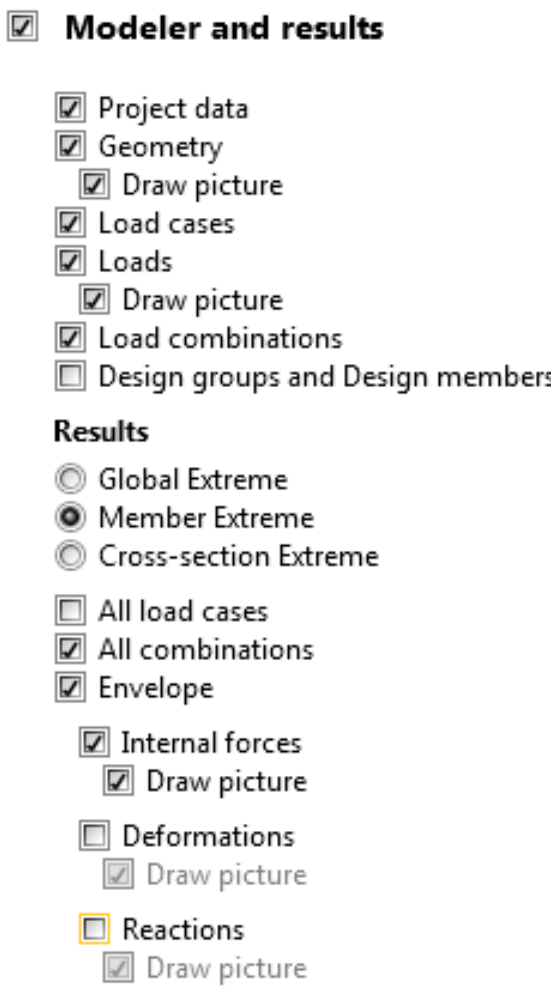
9. Results

- What is an „envelope“?



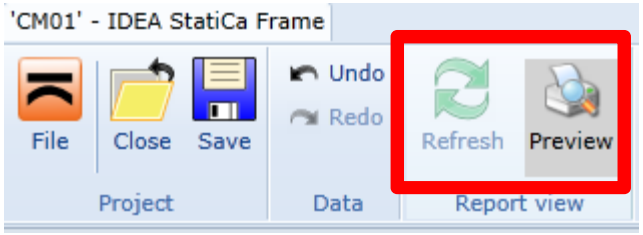
10. Report

- Create Detailed Report: Navigator > Report > Detailed; set as follows:



Refresh, then

Go to Print Preview and print the report

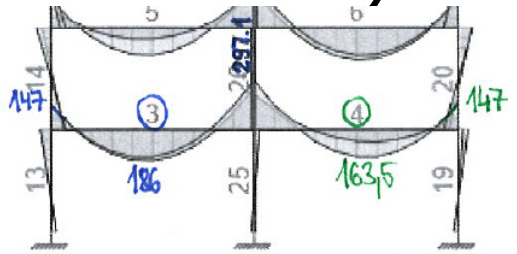


For the next week, **bring the printed report** together with manual calculation of loads.

10. Report

- In the printed report, **manually** add the values of the bending moments to the schemes of the **bottom floor** for **all combinations** (you will find the values in the tables):

The report will **not be accepted** without this manual ammendment !!!



All combinations, My [kNm], Centroidal forces

Internal forces, Member Extreme, Centroidal forces

Member	Combi	Position [m]	N [kN]	V_z [kN]	M_y [kNm]
1	CO1 Full(1)	0.00	-60.2	126.8	-87.9
1	CO2 Checkedboard(2)	0.00	-53.4	91.2	-69.4
1	CO1 Full(1)	6.00	-60.2	-132.5	-105.0
1	CO1 Full(1)	3.00	-60.2	-2.9	98.0
2	CO1 Full(1)	0.00	-60.2	132.5	-105.0
2	CO2 Checkedboard(2)	0.00	-44.4	132.9	-95.3
2	CO1 Full(1)	6.00	-60.2	-126.8	-87.9
2	CO2 Checkedboard(2)	3.00	-44.4	3.3	109.1
3	CO2 Checkedboard(2)	0.00	12.4	238.9	-145.3
3	CO1 Full(1)	0.00	27.9	232.0	-147.0
3	CO1 Full(1)	6.00	27.9	-282.0	-297.1
3	CO2 Checkedboard(2)	3.00	12.4	-18.1	186.0
4	CO1 Full(1)	0.00	27.9	282.0	-297.1
4	CO2 Checkedboard(2)	0.00	31.3	165.4	-210.0
4	CO1 Full(1)	6.00	27.9	-232.0	-147.0
4	CO1 Full(1)	3.00	27.9	25.0	163.5
5	CO1 Full(1)	0.00	1.3	240.5	-173.8