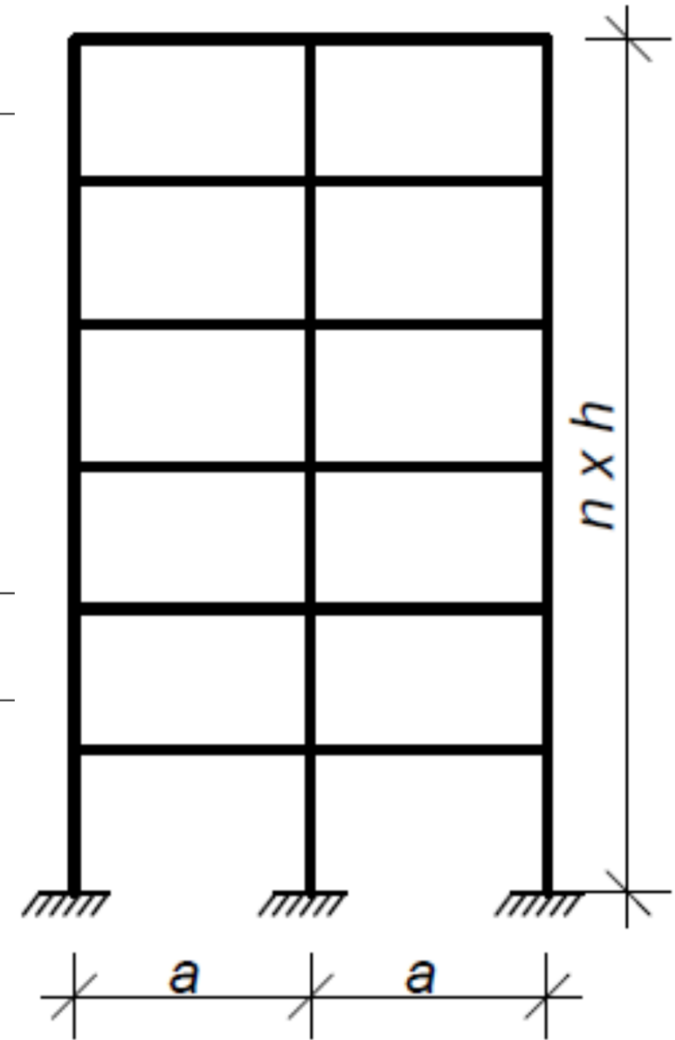
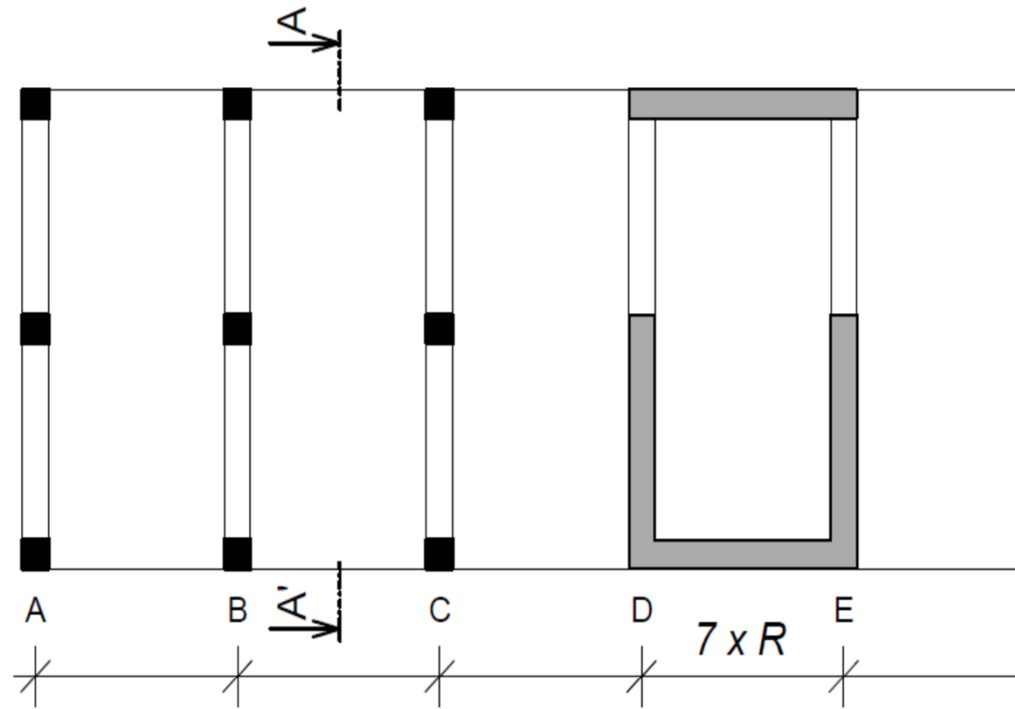


1st task: Frame Structure

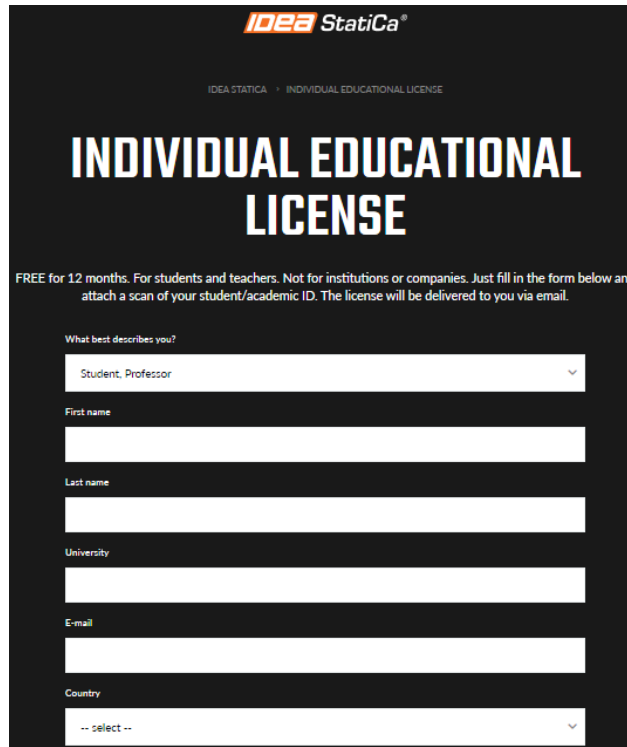


Detailed calculation of 2D frame

- 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)



The image shows a screenshot of the IDEA StatiCa Individual Educational License form. The form is titled "INDIVIDUAL EDUCATIONAL LICENSE" and is set against a dark background. At the top, the IDEA StatiCa logo is visible. Below the logo, the text "IDEA STATICA · INDIVIDUAL EDUCATIONAL LICENSE" is displayed. The main heading "INDIVIDUAL EDUCATIONAL LICENSE" is in large, bold, white letters. Below the heading, a note states: "FREE for 12 months. For students and teachers. Not for institutions or companies. Just fill in the form below and attach a scan of your student/academic ID. The license will be delivered to you via email." The form contains several input fields: a dropdown menu for "What best describes you?" with "Student, Professor" selected; text input fields for "First name", "Last name", and "University"; an "E-mail" field; and a dropdown menu for "Country" with "-- select --" selected.

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 contain the module for calculation of frames that we need
- Install IdeaStatica 21.0 to your computer

PREVIOUS VERSIONS

IDEA StatiCa has major releases in spring and autumn. Here you can download the previous two versions including their latest patches.

IDEA STATICA 21.1.6.0541

Released May 5, 2022

↓ DOWNLOAD

IDEA STATICA 21.0.4.0859

Released September 20, 2021

↓ DOWNLOAD

1. Installation of the program

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

IDEA StatiCa®
Calculate yesterday's estimates

Sign In

User name
petr.bily@fsv.cvut.cz

Password
●●●●●●●●●●●●●●●●

▼ Proxy settings

Sign In

IDEA StatiCa Support center

2. Opening the program

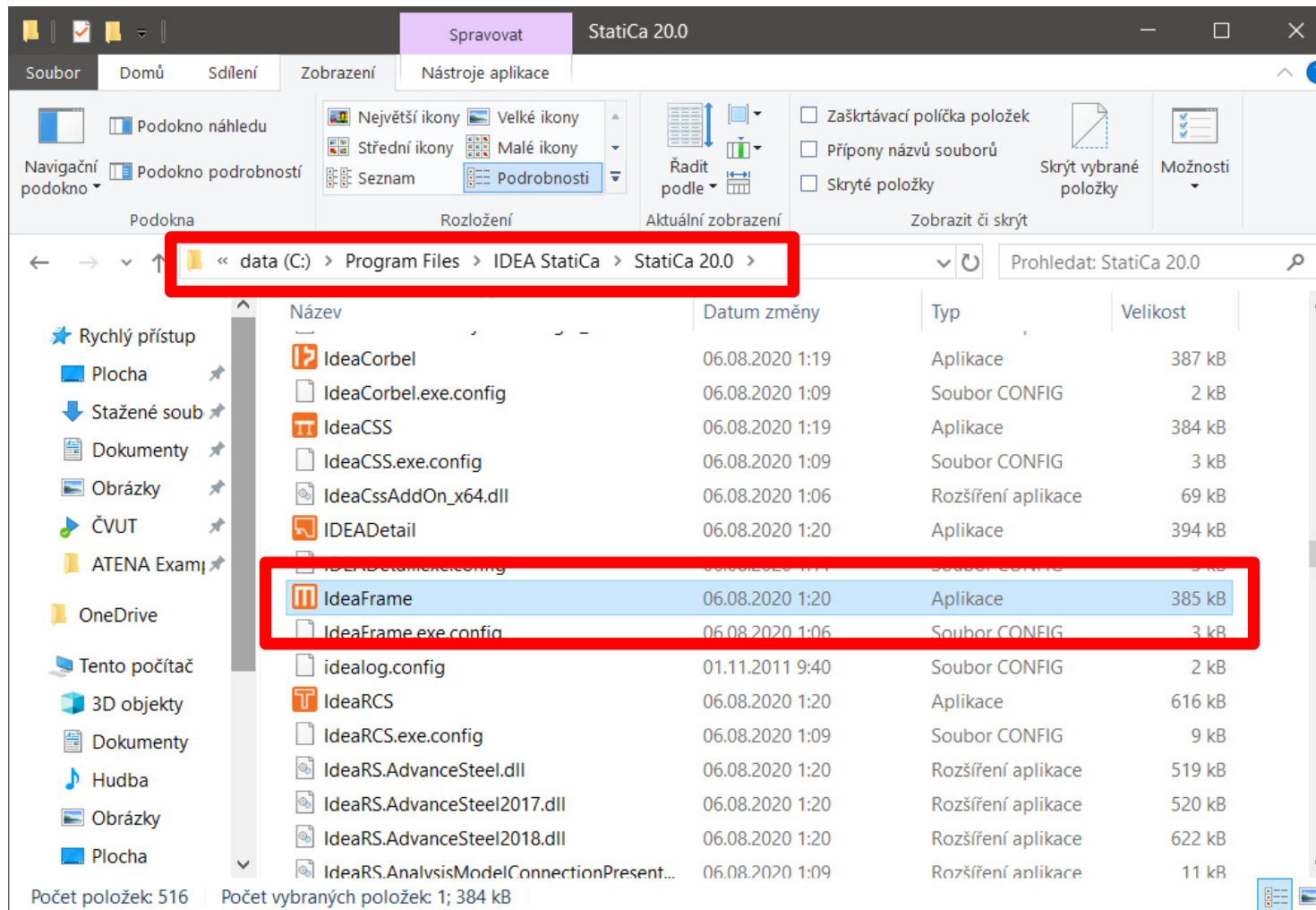
- Frame 2D module is **NOT** accessible from main program window



The screenshot displays the main interface of the IDEA StatiCa software. At the top, the logo "IDEA StatiCa" is visible with the tagline "Calculate yesterday's estimates". Below the logo, there is a navigation bar with three main categories: "STEEL", "CONCRETE", and "BIM". The "CONCRETE" category is highlighted with a red box. To the right of the navigation bar is a "Support center" link. On the left side, there is a vertical menu with four options: "Detail", "RCS", "Beam", and "Column". This menu is also highlighted with a red box. In the center of the interface, there is a section titled "Concrete & Prestressing made simple" with the text: "We solved complexity of design code for you. Design of members, cross-sections and details of reinforced and prestressed concrete structures." Below this text, there are three columns of content: "Precast concrete structures" (with a diagram of a T-section), "Bridges" (with a diagram of a bridge structure), and "Details" (with a diagram of a reinforced concrete detail). At the bottom left, there is a button labeled "Open existing project" and "Import from other program". At the bottom right, there is a footer with the text "Want to get certified?" and "Keep in touch at" followed by social media icons for LinkedIn, Facebook, and YouTube.

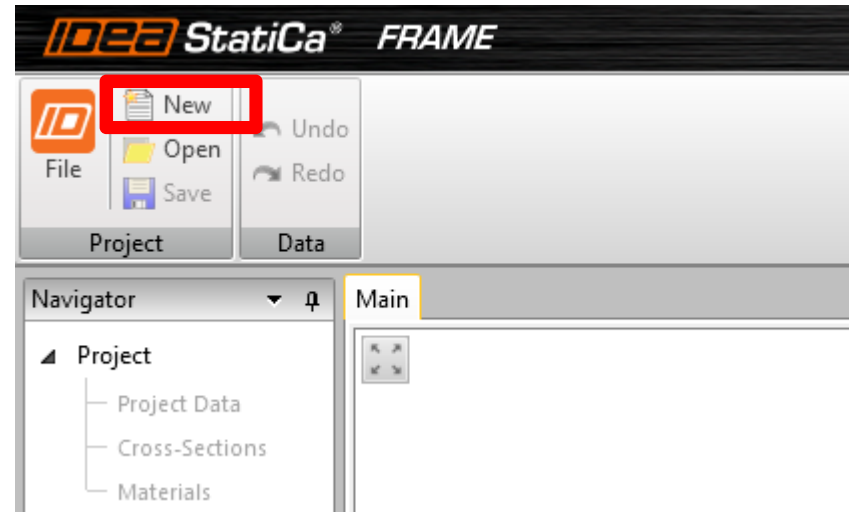
2. Opening the program

- 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

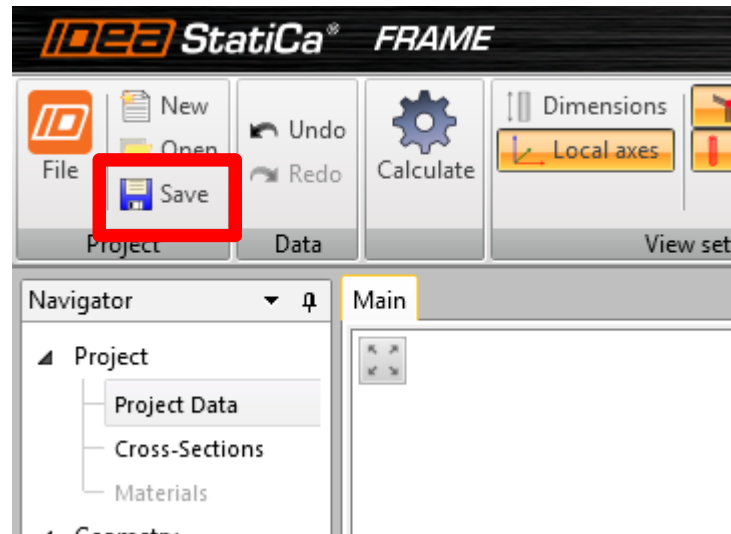
- Use Navigator menu to enter the structure
- Change Project data as follows:

The screenshot displays the 'Data' window in a software application, specifically the 'Project data' section. On the left, a 'Navigator' pane shows a tree view with 'Project Data' selected. The main window is divided into two sections: 'Frame parameters' and 'Identification'. In the 'Frame parameters' section, four dropdown menus are highlighted with red boxes: 'Design code' (set to 'EN'), 'National annex' (set to 'Czech'), 'Type of bridge' (set to 'No bridge'), and 'Type of material' (set to 'Concrete'). Below these are checkboxes for 'EN 1992-2', 'Prestressing', 'Flexible supports', and 'Bridge load rating'. The 'Identification' section contains text input fields for 'Name' (CM01-Frame), 'Number' (01), 'Author' (Petr Bily), 'Description' (Homework), and 'Date' (11.6.2019). The 'Description' field is highlighted in orange.

Project data	
Frame parameters	
Design code	EN
National annex	Czech
EN 1992-2	<input type="checkbox"/>
Type of bridge	No bridge
Type of material	Concrete
Prestressing	<input type="checkbox"/>
Flexible supports	<input type="checkbox"/>
Bridge load rating	<input type="checkbox"/>
Identification	
Name	CM01-Frame
Number	01
Author	Petr Bily
Description	Homework
Date	11.6.2019

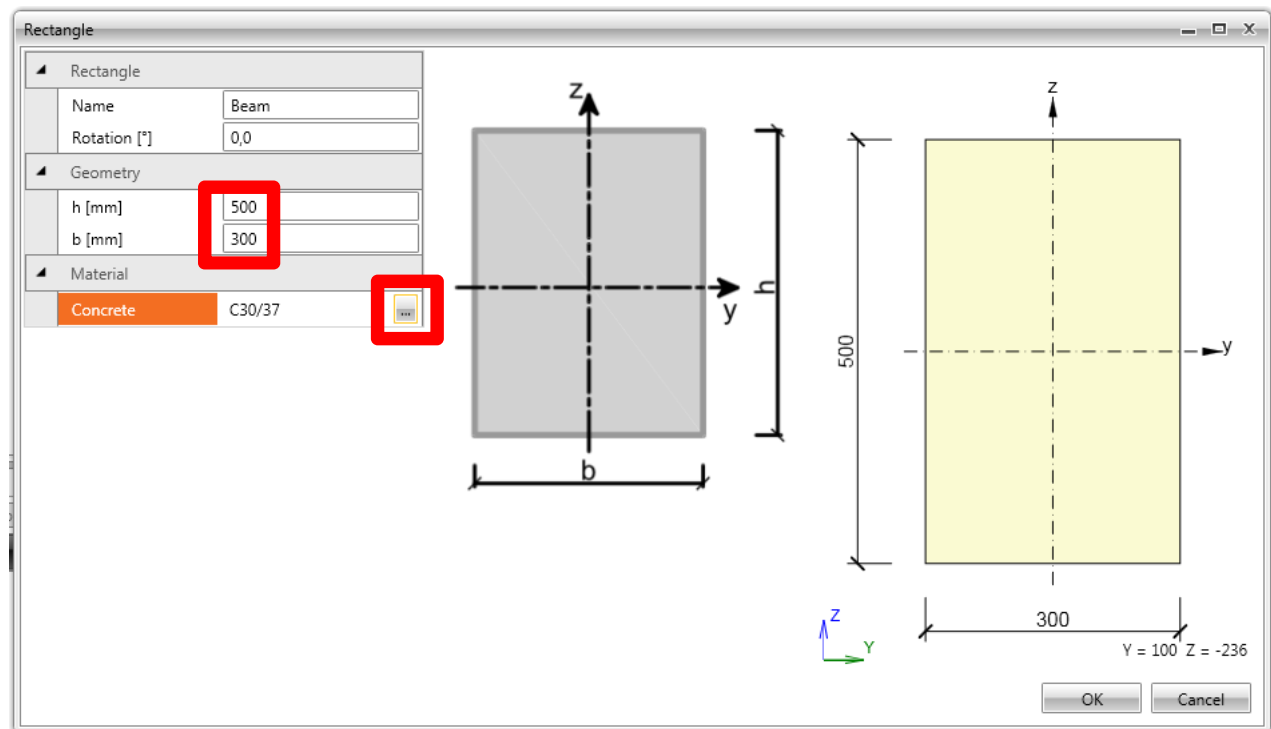
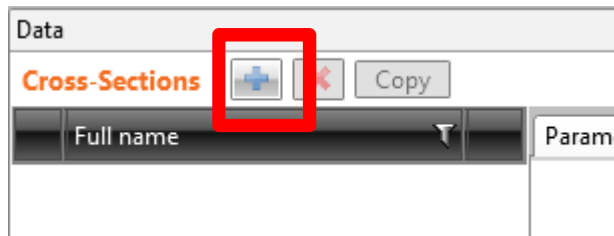
3. Project data

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



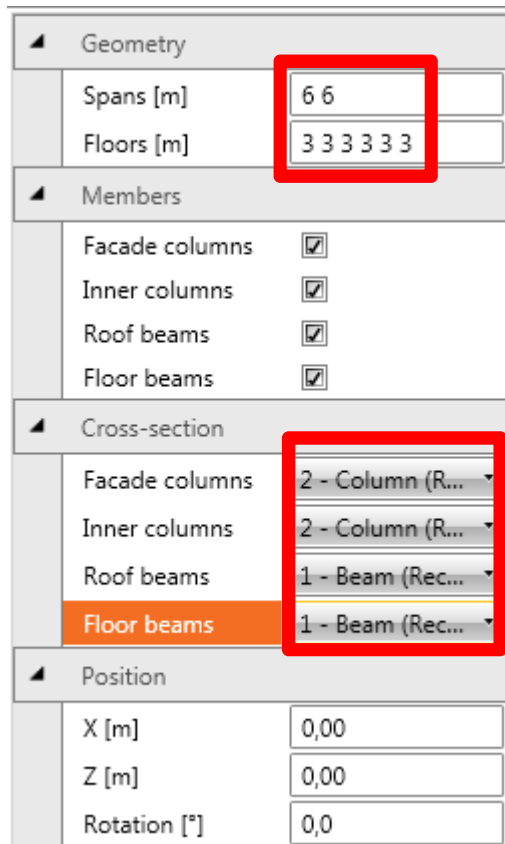
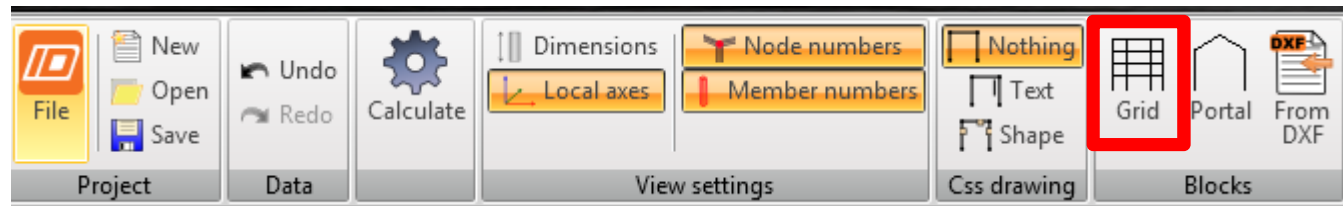
4. Cross-sections

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

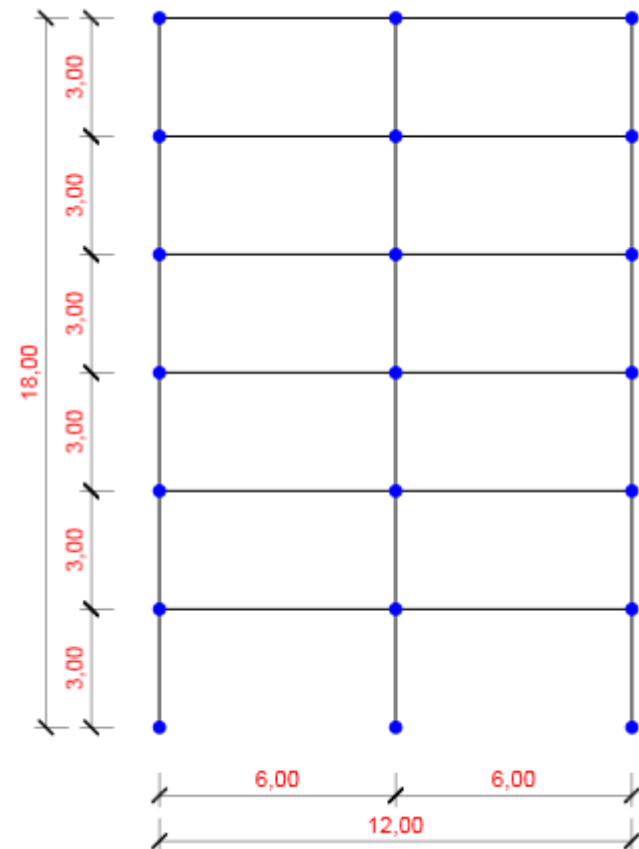


5. Creating the frame

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

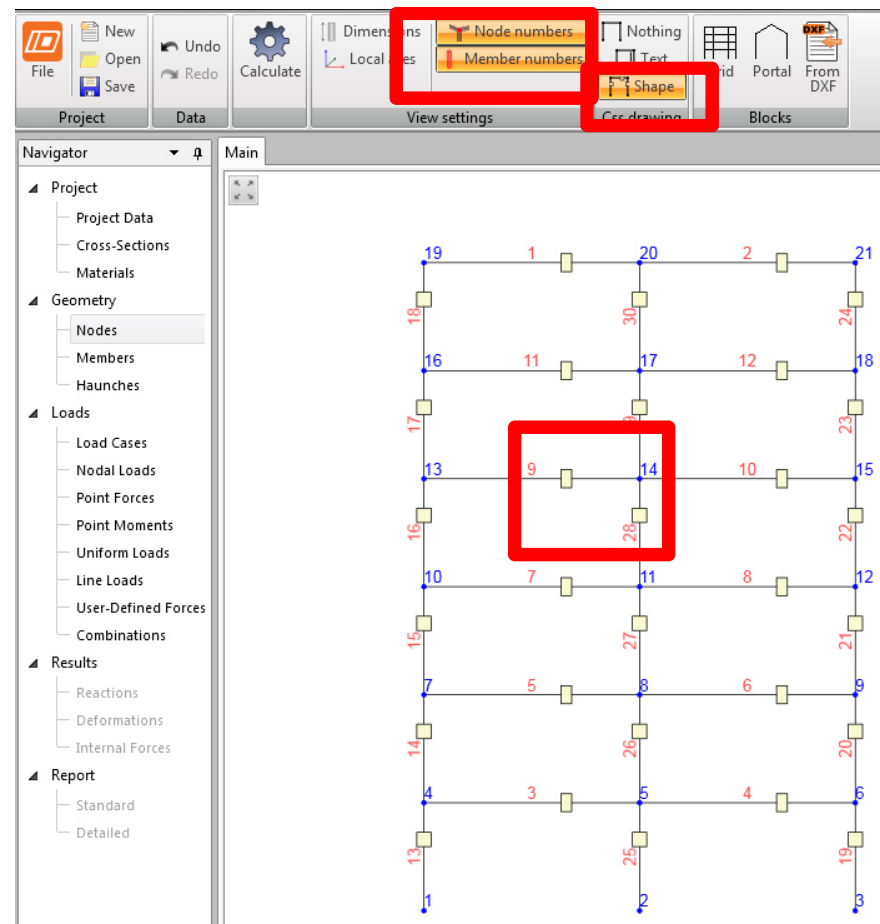


- Set dimensions and number of floors (divide the numbers simply by spacebar)
- Select cross-sections – different for beam/column



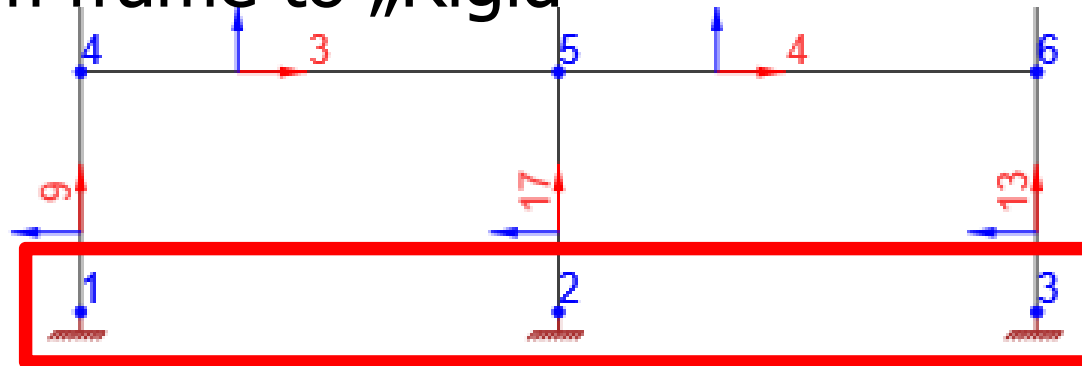
5. Creating the frame

- 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

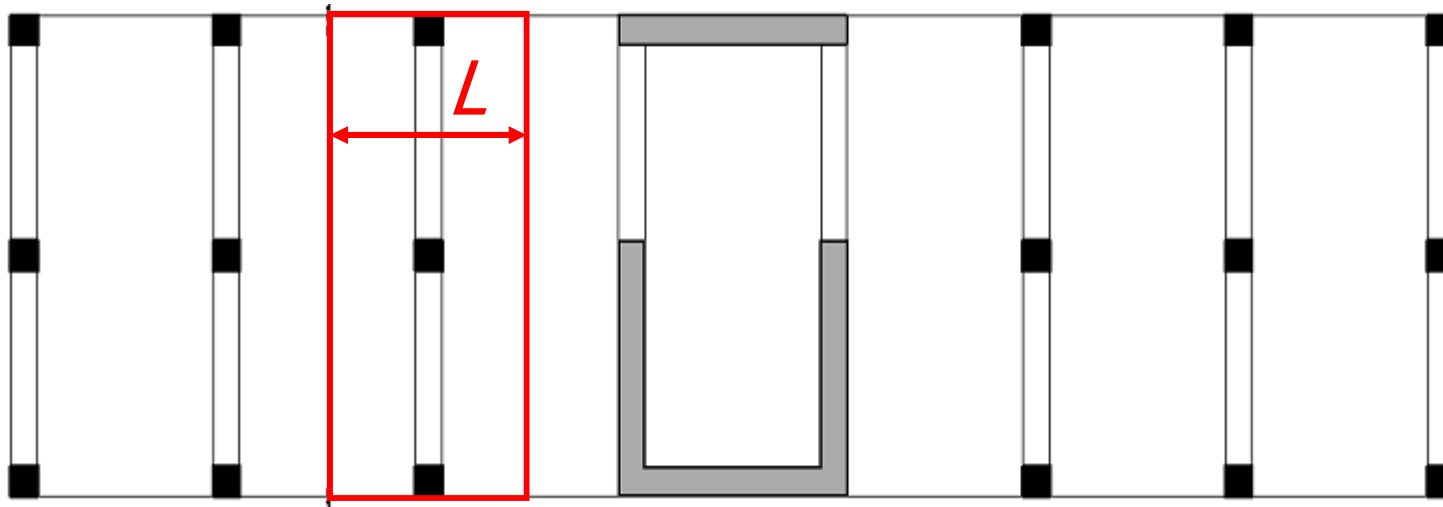
Nodes + Cleanup

Node	X [m]	Z [m]	Nodal Support			
			X	Z	Ry	
1	0,00	0,00	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
2	6,00	0,00	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
> 3	12,00	0,00	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>
4	0,00	3,00	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

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/m²] * L [m]



7. Load cases

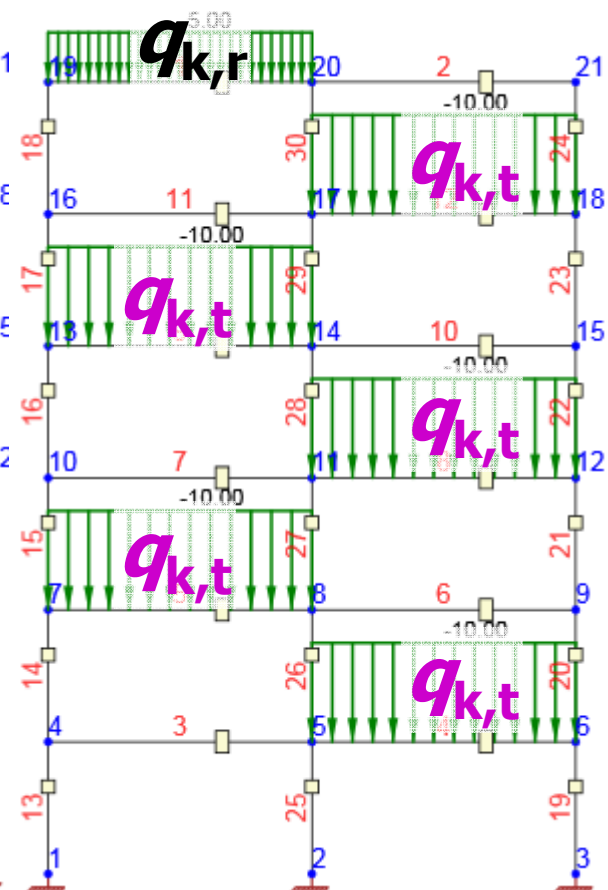
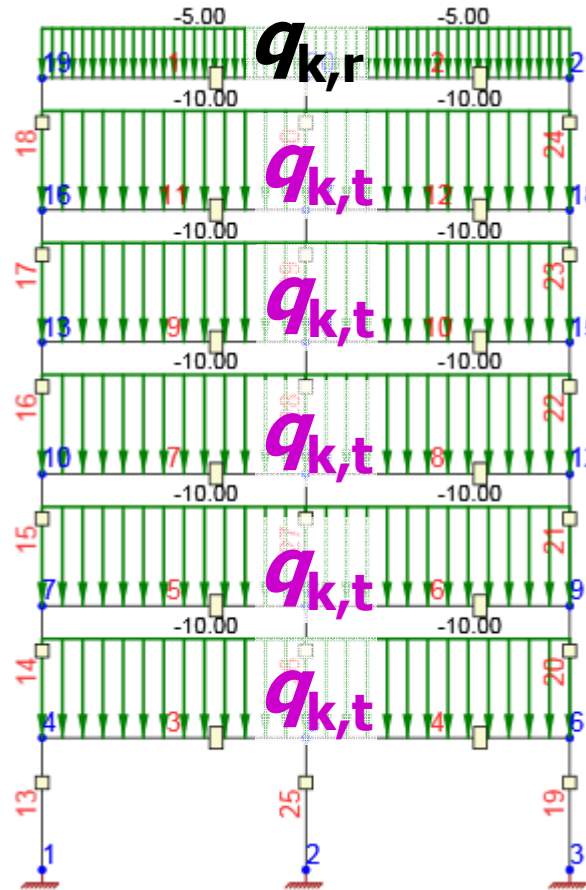
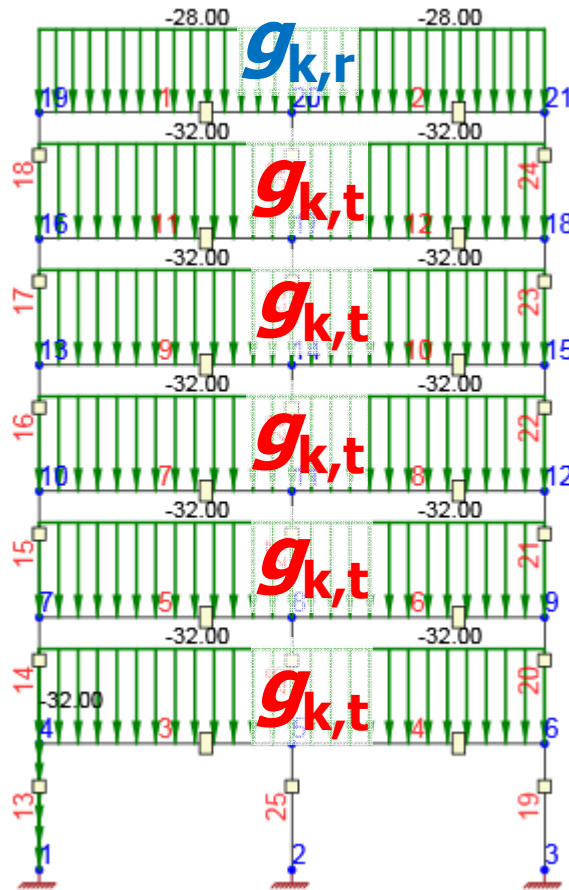
- You should consider 4 load cases:

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:

Data

Load cases Permanent load groups Variable load groups

Load Cases + Copy selected Delete selected

Name	Load Group	Type	
SW	LG1 - Permanent	Permanent	X
LC1 Permanent	LG1 - Permanent	Permanent	X
LC2 Variable full	LG2 - Standard	Variable	X
> LC3 Variable checkerboard	LG2 - Standard	Variable	X


Click here to change

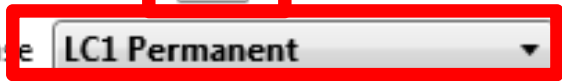
- SW = self-weight of the modeled structure (frame) – automatically generated

6. Loads

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

Data

Uniform Loads  2) Add new member load

Load Case  LC1 Permanent 1) Select Load Case

Membe	Value [kN/m]	Direction	Angle [°]	Location
3	-32,0	Global Z	0,0	Length
4	-32,0	Global Z	0,0	Length
5	-32,0	Global Z	0,0	Length

- Enter member number
- Enter load value (with „minus“ !)
- Select vertical (Z) direction

8. Load combinations

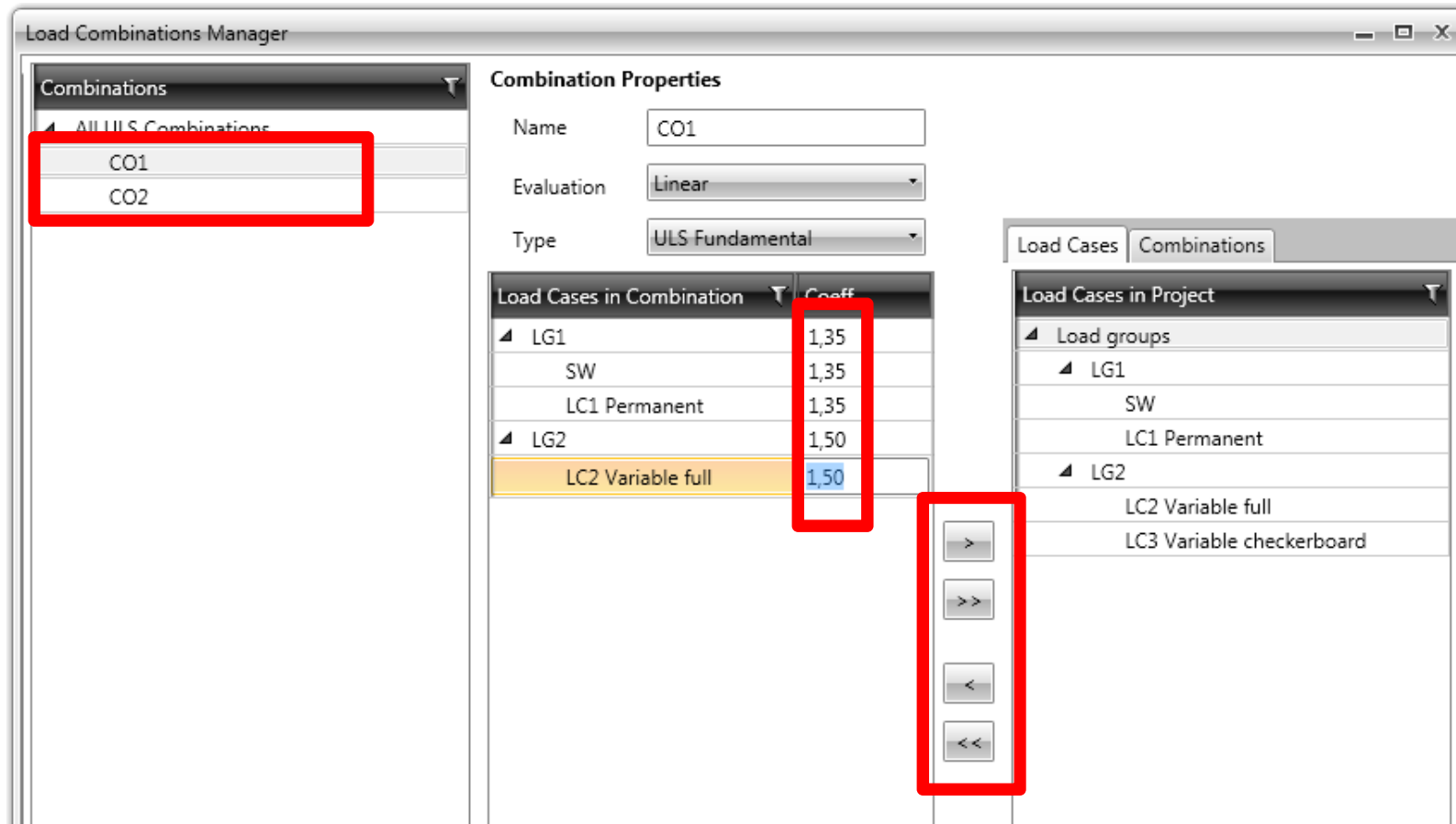
- Create 2 load combinations:
 - CO1 Full = SW+LC1+LC2
 - CO2 Checkerboard = SW+LC1+LC3

Name	Type	Evaluation	Description
CO1	ULS Funda...	Linear	1,35*SW + 1,35*LC1 Permanent + 1,50*LC2 Variable full
CO2	ULS Funda...	Linear	1,35*SW + 1,35*LC1 Permanent + 1,50*LC3 Variable checkerboard

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



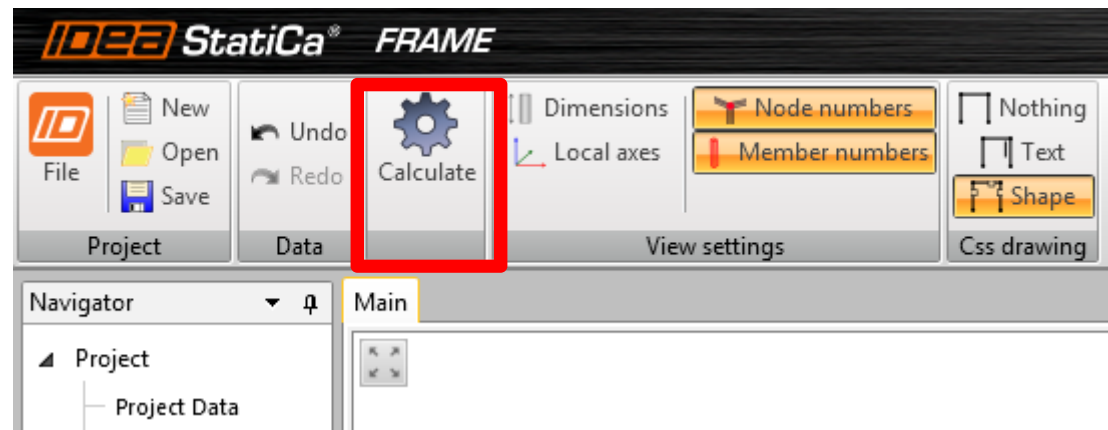
1) Select the combination

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

2) Add/remove load cases to the combination

9. Results

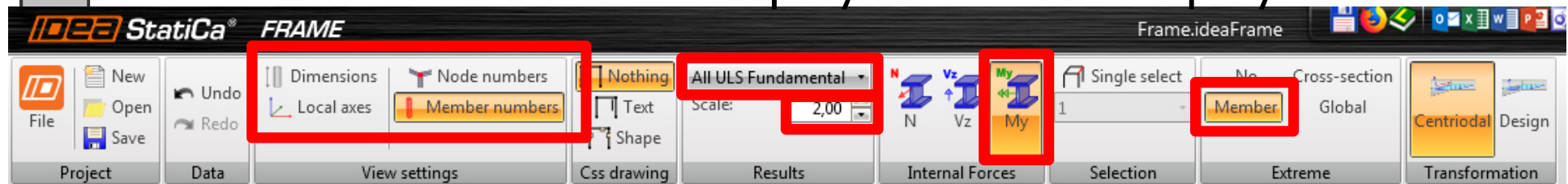
- Calculate results



9. Results

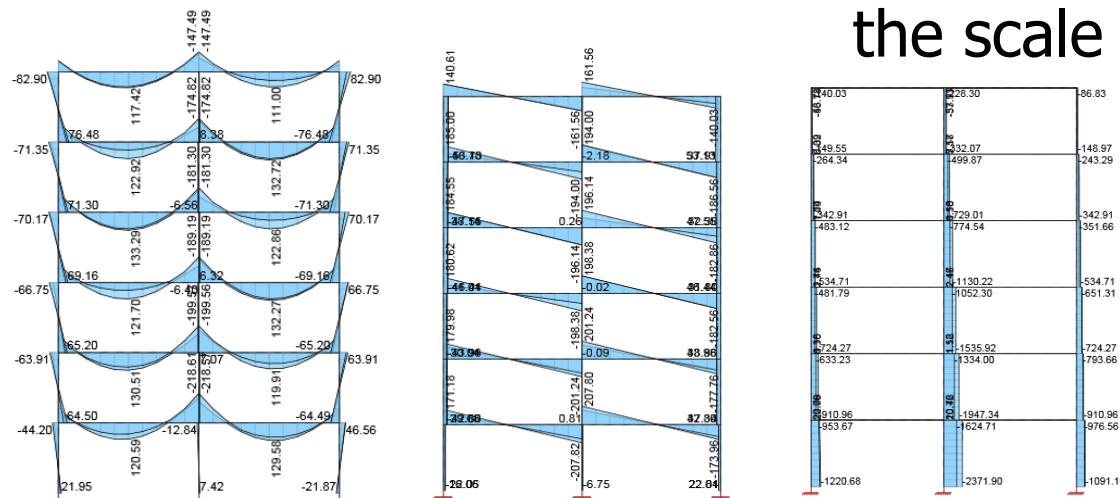
- Go to Results > Internal Forces; check int. forces for CO1, CO2 and „envelope“ (All ULS)

- 1) Turn off all labels **except** Member numbers
- 2) Select load combination to internal force display
- 3) Select combination to internal force to display



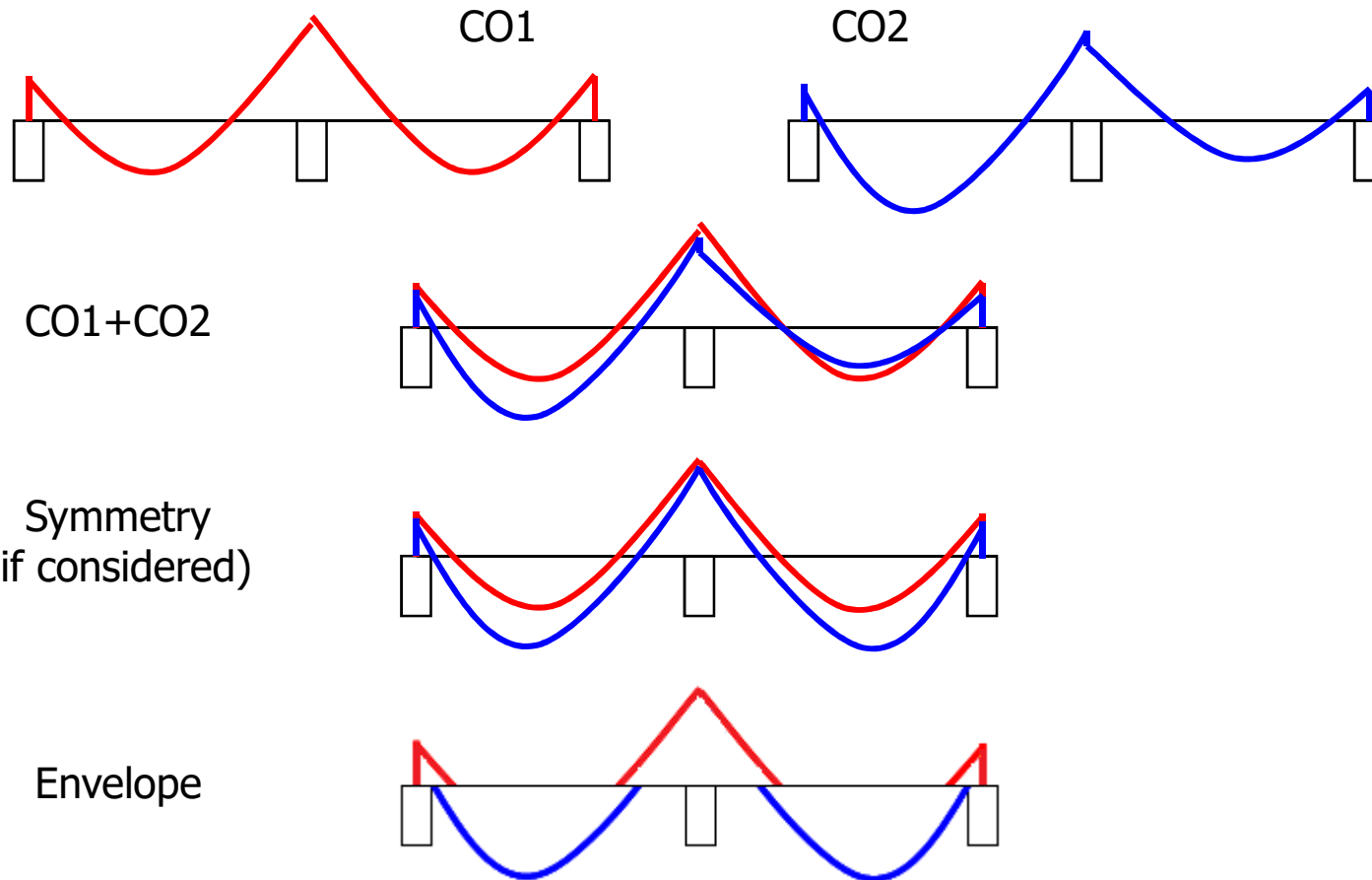
- 4) Adjust the scale

- 5) Display extreme values on members



9. Results

- What is an „envelope“?



10. Report

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

Modeler and results

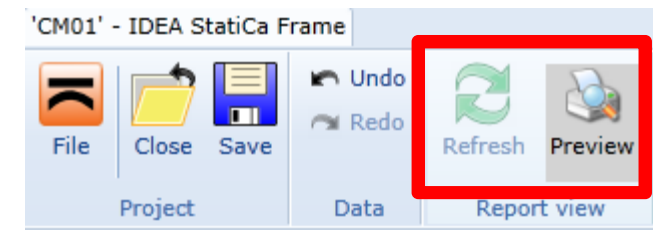
- Project data
- Geometry
 - Draw picture
- Load cases
- Loads
 - Draw picture
- Load combinations
- Design groups and Design members

Results

- Global Extreme
- Member Extreme
- Cross-section Extreme
- All load cases
- All combinations
- Envelope
 - Internal forces
 - Draw picture
 - Deformations
 - Draw picture
 - Reactions
 - Draw picture

Refresh, then

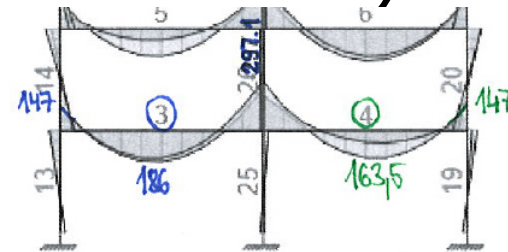
Go to Print Preview and print the report



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):



All combinations, M_y [kNm], Centroidal forces

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

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