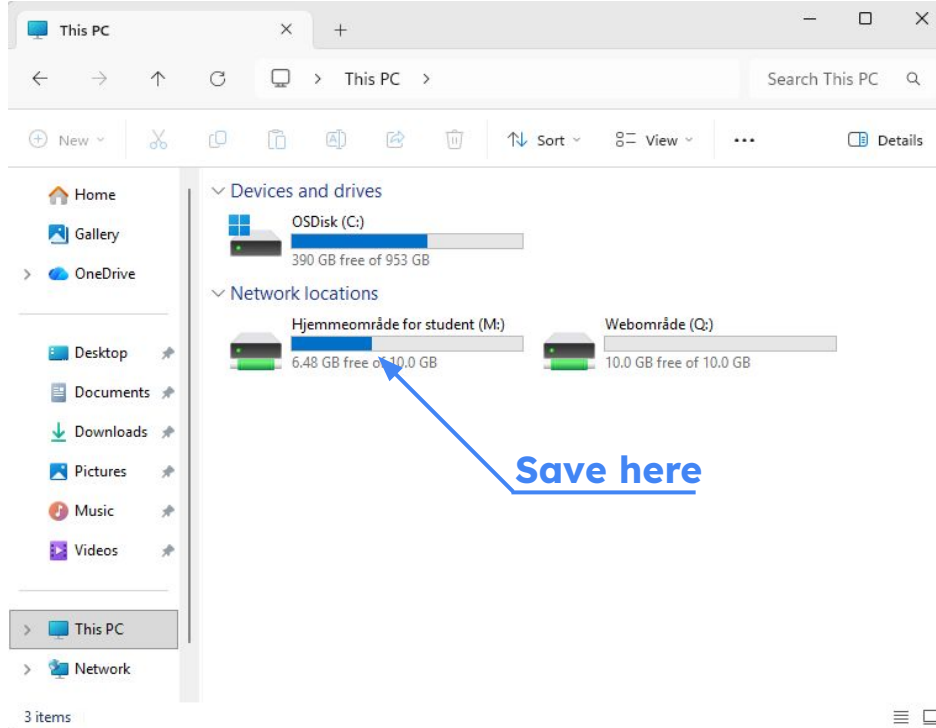


SAP2000®

Introduction

Save file



Or:



Google Drive



OneDrive

Structure/Geometry

Workflow

0. General

1. Grid
2. Materials
3. Frame Sections
4. Frame objects
5. Connections

General

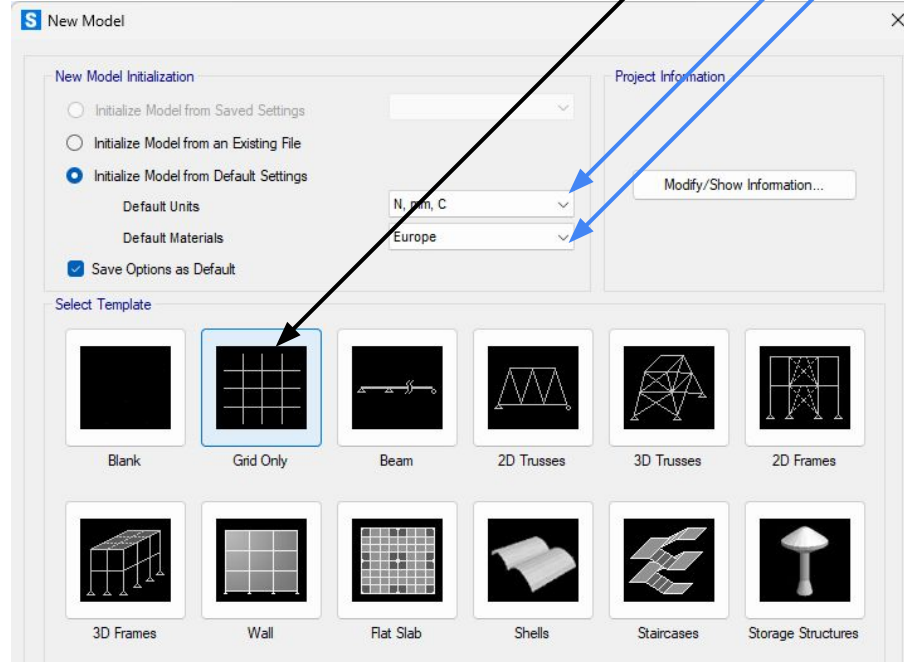
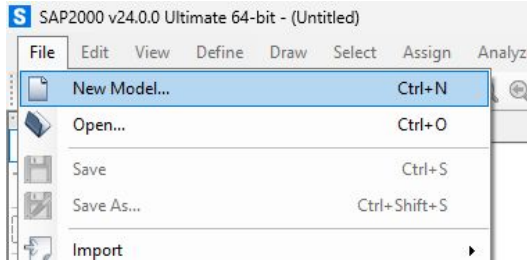
Cursor

- Pan: middle mouse
- Rotate: shift - middle mouse

Display options

- Edit => Display options
 - 3D view:
=> View options=>Extrude

New model



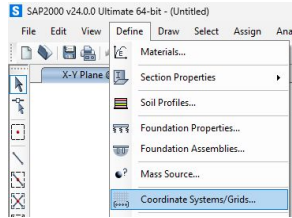
Click Grid Last

N/kN
Europe

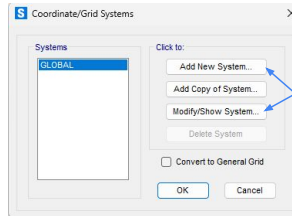
Grid Setup

Modify grid later:

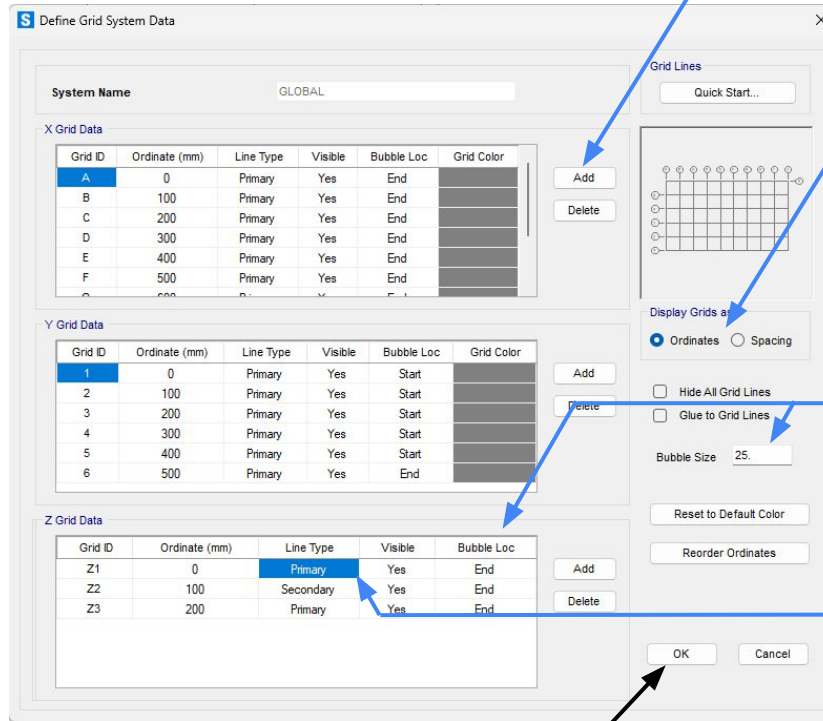
1.



2.



3.



Add Line (same spacing)

Switch Coordinates/Spacing

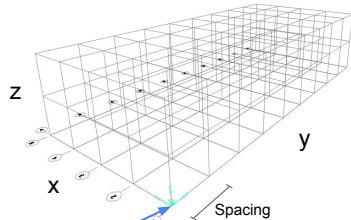
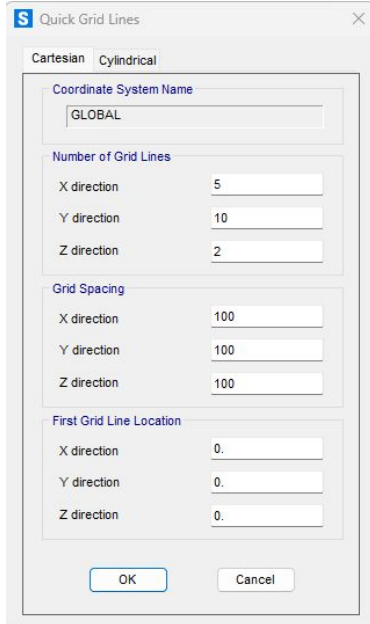
Bubble

Primary (w/ Bubble)

Secondary (No Bubble)

Last: OK

6



First Grid Line Location (Origin)

Frame objects



Select object

Select frame or node to edit

Element drawing options



Draw frame element

Draw a frame or brace element (Ramme-element/stav).



Draw 2D surface or square

Use to draw walls/shear walls

Node snapping options: Choose after element selected



Snap to **Grid** (intersections)

Snaps to defined grid



Snap to perpendicular

Snaps to a point perpendicular to the frame element of the first point



Snap to ends and **midpoints**

Useful for Bracing/fagverk



Snap to closest (Dont use)

Snaps to closest element/node. Only use for ideas or tests.



Snap to element intersections

Intersections between frame elements



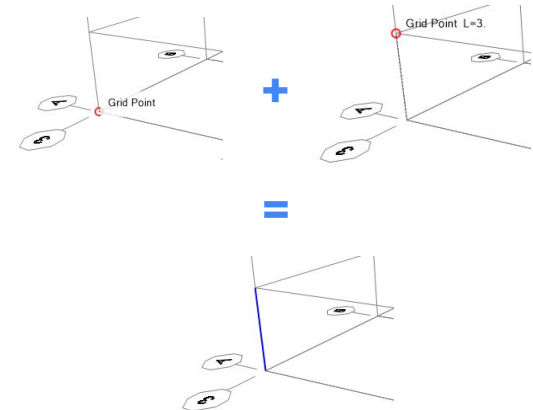
Snap to increment

Snaps to every 0.1cm etc.

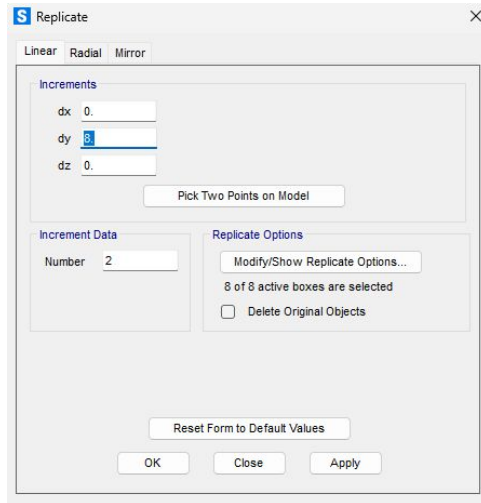
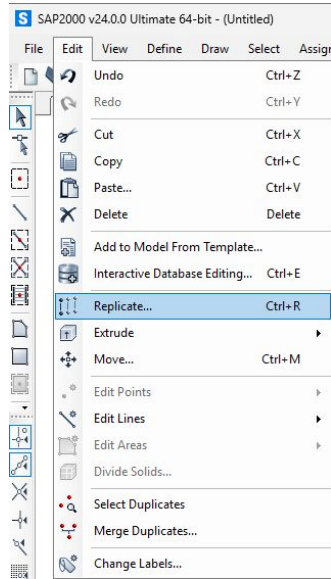
1. Choose object

2. Choose snapping

3. Click on 2 points



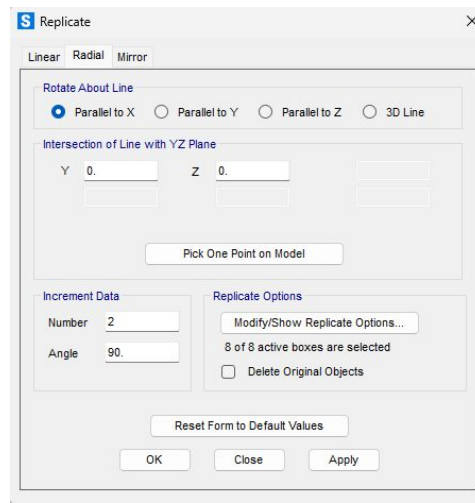
Mirror/rotate/duplicate



Duplicate

Increment: Choose distance between duplicates

Increment Data: Choose number of copies



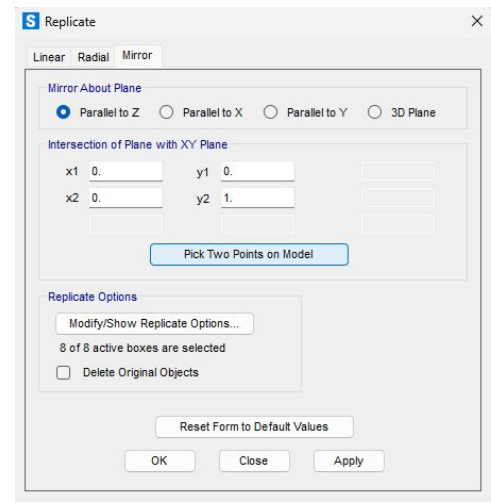
Rotate

About axis: Choose axis to rotate around

Intersection: Choose where the axis is

Increment Data: Choose number of copies

Angle: Choose angle to rotate
Options: Delete original objects if you want to move the selection, instead of making a copy



Mirror

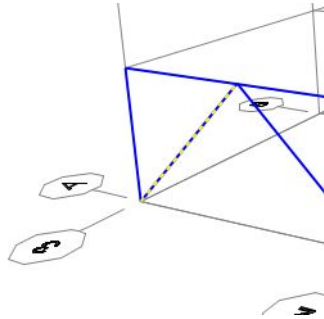
About axis: Choose axis to mirror about

Intersection: Choose where the plane is

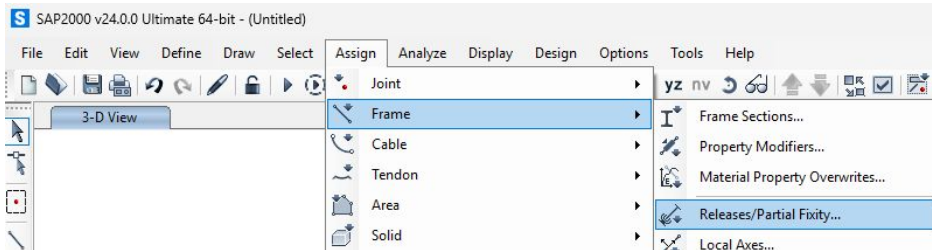
Delete original objects if you want to move the selection, instead of making a copy

Connections

1. Choose drawn element (Turns Blue/Yellow)



2. Assign => Frame => Releases/Partial Fixity

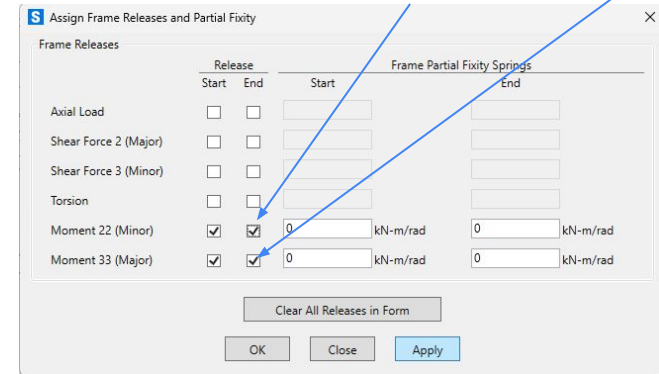
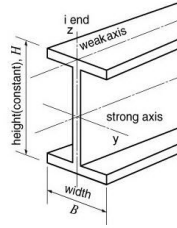


3.

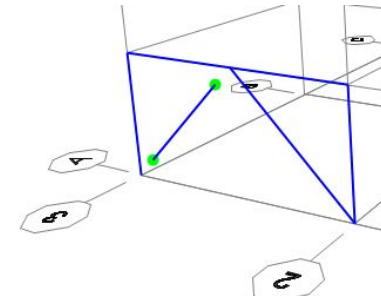
a) Pinned-Pinned

=> Release Moment,

- Major = y axis, Minor = z axis

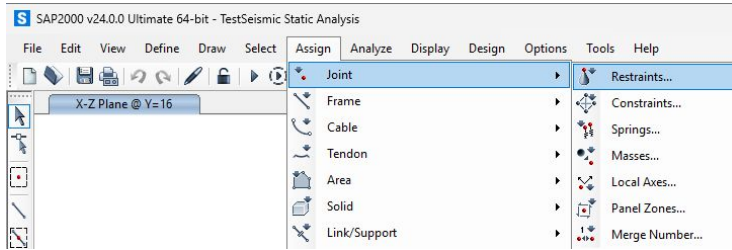


4.

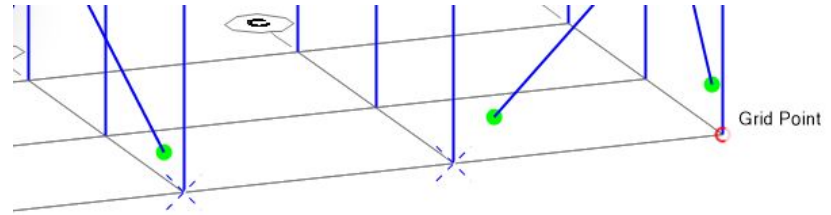


Boundary Conditions

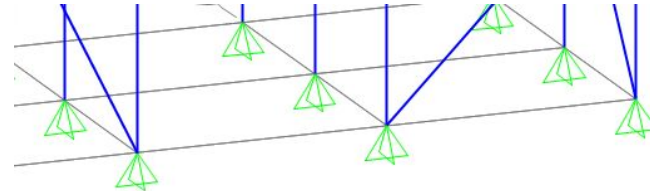
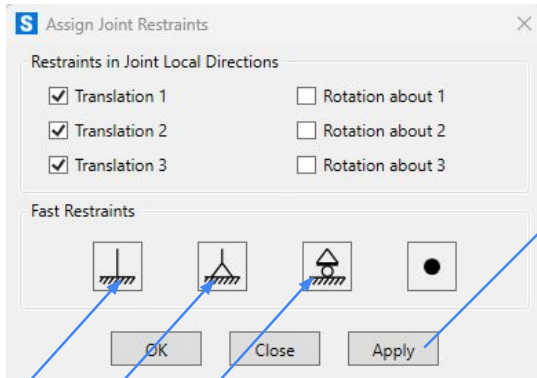
1.



2. Pick all nodes where you want a joint

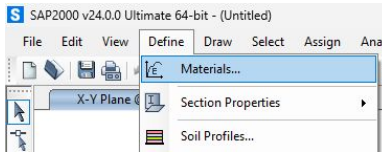


3. Choose Restraint

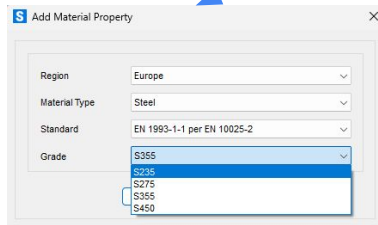
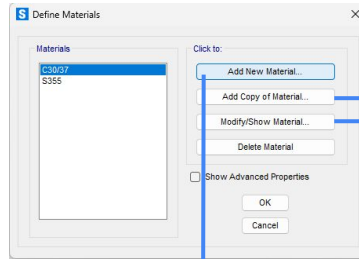


Materials

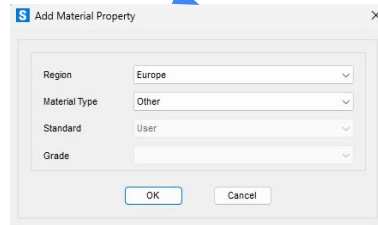
1.



2.

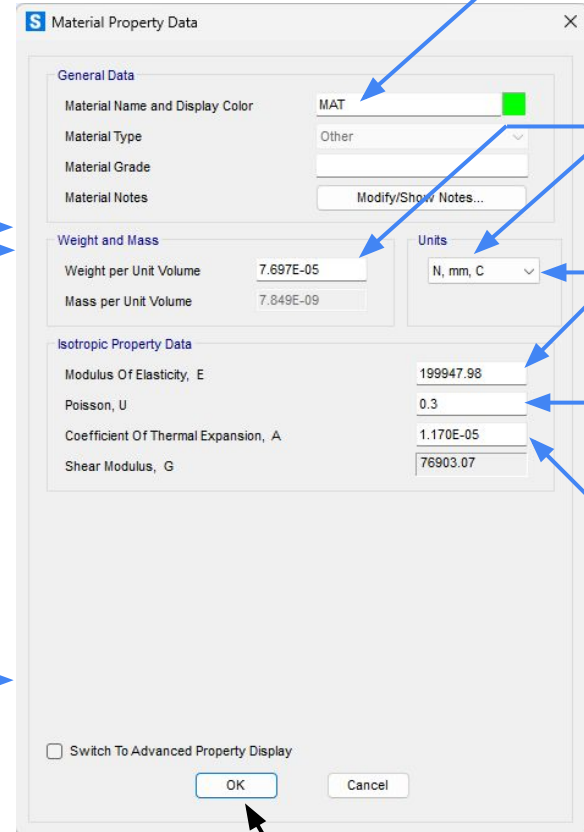


Eurocode Materials



Timber and other materials

3.



Material Name:
GL24h, C24 etc.

Density
(NEWTONS)

Watch units

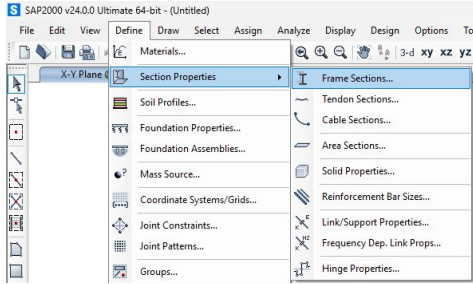
$$\nu = \frac{\epsilon_{\perp}}{\epsilon_{\parallel}}$$

$$\alpha \cdot \Delta T = \epsilon$$

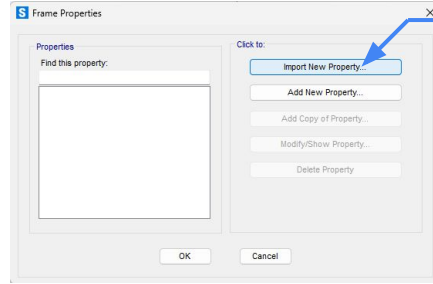
Frame sections (Cross section/tverrsnitt)

Complex sections (I, H, C)

1.

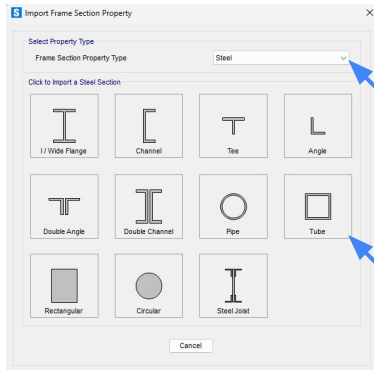


2.



Import
New

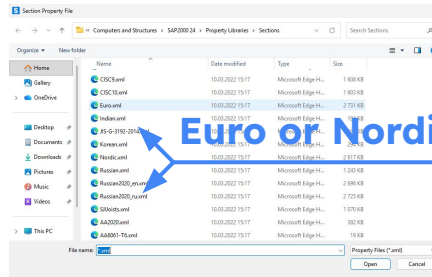
3.



Material

Section

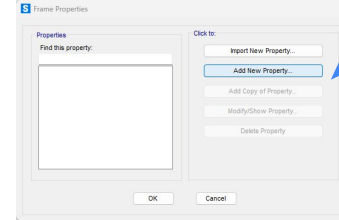
4.



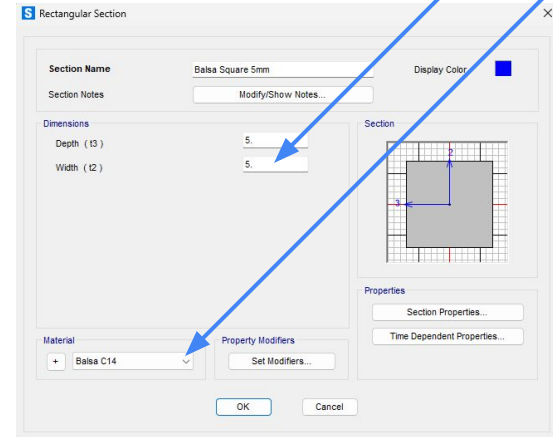
Euro or Nordic

Simple Sections (or non standard)

Add New



Fill out



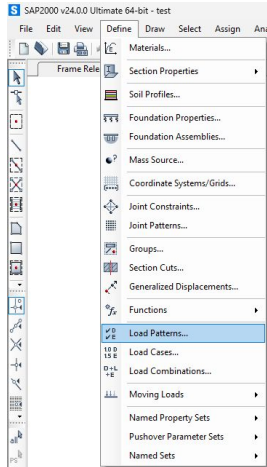
Static Analysis

Workflow

1. Load types
2. Optional: Nonlinear analysis
3. Load Combinations
4. Assign Loads
5. Analysis

Loads

1.



2.

Name

Type

If Dead => 1: Auto adds weight

Last: Click add

Define Load Patterns

Load Pattern Name	Type	Self Weight Multiplier	Auto Lateral Load Pattern
Snow	Snow	0	
DEAD	Dead	1	

Click To:

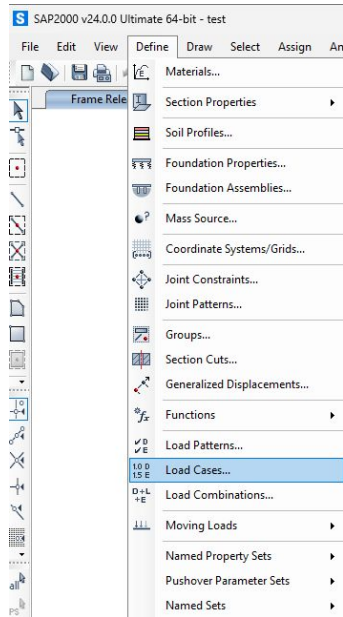
- Add New Load Pattern
- Add Copy of Load Pattern
- Modify Load Pattern
- Modify Lateral Load Pattern...
- Delete Load Pattern
- Show Load Pattern Notes...

OK Cancel

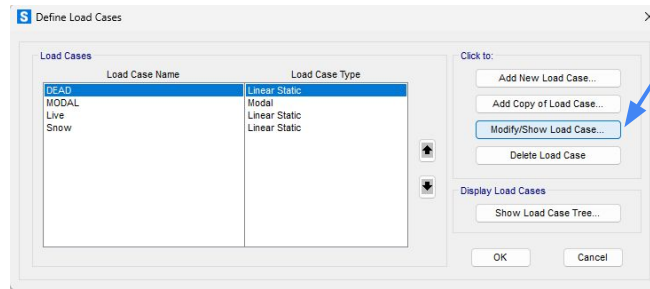
OK

Non-linear analysis

1.

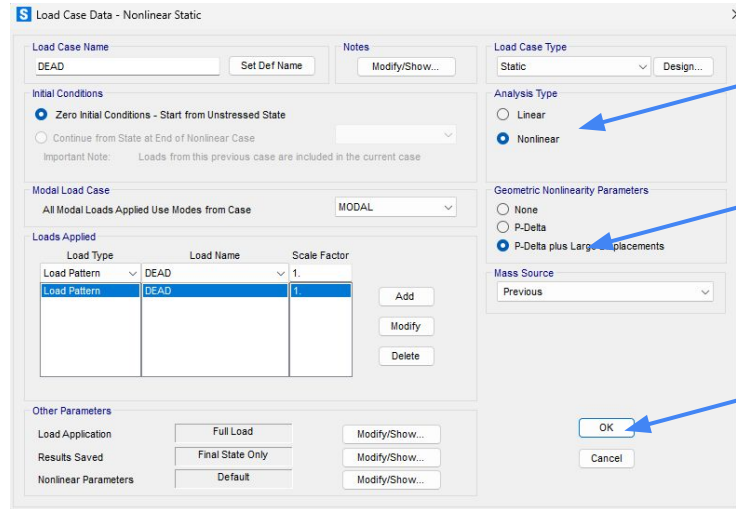


2.



Click Modify

3.



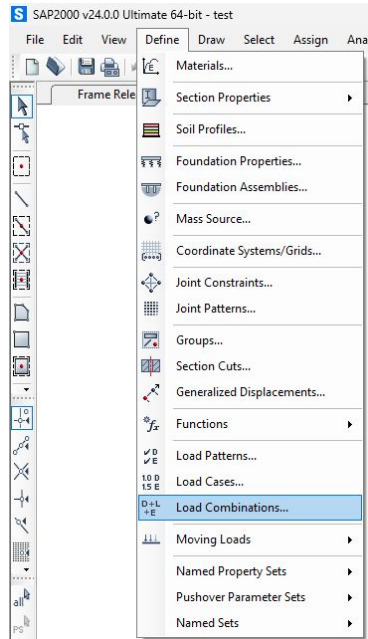
Non-linear

P-Delta

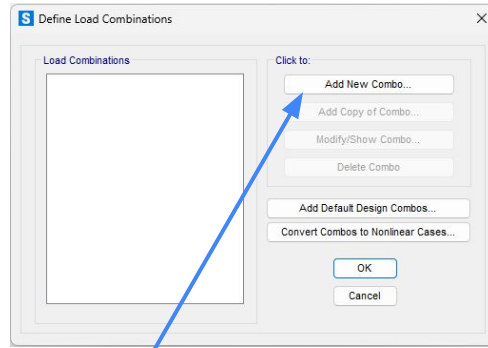
OK

Load combinations

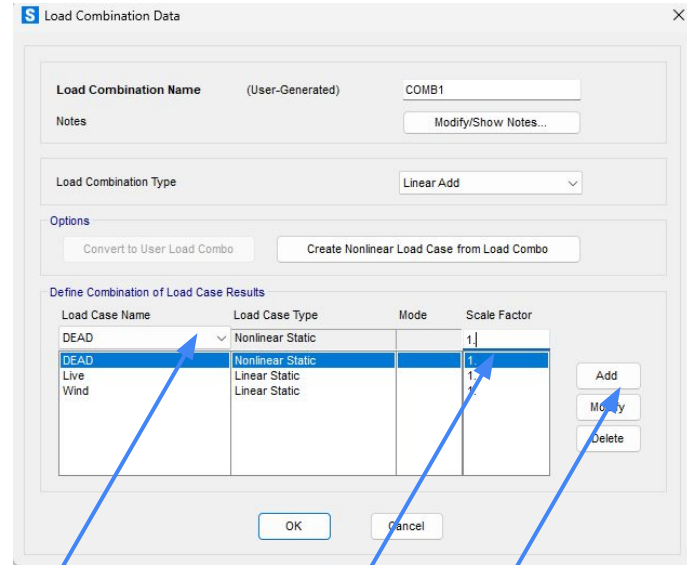
1.



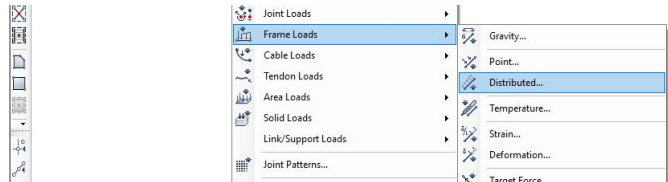
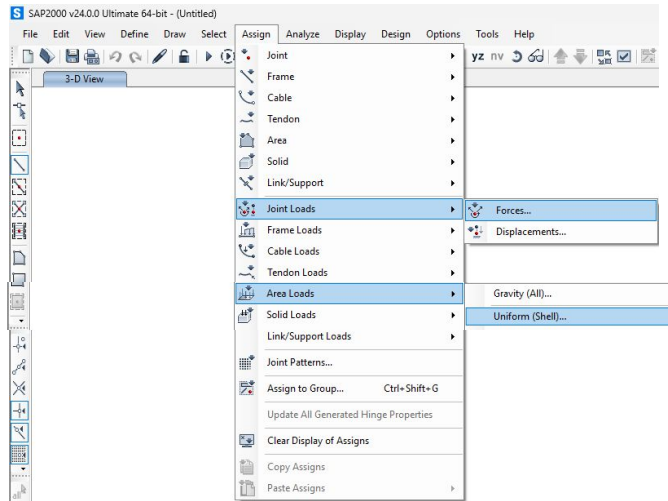
2.



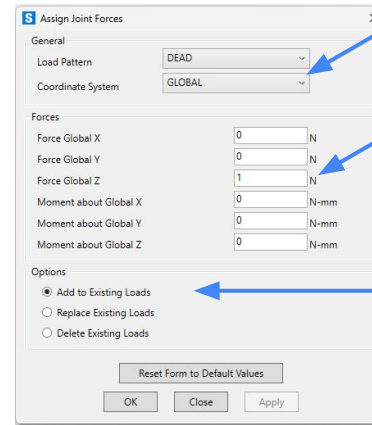
3.



Assign loads



Point Load



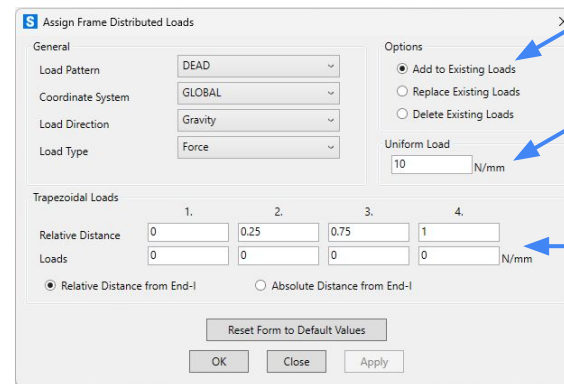
Choose load type

Add Force (Usually Z)

Make sure this is Add

=> Click on all points where you want the load
=> OK

Distributed Load



Make sure this is Add

Add distributed load

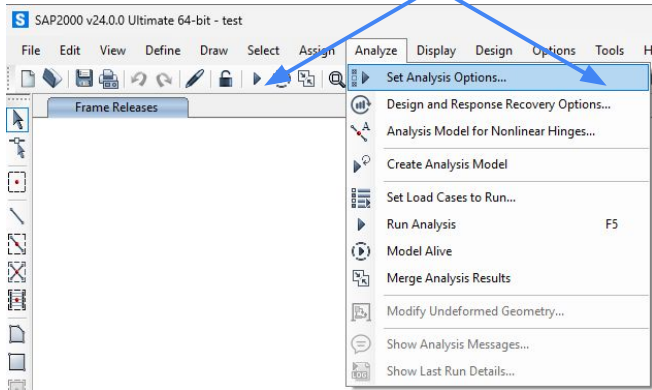
Only for non-uniform
(Don't need to touch)

=> Click on all elements where you want the load
=> OK

Analysis

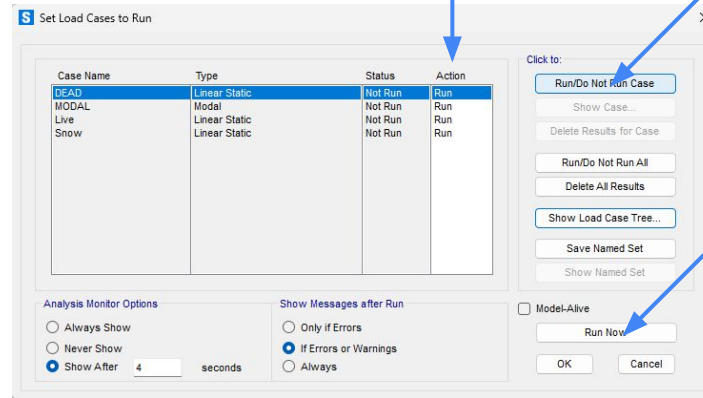
1.

Either



The ones that say
"Run" will run

Click Run for all cases you
want to see the results of

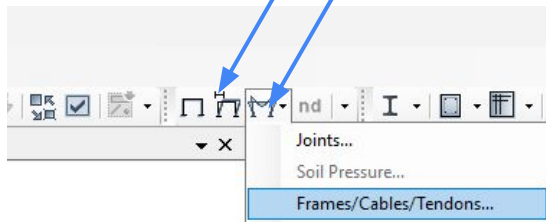


Click Run Now

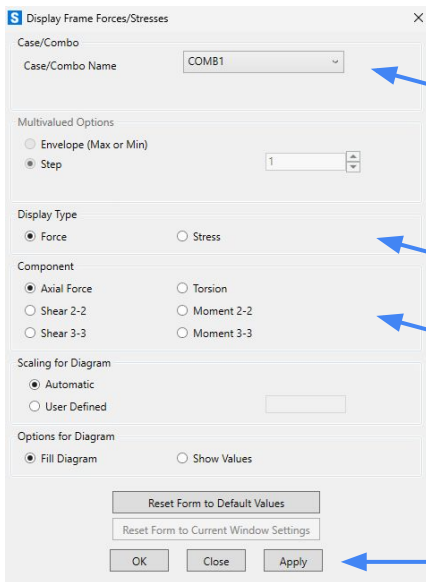
View

Show Deformed shape

Show Internal Forces



Forces at Joint or Frame



Choose load/loadcombo

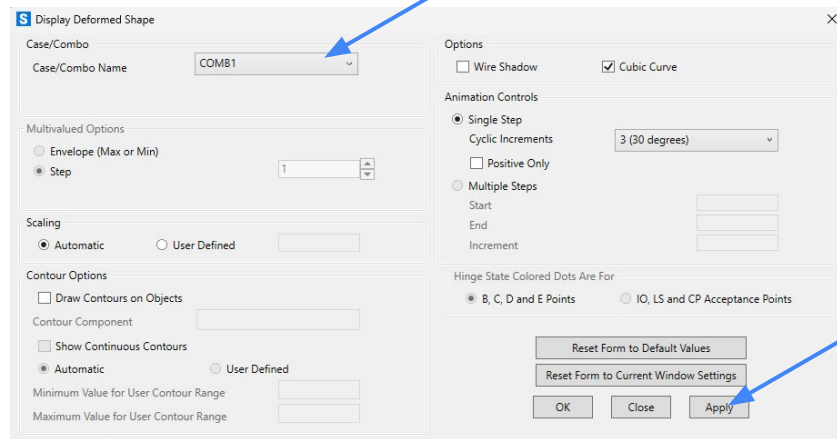
Force/Stress

Choose Force/Moment type

Apply

Deformed shape

Choose load/loadcombo



Apply

Dynamic Analysis

Workflow RHA

1. Mass
2. Time history
3. Load Case

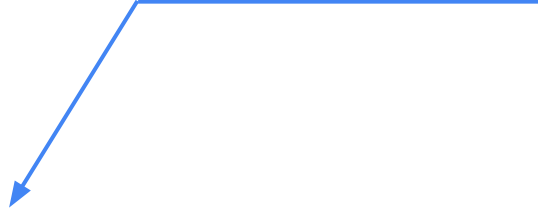
Video on Response Time History (RHA) analysis

https://www.youtube.com/watch?v=VjkqA4_hIMk

Define Mass Source

1.

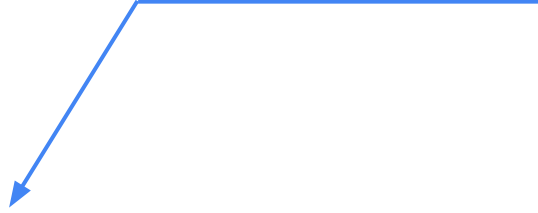
TEXT



Time History

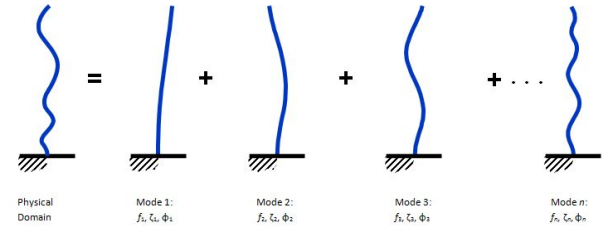
1.

TEXT



Modal Load Case

Recall:



Modes refer to the structures oscillating behaviour.

1.

TEXT

