



9th Convention on Advances and Applications of GiD

GiD+OpenSees Interface

An Introductory Course

Vassilis K. Papanikolaou

Assistant Professor, School of Civil Engineering

Aristotle University of Thessaloniki, Greece



GiDOpenSees

Barcelona, June 2018

Interface Installation and Walkthrough

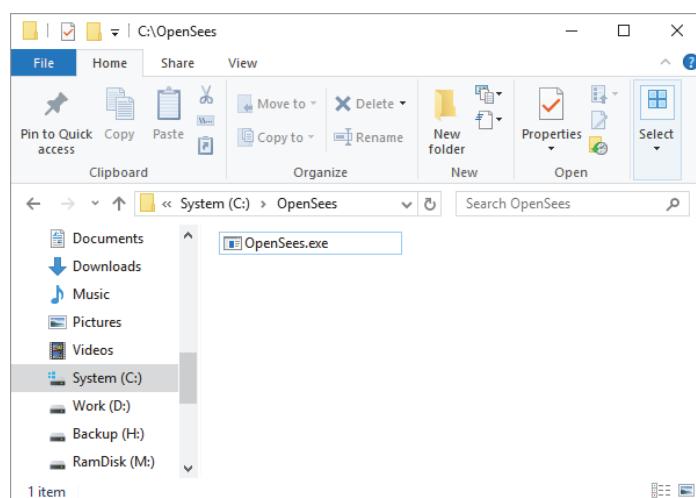
This section covers the essentials for installing GiD+OpenSees interface and displays its basic features. The installation steps are the following :

- 1) Download and install GiD (32 or 64 bit Windows) from the official GiD website

<https://www.gidhome.com/download/official-versions>

- 2) Download OpenSees and the associated tcl framework (32 or 64 bit Windows) from the official OpenSees website. Install the tcl framework (recommended C:\tcl) and copy OpenSees.exe to any folder (recommended C:\OpenSees)

<http://opensees.berkeley.edu/OpenSees/user/download.php>

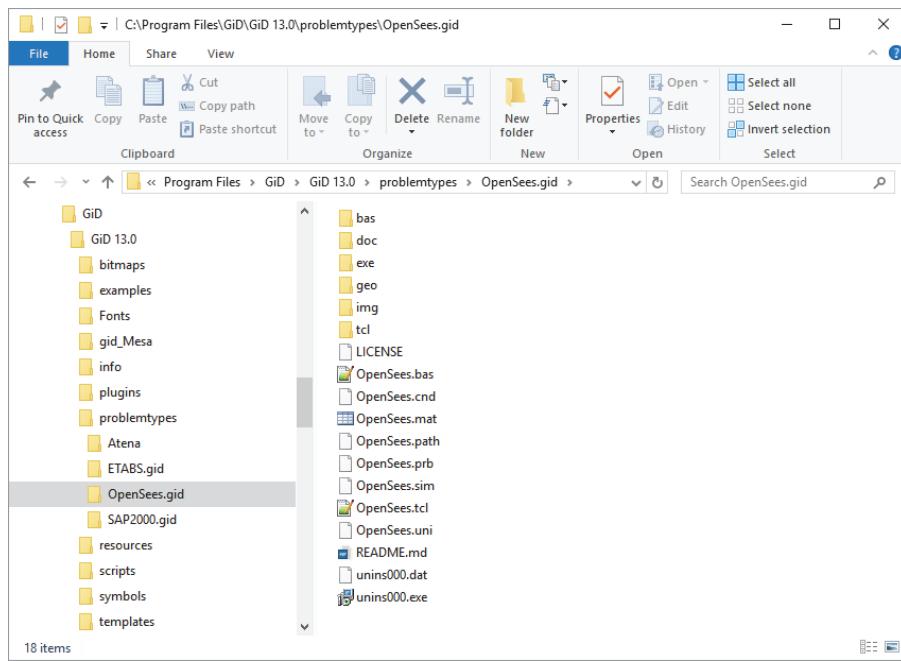


- 3) Download GiD+OpenSees interface from the official site by selecting the Download option (redirects to GitHub). Run the installer and select the previously defined OpenSees path when prompted.

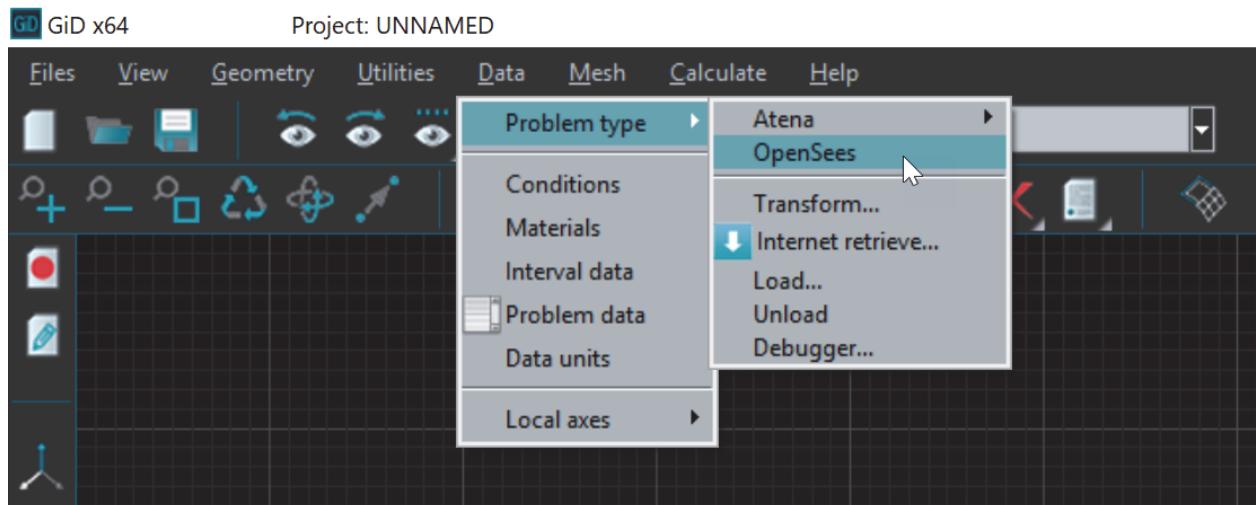
<http://gidopensees.rclab.civil.auth.gr>

After installing the interface, note that all the relevant files are located in :

<GiD installation folder>\problemTypes\OpenSees.gid\



The installation is now completed. The GiD+OpenSees Interface can be invoked by running GiD and select Data → Problem type → OpenSees



After loading the interface, a side toolbar appears containing all the essential functionality for building and running a finite element model with OpenSees. Moreover, the 'OpenSees' menu contains additional options for executing analysis and getting Help. The toolbar and menu commands are displayed in the following tables :

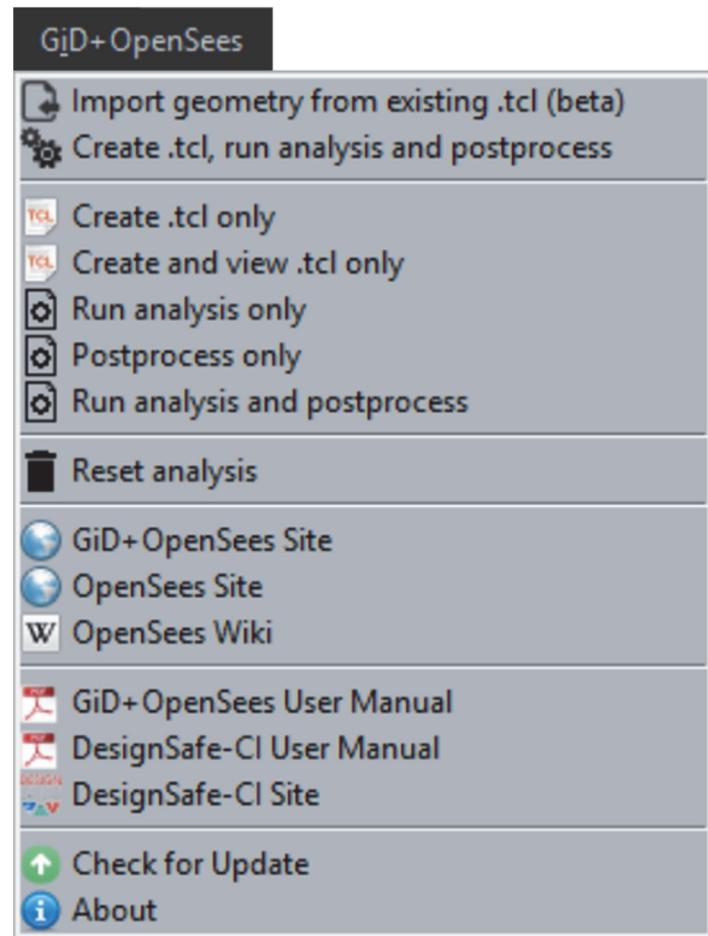
Toolbar commands

Standard Uniaxial Materials		Zero-Length Elements	
Concrete Uniaxial Materials		Truss Elements	
Steel Uniaxial Materials		Beam-Column Elements	
Multidimensional Materials		Surface Elements	
Section Force-Deformation		Solid Elements	
Combined Materials		General Data	
Records		Output Options	
Restraints		Interval Data	
Constraints		Generate Mesh	
Masses/Damping		Run Analysis	
Loads		Show/Hide Local Axes	
		Show/Hide Elements	
		Show/Hide Conditions	
		Set Active Interval	

All toolbar buttons in **bold** will open a respective dialog window for data definition and associations.

Detailed reference for all toolbar commands with examples can be found in the User Manual located in the GiD+OpenSees menu

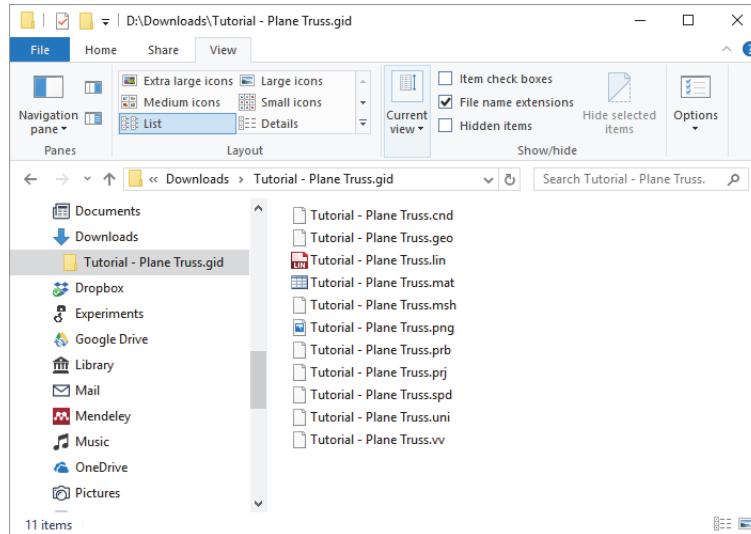
Menu commands



1. Select an existing OpenSees model file (.tcl) and import its geometry in GiD
2. Run analysis and postprocess (same to toolbar button)
3. Just create the .tcl file
4. Create the .tcl file and view it with default text editor
5. Just run the analysis
6. Just translate the output files for postprocessing
7. Combined options 5+6
8. Delete all generated results from previous analysis
9. Official GiD+OpenSees site
10. Official OpenSees site
11. Official OpenSees wiki
12. GiD+OpenSees user manual
13. DesignSafe-CI user manual
14. DesignSafe-CI site
15. Check online for newer versions
16. About screen

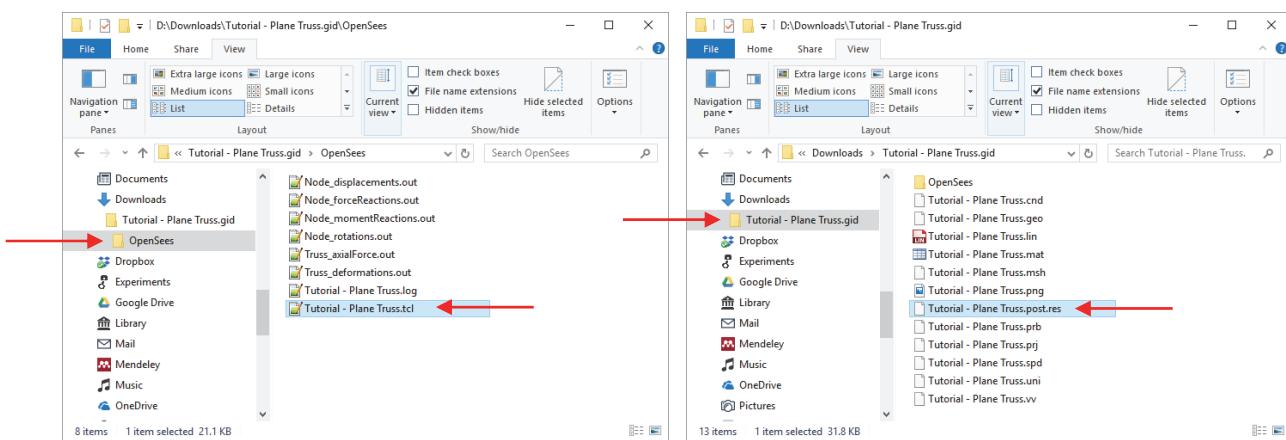
How it (actually) works

1. Model is created in GiD and saved in folder <path>\<model name>.gid
e.g. D:\Downloads\Tutorial - Plane Truss.gid



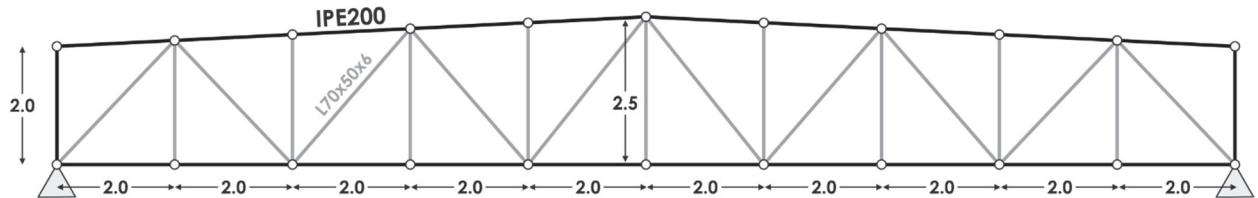
2. After pressing the analysis button :

- a) <model name>.tcl is created in <path>\<model name>.gid\OpenSees
- b) OpenSees.exe is executed and all analysis output and log files are written in <path>\<model name>.gid\OpenSees
- c) Output files are automatically translated and <model name>.post.res is written in <path>\<model name>.gid. This only file is used by GiD for postprocessing.
- d) Resetting analysis option deletes <path>\<model name>.gid\OpenSees and <model name>.post.res



Example A – Elastic Truss Structure

The first example is an elastic truss structure, modeled in the XY plane. The truss geometry is depicted below :



Material properties

Steel S450 : $E_s = 200 \text{ GPa}$

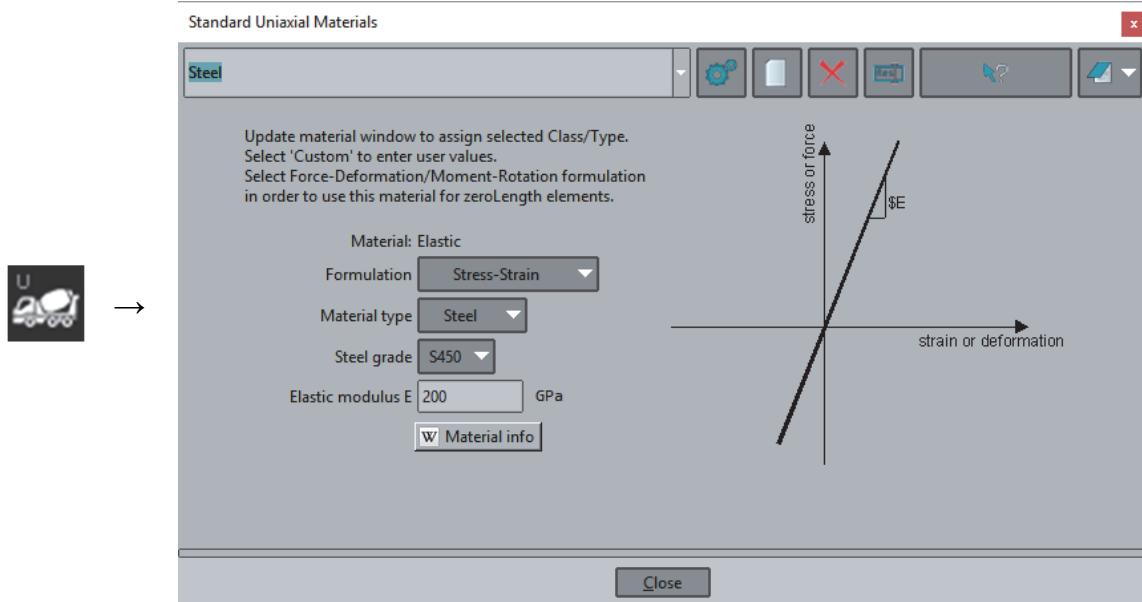
Section properties

IPE200 : $A = 28.5 \text{ cm}^2$

L70x50x6 : $A = 6.89 \text{ cm}^2$

A. Preprocessing

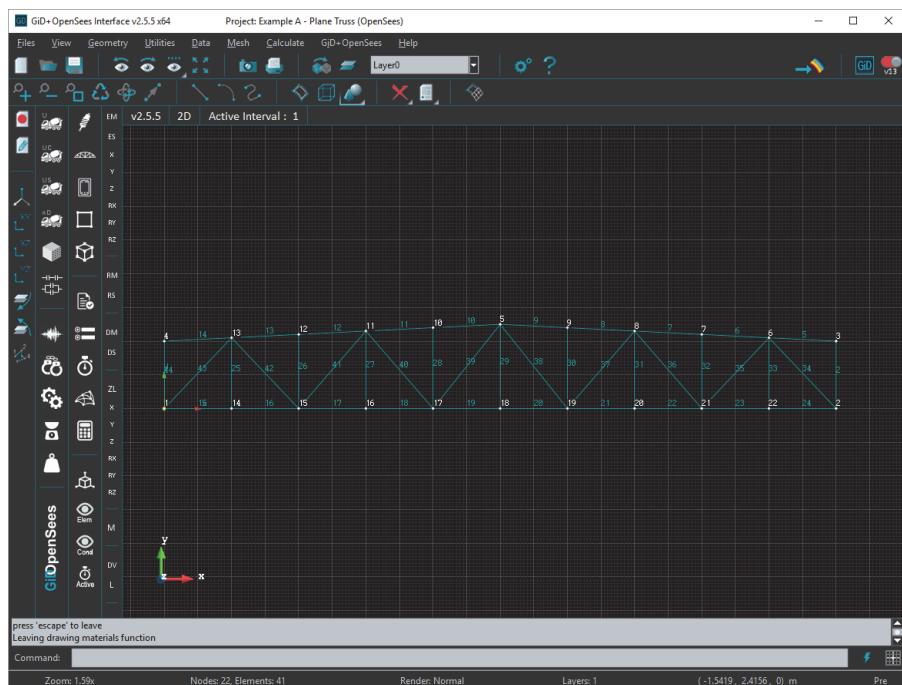
1. Define steel material



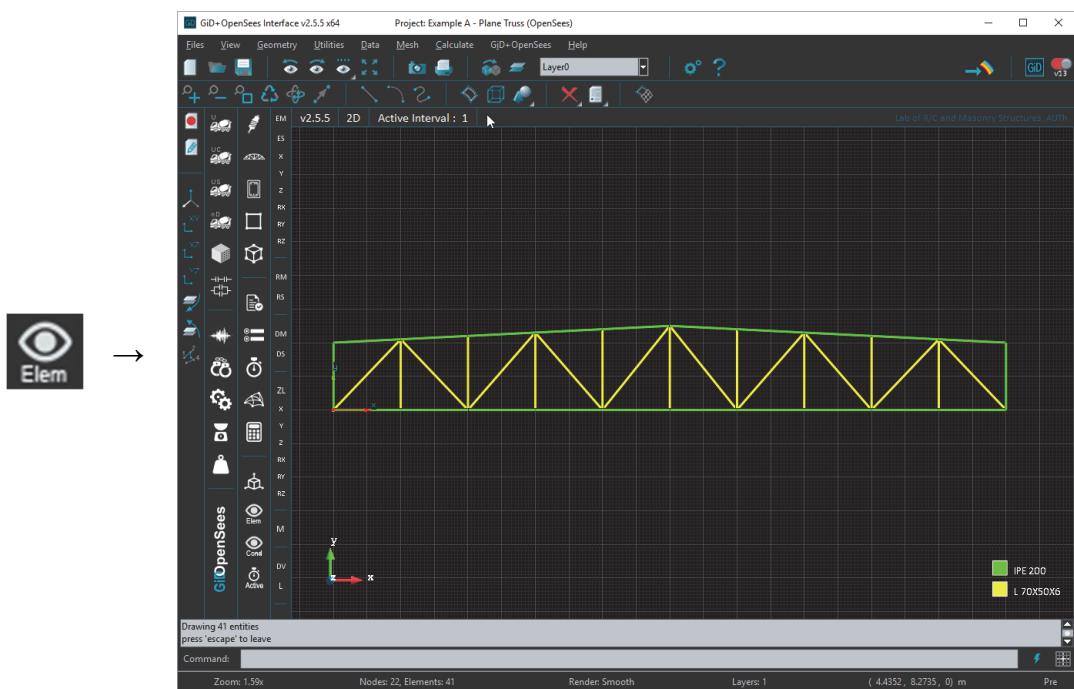
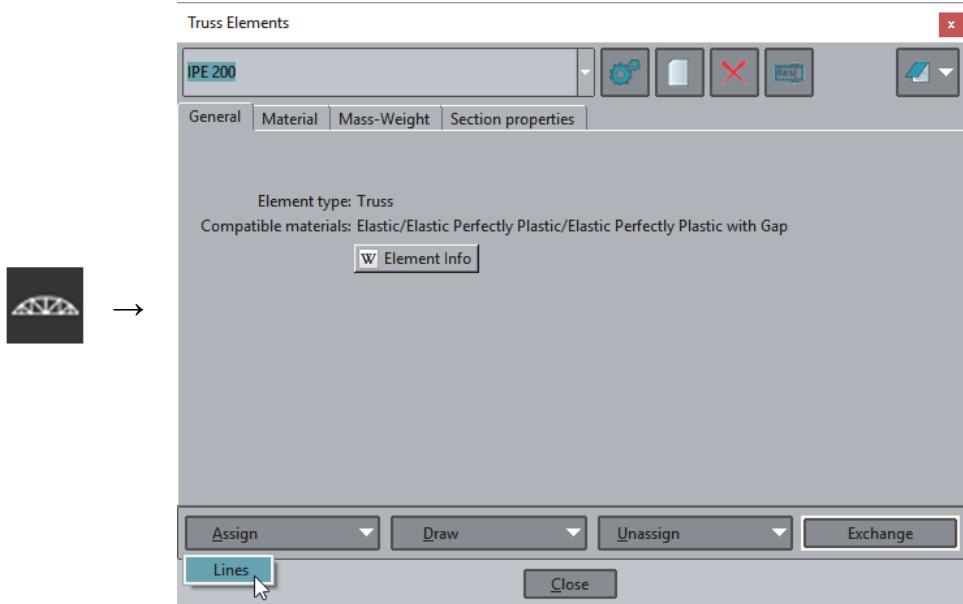
2. Define truss elements



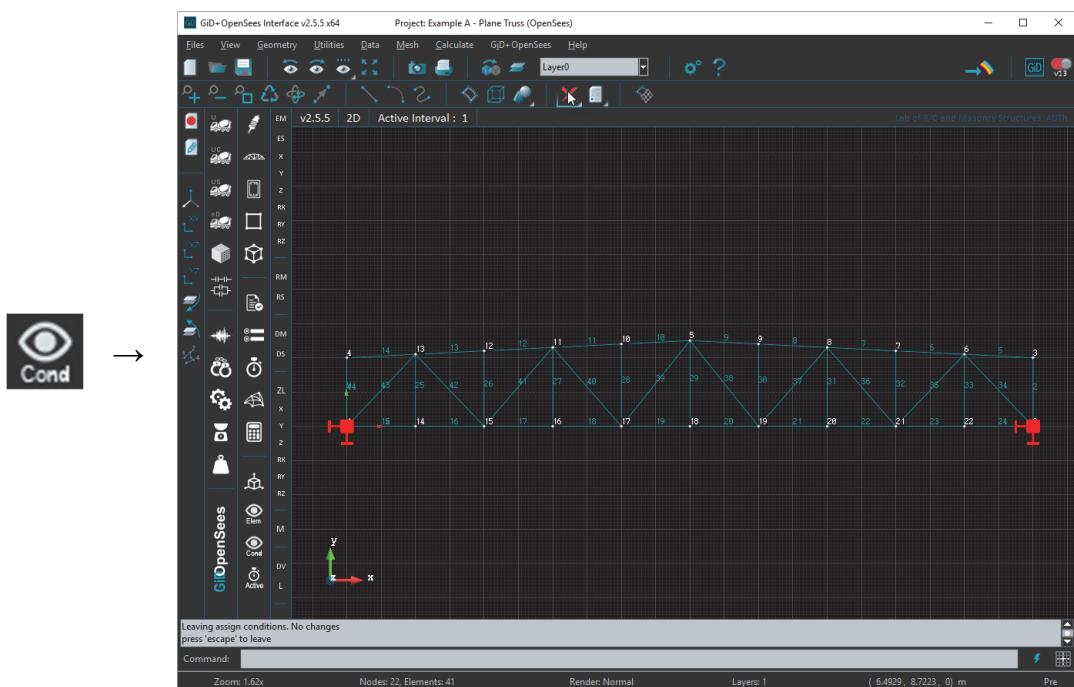
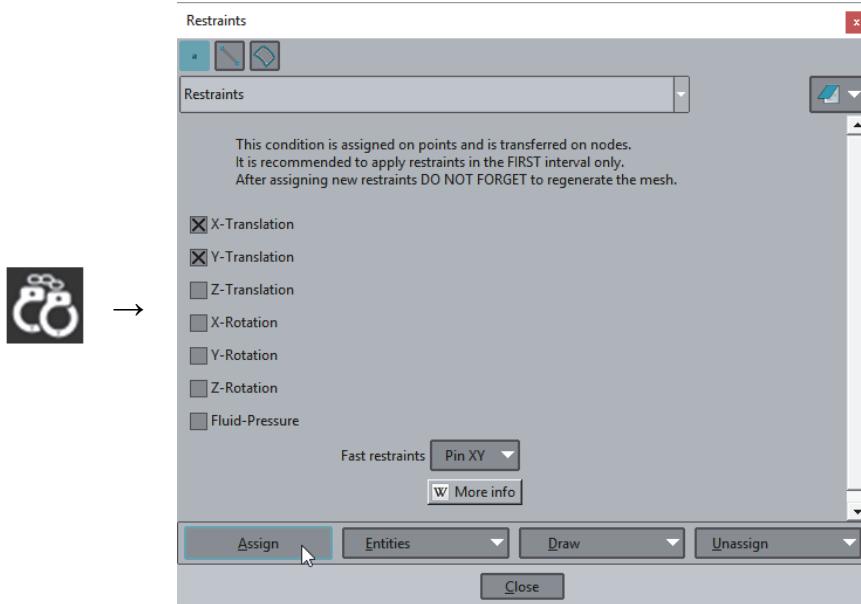
3. Draw geometry in GiD



4. Assign truss elements to geometry lines

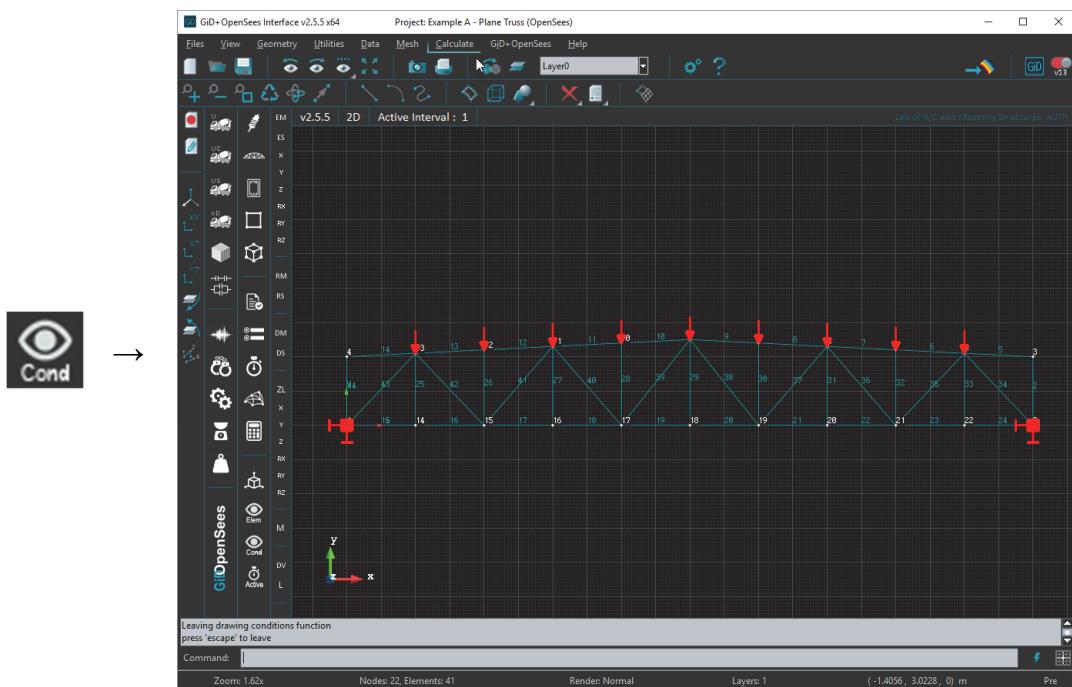
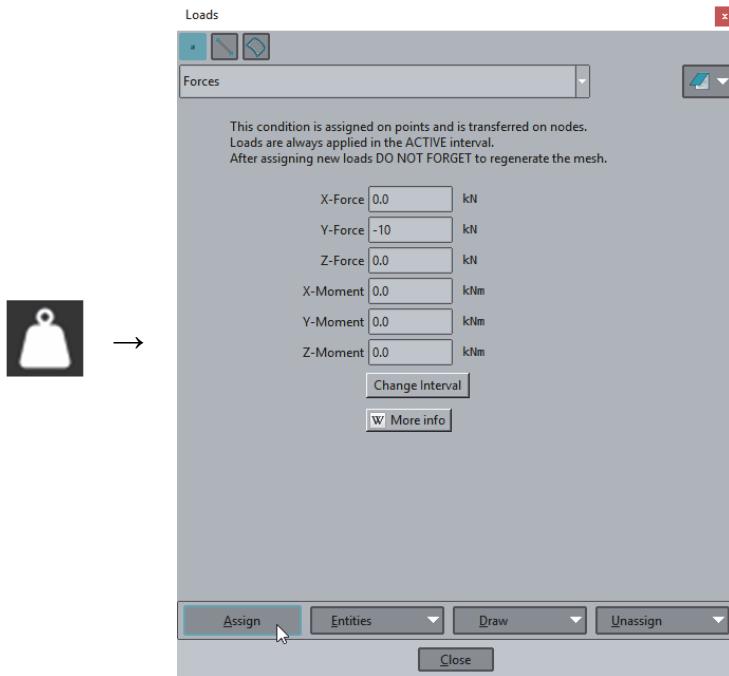


5. Assign restraints (pin XY) to truss support nodes



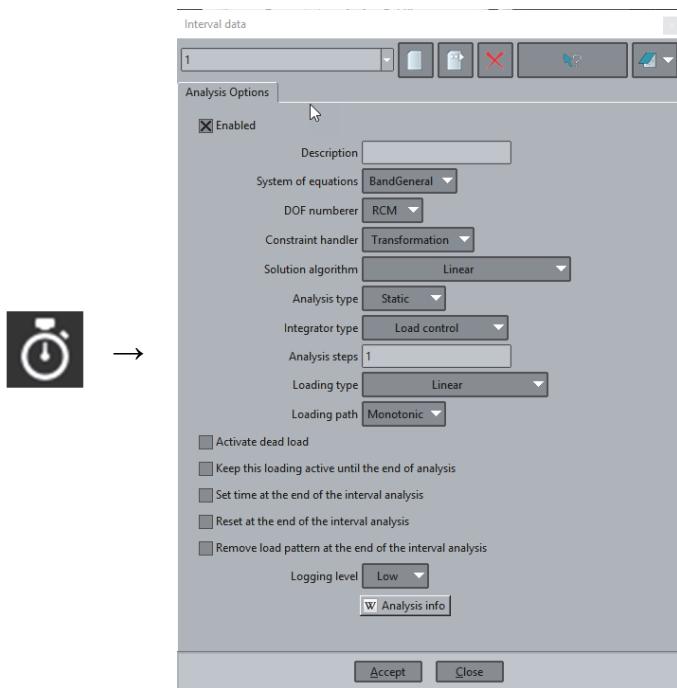
6. Assign loads

Assign -10 kN point forces to the nine truss nodes depicted below



7. Set analysis options

For this simple elastic analysis, no changes to the default interval settings are needed.



8. Set mesh size

Set one truss element per line : Mesh → Structured → Lines → Assign Number of Cells → 1 to all elements.

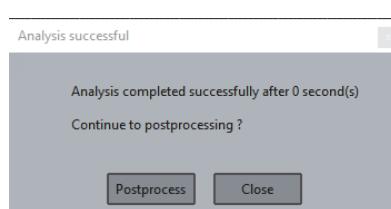
Then generate mesh from



Nodes: 22, Elements: 41

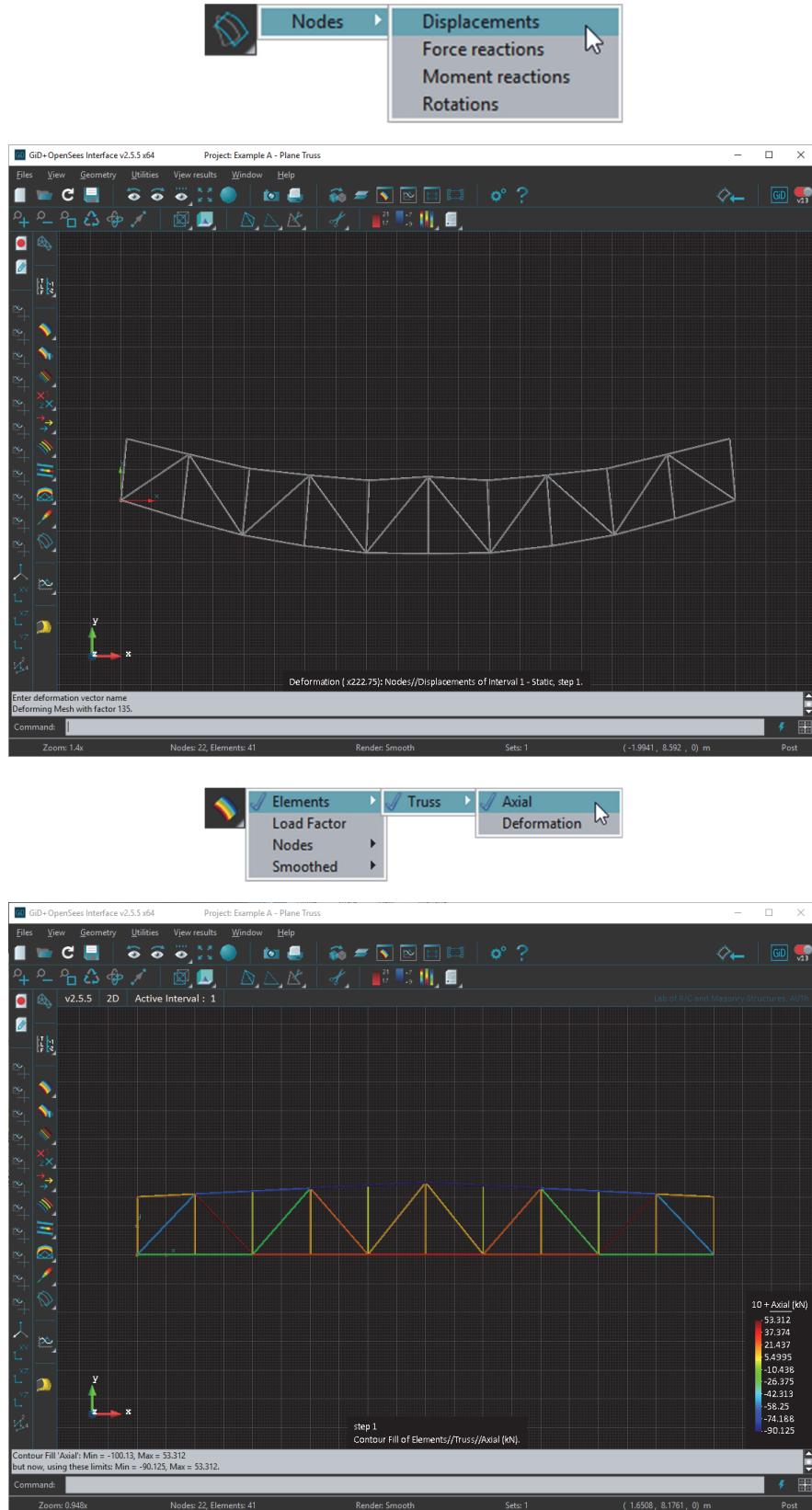
B. Analysis

Click and proceed to postprocessing when prompted.



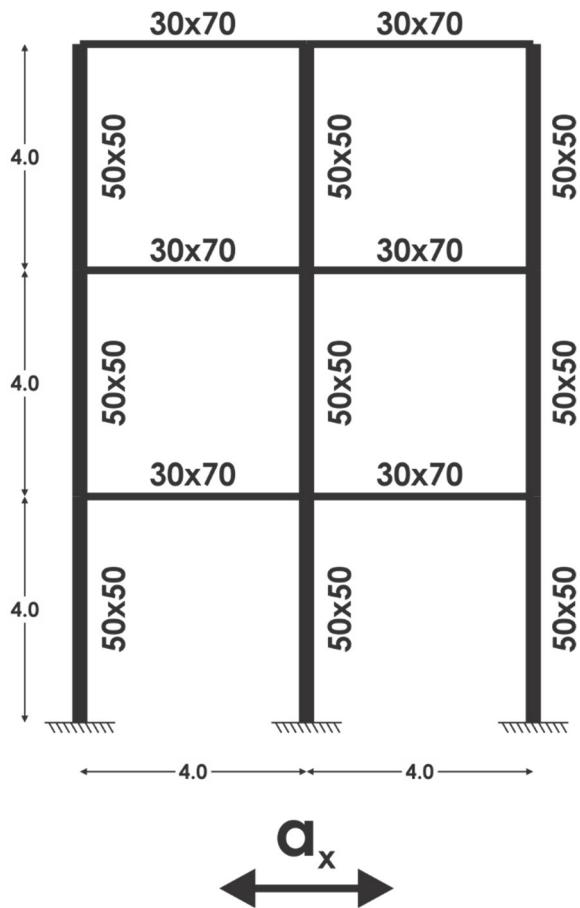
C. Postprocessing

From the GiD postprocessor, various result types can be viewed, i.e. deformed shape and truss axial forces :



Example B1 – Inelastic Plane Frame

The second example is a three-story, two-bay inelastic frame structure, modeled in the XY plane. The frame geometry is depicted below :



Material properties Concrete C20/25, Steel B500C

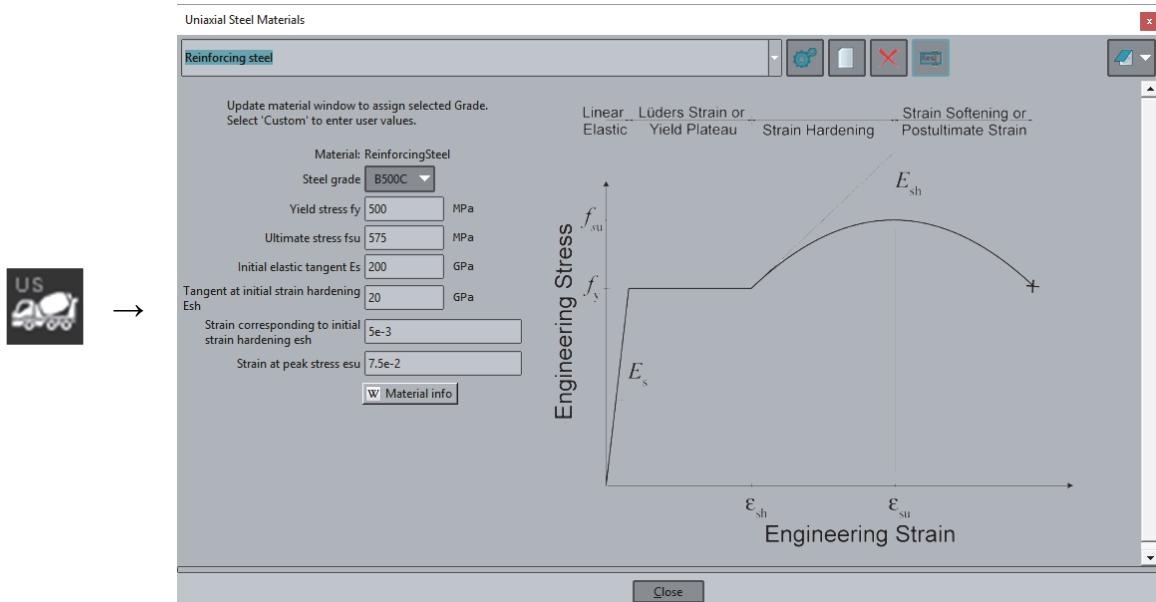
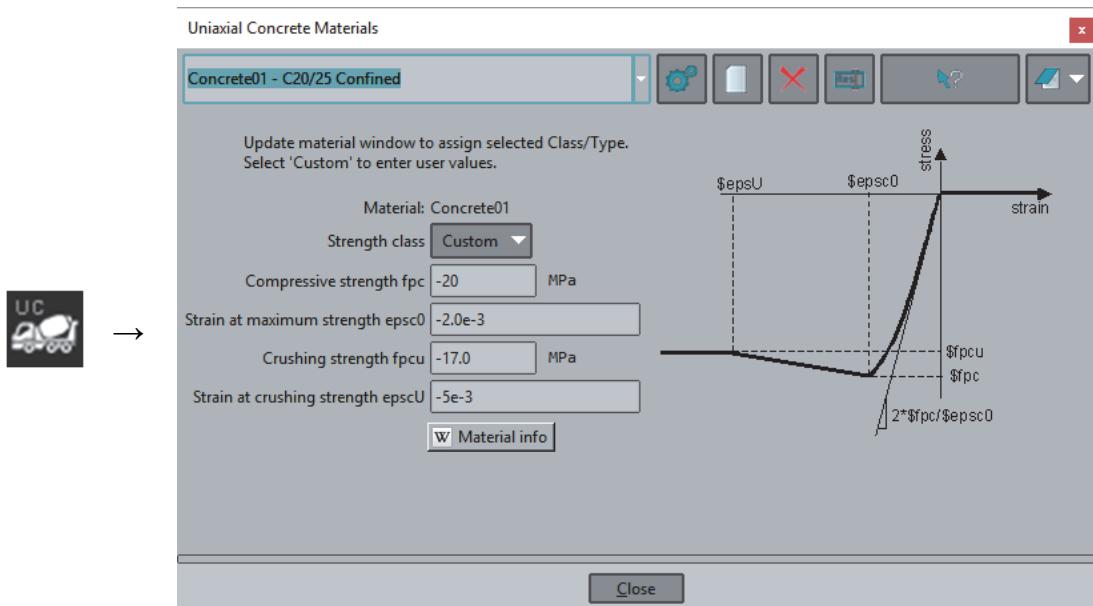
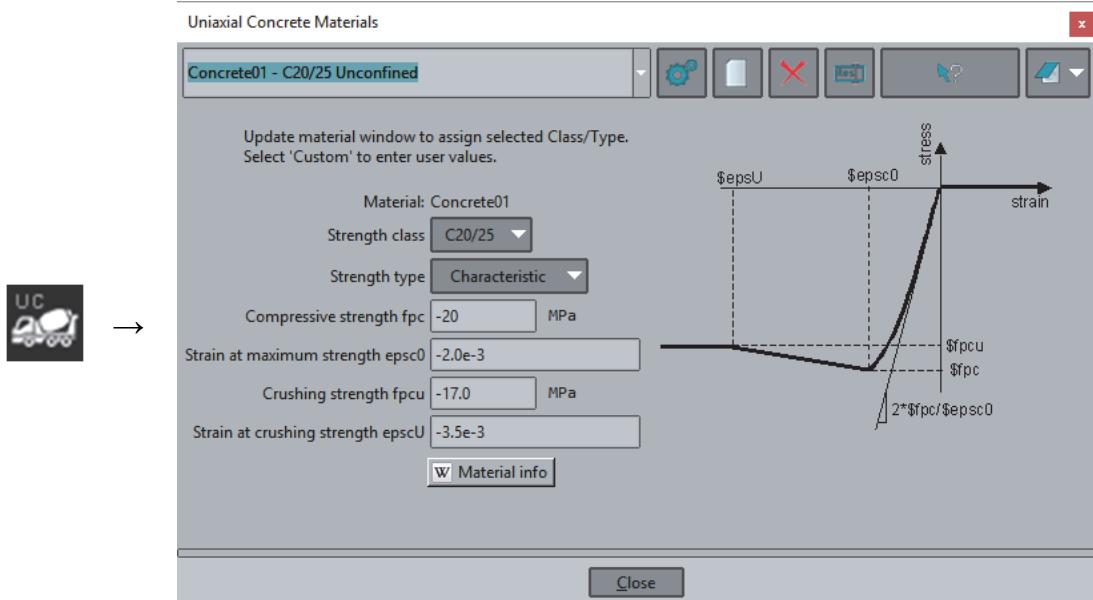
Section properties Columns 50x50, Beams 30x70

A. Preprocessing

1. Define concrete material as Concrete01

- Unconfined : C20/25 default values
- Confined : C20/25 default values with -5e-3 ultimate crushing strain

Define steel material as Reinforcing steel with B500C default values



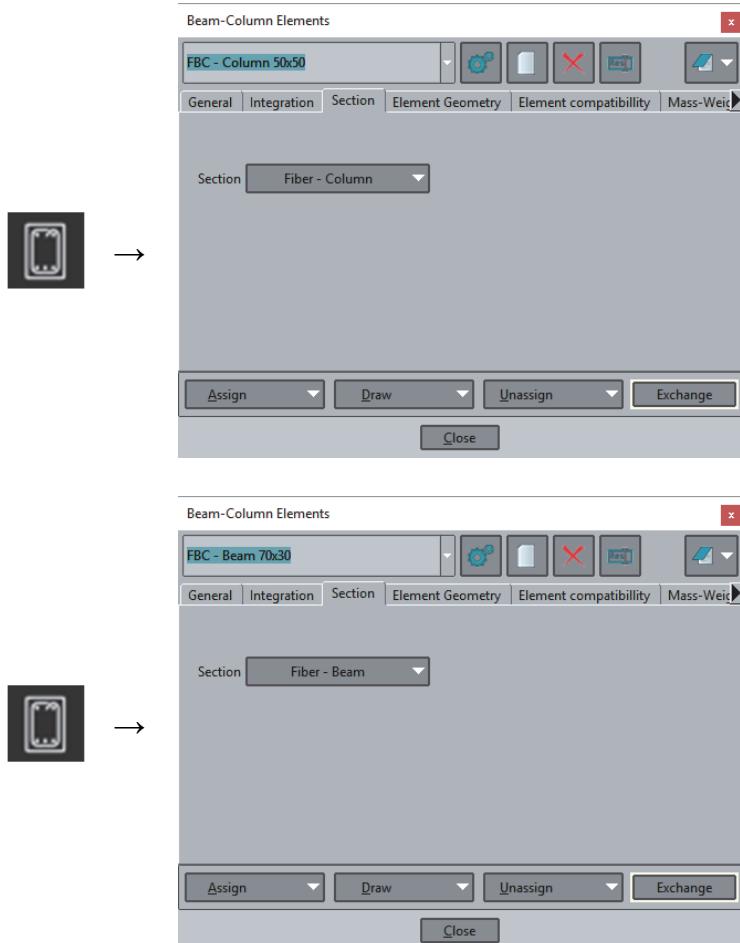
2. Define fiber sections for beams and columns

- Columns $4\varnothing 20+4\varnothing 18$
- Beams $3\varnothing 16$ top & bottom

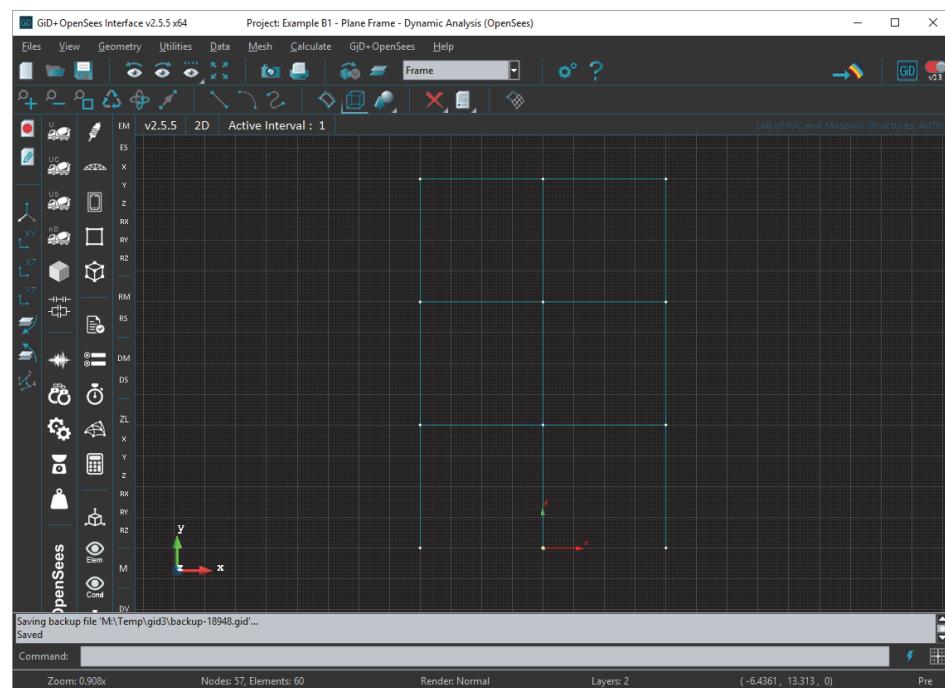
The diagram illustrates the process of defining fiber sections for concrete structures. It starts with a 3D model icon of a rectangular column or beam, followed by a right-pointing arrow, and then four corresponding software dialog boxes:

- Fiber - Column:** This dialog is used for columns. It shows a 2D cross-section diagram with height h and width b . Reinforcement details include 4 bars along the z-axis face and 4 bars along the y-axis face. Corner bar size is 20 mm and middle bar size is 18 mm. Material properties are set to Core material: Concrete01 - C20/25 Confined, Cover material: Concrete01 - C20/25 Unconfined, and Reinforcing bar material: Reinforcing steel.
- Fiber - Beam:** This dialog is used for beams. It shows a 2D cross-section diagram with height h and width b . Reinforcement details include 3 top bars and 3 bottom bars, both with a size of 16 mm. Material properties are set to Core material: Concrete01 - C20/25 Confined, Cover material: Concrete01 - C20/25 Unconfined, and Reinforcing bar material: Reinforcing steel.
- General:** A tab in the top-left corner of both dialogs.
- Section properties:** A tab in the top-left corner of both dialogs.
- Materials:** A tab in the top-left corner of both dialogs.
- Close:** A button at the bottom-right of each dialog.

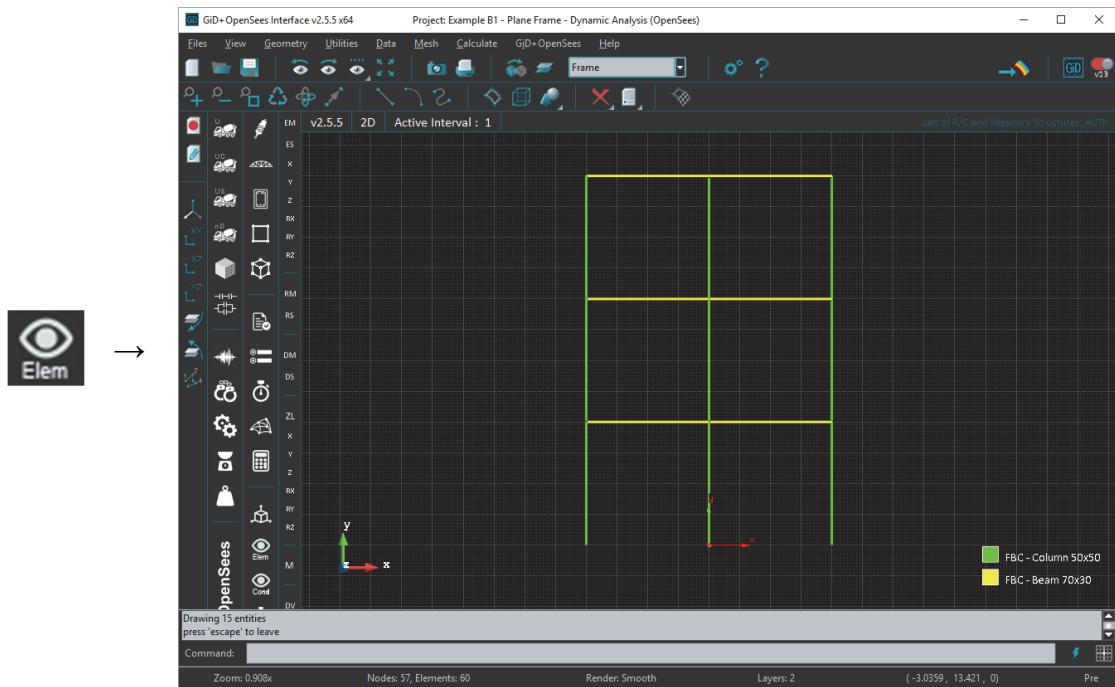
3. Define force-based beam-column elements for beams and columns



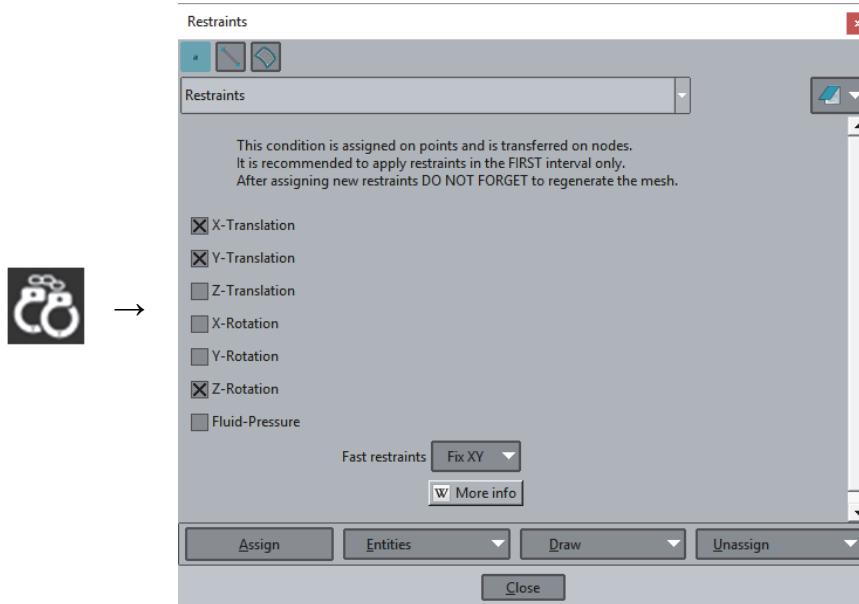
4. Draw geometry in GiD

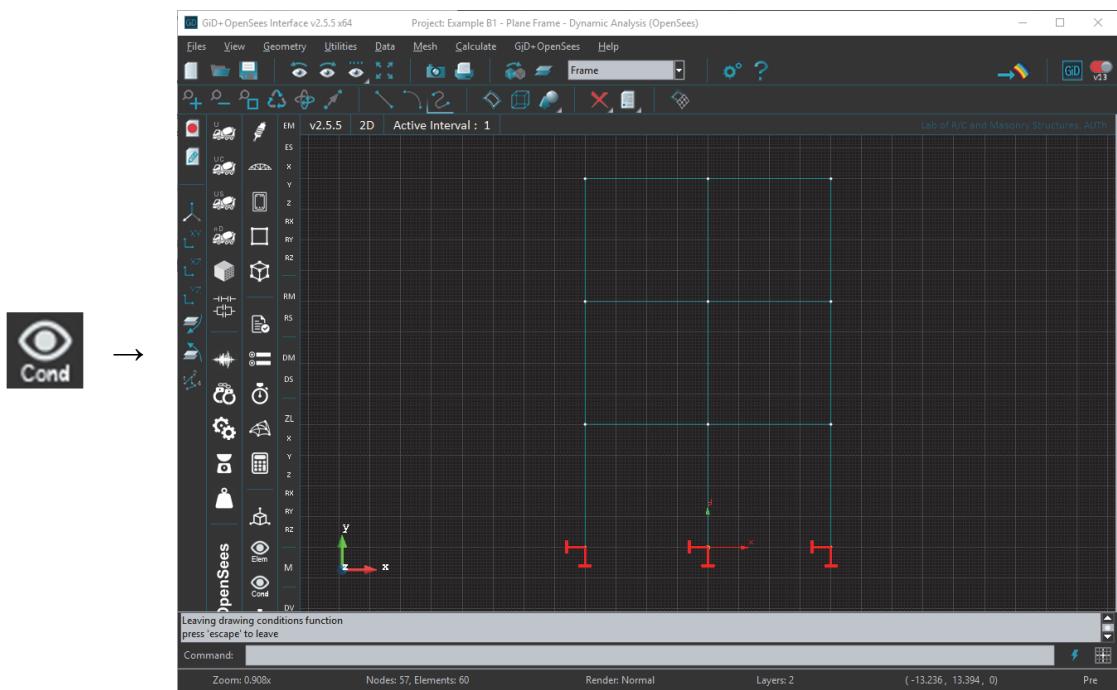


5. Assign beam and column force-deformation elements to geometry lines



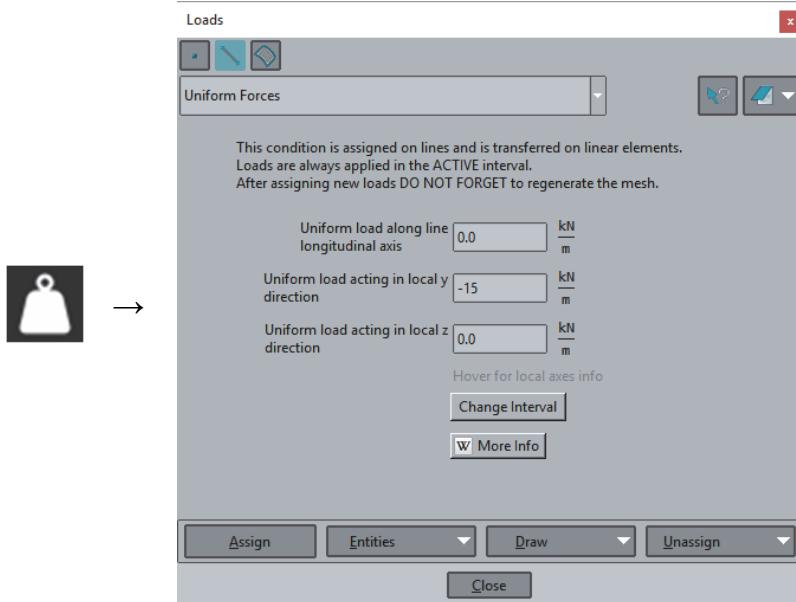
6. Assign restraints (fix XY) to truss support nodes (Active Interval : 1)

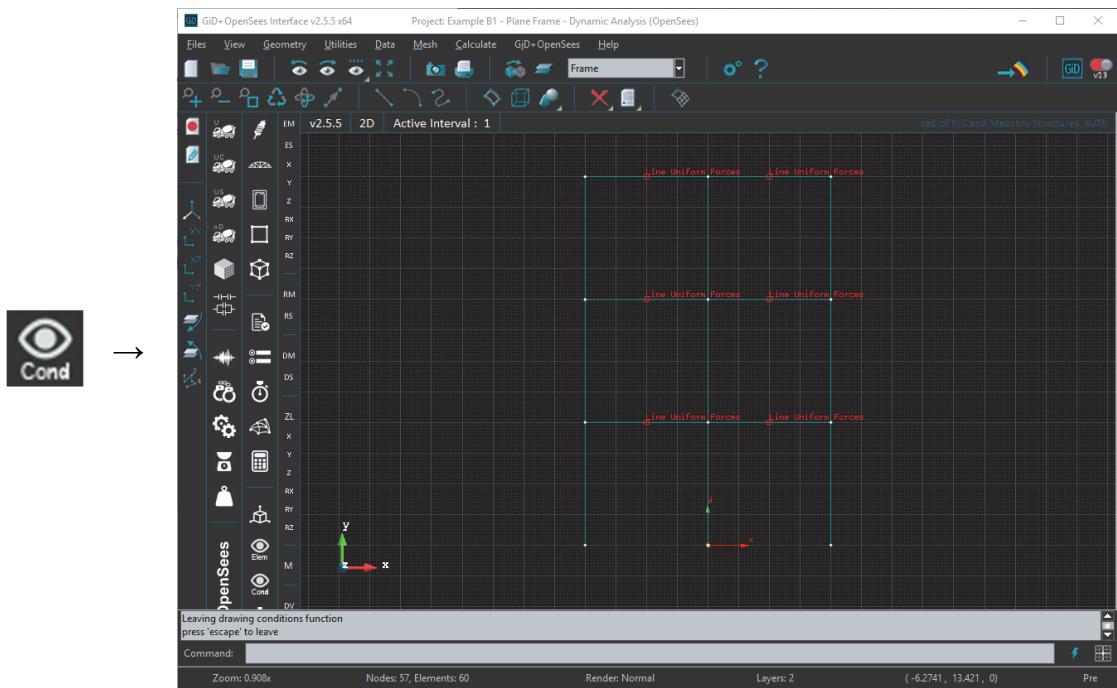




7. Assign static loads

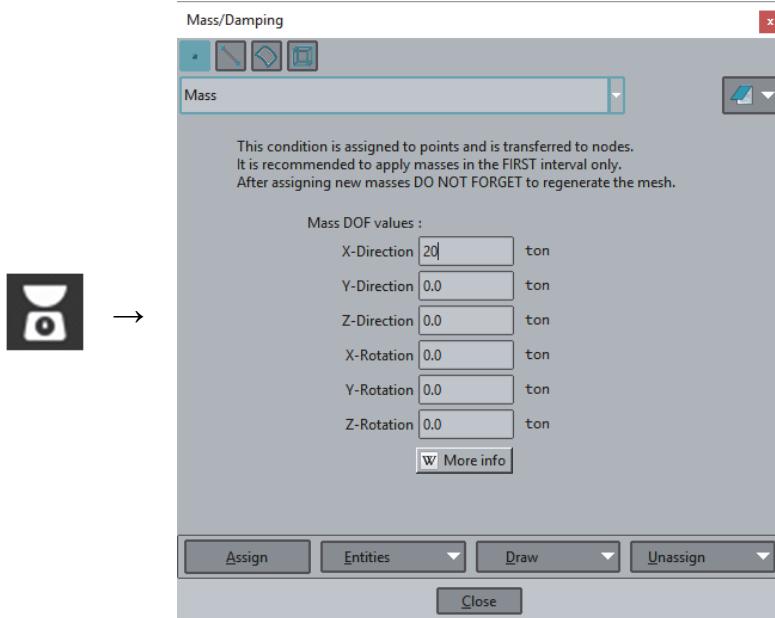
Assign -15 kN/m line uniform forces to all beams (Active Interval : 1)

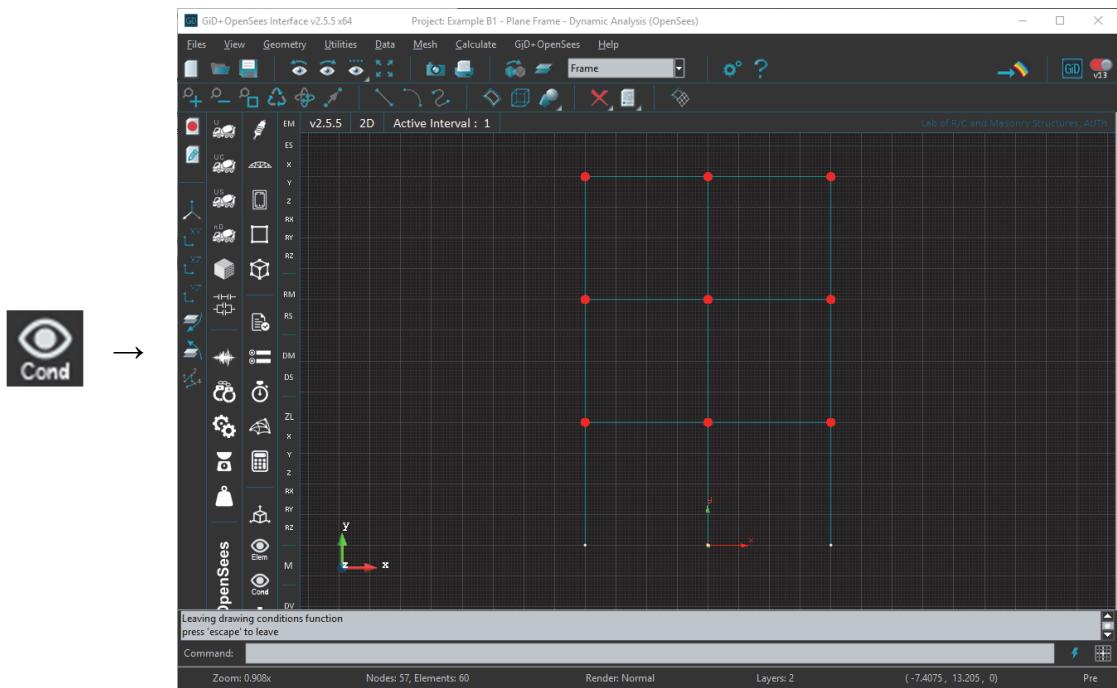




8. Assign masses

Assign 20 tons of mass in x-direction to all nodes in all stories (Active Interval : 1)





9. Define strong motion record

Record file : Loma_Prieta.dat

Time column : 1

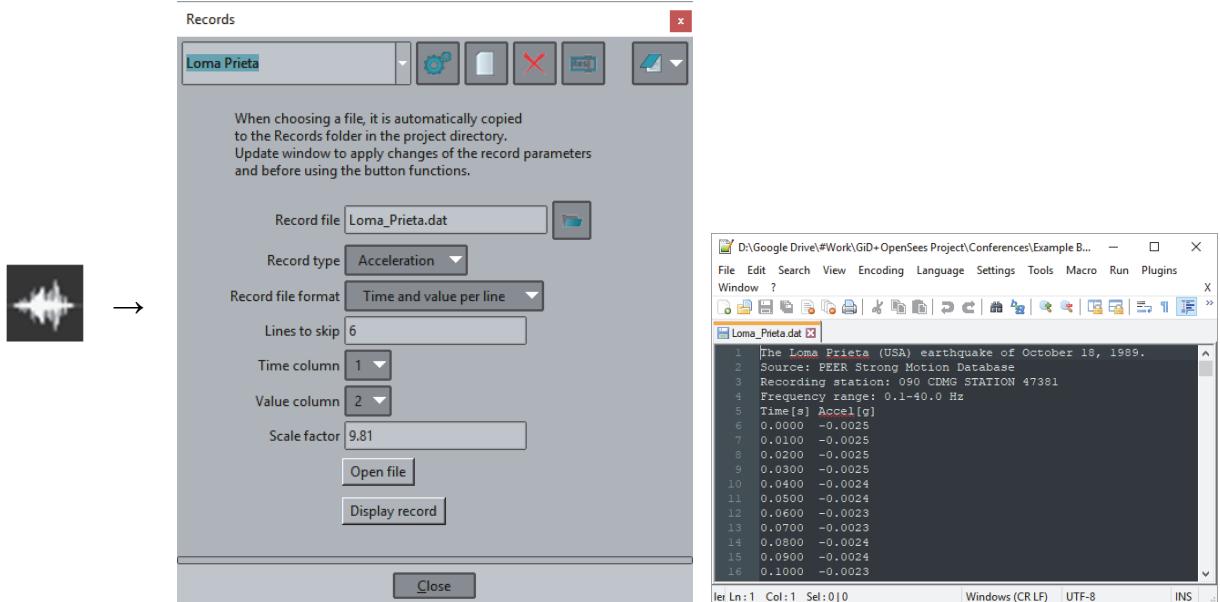
Type : Acceleration

Value column : 2

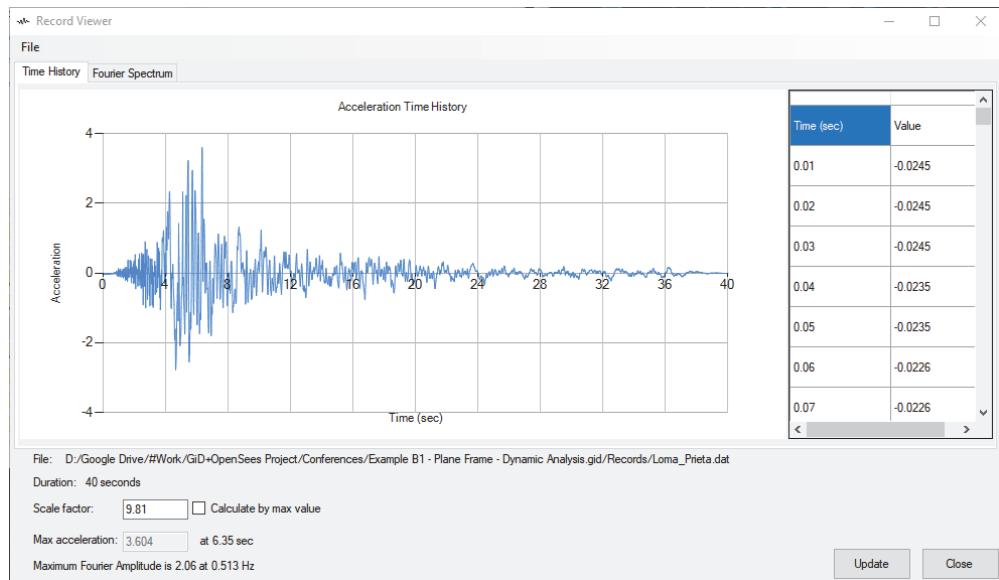
Format : Time and value per line

Scale factor : 9.81 (convert %g to m/s²)

Lines to skip : 6

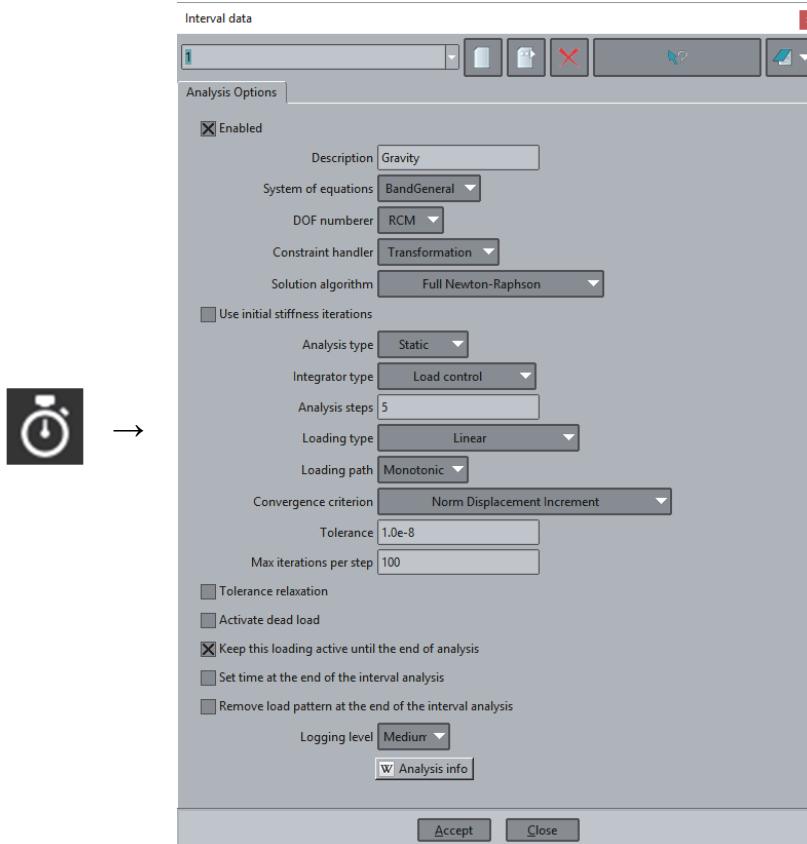


The selected strong motion record can be viewed by pressing ‘Display record’

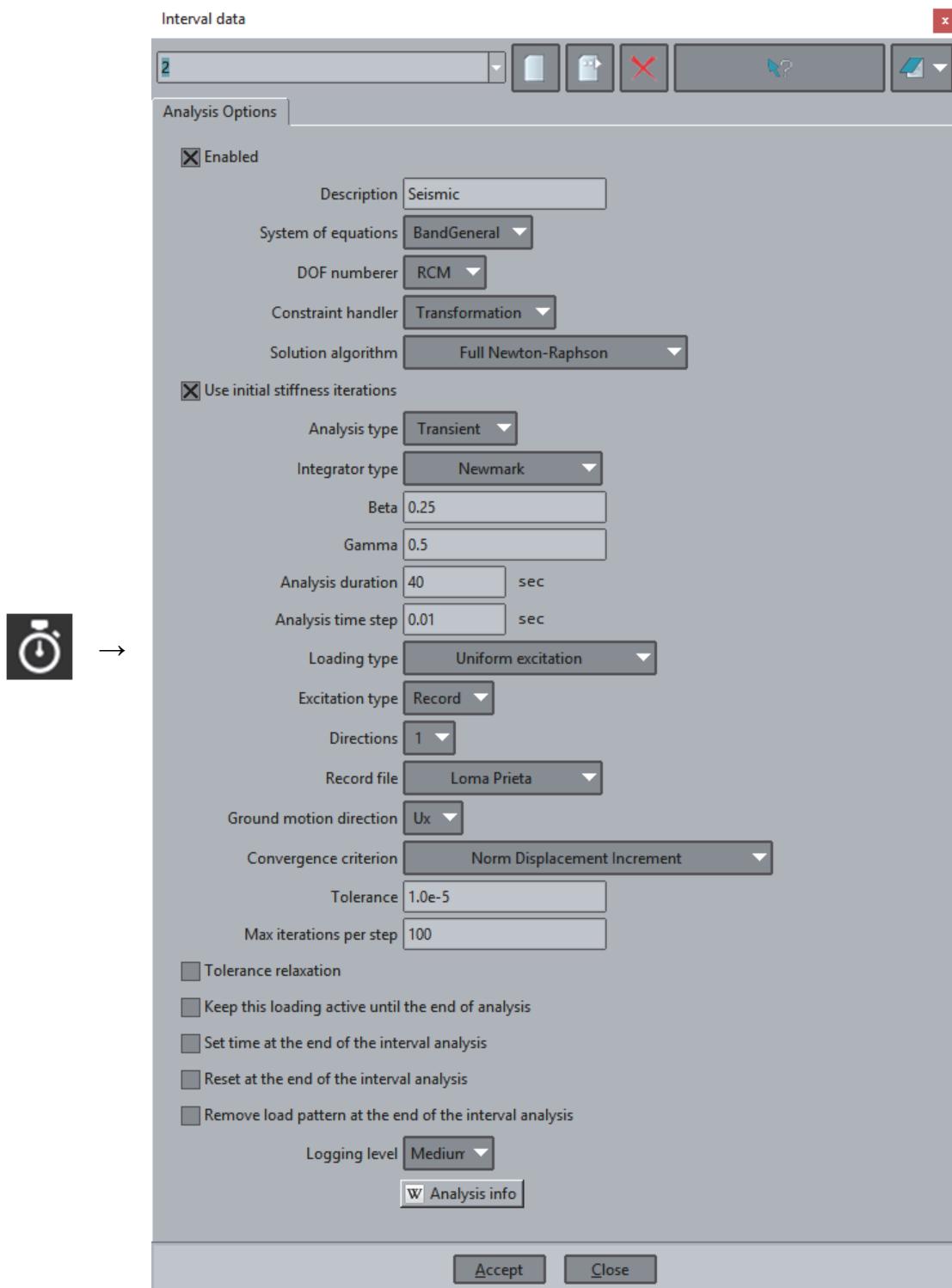


10. Set analysis options

The first interval contains boundary conditions, masses and gravity loading. It is defined as follows (Static analysis, Load Control, 5 steps)



After the end of gravity loading, another interval (2) is added for strong motion (inelastic dynamic analysis), as follows :



Analysis type is Transient, using the Newmark Integration Scheme ($\beta=0.5$, $\gamma=0.25$), the analysis duration is 40 seconds with a time step of 0.01 (exactly as specified in the record file) and the excitation type is uniform, based on the already defined record (Loma_Prieta) along the Ux direction (horizontal excitation).

11. Set mesh size

Set four force-based elements per line : Mesh → Structured → Lines → Assign Number of Cells → 4 to all elements.

Then generate mesh from 

Nodes: 57, Elements: 60

B. Analysis

Click  and proceed to postprocessing when prompted.

```
C:\OpenSees\OpenSees.exe

OpenSees -- Open System For Earthquake Engineering Simulation
          Pacific Earthquake Engineering Research Center
          Version 2.5.0 (rev 6345) 64-Bit

(c) Copyright 1999-2010 The Regents of the University of California
All Rights Reserved
(Copyright and Disclaimer @ http://www.berkeley.edu/OpenSees/copyright.html)

-----  

GID+OpenSees Interface v2.5.5  

Analysis summary  

Interval 1 : Static - 6 steps  

Interval 2 : Transient - 4000 steps x 0.01 s  

WARNING: invalid modelid: C:/Tcl/lib/tcl8.5/reg1.2/tclreg12.dll - load modelid 1 forces  

WARNING: invalid modelid: C:/Tcl/lib/tcl8.5/reg1.2/tclreg12.dll - load modelid 1 forces  

Starting analysis at 13:59:29  

Running interval 1 ...  

Ls (1)0.20000 CtestNormDispIncr::test() - iteration: 6 current Norm: 9.26279e-010 (max: 1e-008, Norm deltaR: 1.2615e-010)  

Ls (1)0.40000 CtestNormDispIncr::test() - iteration: 5 current Norm: 1.91783e-009 (max: 1e-008, Norm deltaR: 2.10208e-010)  

Ls (1)0.60000 CtestNormDispIncr::test() - iteration: 4 current Norm: 2.97271e-009 (max: 1e-008, Norm deltaR: 3.84317e-010)  

Ls (1)0.80000 CtestNormDispIncr::test() - iteration: 4 current Norm: 4.09987e-009 (max: 1e-008, Norm deltaR: 7.35938e-010)  

Ls (1)1.00000 CtestNormDispIncr::test() - iteration: 4 current Norm: 5.28431e-009 (max: 1e-008, Norm deltaR: 1.14817e-009)  

Analysis completed SUCCESSFULLY  

Committed steps : 6  

Running interval 2 ...  

Time (2)0.01000 CtestNormDispIncr::test() - iteration: 1 current Norm: 4.26279e-006 (max: 1e-005, Norm deltaR: 0.0365975)  

Time (2)0.02000 CtestNormDispIncr::test() - iteration: 2 current Norm: 2.33546e-007 (max: 1e-005, Norm deltaR: 0.142695)  

Time (2)0.03000 CtestNormDispIncr::test() - iteration: 2 current Norm: 4.7856e-007 (max: 1e-005, Norm deltaR: 0.3427)  

Time (2)0.04000 CtestNormDispIncr::test() - iteration: 2 current Norm: 7.3306e-007 (max: 1e-005, Norm deltaR: 0.60121)  

Time (2)0.05000 CtestNormDispIncr::test() - iteration: 2 current Norm: 1.08223e-006 (max: 1e-005, Norm deltaR: 0.76075)  

Time (2)0.06000 CtestNormDispIncr::test() - iteration: 2 current Norm: 1.24888e-006 (max: 1e-005, Norm deltaR: 1.2559)  

Time (2)0.07000 CtestNormDispIncr::test() - iteration: 2 current Norm: 1.49601e-006 (max: 1e-005, Norm deltaR: 1.61242)  

Time (2)0.08000 CtestNormDispIncr::test() - iteration: 2 current Norm: 1.73342e-006 (max: 1e-005, Norm deltaR: 1.97884)  

Time (2)0.09000 CtestNormDispIncr::test() - iteration: 3 current Norm: 1.97204e-006 (max: 1e-005, Norm deltaR: 2.34255)  

Time (2)0.10000 CtestNormDispIncr::test() - iteration: 3 current Norm: 2.22424e-006 (max: 1e-005, Norm deltaR: 2.11609)  

Time (2)0.11000 CtestNormDispIncr::test() - iteration: 3 current Norm: 2.44132e-006 (max: 1e-005, Norm deltaR: 1.05809)  

Time (2)0.12000 CtestNormDispIncr::test() - iteration: 2 current Norm: 2.66226e-006 (max: 1e-005, Norm deltaR: 3.41243)  

Time (2)0.13000 CtestNormDispIncr::test() - iteration: 2 current Norm: 2.87941e-006 (max: 1e-005, Norm deltaR: 3.76812)  

Time (2)0.14000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.07283e-006 (max: 1e-005, Norm deltaR: 4.08228)  

Time (2)0.15000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.26524e-006 (max: 1e-005, Norm deltaR: 4.39677)  

Time (2)0.16000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.4131e-006 (max: 1e-005, Norm deltaR: 4.65203)  

Time (2)0.17000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.56376e-006 (max: 1e-005, Norm deltaR: 4.92228)  

Time (2)0.18000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.69279e-006 (max: 1e-005, Norm deltaR: 5.15656)  

Time (2)0.19000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.80161e-006 (max: 1e-005, Norm deltaR: 5.39486)  

Time (2)0.20000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.90049e-006 (max: 1e-005, Norm deltaR: 5.63315)  

Time (2)0.21000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.92493e-006 (max: 1e-005, Norm deltaR: 5.76413)  

Time (2)0.22000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.92571e-006 (max: 1e-005, Norm deltaR: 5.89229)  

Time (2)0.23000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.89294e-006 (max: 1e-005, Norm deltaR: 5.97076)  

Time (2)0.24000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.82782e-006 (max: 1e-005, Norm deltaR: 6.00112)  

Time (2)0.25000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.76004e-006 (max: 1e-005, Norm deltaR: 5.99773)  

Time (2)0.26000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.63142e-006 (max: 1e-005, Norm deltaR: 5.97773)  

Time (2)0.27000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.48744e-006 (max: 1e-005, Norm deltaR: 5.85281)  

Time (2)0.28000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.44541e-006 (max: 1e-005, Norm deltaR: 5.703844)  

Time (2)0.29000 CtestNormDispIncr::test() - iteration: 2 current Norm: 3.17938e-006 (max: 1e-005, Norm deltaR: 5.56575)  

Time (2)0.30000 CtestNormDispIncr::test() - iteration: 2 current Norm: 2.98056e-006 (max: 1e-005, Norm deltaR: 5.35198)  

Time (2)0.31000 CtestNormDispIncr::test() - iteration: 2 current Norm: 2.75454e-006 (max: 1e-005, Norm deltaR: 5.08983)  

Time (2)0.32000 CtestNormDispIncr::test() - iteration: 2 current Norm: 2.52678e-006 (max: 1e-005, Norm deltaR: 4.79492)
```

Analysis successful

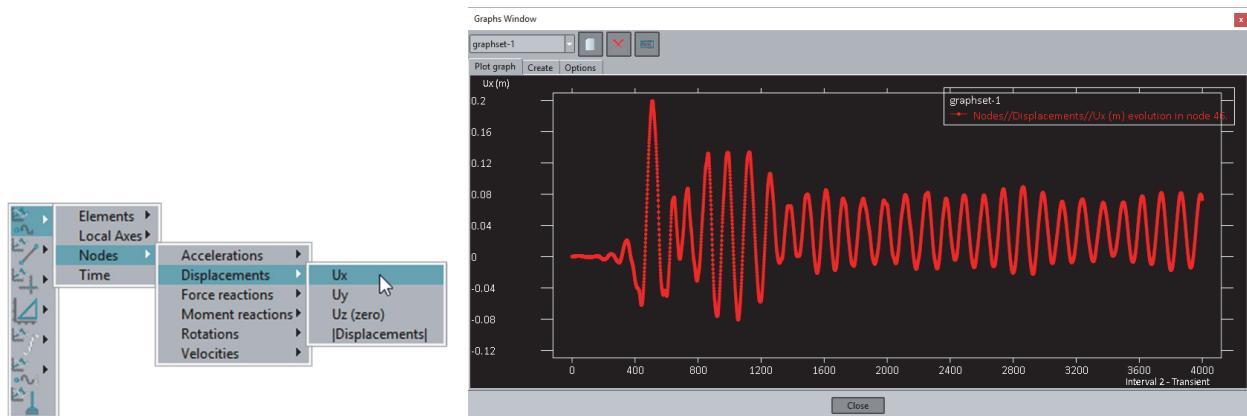
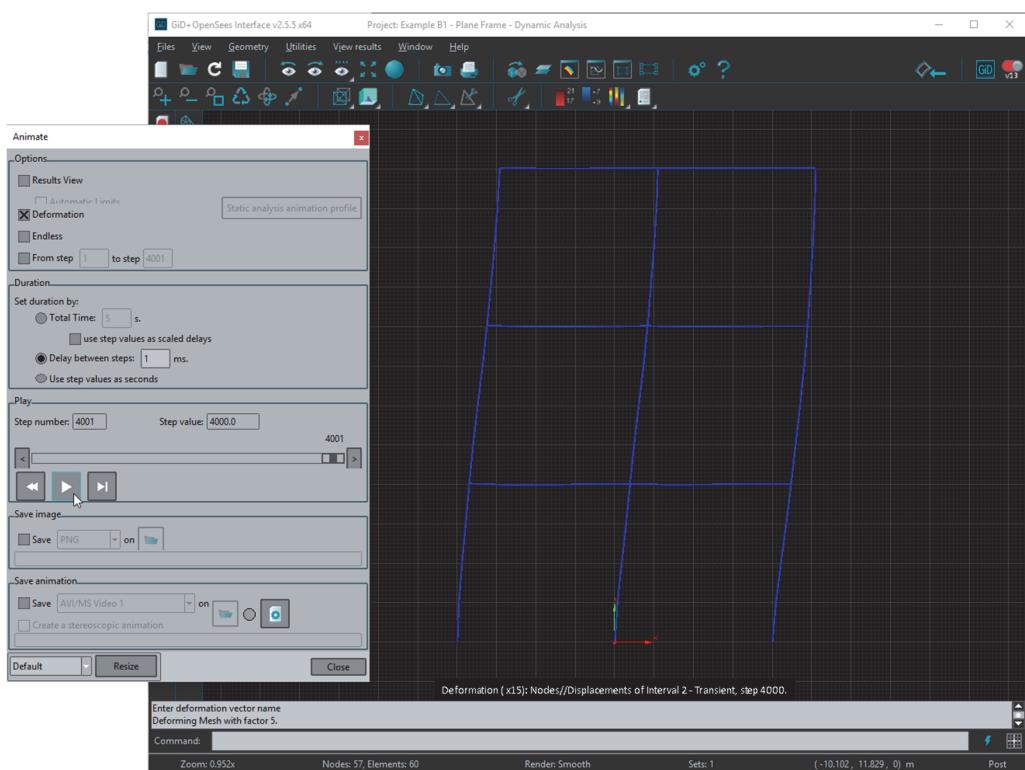
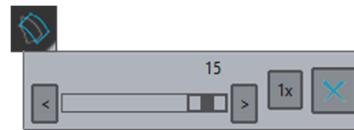
Analysis completed successfully after 2 minute(s) and 35 second(s)

Continue to postprocessing ?

C. Postprocessing

For this dynamic analysis, apart from the usual result types that were discussed in the previous example, it is interesting to show a displacement animation under strong motion and the roof horizontal displacement response graph in time.

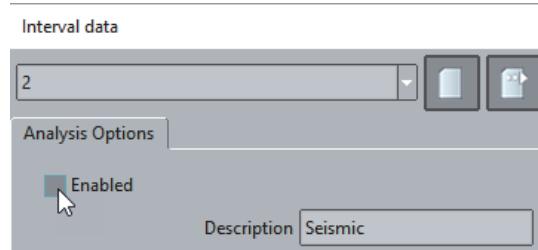
Set displacements to 15x select Animate and press play



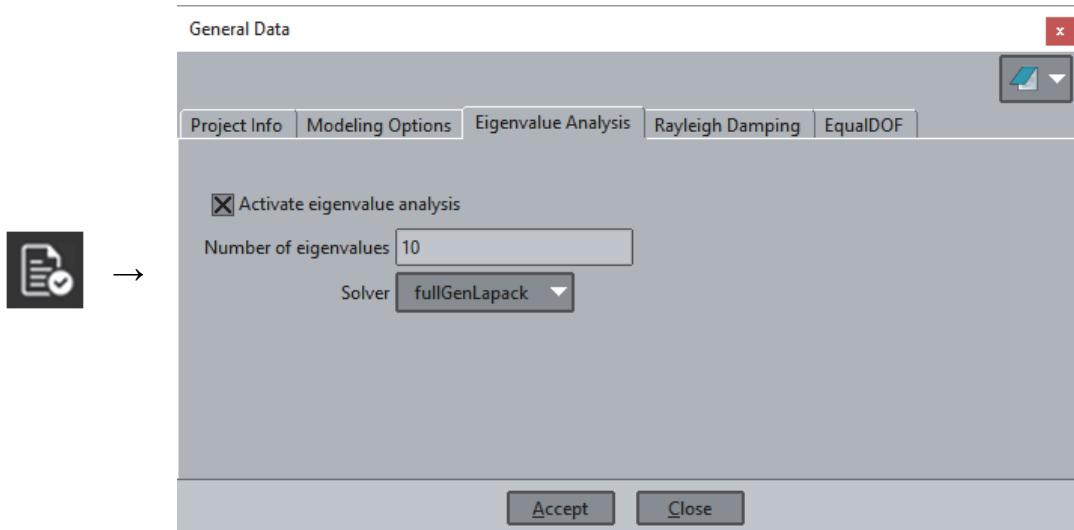
Note : No damping was considered in the analysis

How to include Rayleigh Damping

1. Deactivate Interval 2 (dynamic analysis)



2. Activate eigenvalue analysis

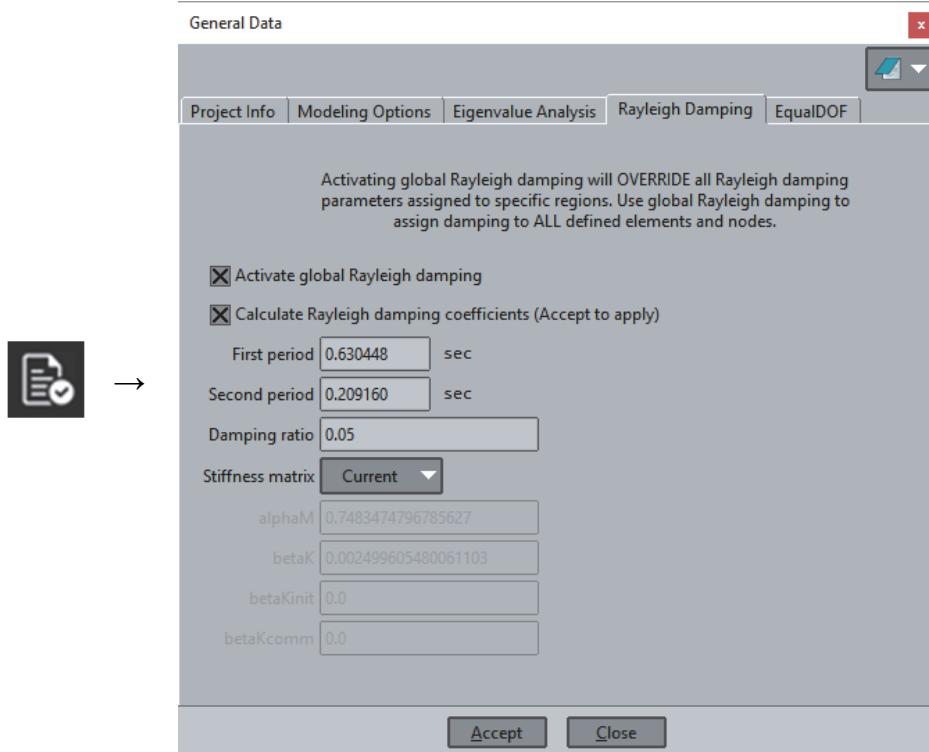


3. Run analysis, postprocess and get period values

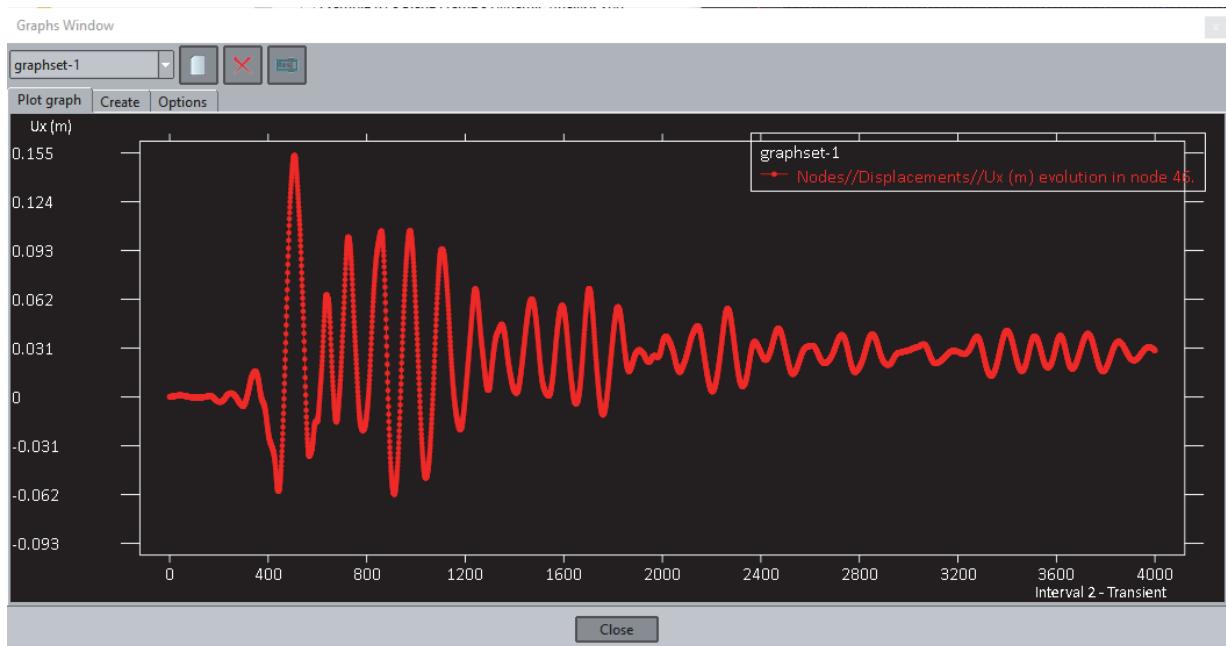
✓ Mode 1 (T1 = 0.630448 s)
Mode 2 (T2 = 0.209160 s)
Mode 3 (T3 = 0.131403 s)
Mode 4 (T4 = 0.026645 s)
Mode 5 (T5 = 0.026481 s)
Mode 6 (T6 = 0.026150 s)
Mode 7 (T7 = 0.015390 s)
Mode 8 (T8 = 0.015359 s)
Mode 9 (T9 = 0.015294 s)
Mode 10 (T10 = 4.686213 s)

4. Back to preprocessing, activate Interval 2 again.

5. Enter T1 and T2 into Rayleigh Damping window and calculate parameters for 5% damping (common value for buildings)

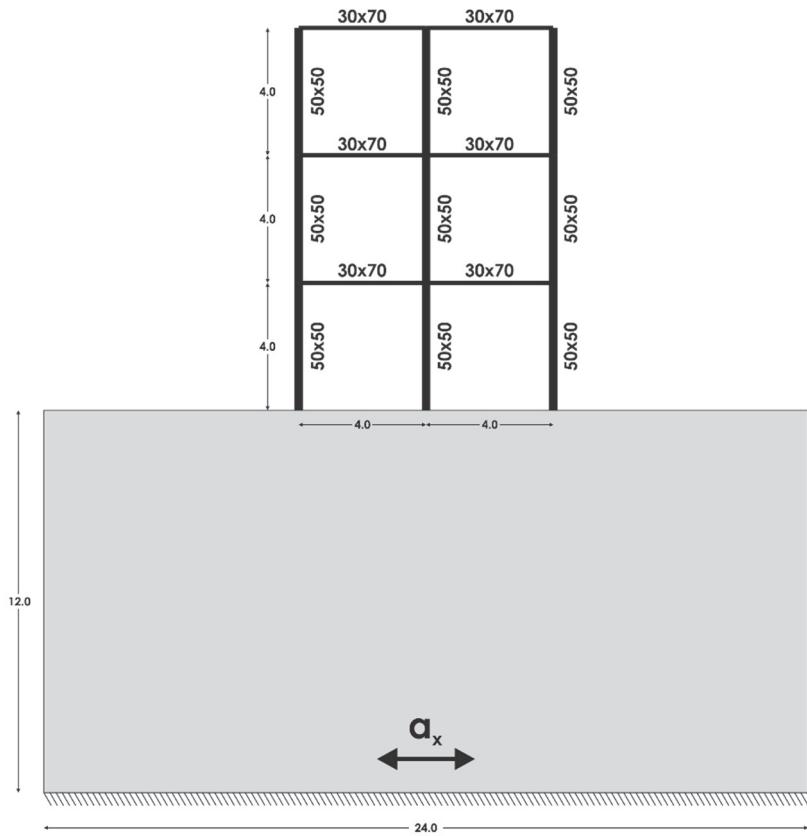


6. Run analysis again and display results. Now the response is clearly damped.



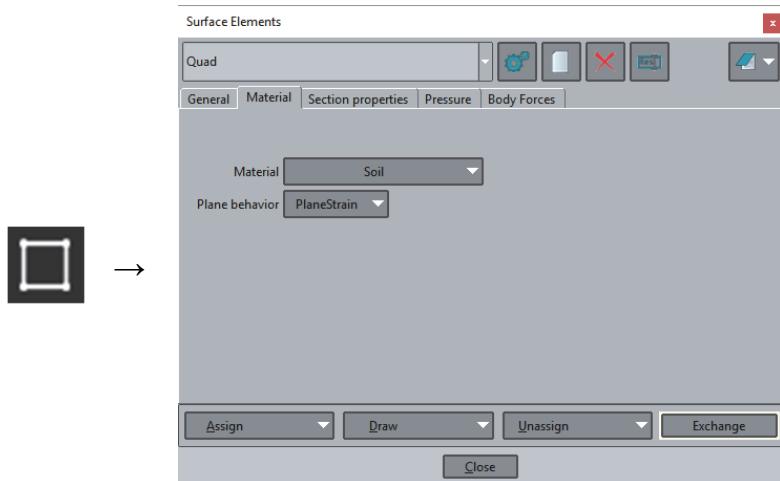
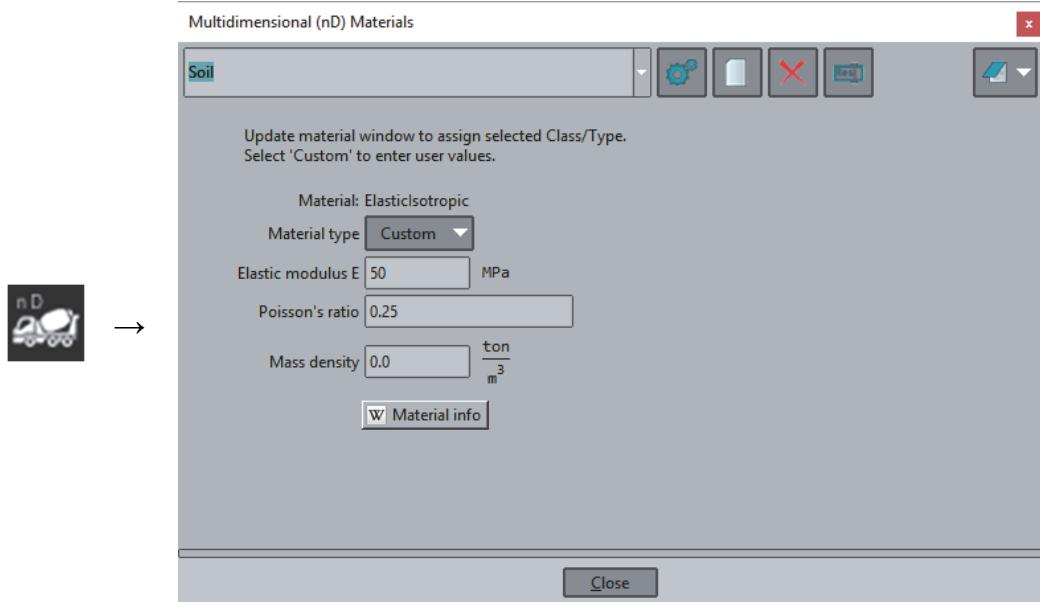
Example B2 – Inelastic Plane Frame on Elastic Soil (SSI)

The objective of the last example is to add an elastic soil medium under the previous inelastic frame structure and observe the differences in response due to soil-structure interaction (SSI) effects. The updated geometry is depicted below :

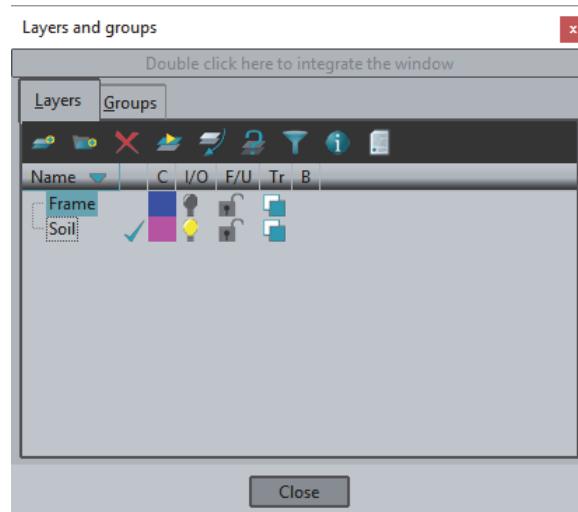


A. Preprocessing

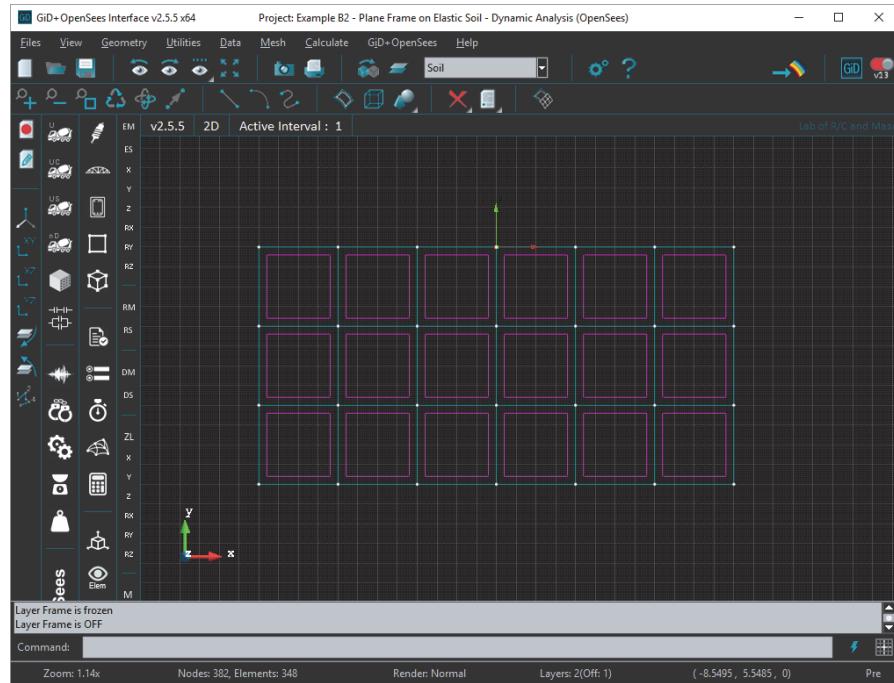
1. Define soil material as an Elastic Isotropic nD material with $E = 50$ MPa and $\nu = 0.25$ (unsaturated medium to hard clay) and assign this material to a quadrilateral surface element type under Plane Strain conditions.



2. Hide the current layer containing the frame model and add a new empty 'Soil' layer for adding the additional geometry.

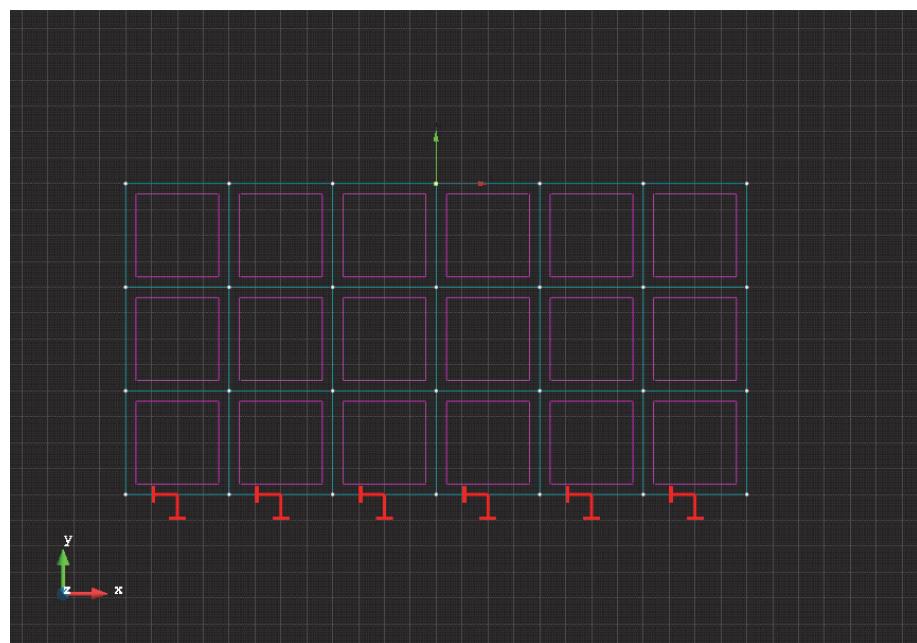


3. Draw soil geometry in GiD. Draw a 6x3 matrix 4x4 m square surfaces, using the 'Soil' layer as depicted. The axes origin (0,0) should coincide with the base of the middle frame column.

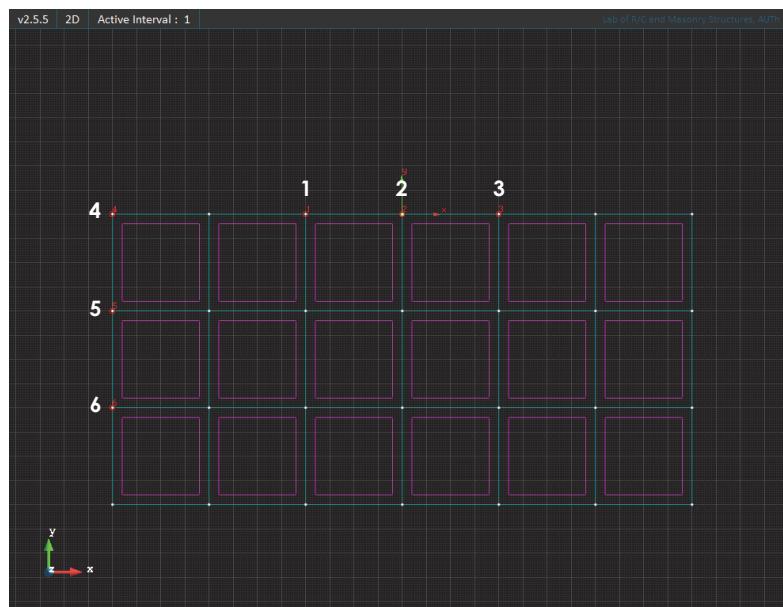
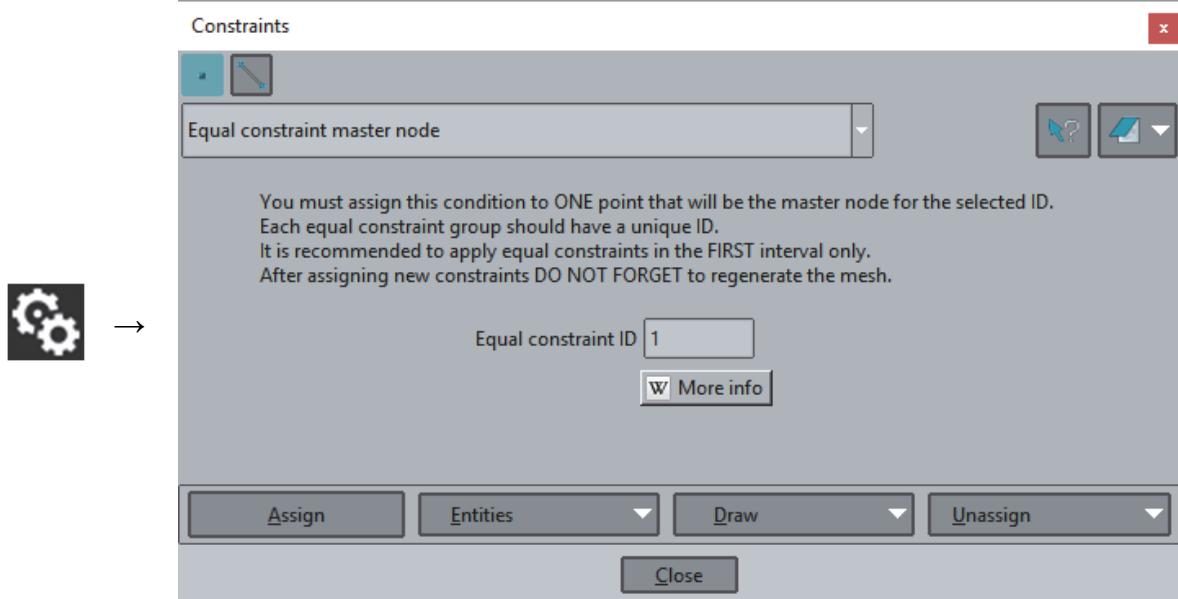


4. Assign the soil quadrilateral element defined earlier to all surfaces.

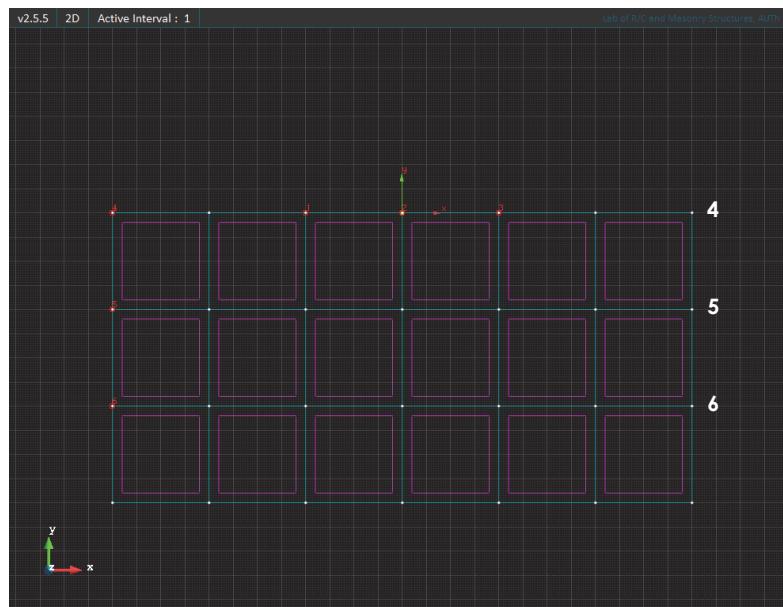
5. Assign a pin-XY restraint to the bottom boundary of the soil medium.



6. Assign equal constraint master nodes with ID = 1,2,3,4,5,6 to the following points :

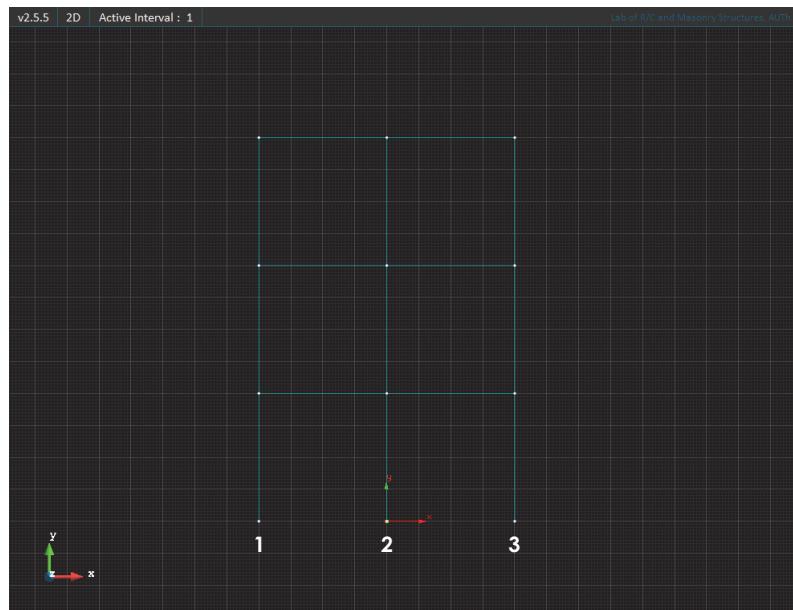


Then assign equal constraint slave nodes with ID = 4,5,6 with X-Translation only to the following points :

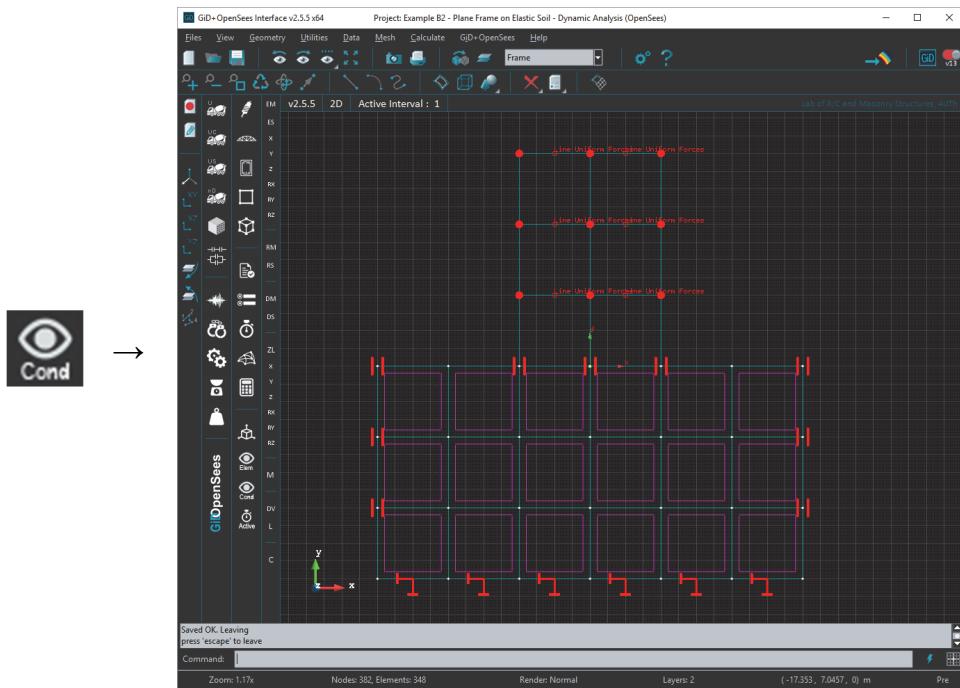


This condition enforces an approximate shear-type horizontal response for the soil substrate.

Hide the Soil layer and show the Frame layer. Then assign the remaining equal constraint slave nodes with ID = 1,2,3 with X & Y translation to the three frame column bases.



This assignment will kinematically connect the inelastic frame with the elastic soil substrate. The final loads and boundary conditions are depicted below :



11. Set mesh size

Set a 4x4 mesh for all surfaces : Mesh → Structured → Lines → Assign Number of Cells → 4 to all surface boundary lines.

Then generate mesh from 

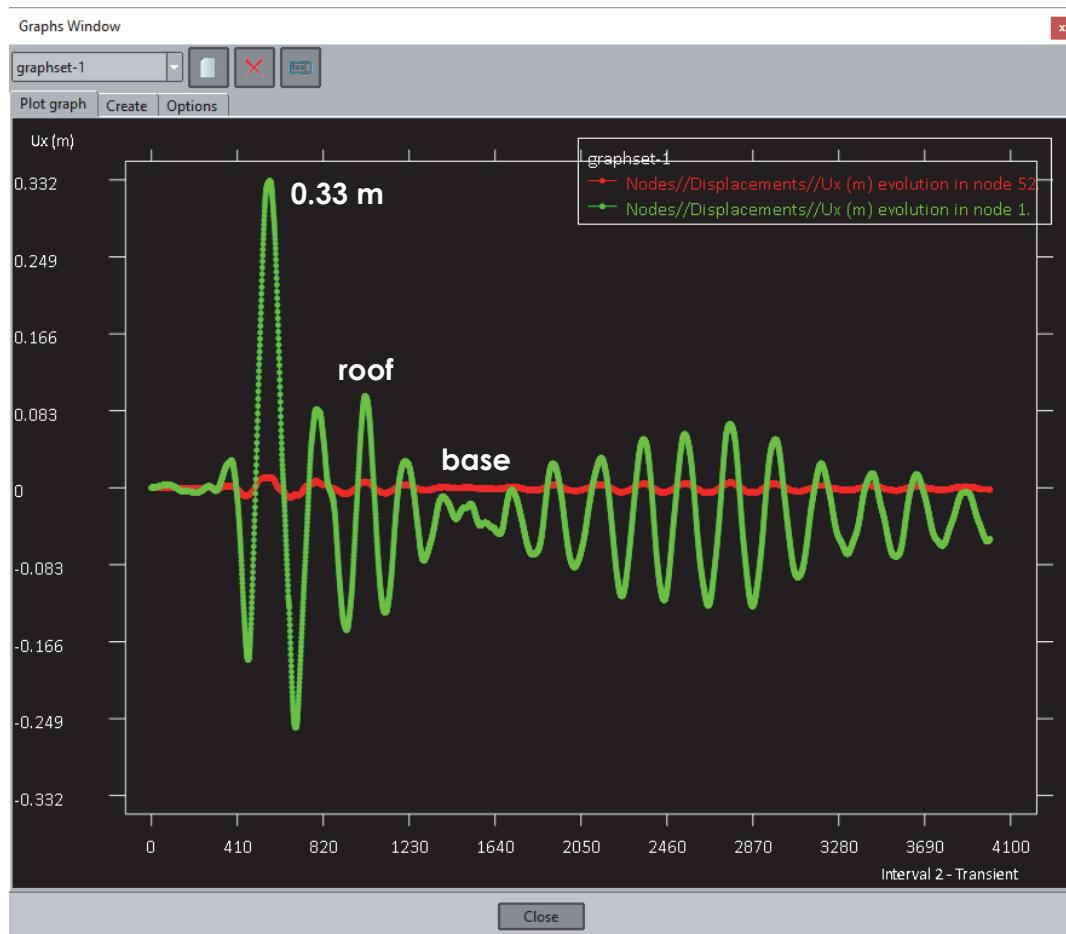
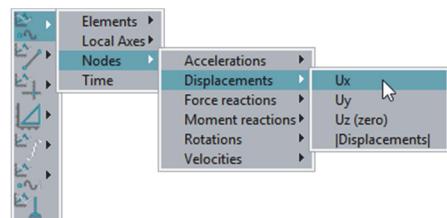
Nodes: 382, Elements: 348

B. Analysis

Click  and proceed to postprocessing when prompted.

C. Postprocessing

For this SSI analysis, it is interesting to show the horizontal displacement response plot on both the base and roof nodes of the frame structure. It is observed that the response is significantly different than the previous example (without damping).



No SSI, no damping →

