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

Barcelona, June 2018

Interface Installation and Walkthrough

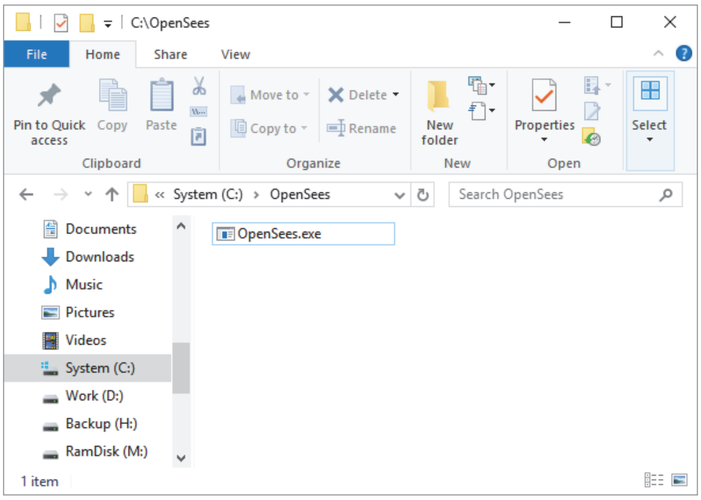
This section covers the essentials for installing GiD+OpenSees interface and displays its basic features. The installation steps are the following :

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

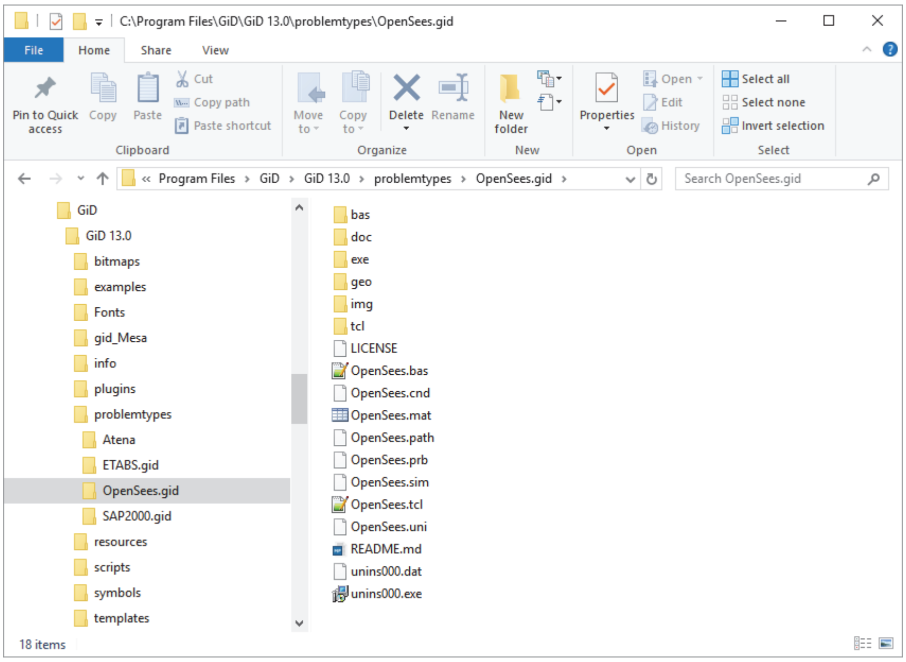


3) Download GiD+OpenSees interface from the official site by selecting the Download option (redirects to GitHub). Run the installer and select the previously defined OpenSees path when prompted.

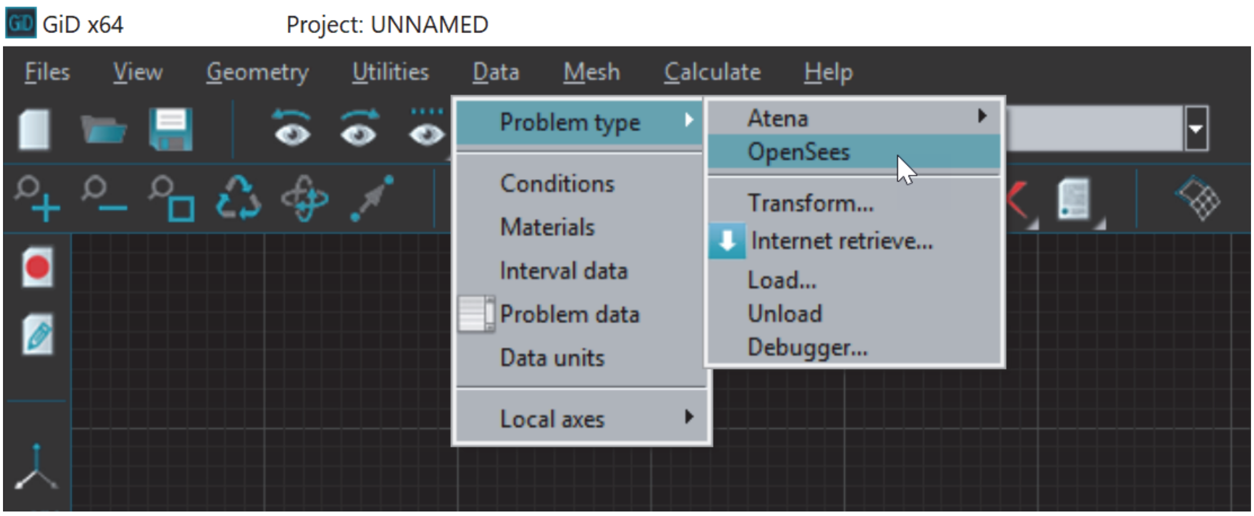
http://gidopensees.rclab.civil.auth.gr

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 :

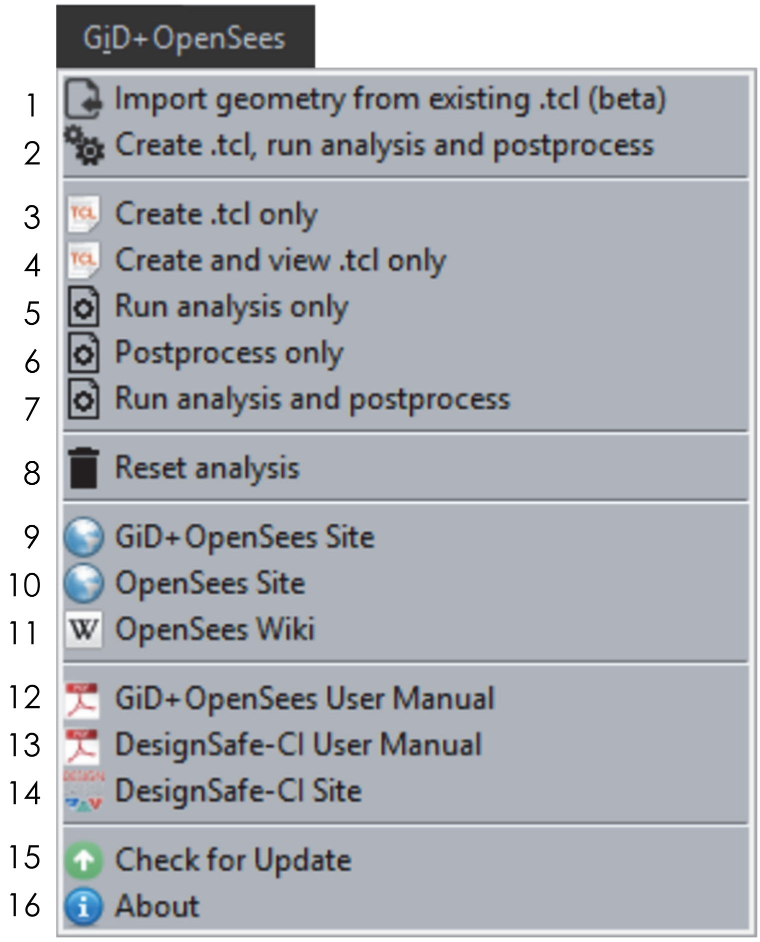
Toolbar commands

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

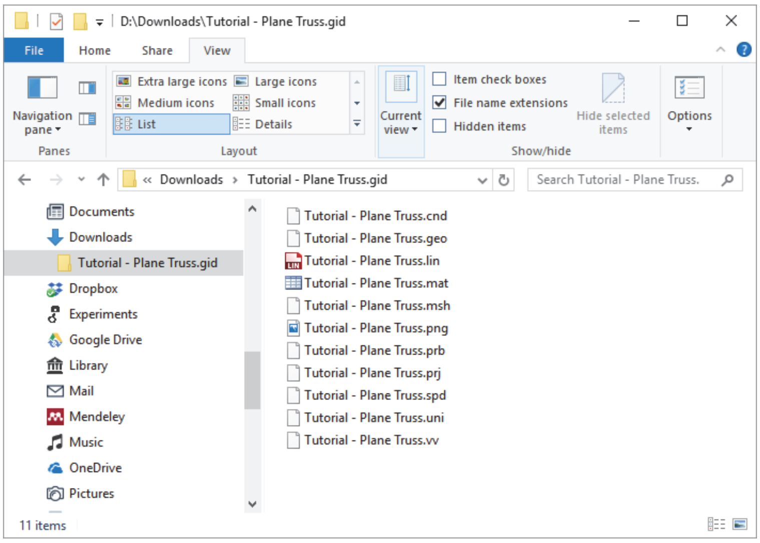
Menu commands



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



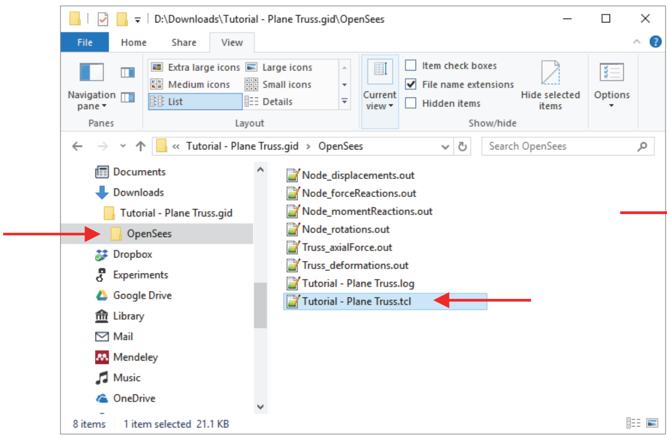
2. After pressing the analysis button :

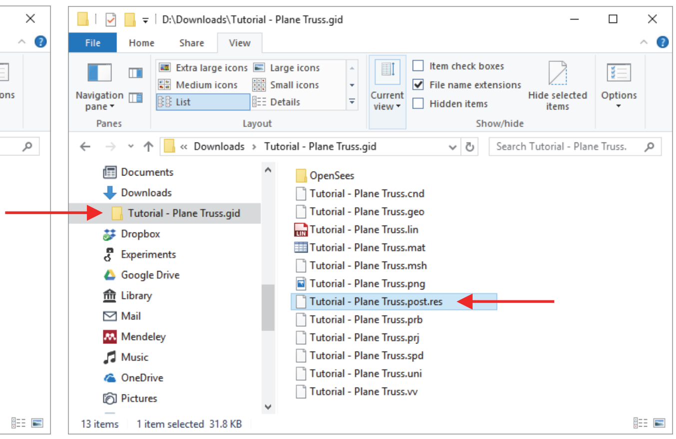
a) *<model name>.tcl* is created in *<path>**<model name>.gid**OpenSees*

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.

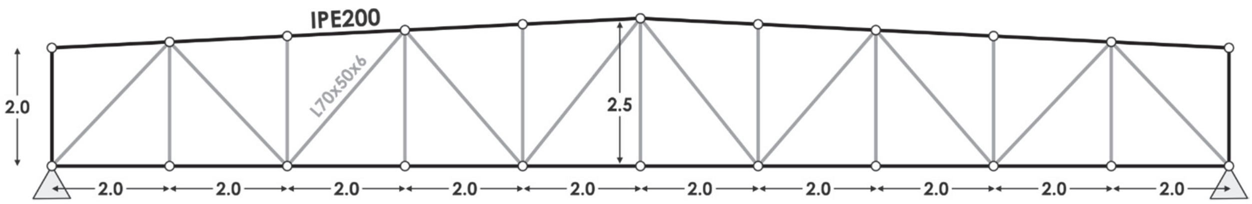
d) Resetting analysis option deletes *<path>**<model name>.gid**OpenSees* and <model name>.post.res





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 GPa

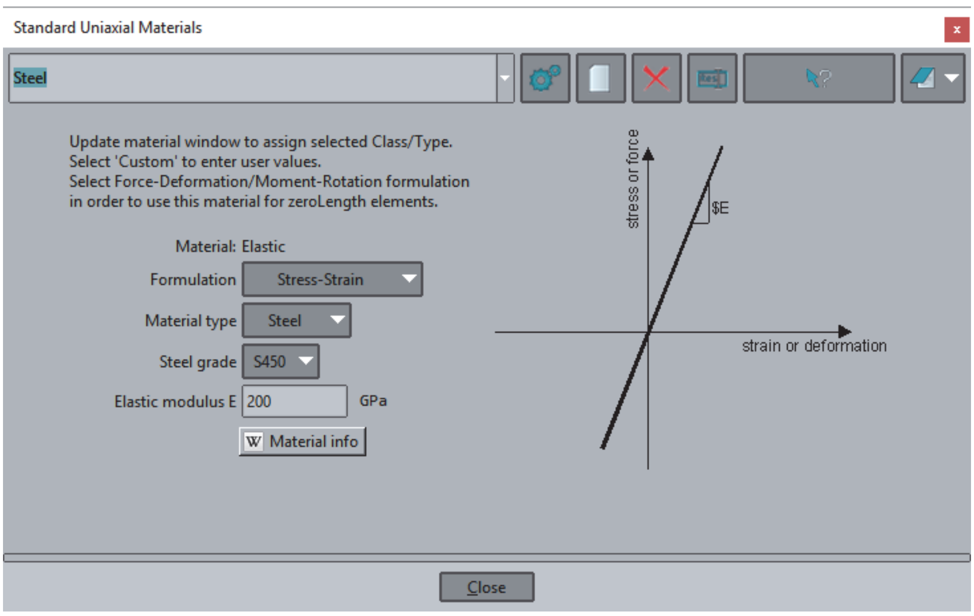
Section properties — IPE200 : A = 28.5 cm^2

L70x50x6 : A = 6.89 cm^2

A. Preprocessing

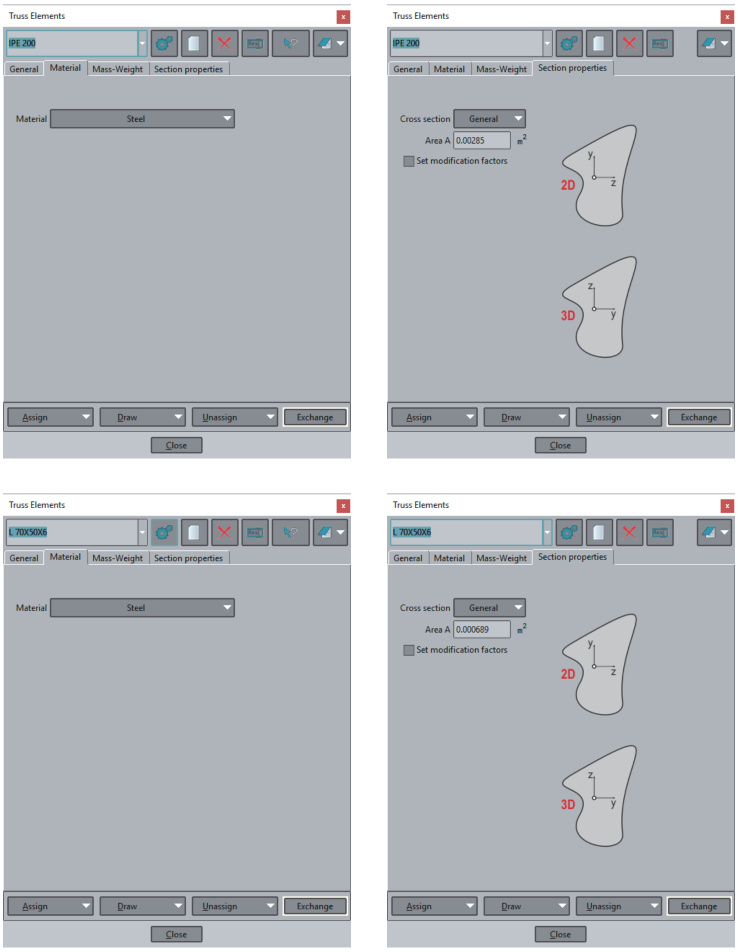
1. Define steel material

→

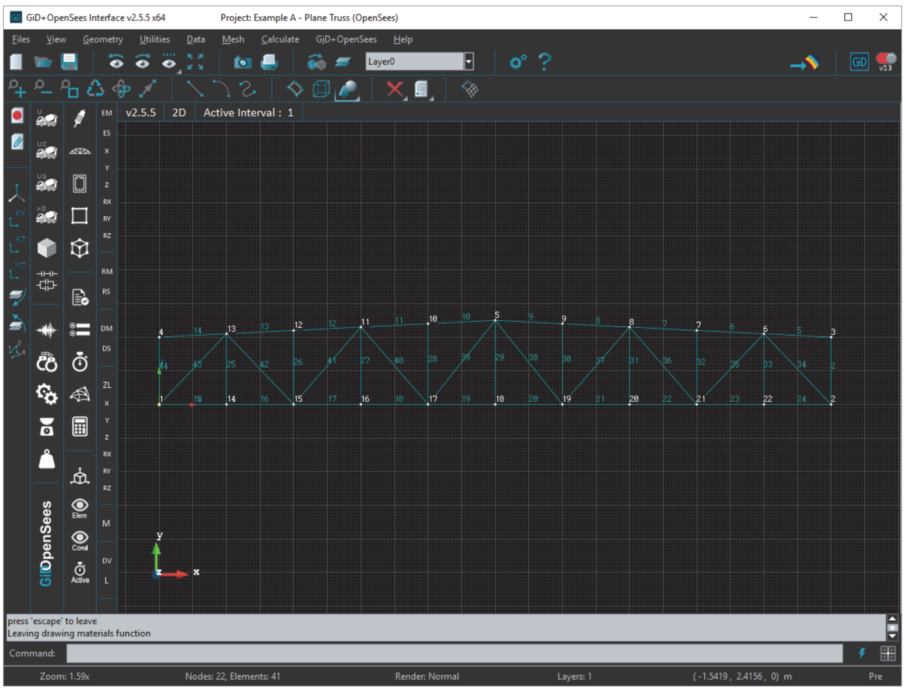


2. Define truss elements

→

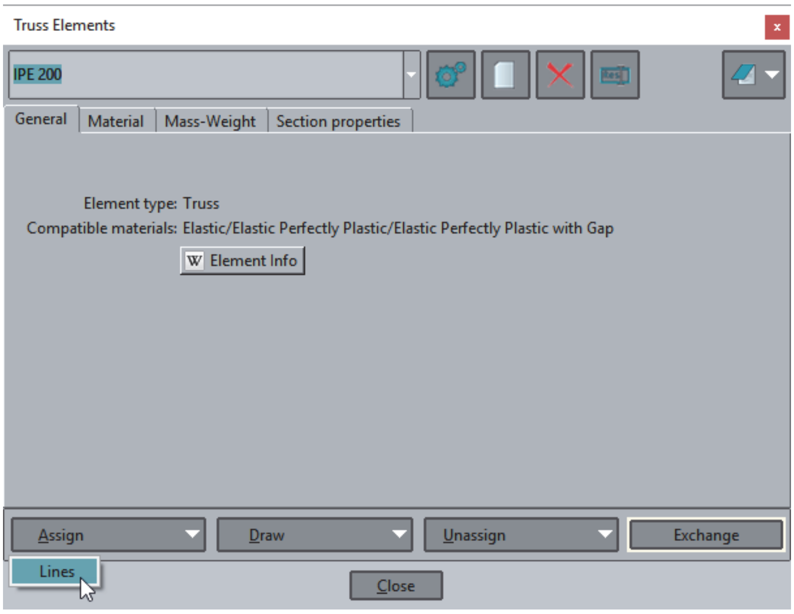


3. Draw geometry in GiD

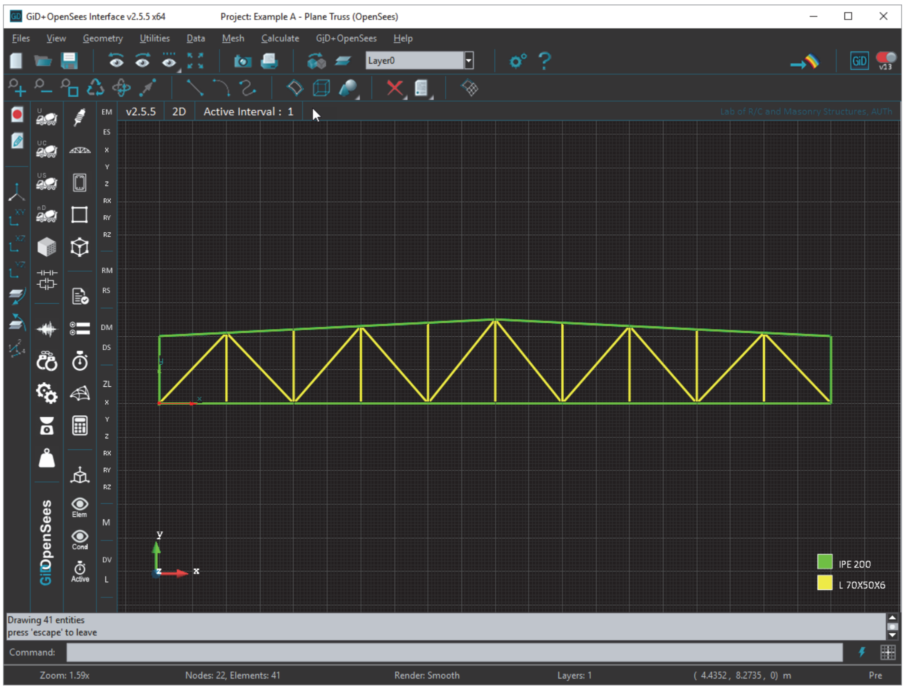


4. Assign truss elements to geometry lines

→

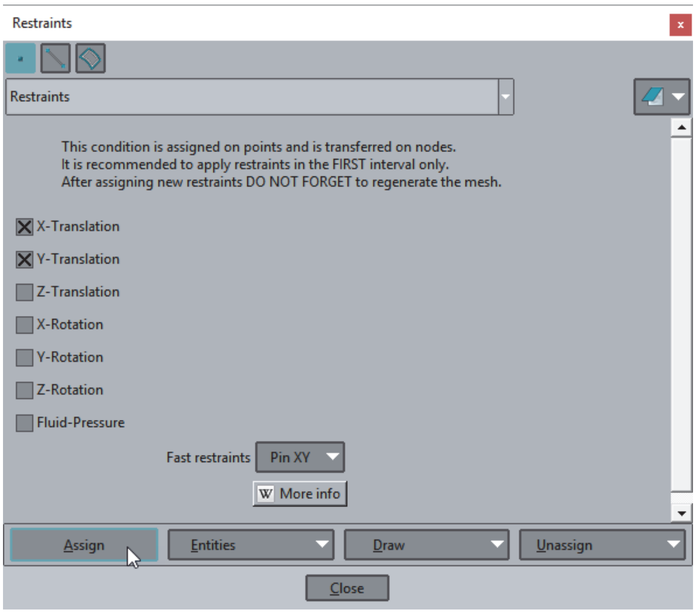


→

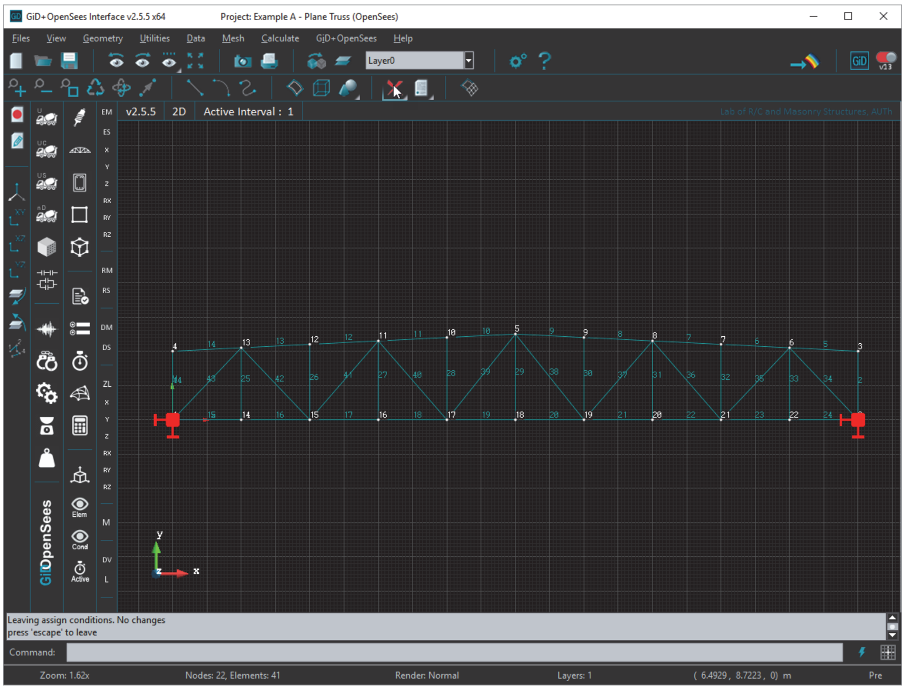


5. Assign restraints (pin XY) to truss support nodes

→



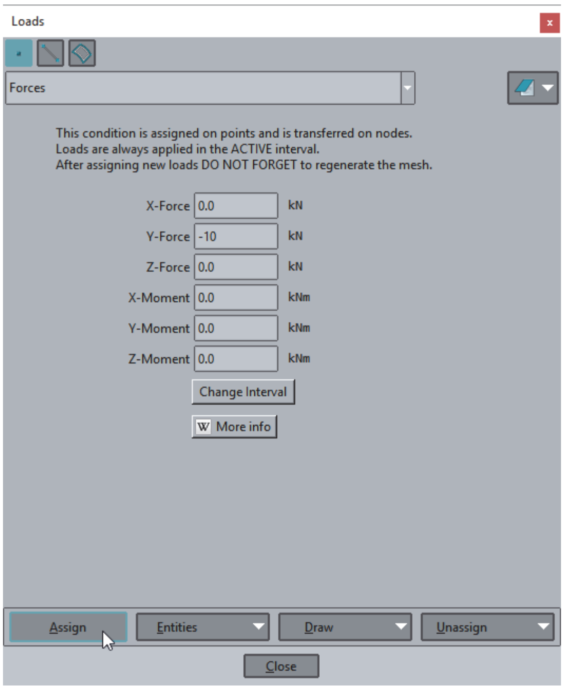
→



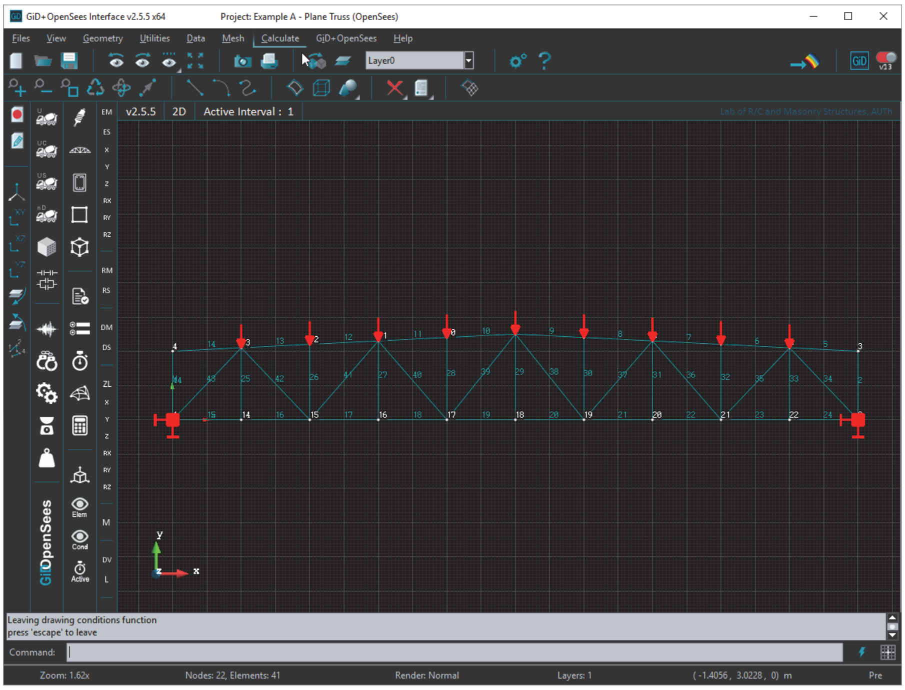
6. Assign loads

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

→

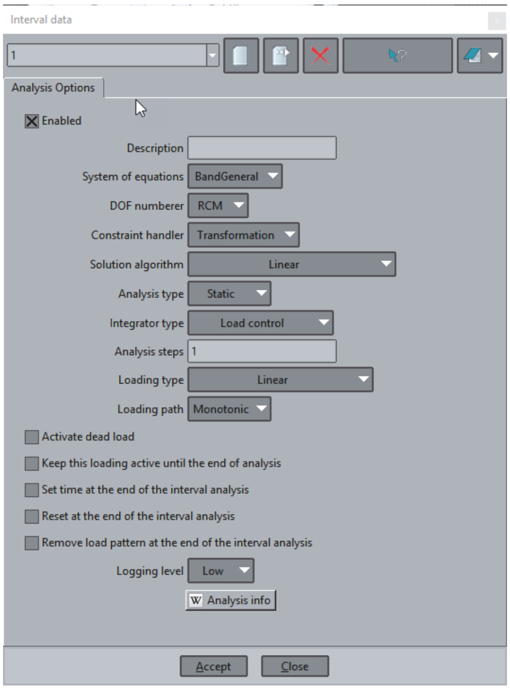


→



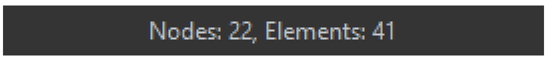
7. Set analysis options

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

 → 

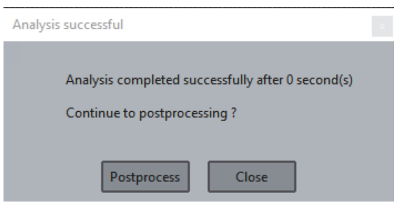
8. Set mesh size

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

Then generate mesh from  

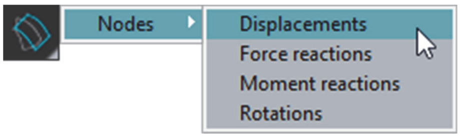
B. Analysis

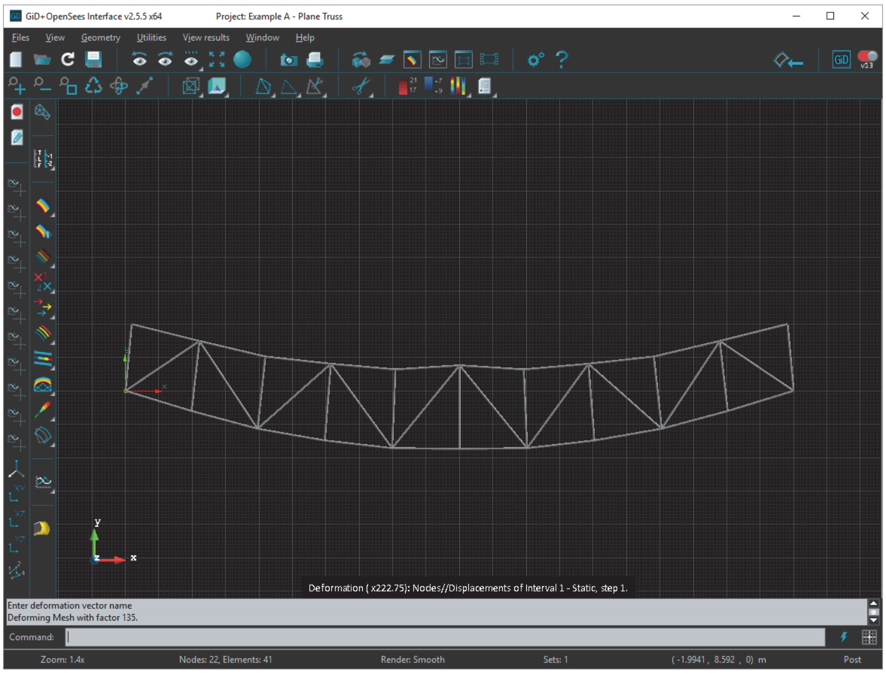
Click  and proceed to postprocessing when prompted.

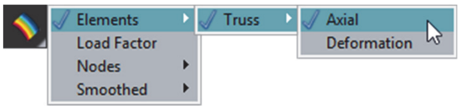


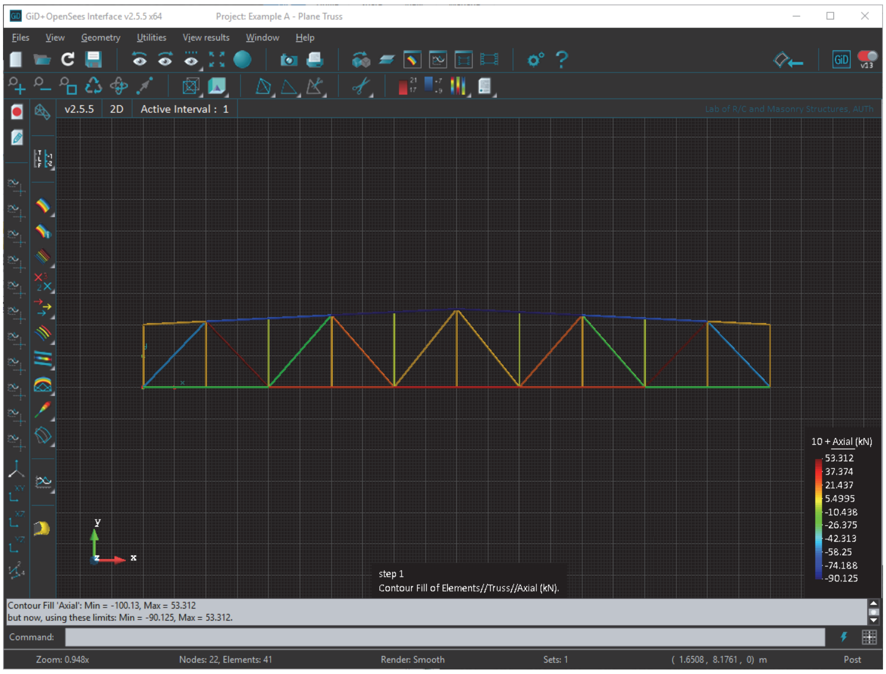
C. Postprocessing

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



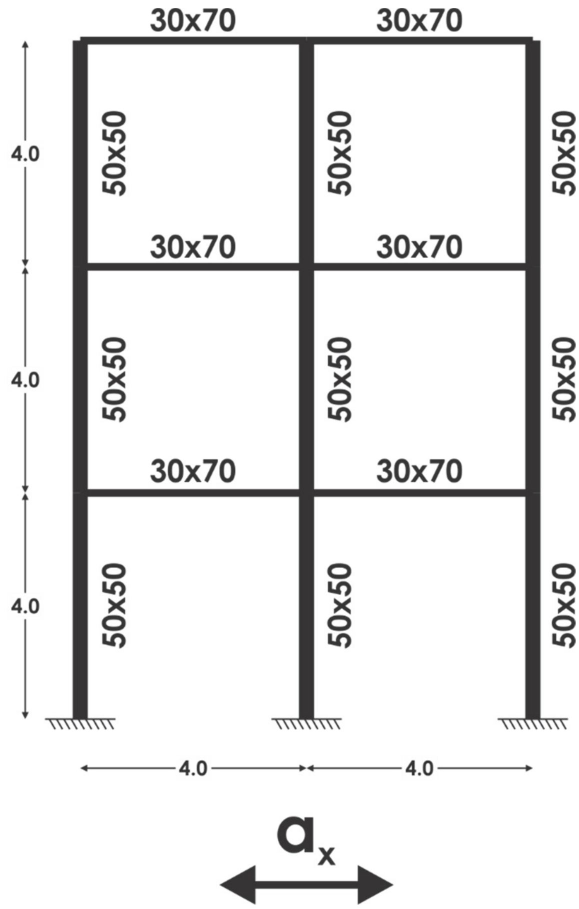






Example B1 – Inelastic Plane Frame

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

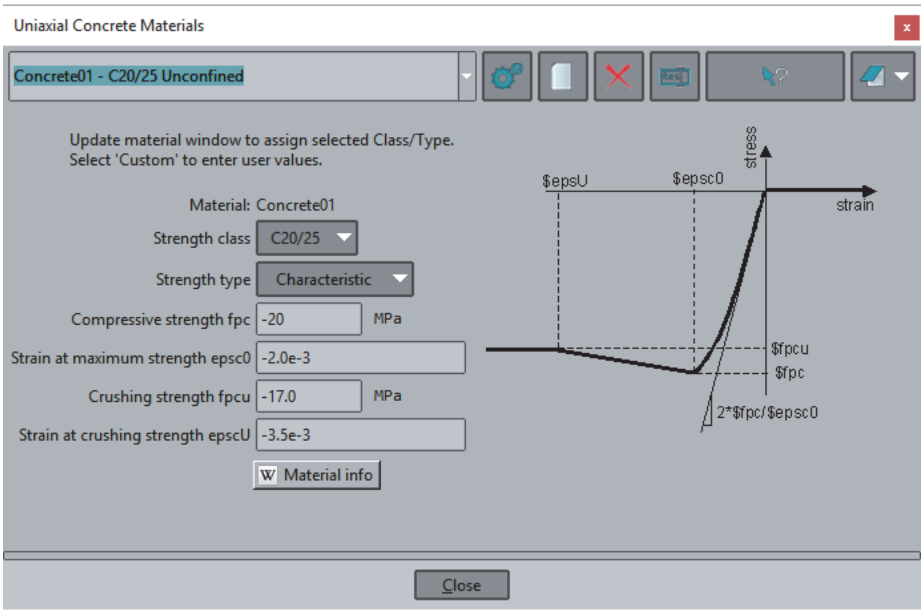
Section properties Columns 50x50, Beams 30x70

A. Preprocessing

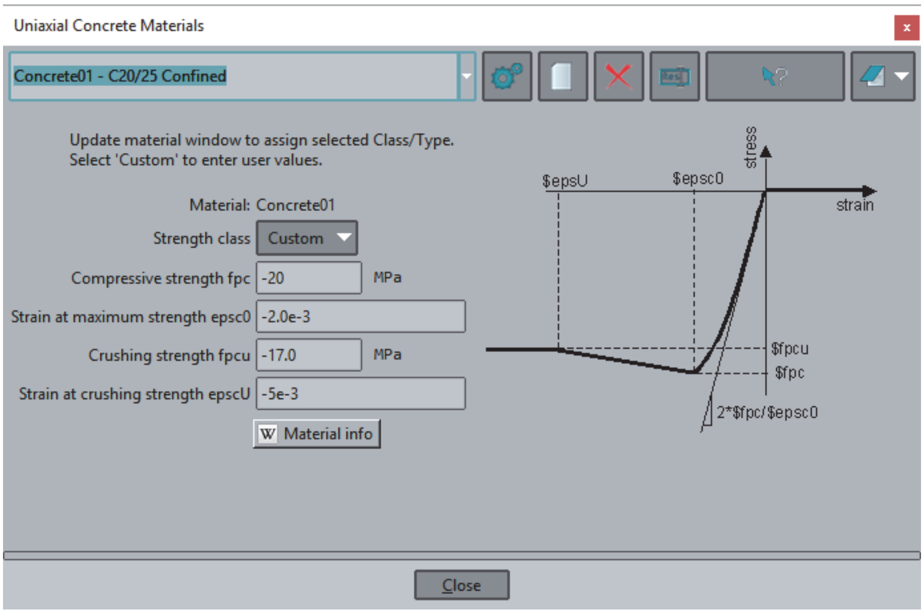
1. Define concrete material as Concrete01
   - Unconfined : C20/25 default values
   - Confined : C20/25 default values with -5e-3 ultimate crushing strain

   Define steel material as Reinforcing steel with B500C default values

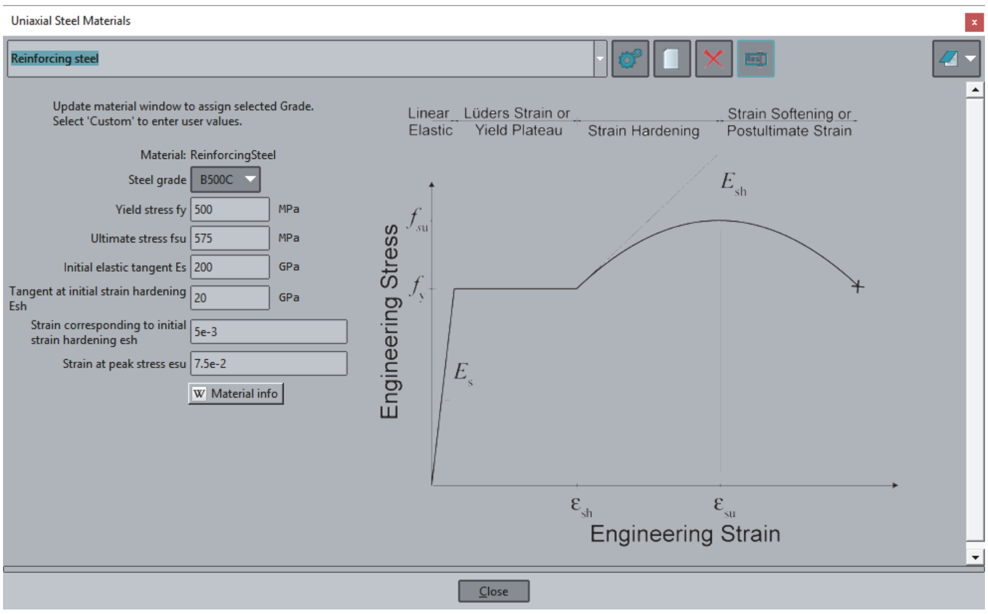

UC
→


Uniaxial Concrete Materials
Concrete01 - C20/25 Unconfined
Update material window to assign selected Class/Type.
Select 'Custom' to enter user values.
Material: Concrete01
Strength class C20/25
Strength type Characteristic
Compressive strength fpc -20 MPa
Strain at maximum strength epsc0 -2.0e-3
Crushing strength fpcu -17.0 MPa
Strain at crushing strength epscU -3.5e-3
Material info
stress
$epsU
$epsc0
strain
$fpcu
$fpc
2*$fpc/$epsc0
Close


UC
→


Uniaxial Concrete Materials
Concrete01 - C20/25 Confined
Update material window to assign selected Class/Type.
Select 'Custom' to enter user values.
Material: Concrete01
Strength class Custom
Compressive strength fpc -20 MPa
Strain at maximum strength epsc0 -2.0e-3
Crushing strength fpcu -17.0 MPa
Strain at crushing strength epscU -5e-3
Material info
stress
$epsU
$epsc0
strain
$fpcu
$fpc
2*$fpc/$epsc0
Close

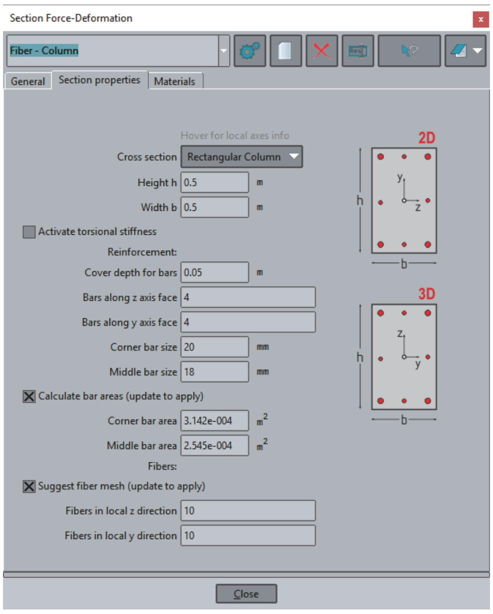

US
→

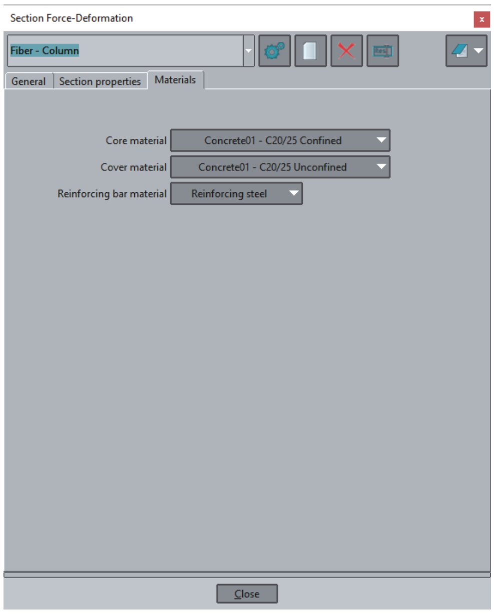

Uniaxial Steel Materials
Reinforcing steel
Update material window to assign selected Grade.
Select 'Custom' to enter user values.
Material: ReinforcingSteel
Steel grade B500C
Yield stress fy 500 MPa
Ultimate stress fsu 575 MPa
Initial elastic tangent Es 200 GPa
Tangent at initial strain hardening Esh 20 GPa
Strain corresponding to initial strain hardening esh 5e-3
Strain at peak stress esu 7.5e-2
Material info
Linear Elastic
Lüders Strain or Yield Plateau
Strain Hardening
Strain Softening or Postultimate Strain
Engineering Stress
Engineering Strain
Close

2. Define fiber sections for beams and columns

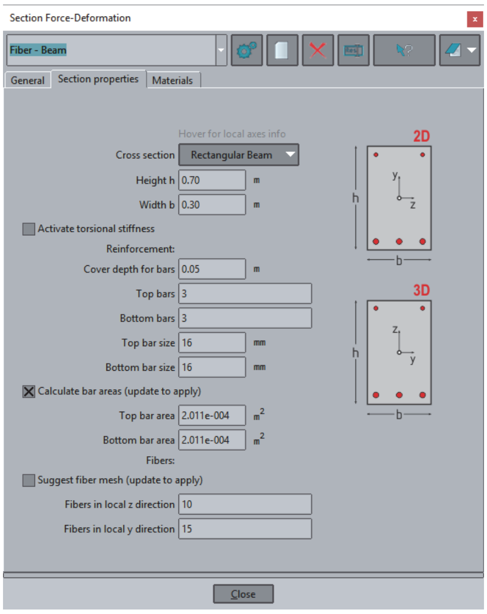
- Columns 4Ø20+4Ø18
- Beams 3Ø16 top & bottom

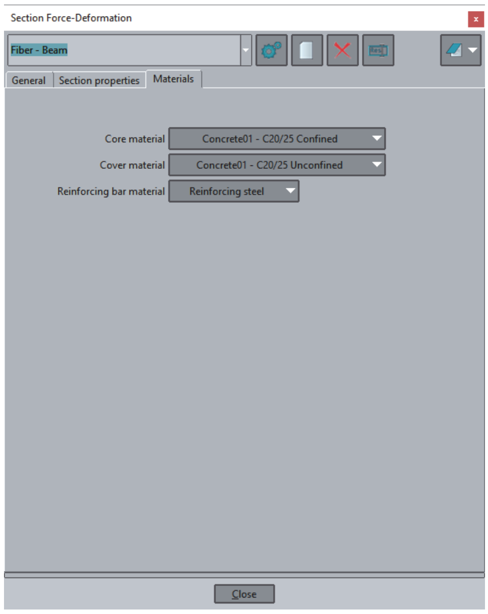

→





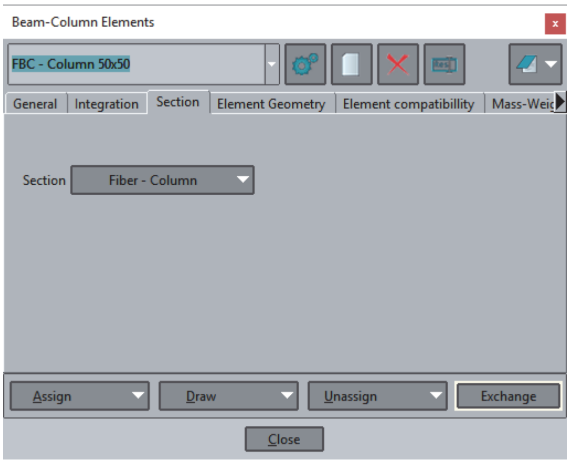

→



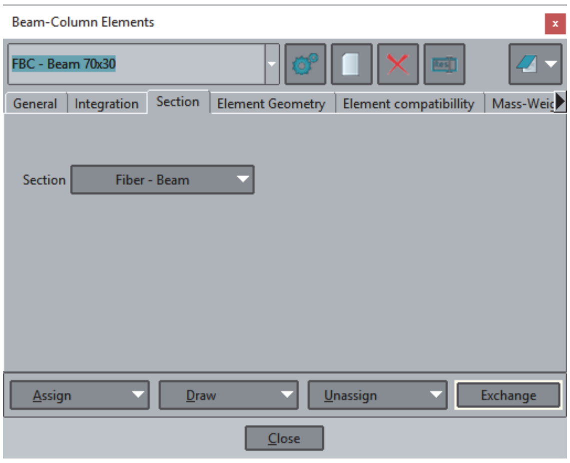


3. Define force-based beam-column elements for beams and columns

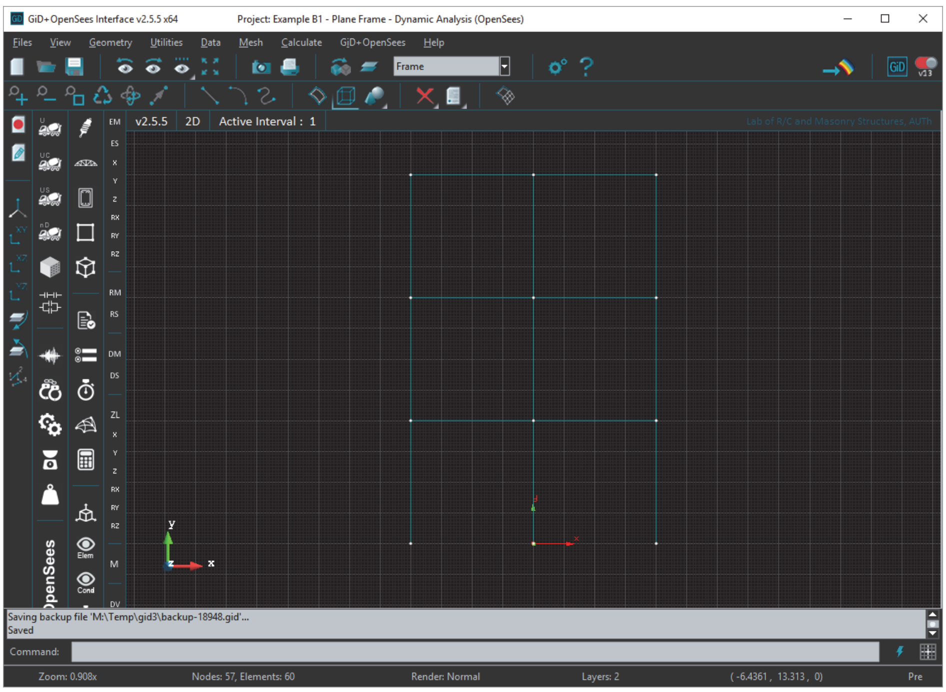
→



→

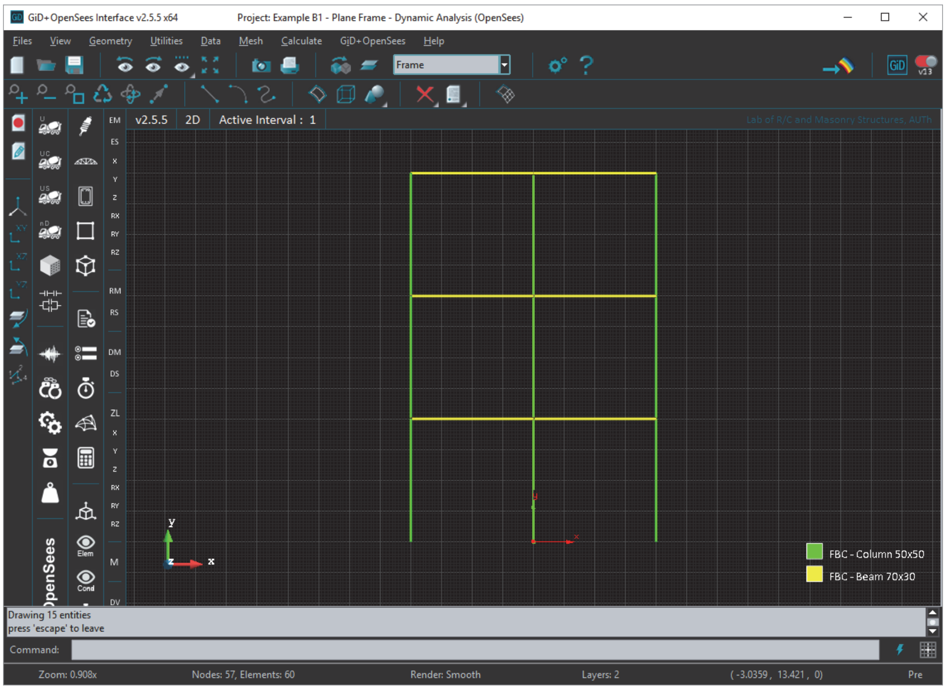


4. Draw geometry in GiD



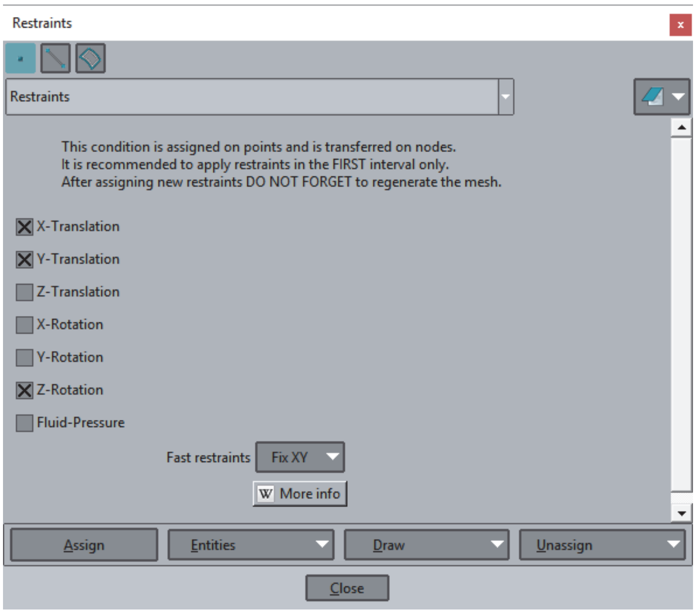
5. Assign beam and column force-deformation elements to geometry lines

→

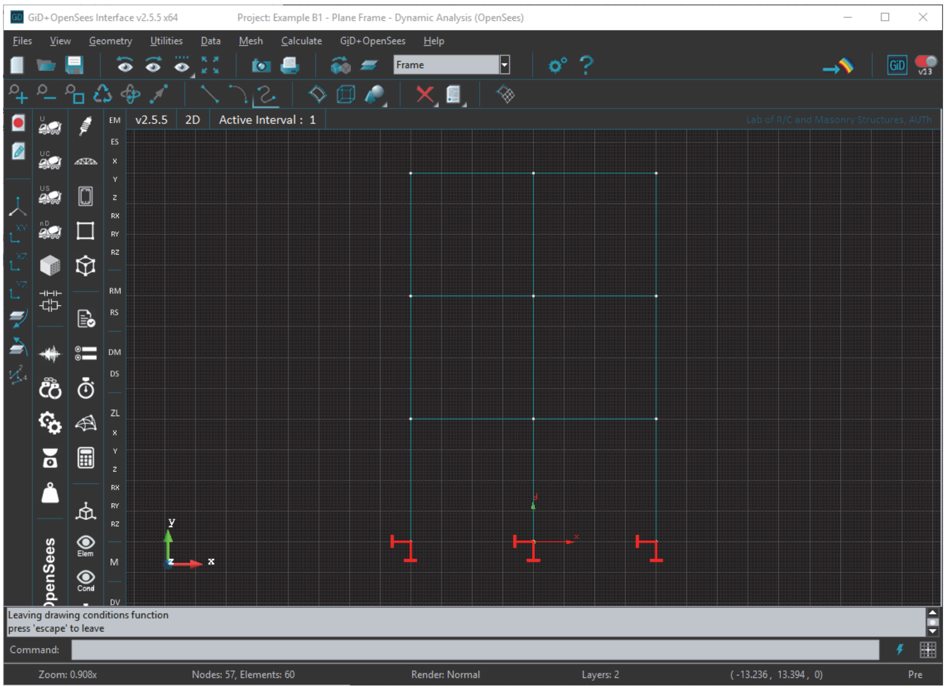


6. Assign restraints (fix XY) to truss support nodes (Active Interval : 1)

→



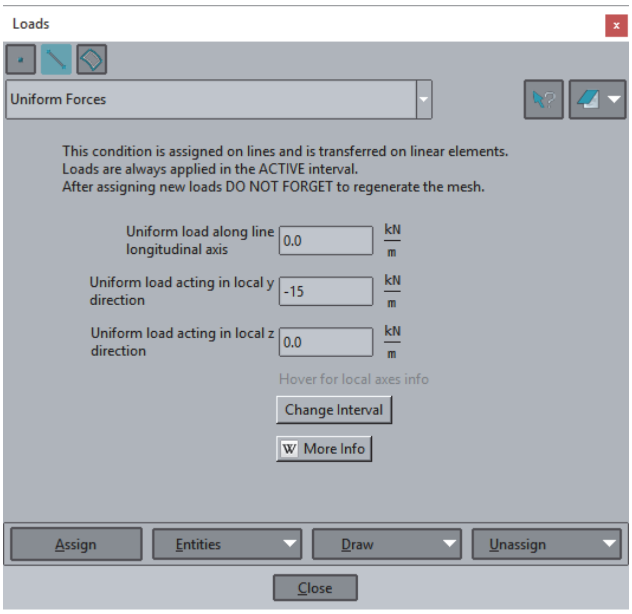
→

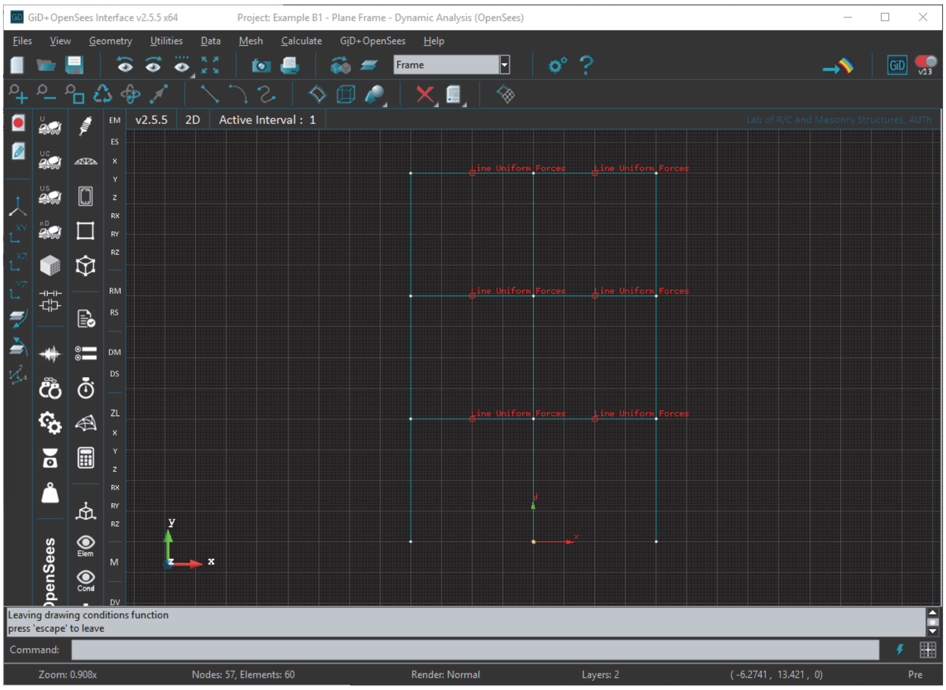


7. Assign static loads

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

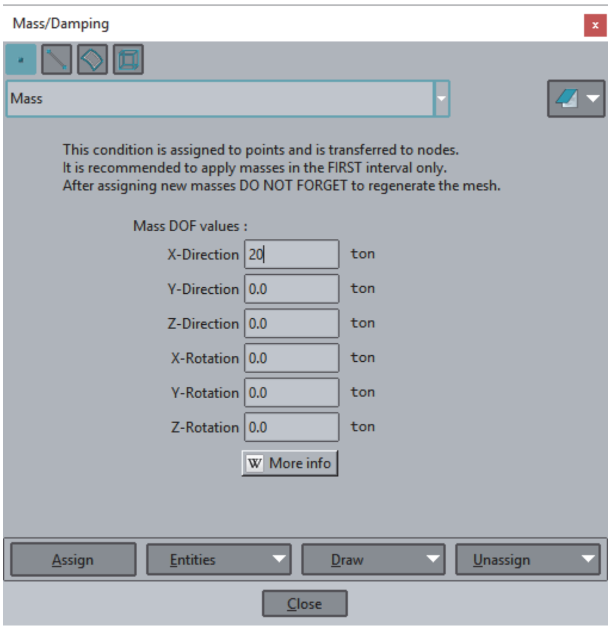
→



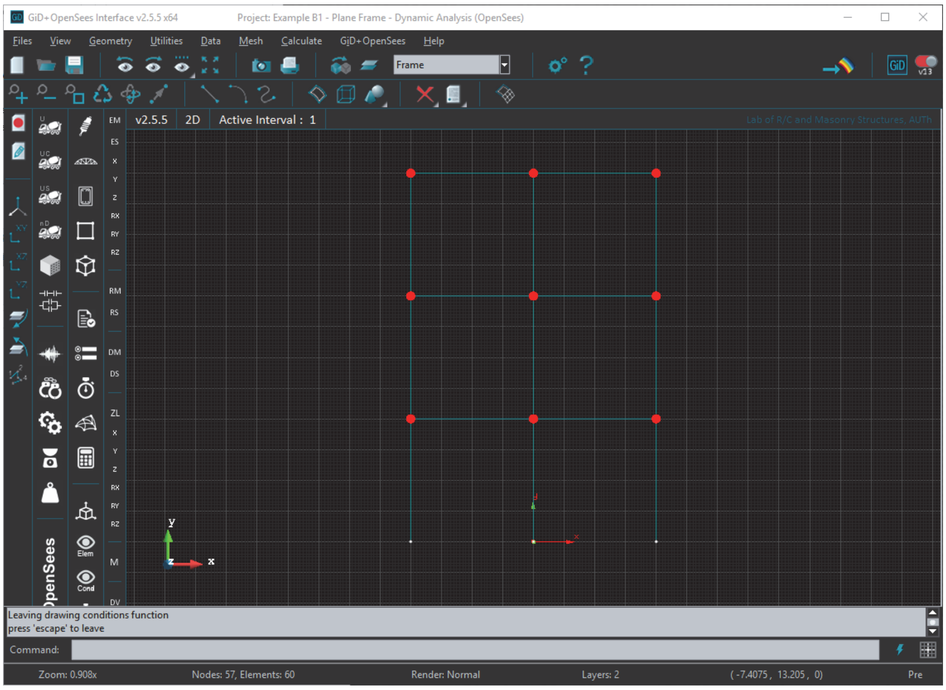
→


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

Type : Acceleration

Format : Time and value per line

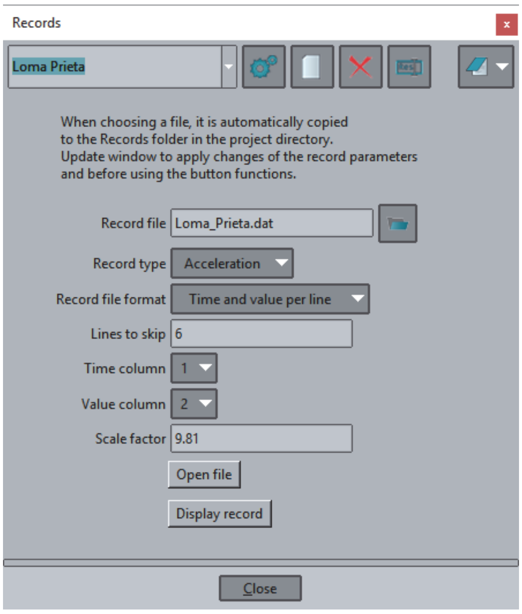
Lines to skip : 6

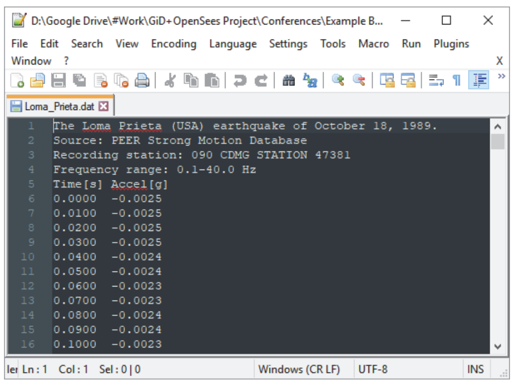
Time column : 1

Value column : 2

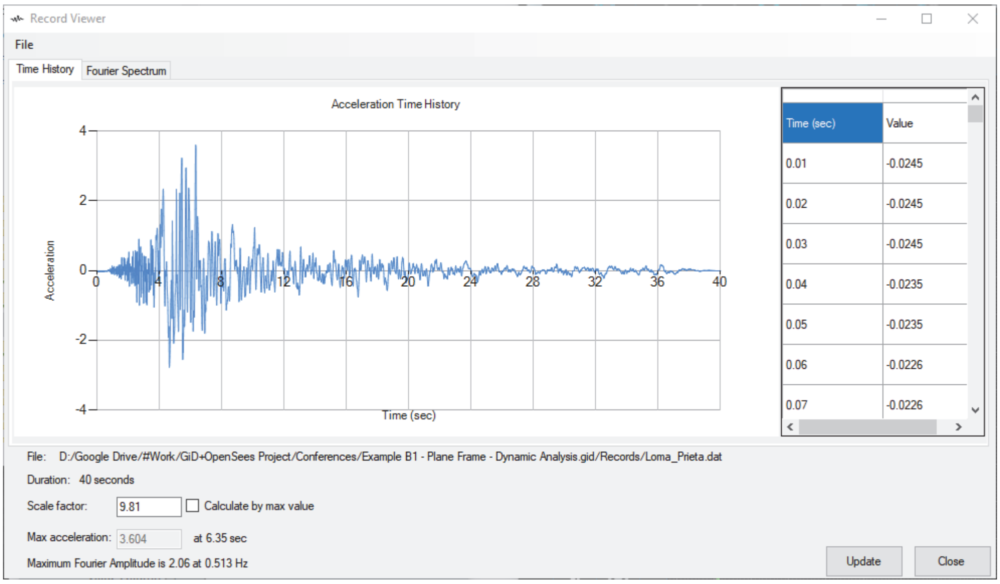
Scale factor : 9.81 (convert %g to m/s²)

→





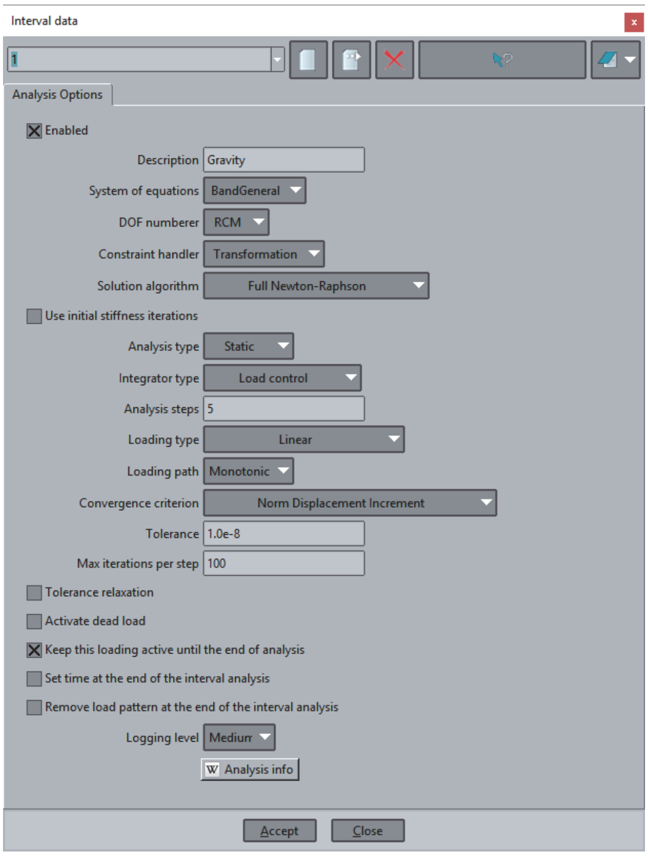
The selected strong motion record can be viewed by pressing 'Display record'



10. Set analysis options

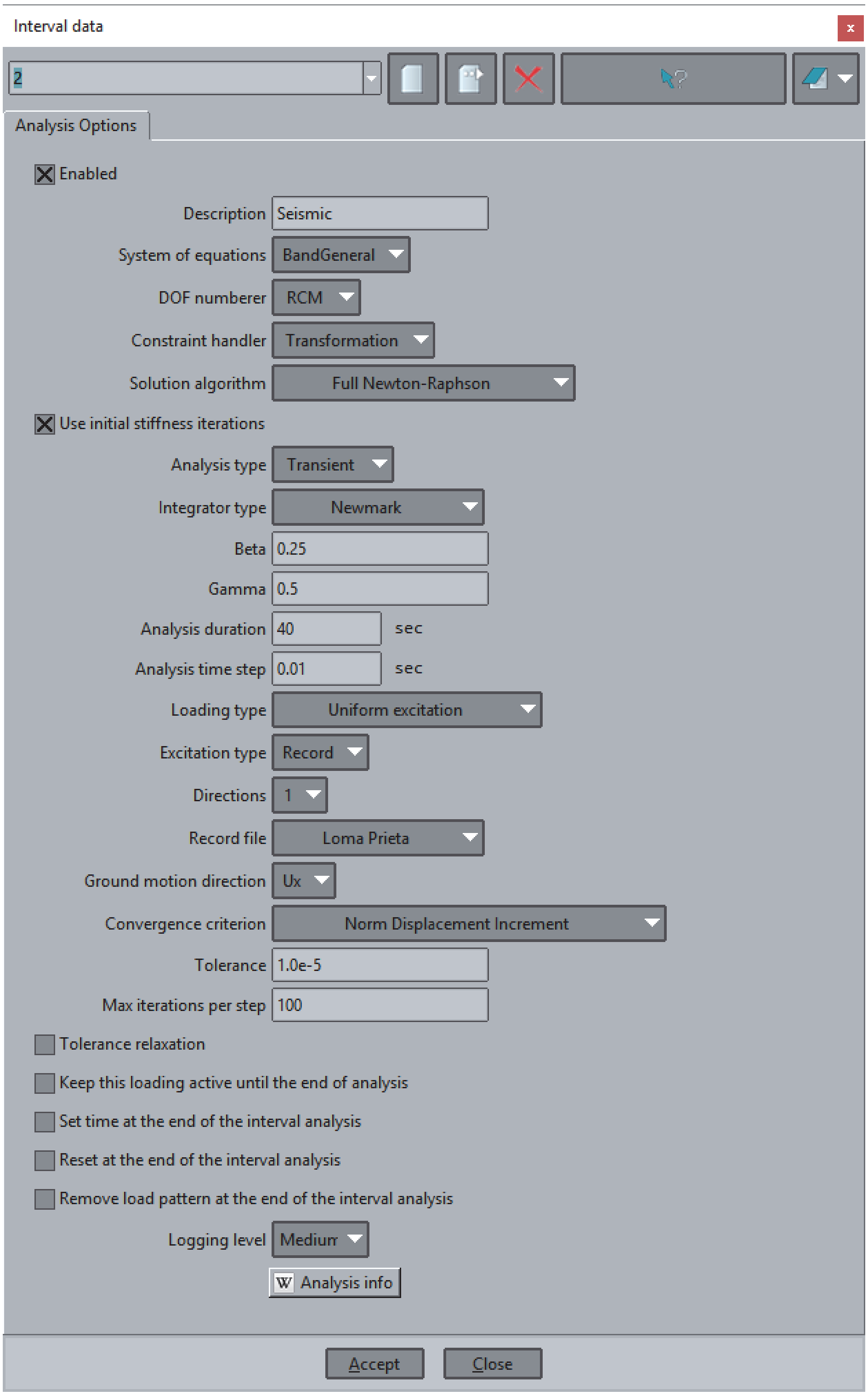
The first interval contains boundary conditions, masses and gravity loading. It is defined as follows (Static analysis, Load Control, 5 steps)

 →



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

 →

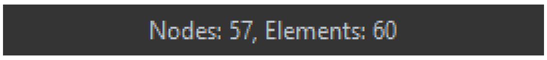


Analysis type is Transient, using the Newmark Integration Scheme (β=0.5, γ=0.25), the analysis duration is 40 seconds with a time step of 0.01 (exactly as specified in the record file) and the excitation type is uniform, based on the already defined record (Loma_Prieta) along the Ux direction (horizontal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

11. Set mesh size

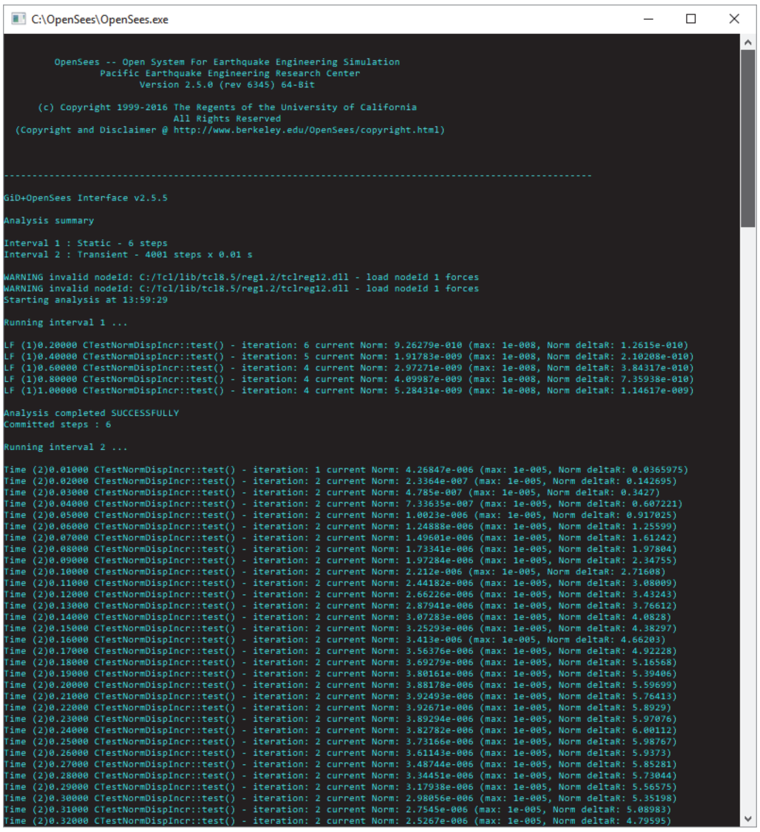
Set four force-based elements per line : Mesh → Structured → Lines → Assign Number of Cells → 4 to all elements.

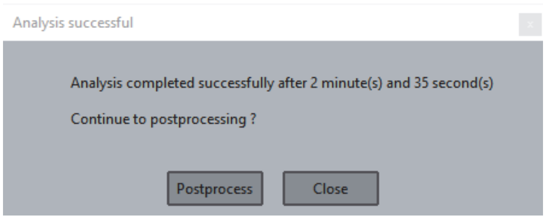
Then generate mesh from 



B. Analysis

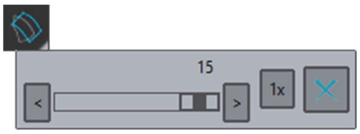
Click  and proceed to postprocessing when prompted.



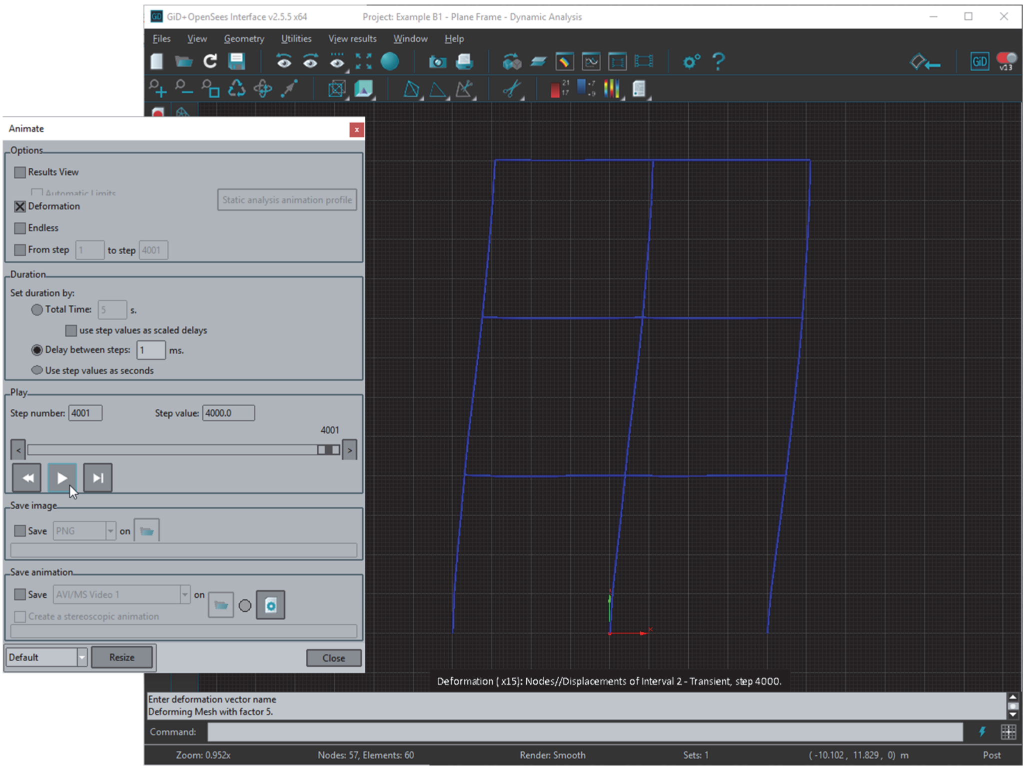


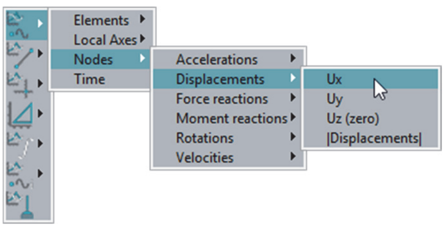
C. Postprocessing

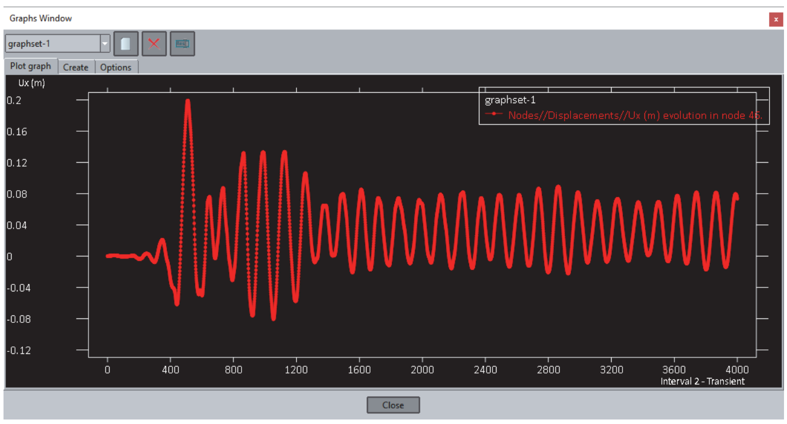
For this dynamic analysis, apart from the usual result types that were discussed in the previous example, it is interesting to show a displacement animation under strong motion and the roof horizontal displacement response graph in time.

Set displacements to 15x 

select Animate and press play



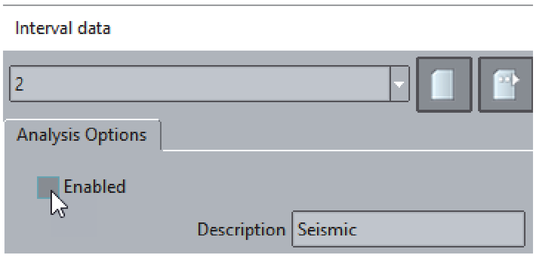




Note : No damping was considered in the analysis

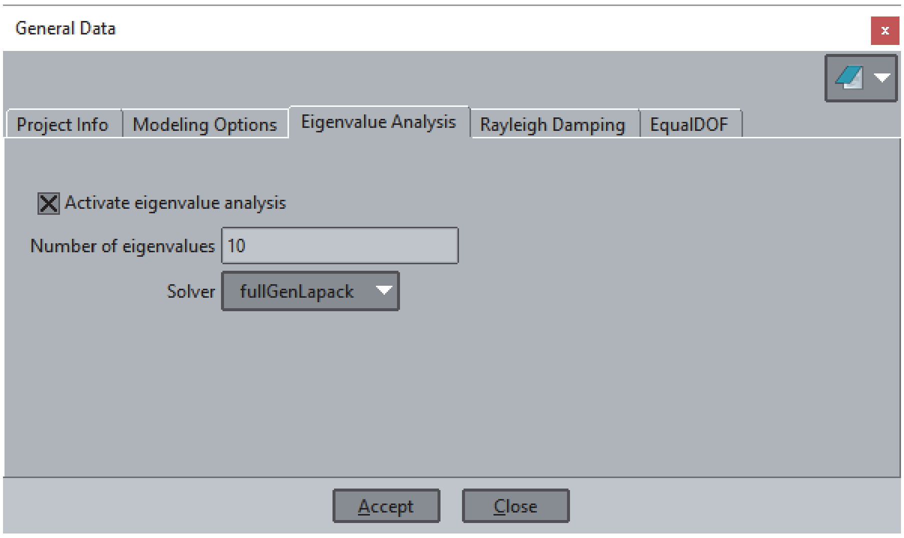
How to include Rayleigh Damping

1. Deactivate Interval 2 (dynamic analysis)

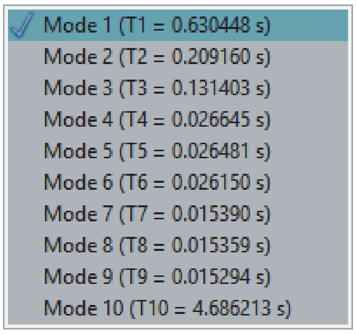


2. Activate eigenvalue analysis

→



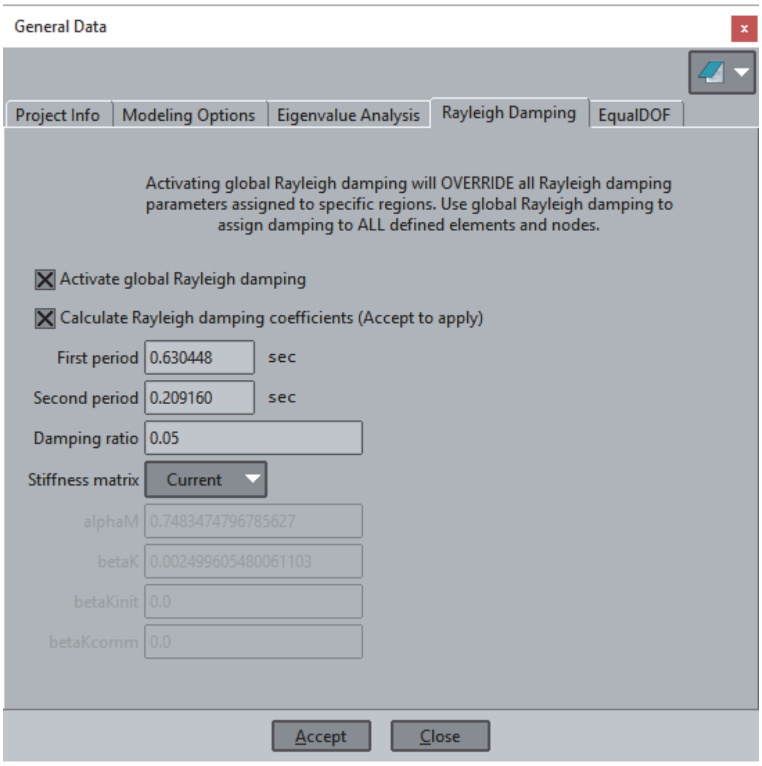
3. Run analysis, postprocess and get period values



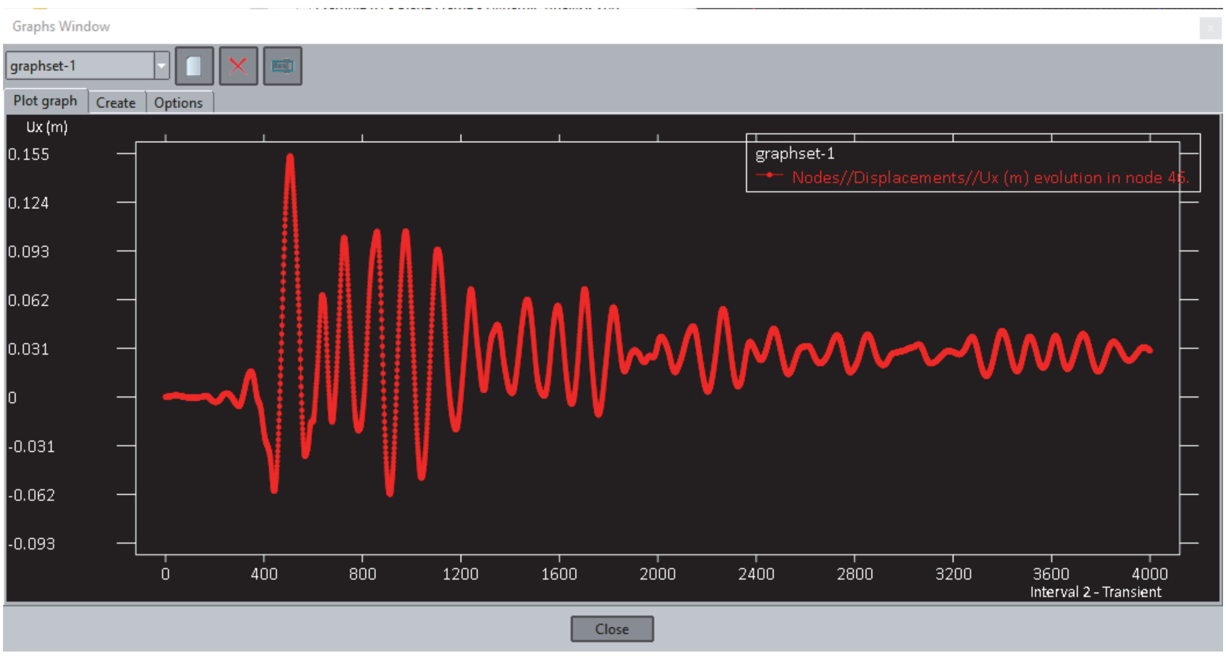
4. Back to preprocessing, activate Interval 2 again.

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

→

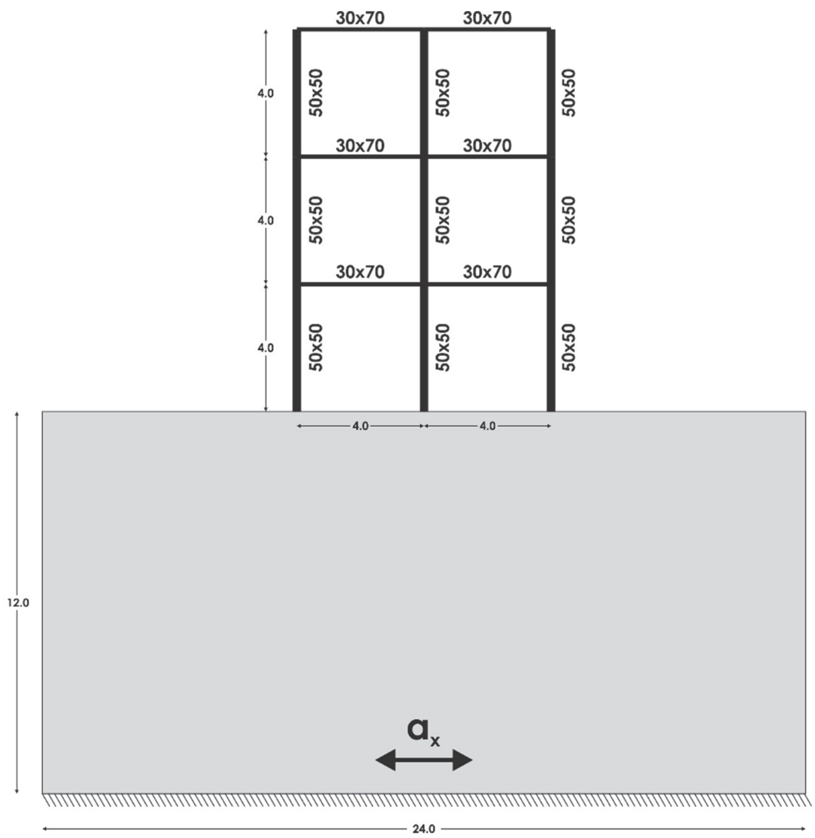


6. Run analysis again and display results. Now the response is clearly damped.



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

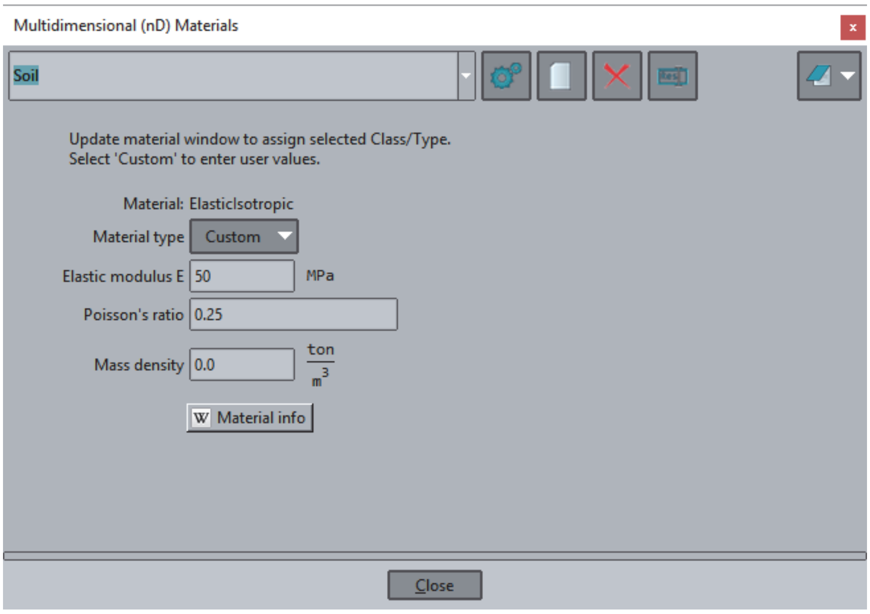
The objective of the last example is to add an elastic soil medium under the previous inelastic frame structure and observe the differences in response due to soil-structure interaction (SSI) effects. The updated geometry is depicted below :



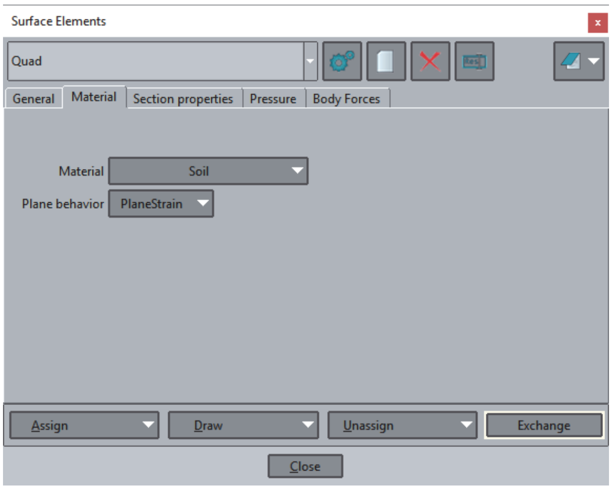
A. Preprocessing

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

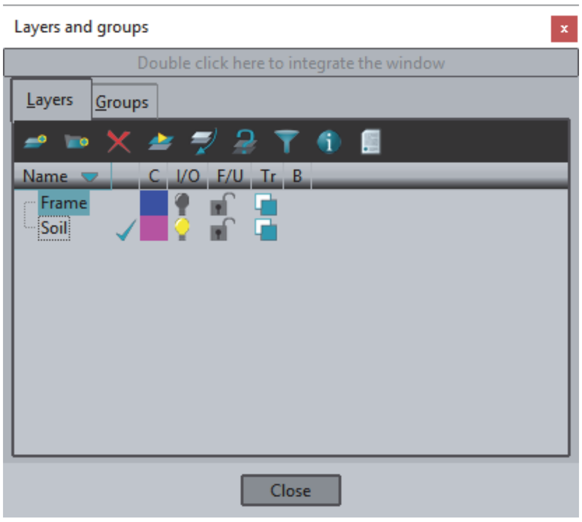
→



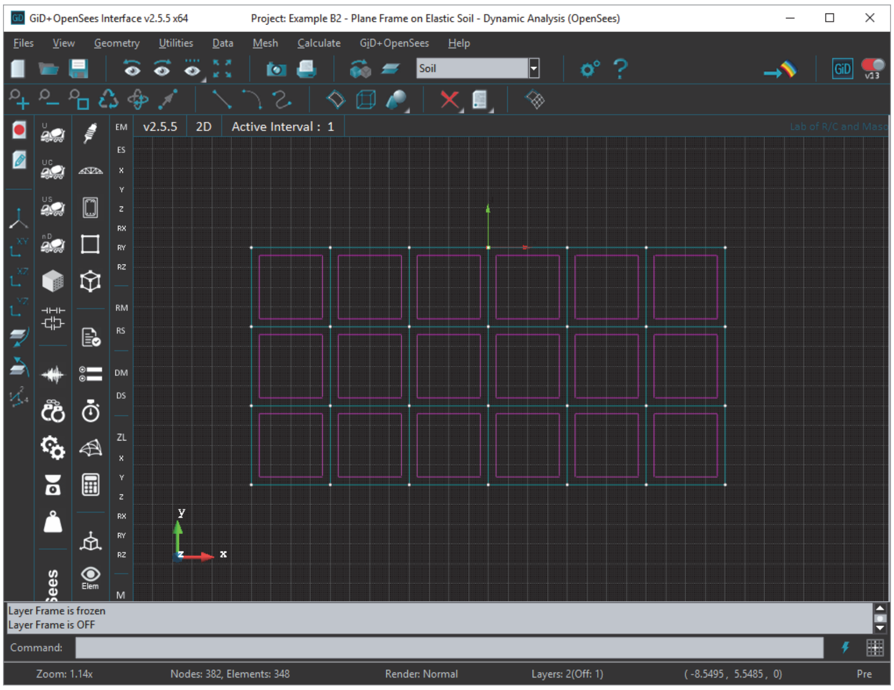
→



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

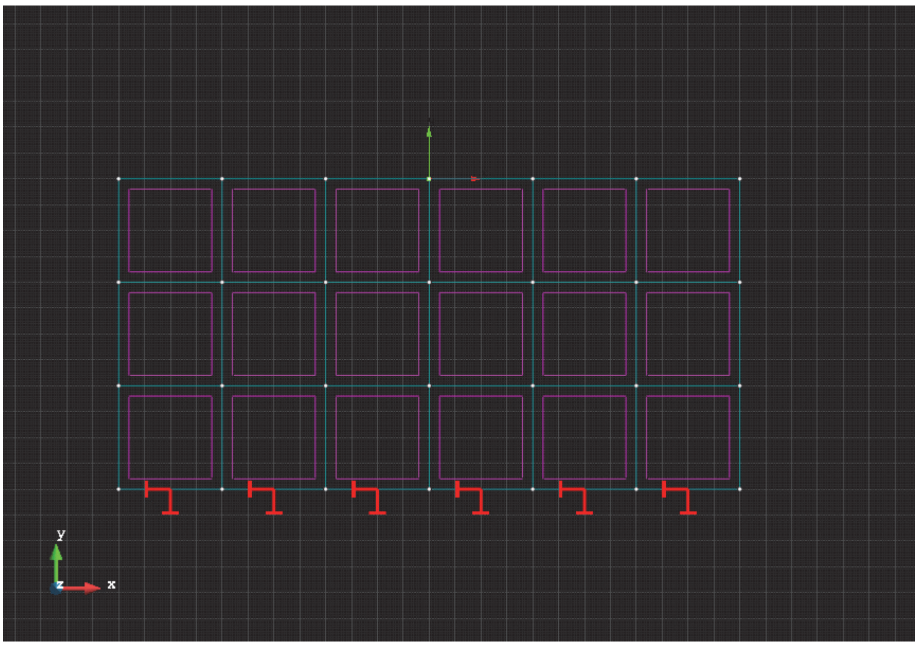


3. Draw soil geometry in GiD. Draw a 6x3 matrix 4x4 m square surfaces, using the ‘Soil’ layer as depicted. The axes origin (0,0) should coincide with the base of the middle frame column.



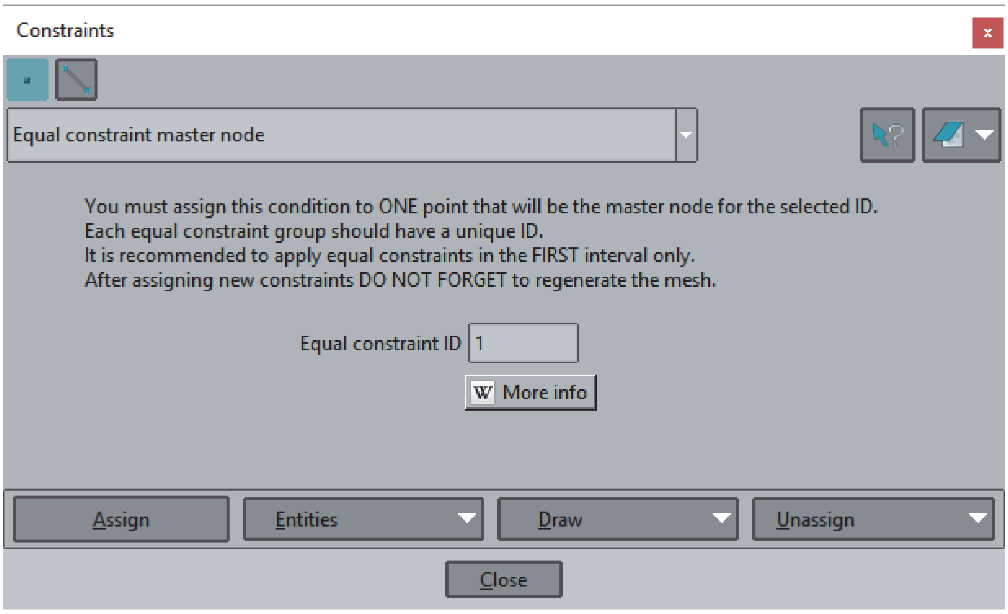
4. Assign the soil quadrilateral element defined earlier to all surfaces.

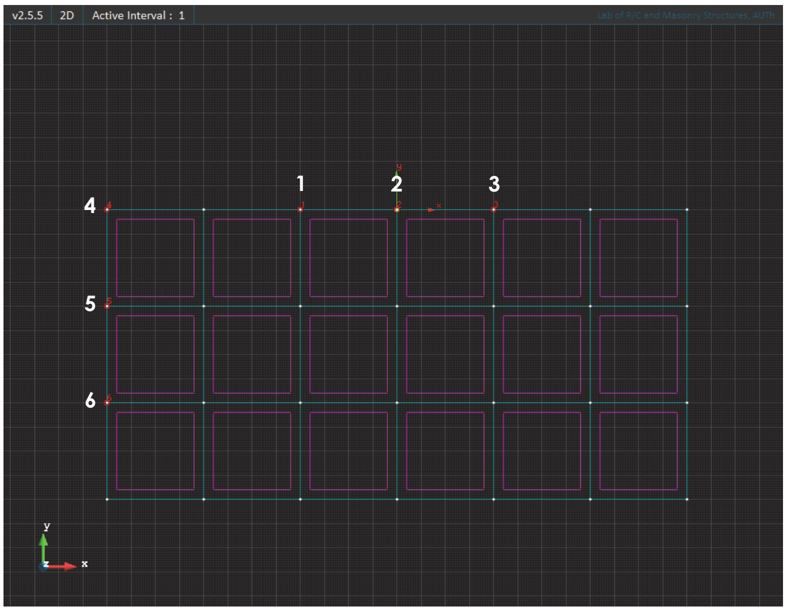
5. Assign a pin-XY restraint to the bottom boundary of the soil medium.



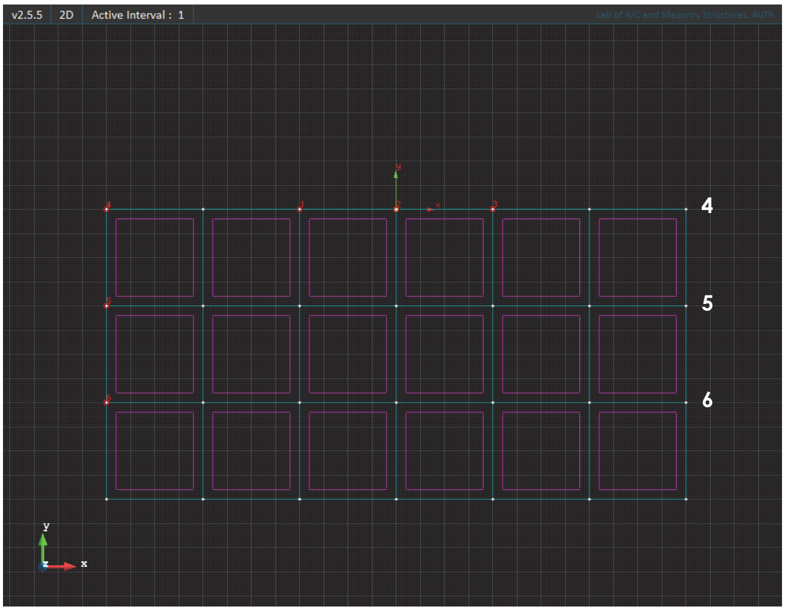
6. Assign equal constraint master nodes with ID = 1,2,3,4,5,6 to the following points :

 →



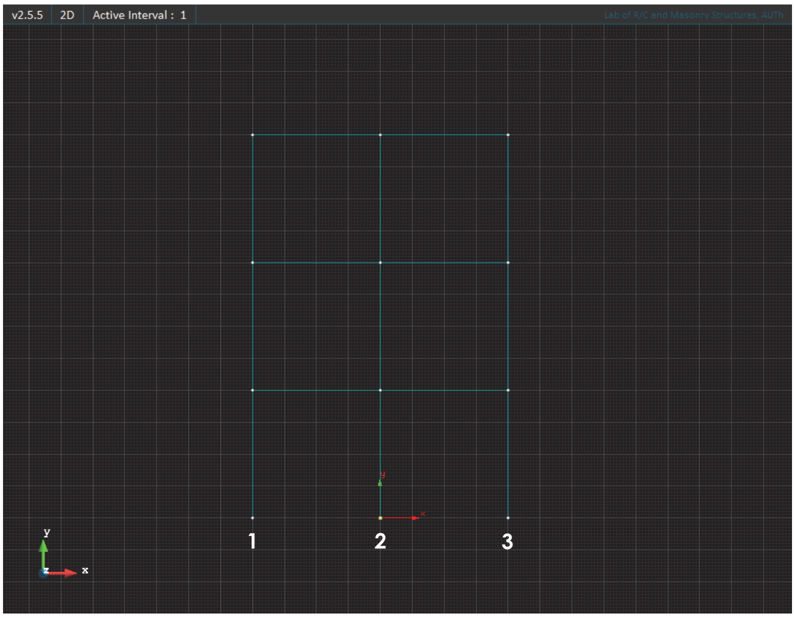


Then assign equal constraint slave nodes with ID = 4,5,6 with X-Translation only to the following points :



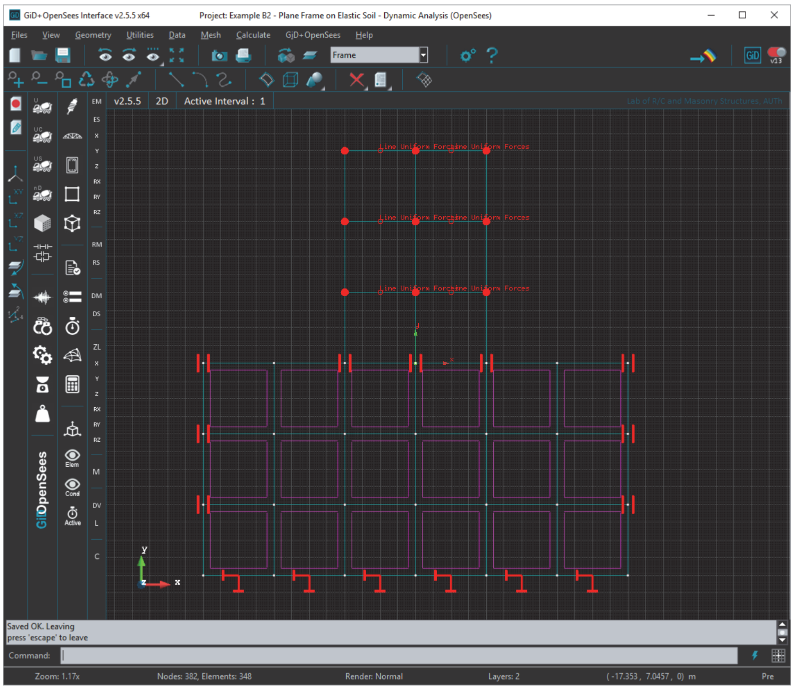
This condition enforces an approximate shear-type horizontal response for the soil substrate.

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 :

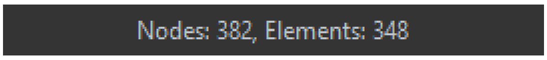
→



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 

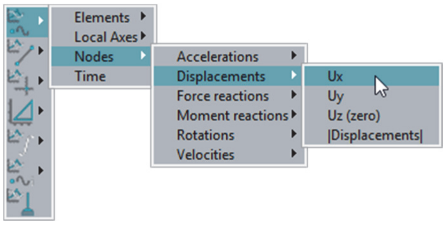


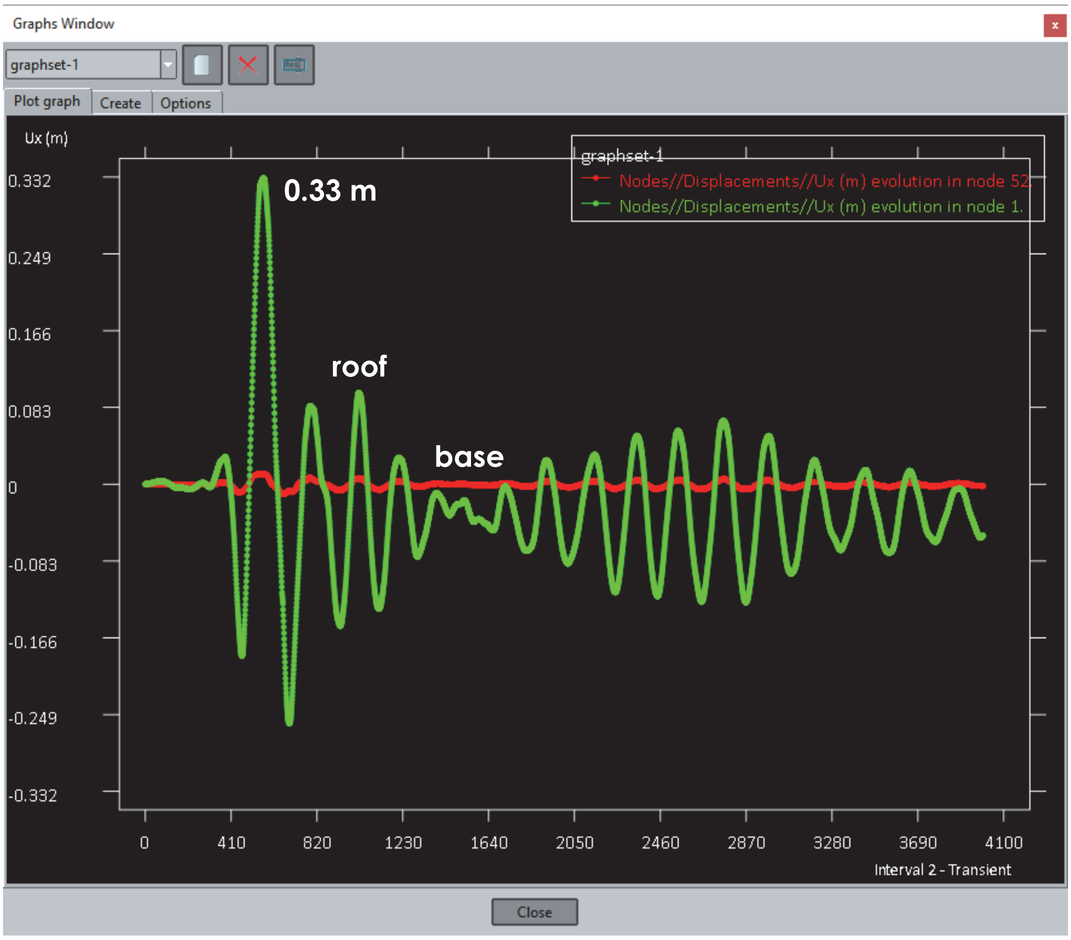
B. Analysis

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 →