OpenSees Navigator

Andreas Schellenberg
Research Engineer
Pacific Earthquake Engineering Research Center

Tony Yang
Assistant Professor
Department of Civil Engineering
University of British Columbia, Vancouver, Canada

Eiji Kohama
Head of Group
Earthquake and Structural Dynamics Group
Port and Airport Research Institute, Japan



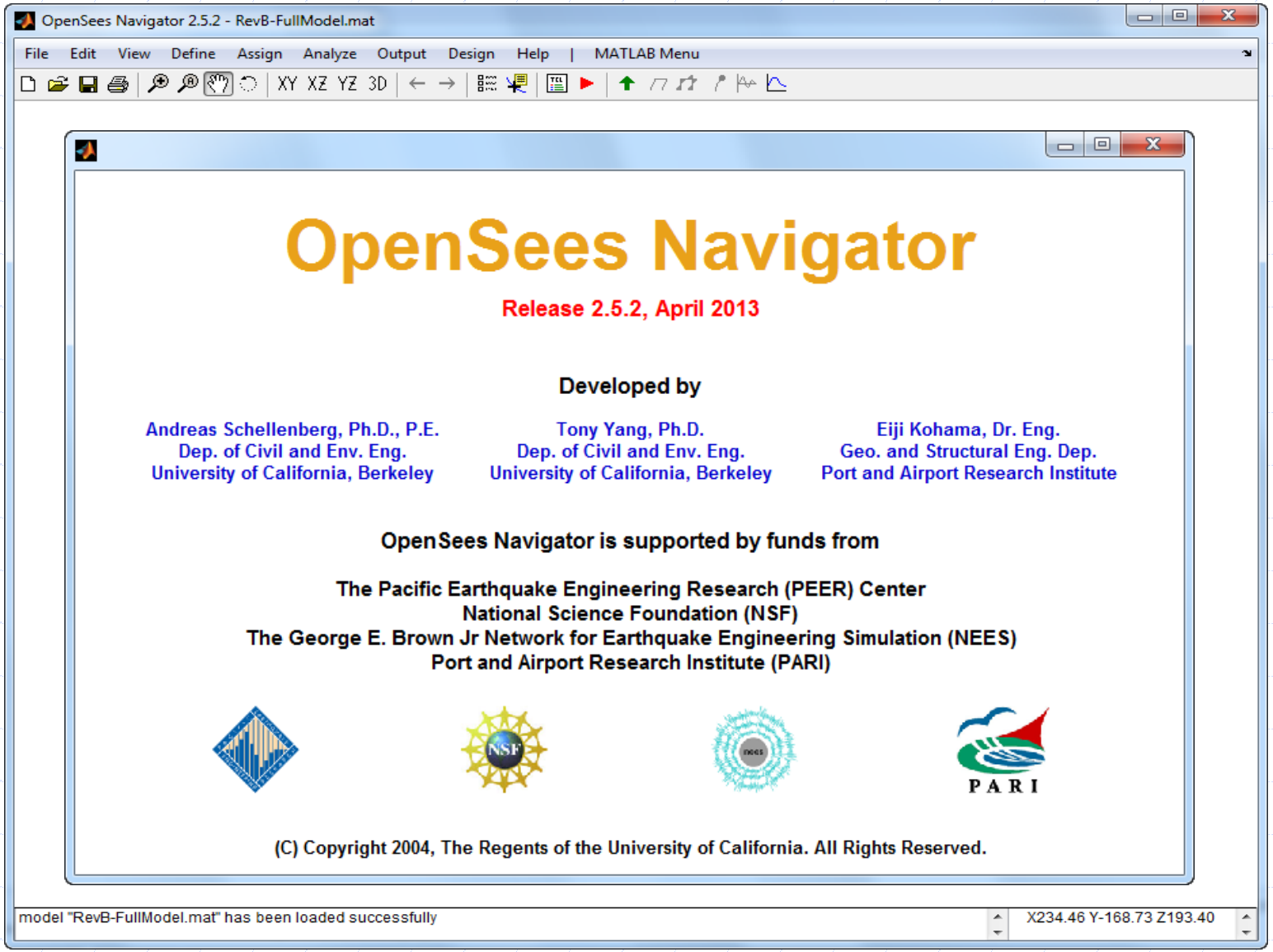
Introduction

- MATLAB based graphical user interface (GUI).
- Pre- and post-processing for OpenSees and OpenFresco.
- Design toolboxes: NSP, PBEE, AISC design checks, AISC database, response spectra for linear and bilinear systems and signal filtering.
- Both MATLAB Pcode and self-executable versions are available for Windows & Mac.
- Being used by researchers from Asia, US, Canada, South America and Europe.

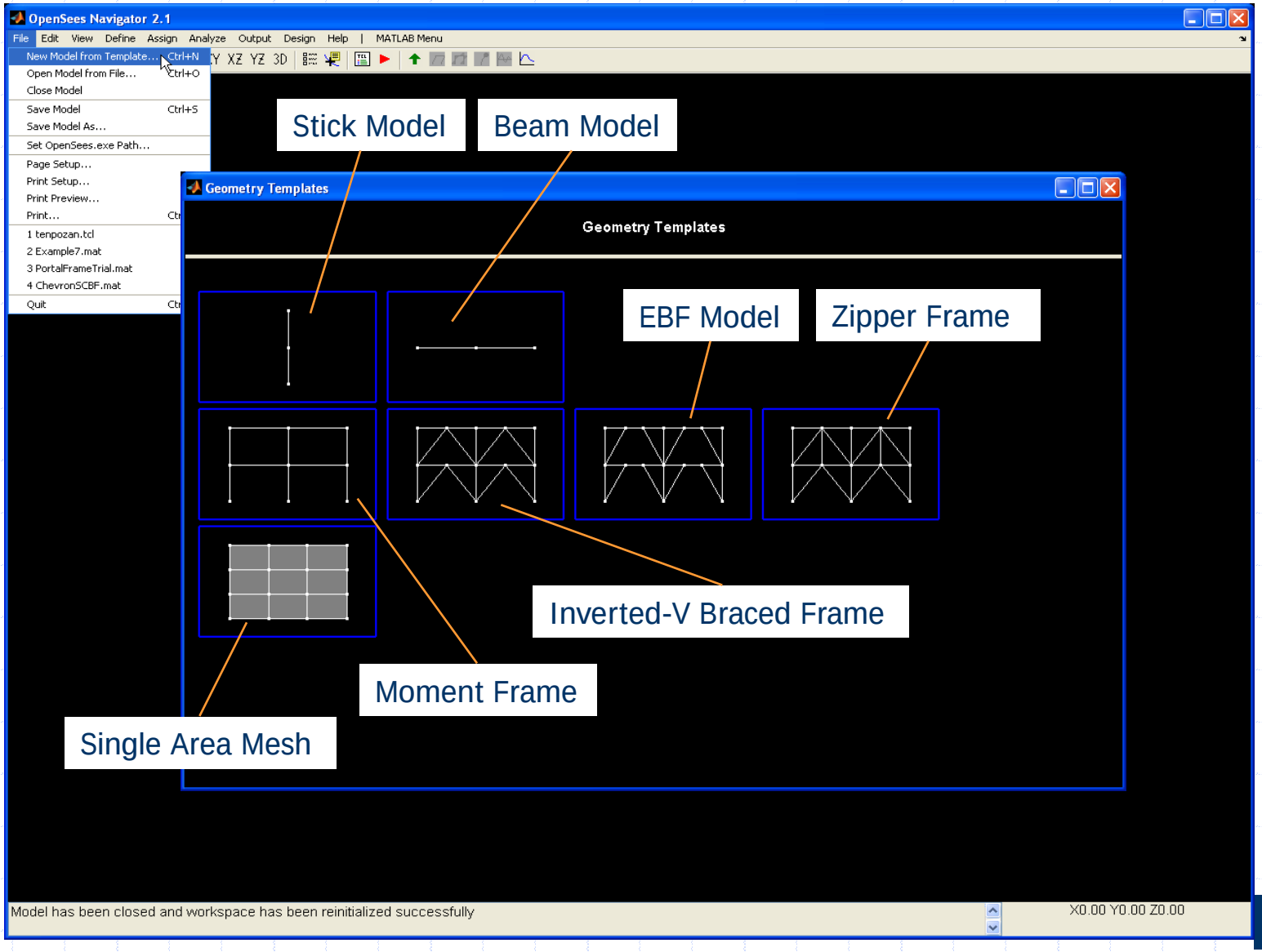
Motivations

- Replace the TCL text input with graphical input.
- Most researchers use MATLAB to do the post-processing, and MATLAB/Simulink is the typical framework for implementing hybrid simulation tests.
- OpenSees Navigator will create the OpenSees (analytical/hybrid) model and graphically display the results before, during or after a test.
- Provides many robust plotting algorithms and is very effective in generating the plots for engineering applications.
- Flexible to use and requires no programming skill.

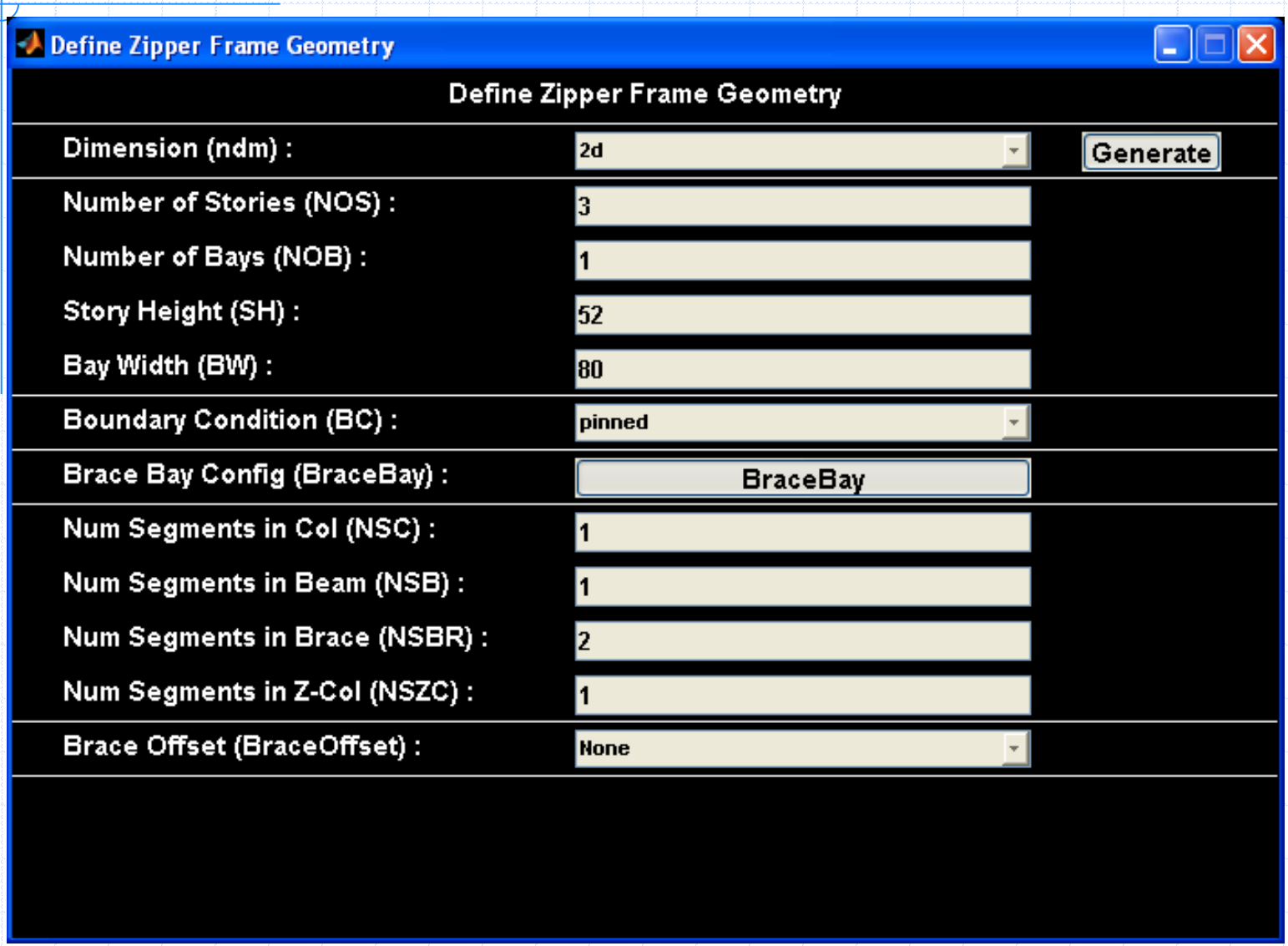
OpenSees Navigator



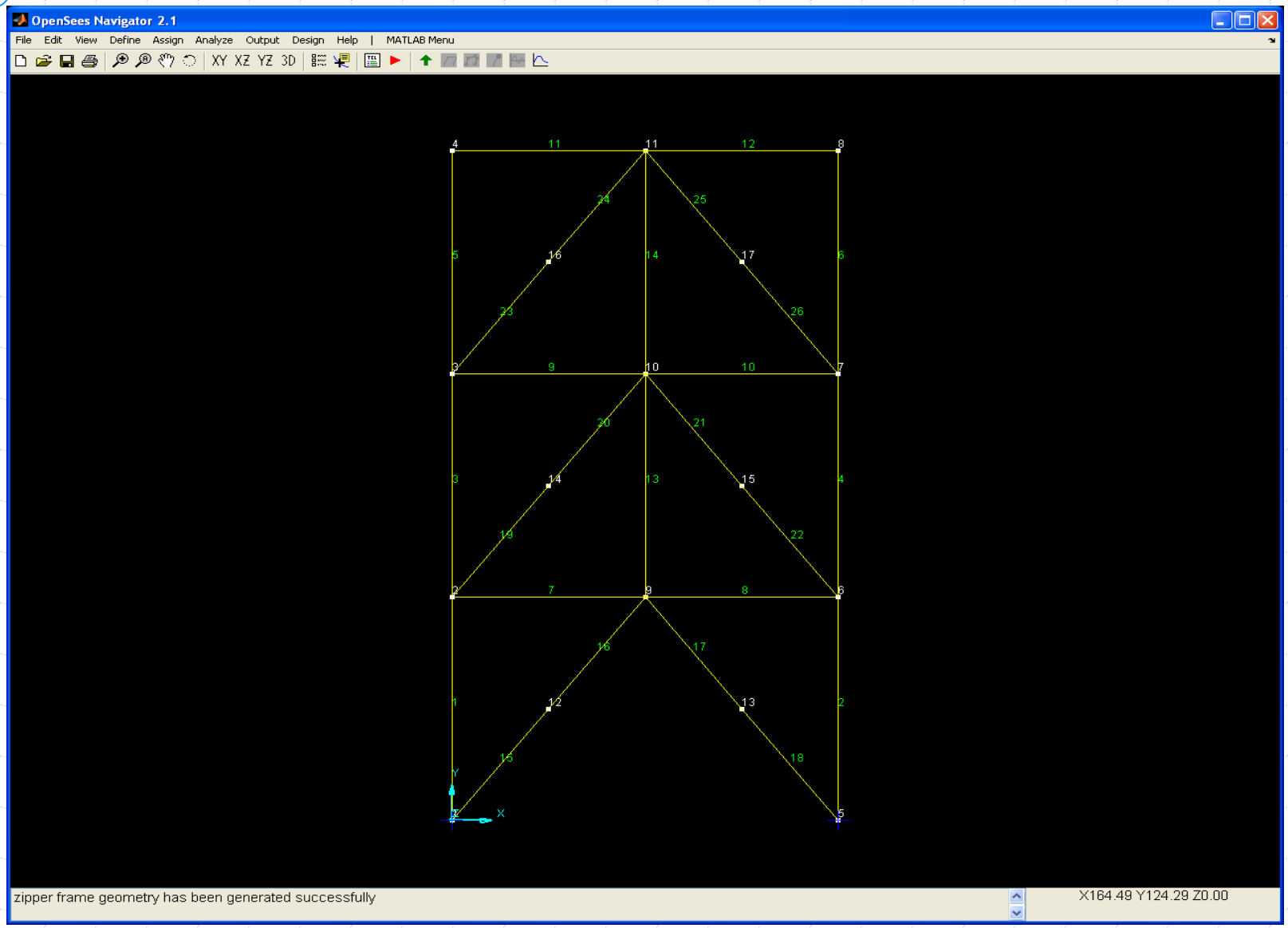
Define geometry: new model template



Define geometry: Zipper braced frame



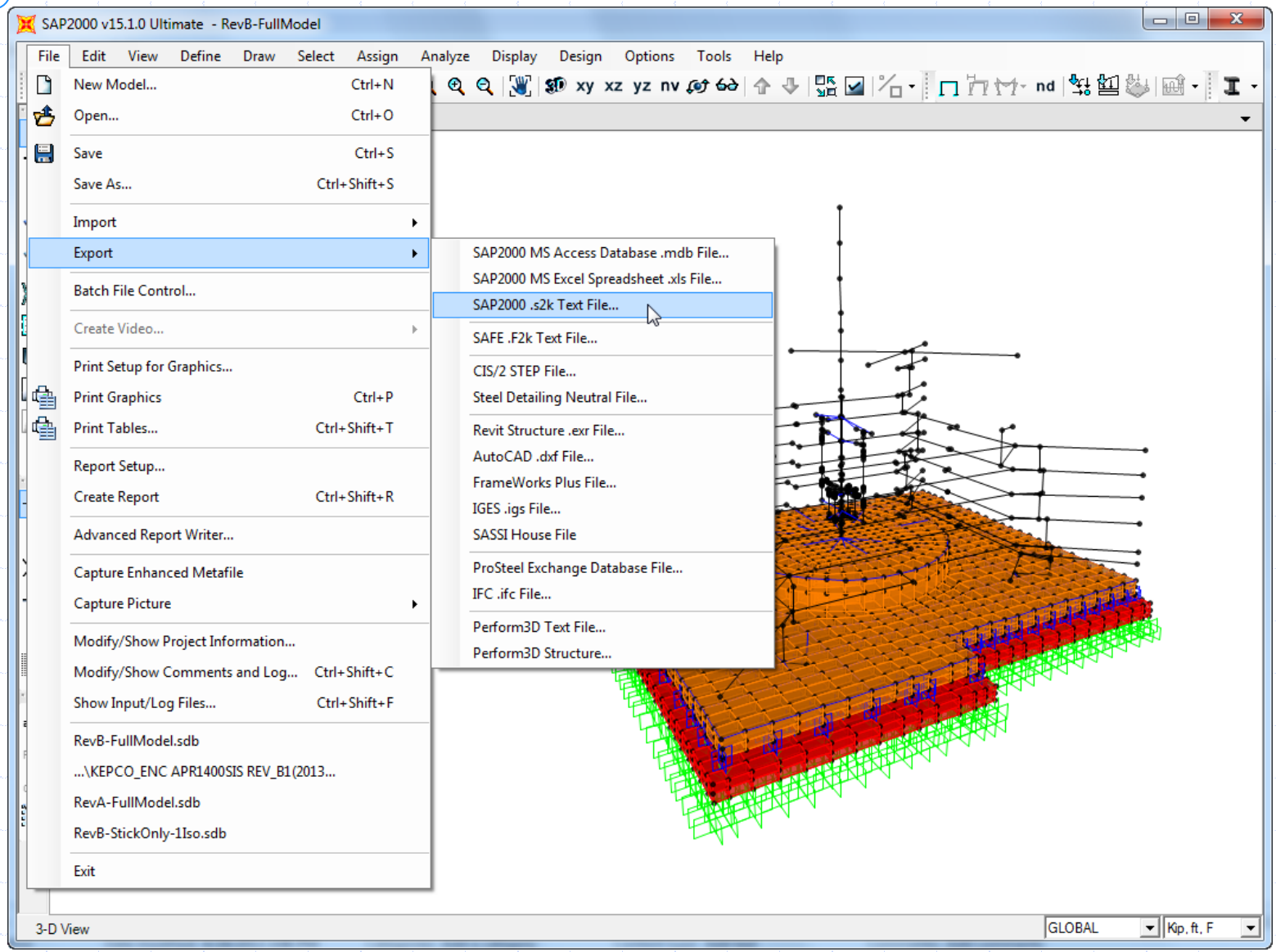
View geometry: display



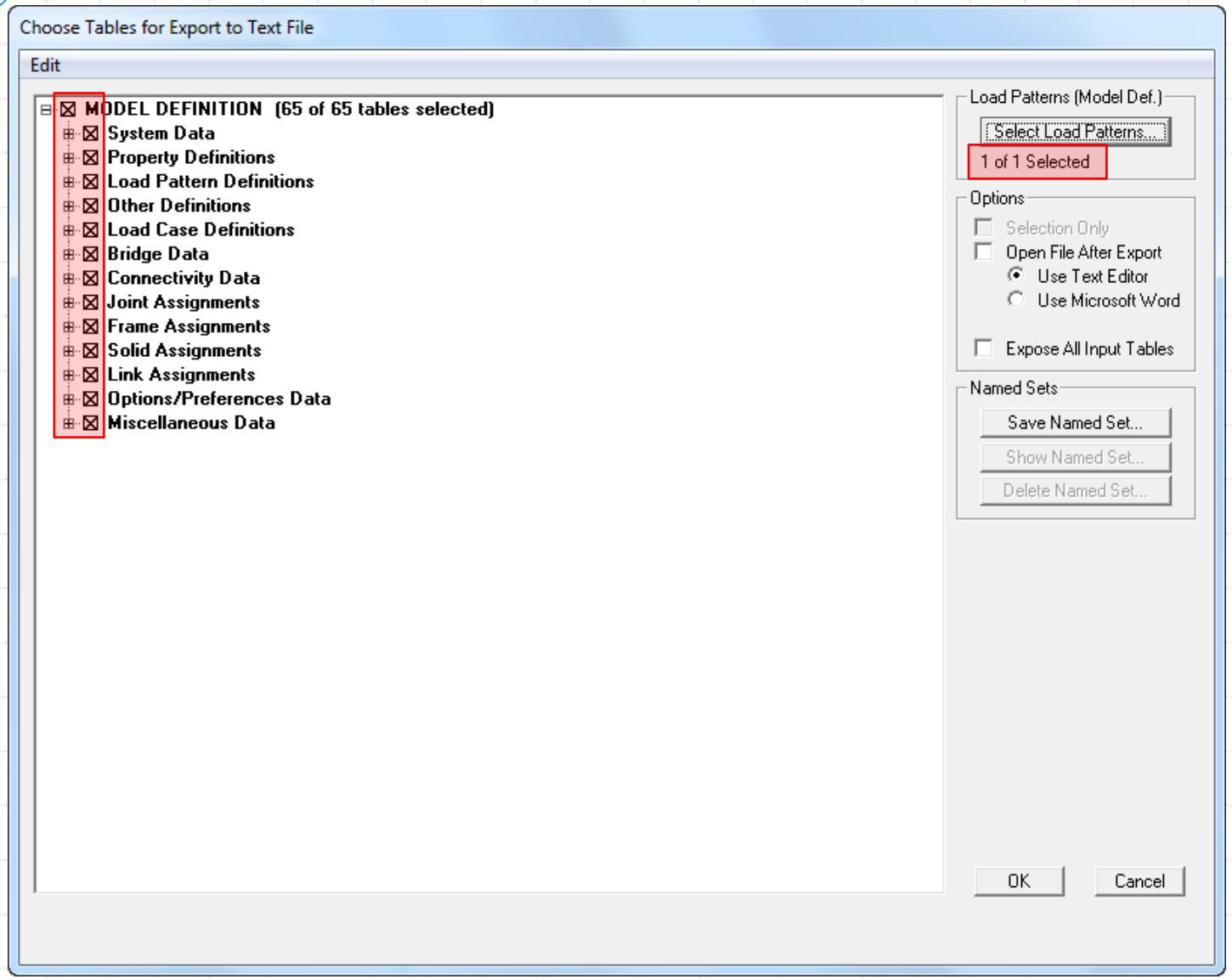
Conversion: SAP2000 to OpenSees

- Export SAP2000 model to .s2k file
- Start OpenSees Navigator program
- Select “Open Model from File…” and choose the earlier exported .s2k file
- There is no one-to-one relationship between SAP2000 and OpenSees objects, therefore
- Carefully check the converted model
- Element loads and nodal constraints are currently not converted yet

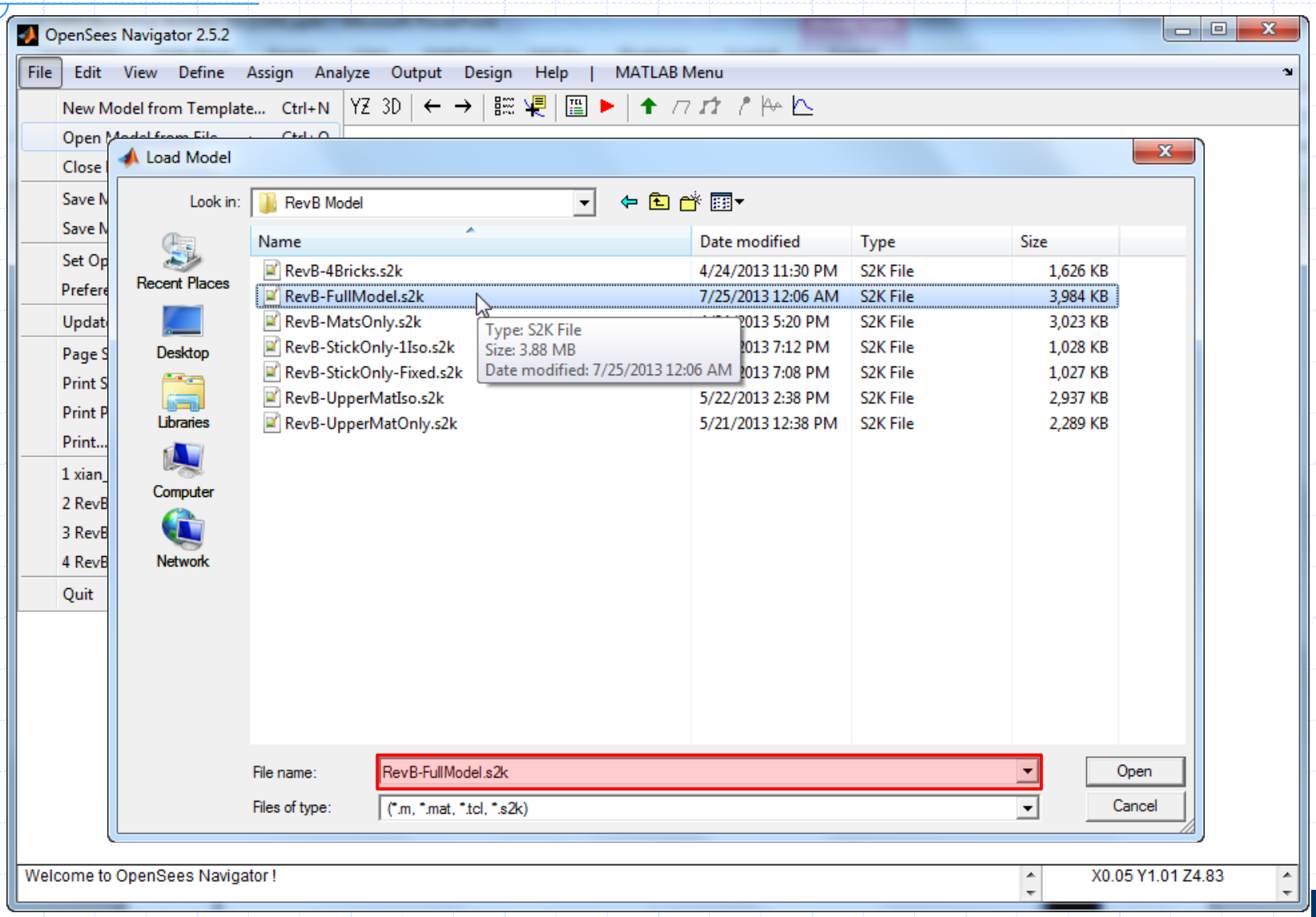
Export to s2k file



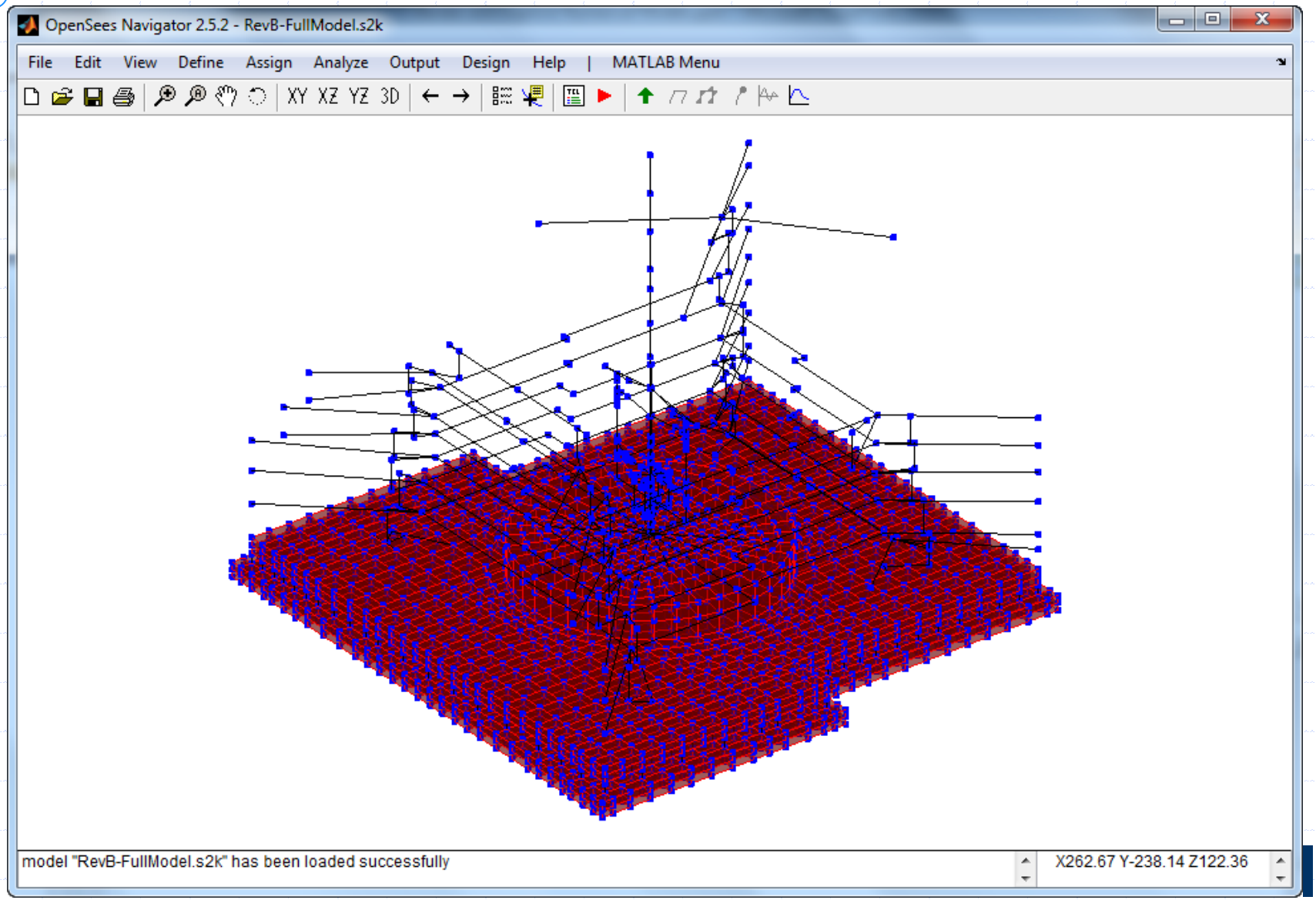
Export to s2k file



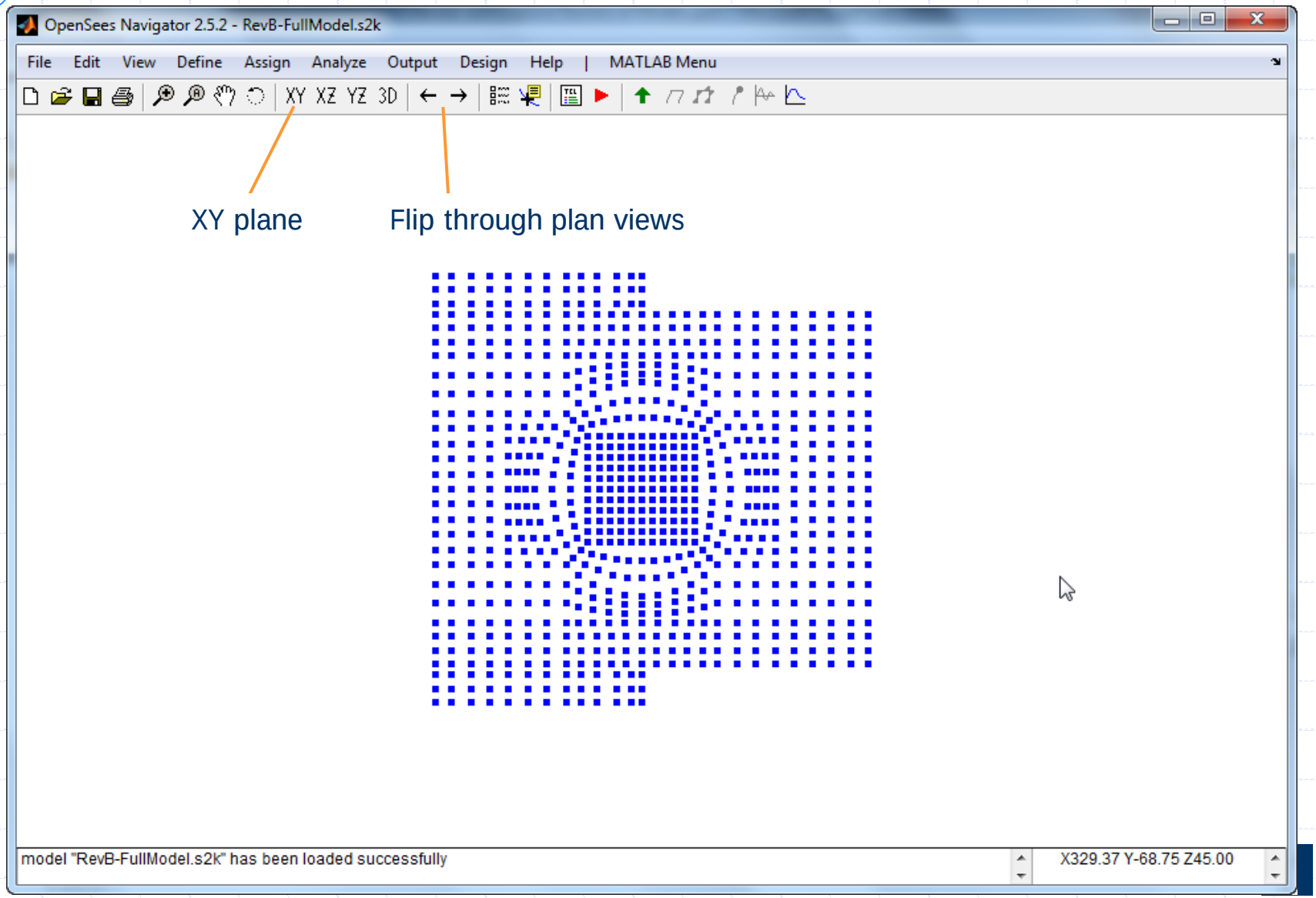
Define geometry: import from SAP2000



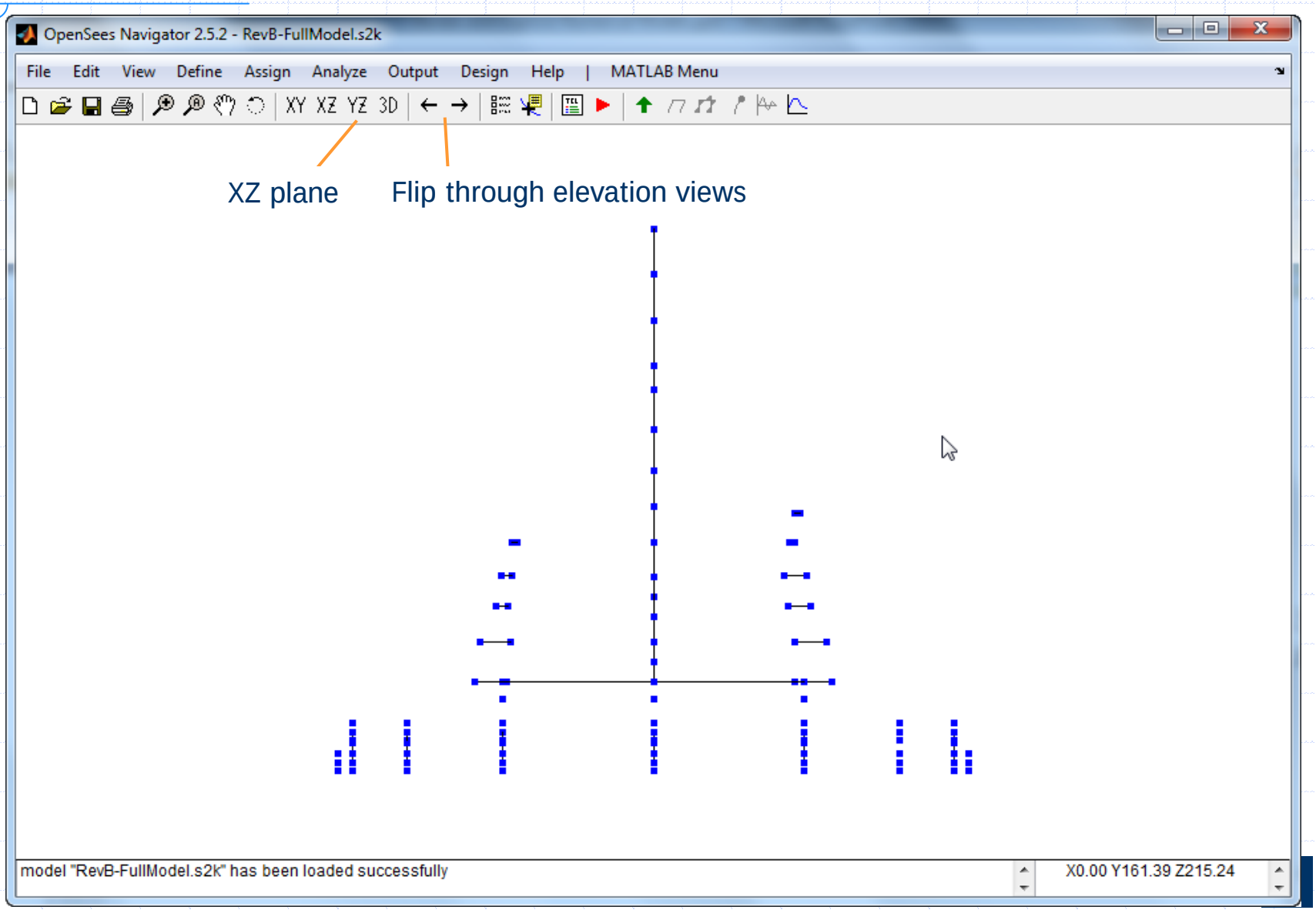
Define geometry: import from SAP2000



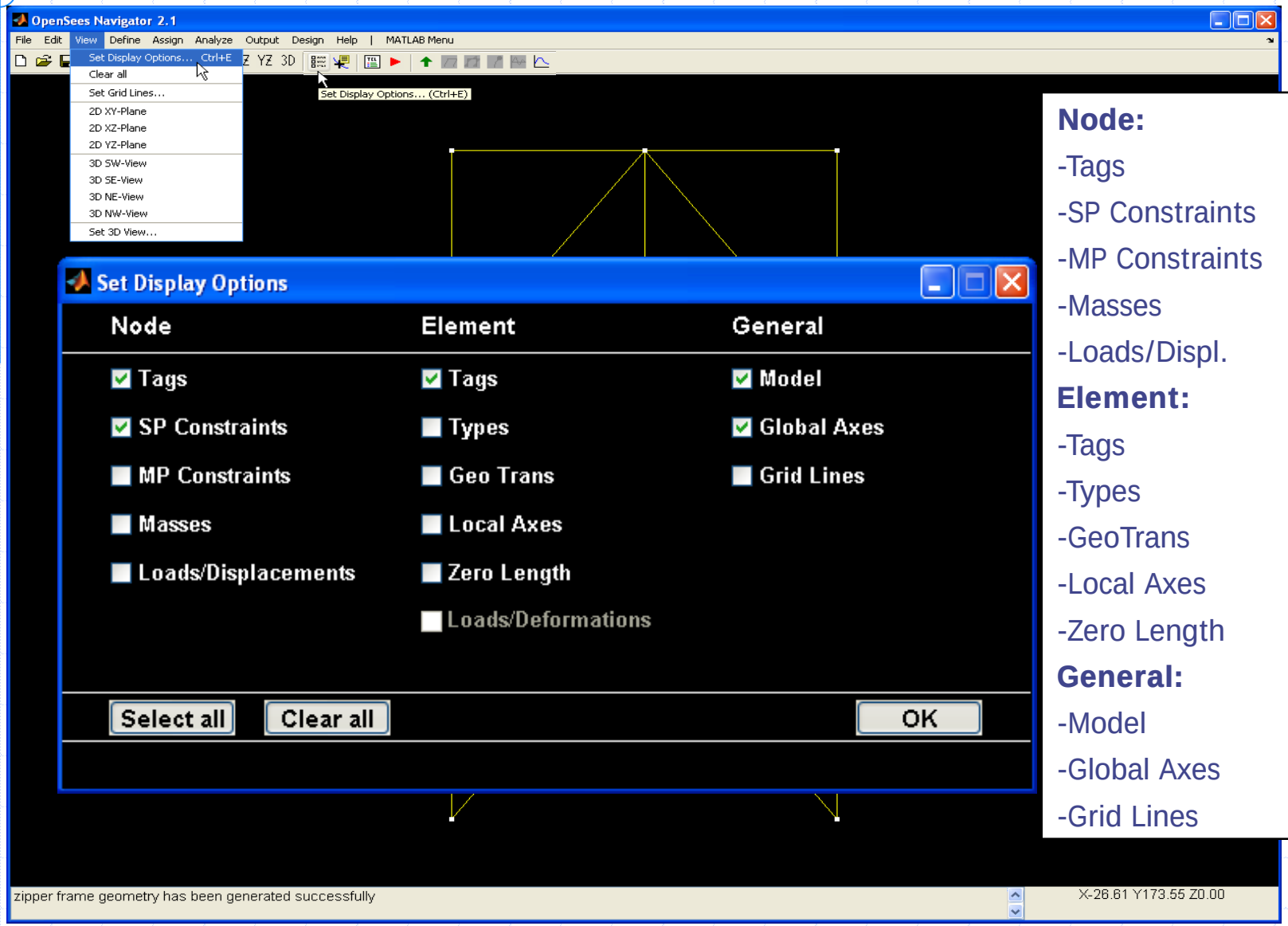
Define geometry: import from SAP2000



Define geometry: import from SAP2000



View geometry: set display options



Node:

-Tags

-SP Constraints

-MP Constraints

-Masses

-Loads/Displ.

Element:

-Tags

-Types

-GeoTrans

-Local Axes

-Zero Length

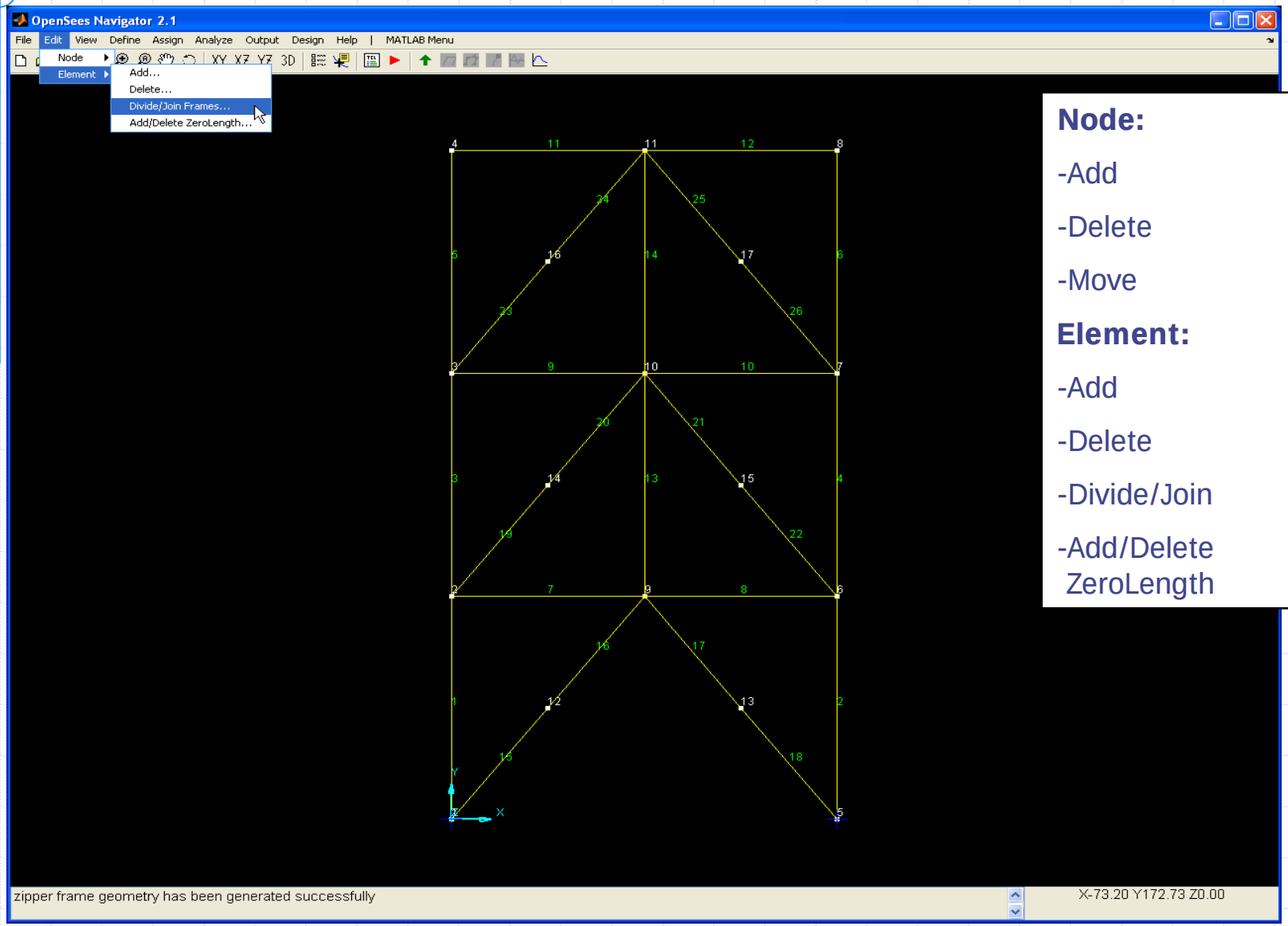
General:

-Model

-Global Axes

-Grid Lines

Edit geometry



Node:

-Add

-Delete

-Move

Element:

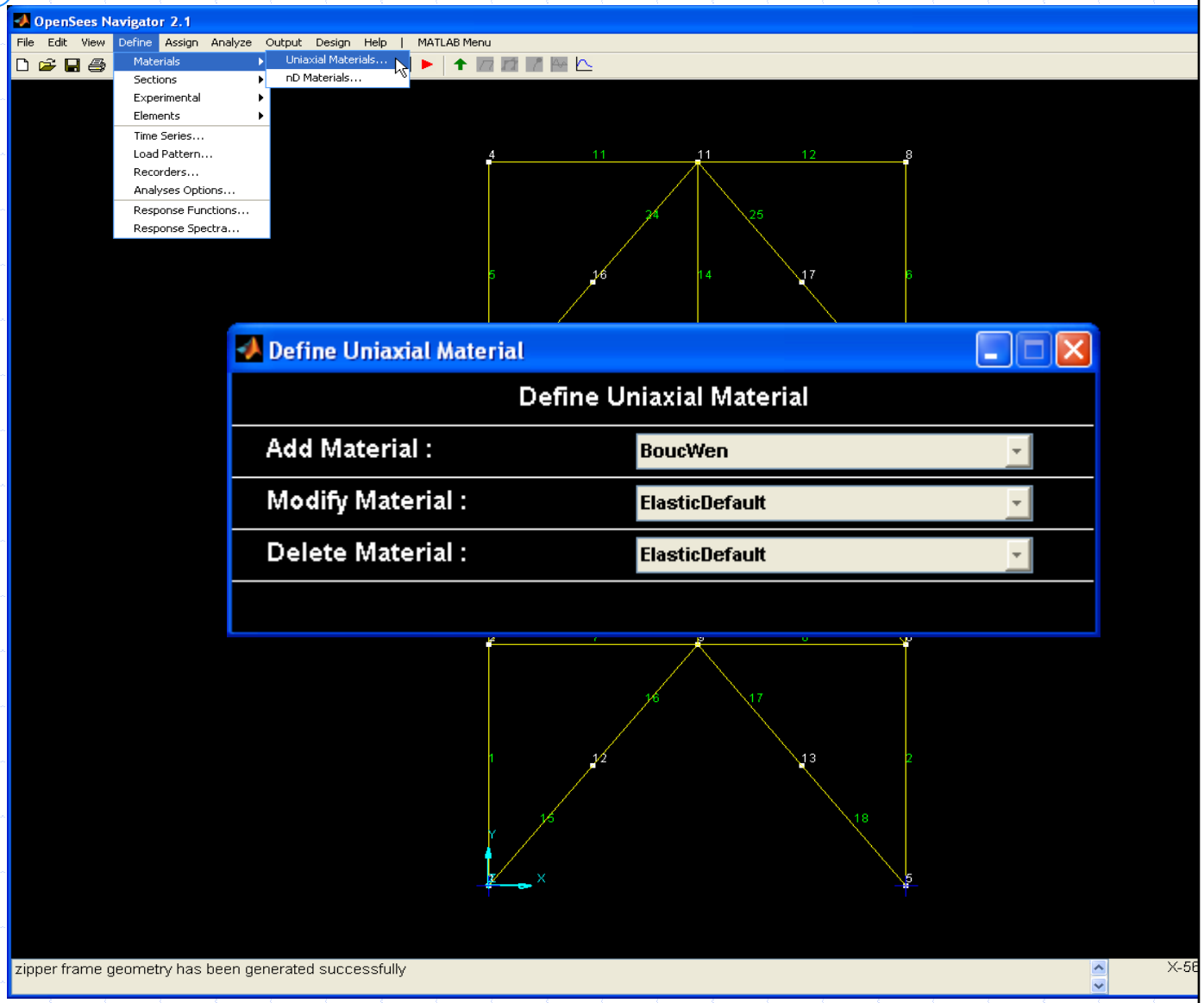
-Add

-Delete

-Divide/Join

-Add/Delete ZeroLength

Define material: uniaxial materials



Templates:

-BoucWen

-Concrete01

-Concrete02

-Concrete04

-Elastic

-ElasticNoTension

-ElasticPP

-ElasticPPGap

-Fatigue

-Hardening

-Hysteretic

-MinMax

-Parallel

-ReinforcingSteel

-Series

-Steel01

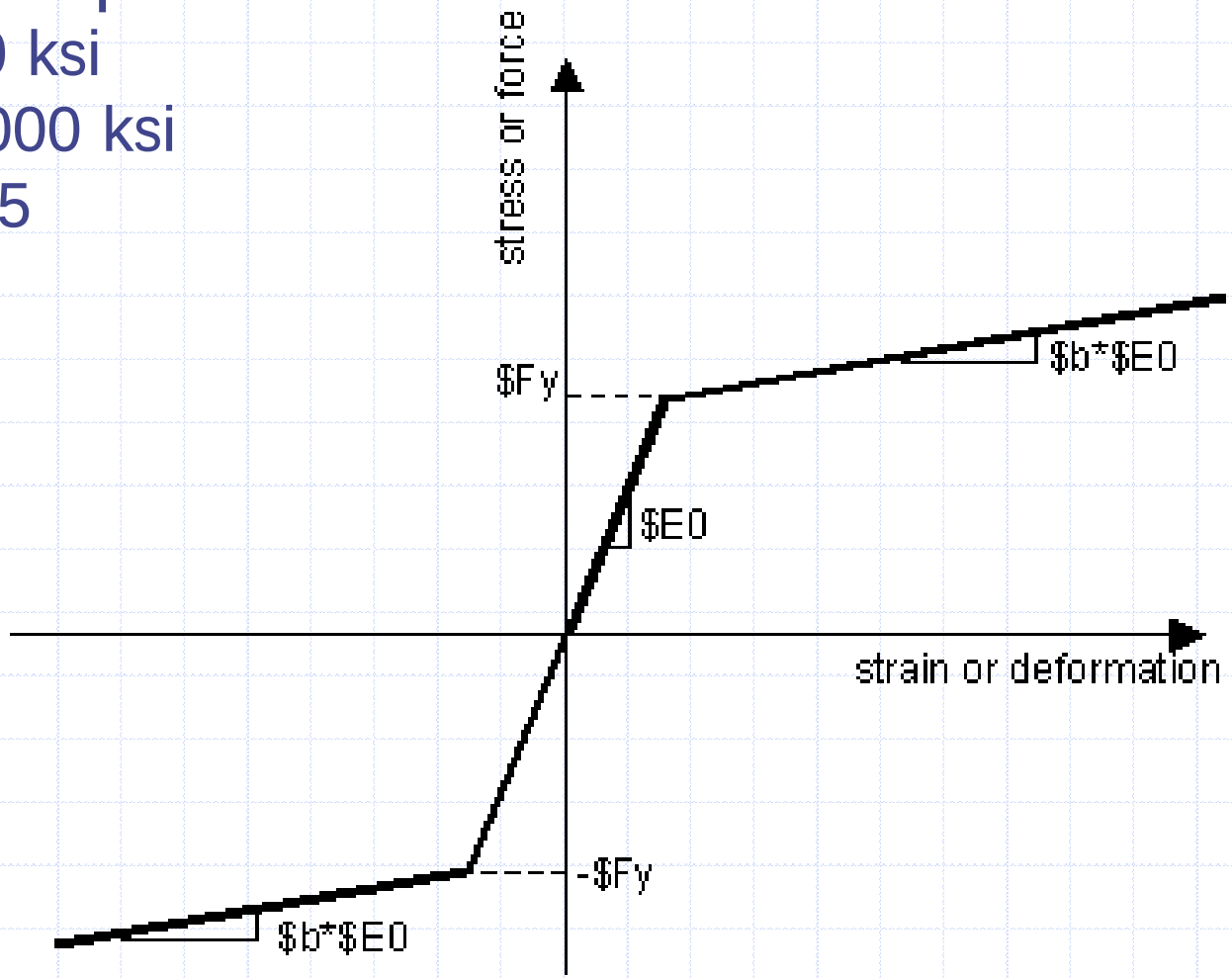
-Steel02

-Viscous

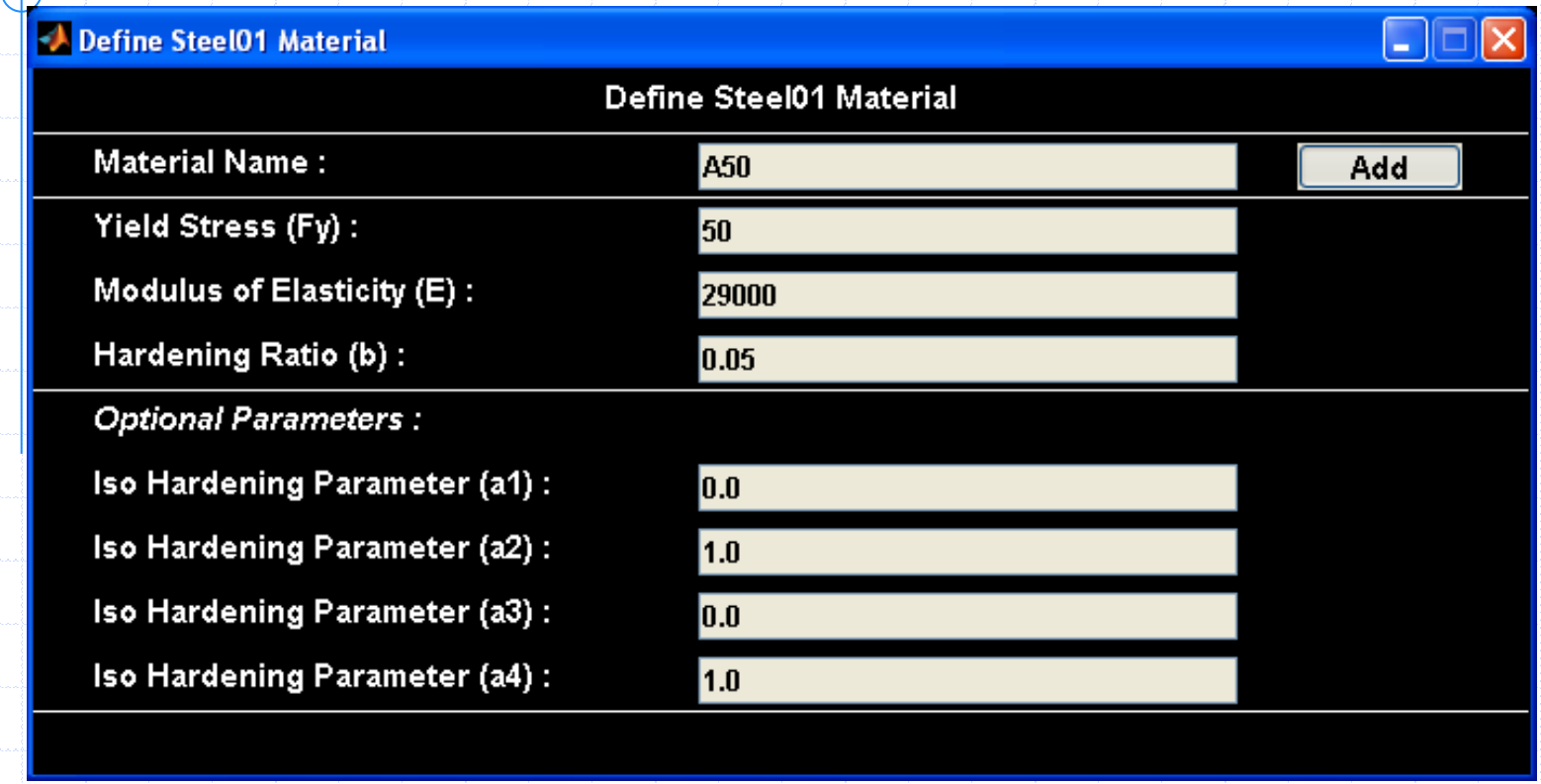
-...

Define uniaxial material: Steel01

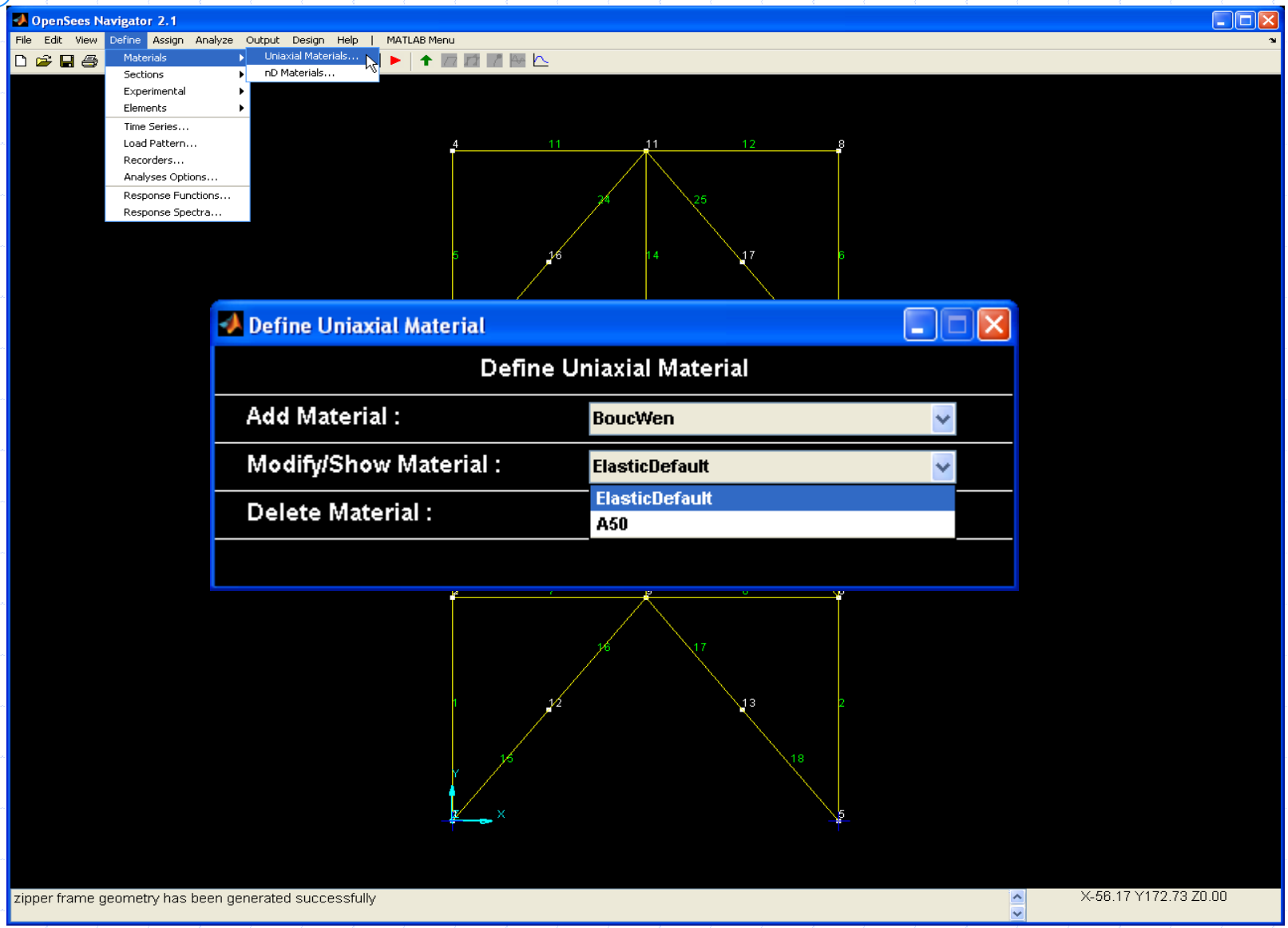
- Material properties
  - Fy = 50 ksi
  - E = 29000 ksi
  - b = 0.05



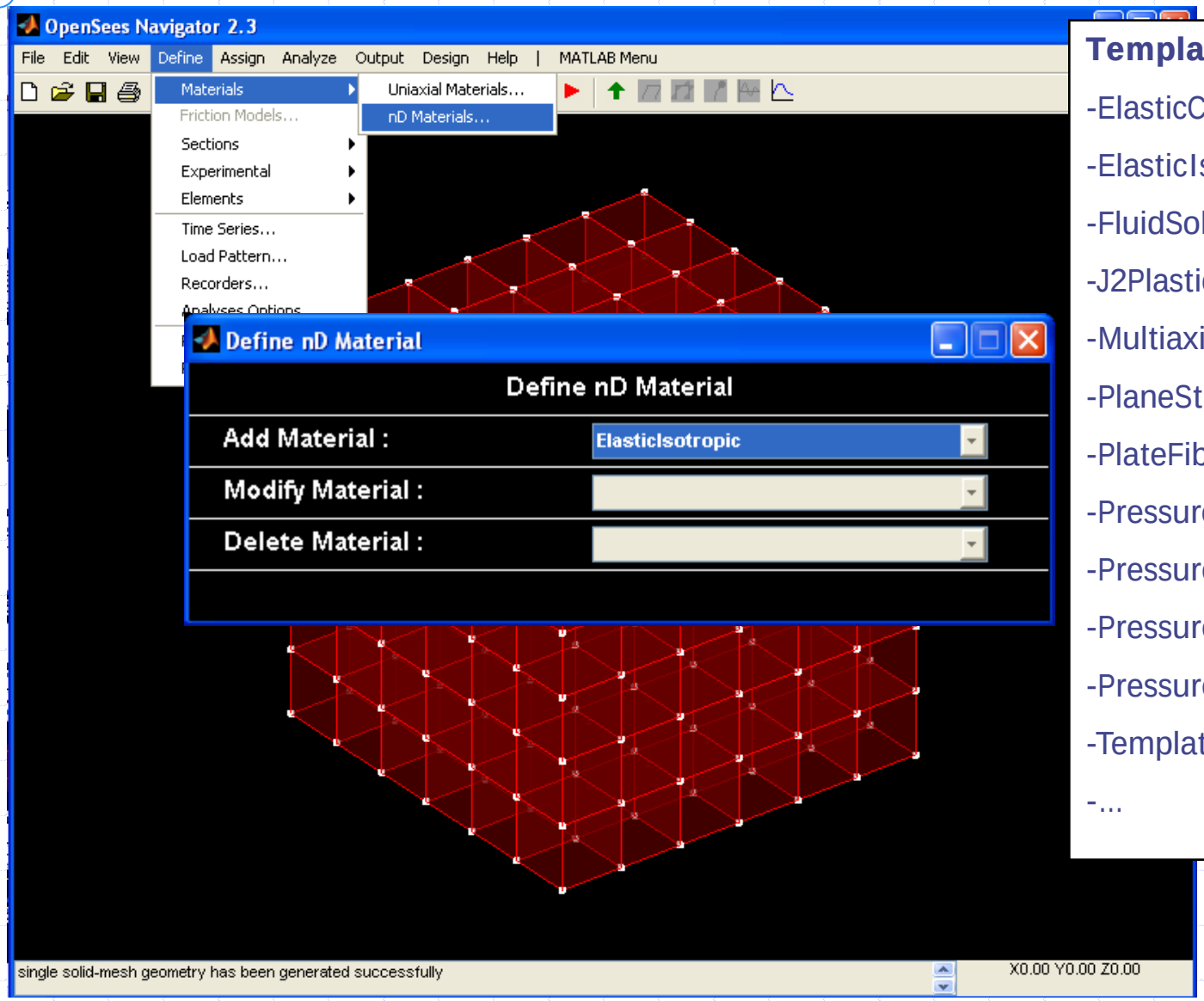
Define uniaxial material: Steel01



Define material: uniaxial materials



Define material: nD materials



Templates:

-ElasticCrossAnisotropic3D

-ElasticIsotropic

-FluidSolidPorous

-J2Plasticity

-MultiaxialCyclicPlasticity

-PlaneStress

-PlateFiber

-PressureDependMultiYield

-PressureDependMultiYield02

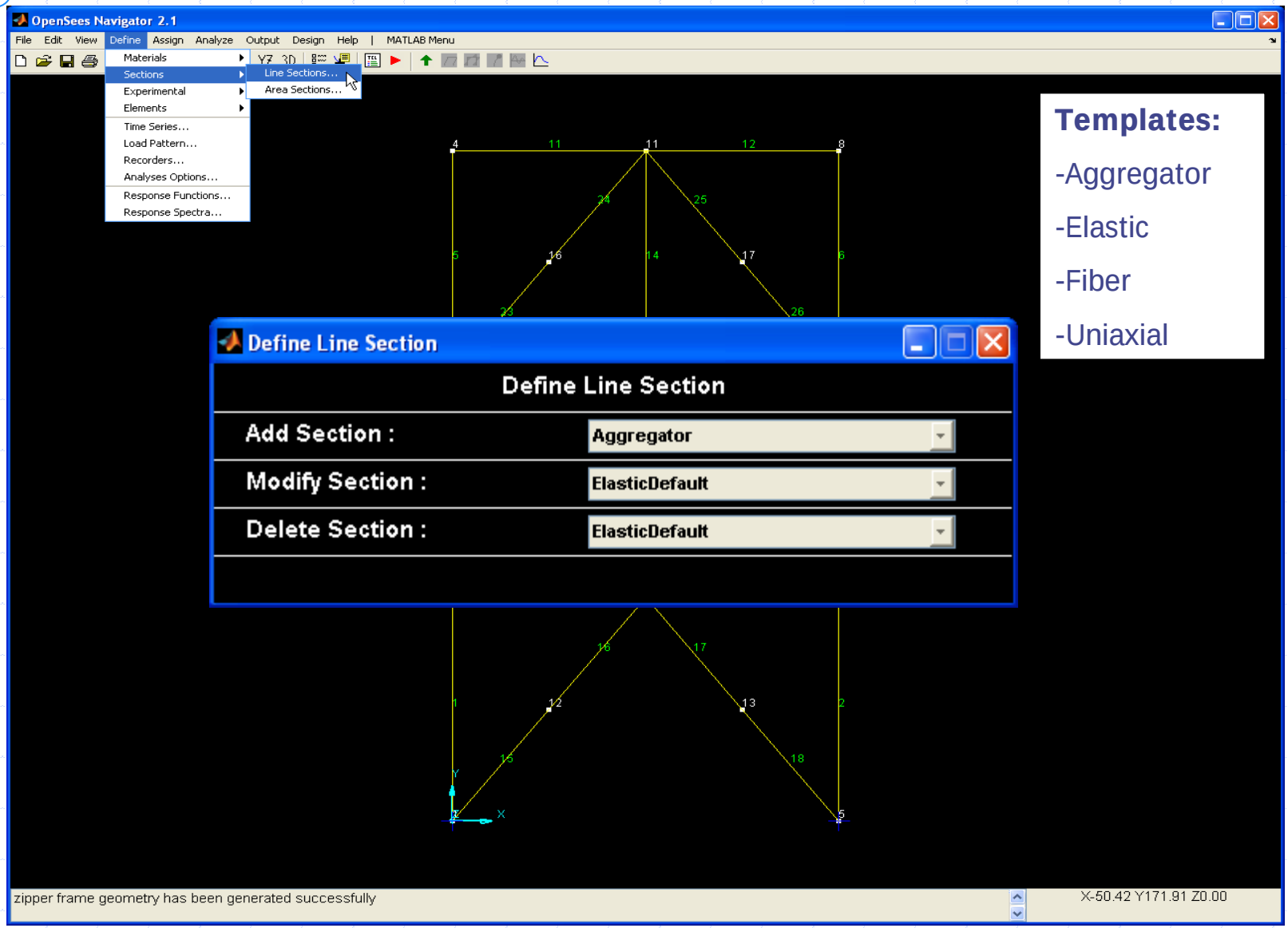
-PressureDependentElastic3D

-PressureIndependMultiYield

-Template3DElastoPlastic

-...

Define section: line sections



Templates:

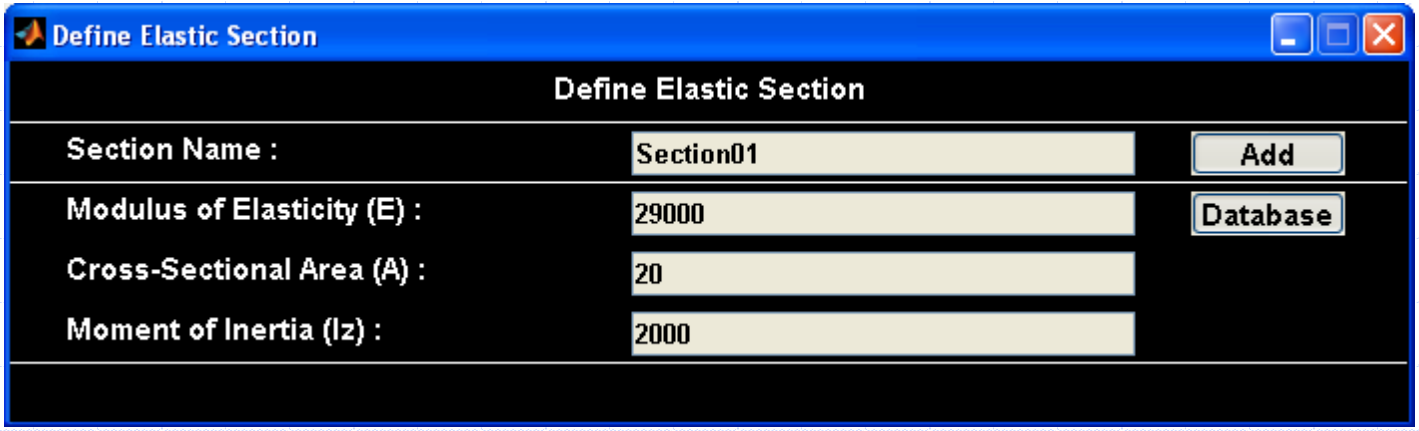
-Aggregator

-Elastic

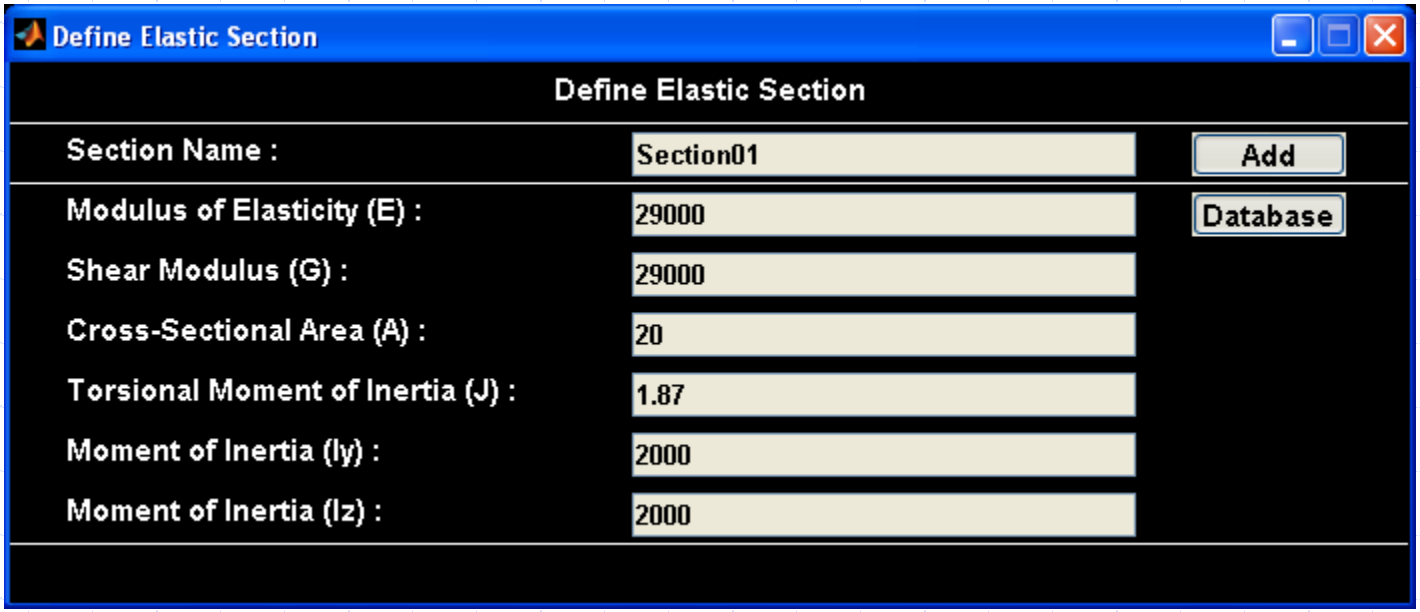
-Fiber

-Uniaxial

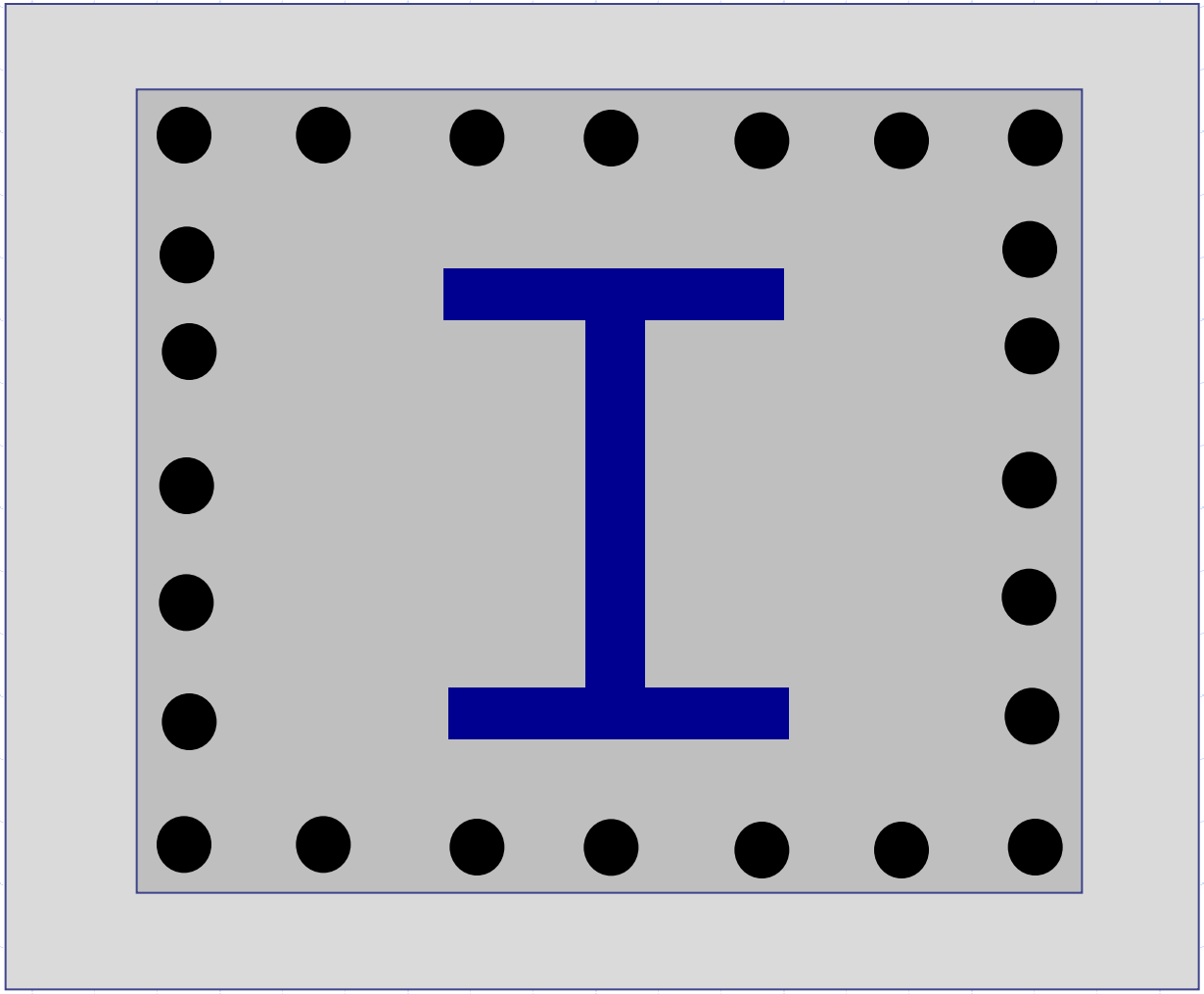
Define line section: elastic section



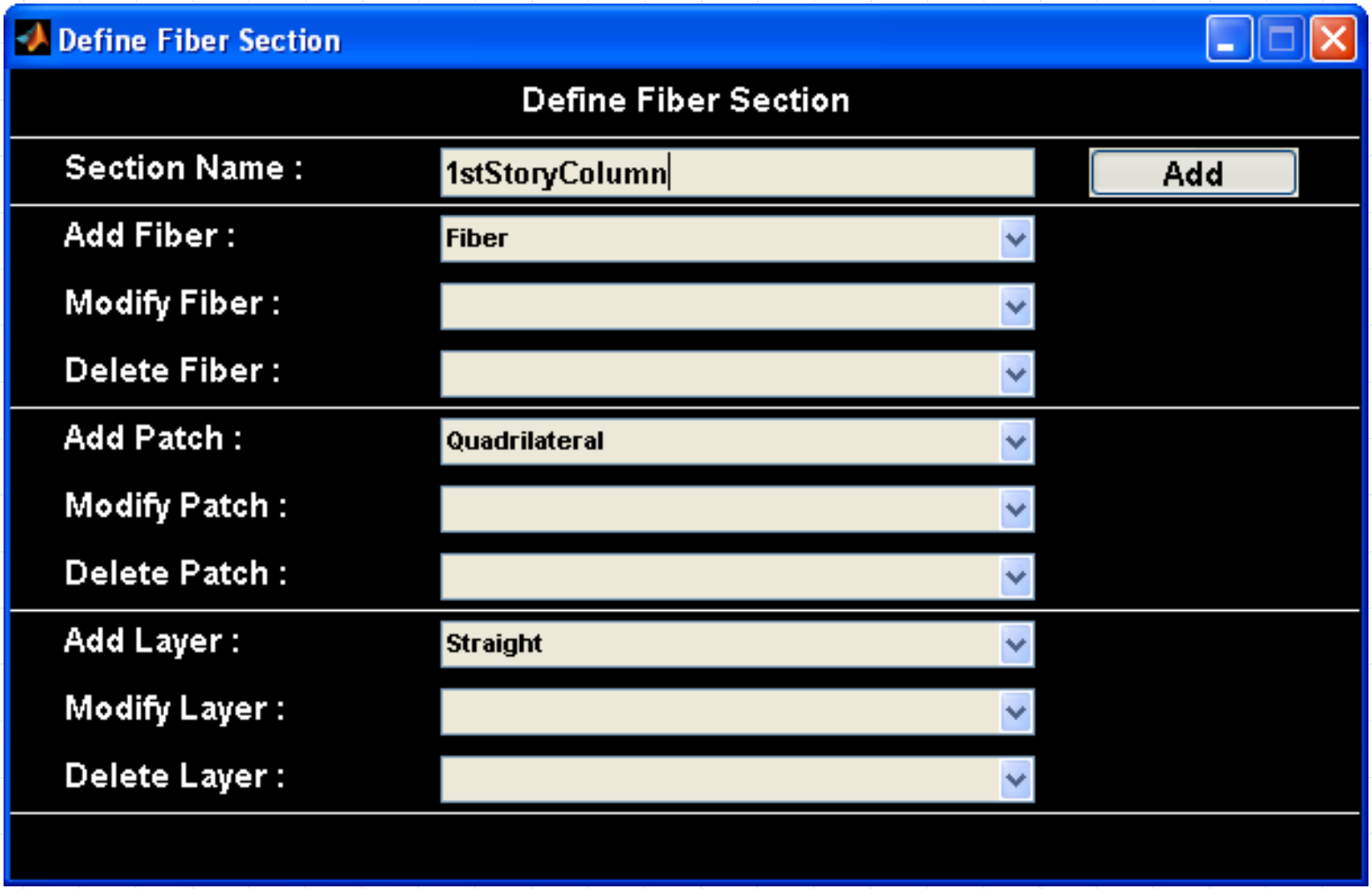
If the model is 3D



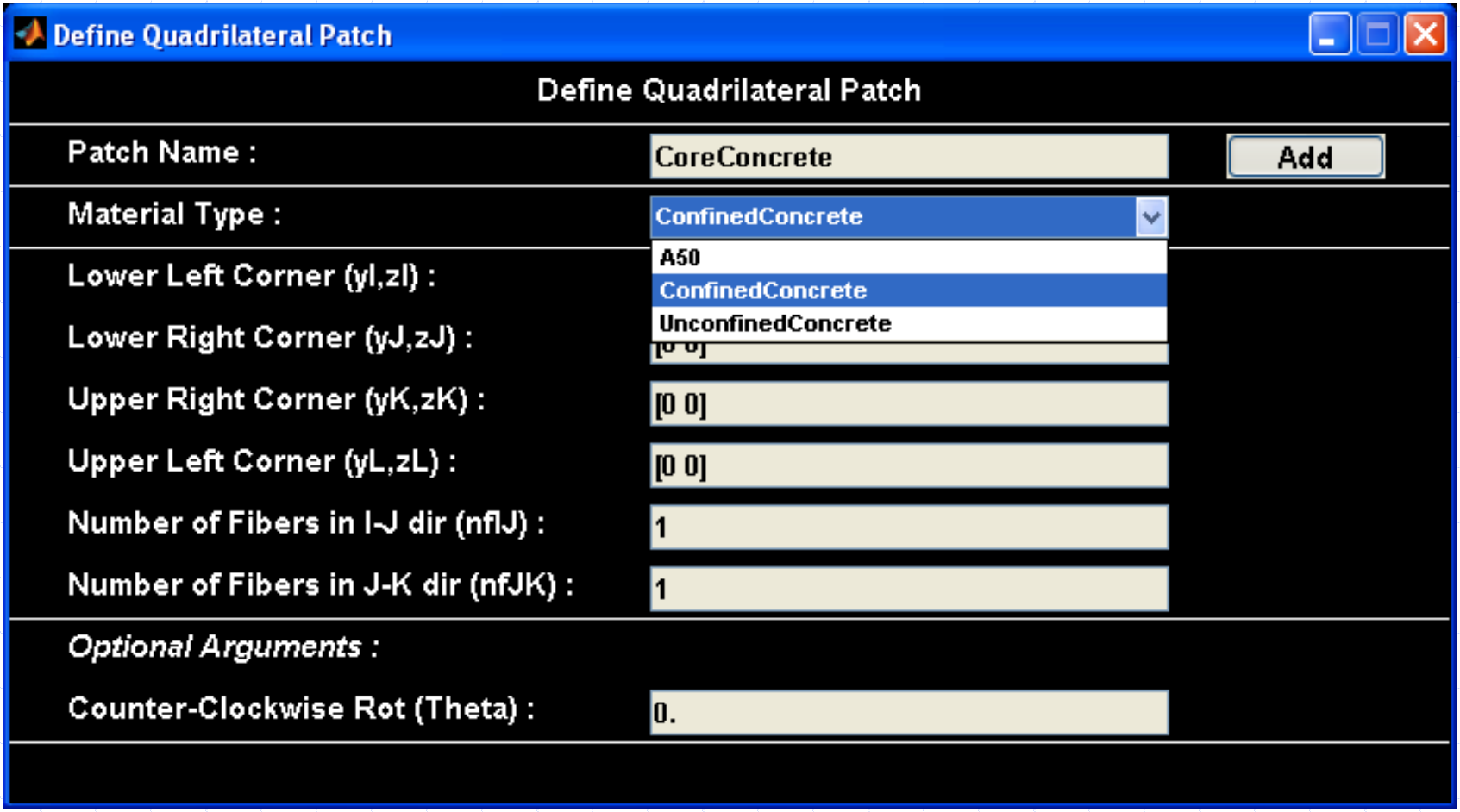
Define fiber section: Composite patch



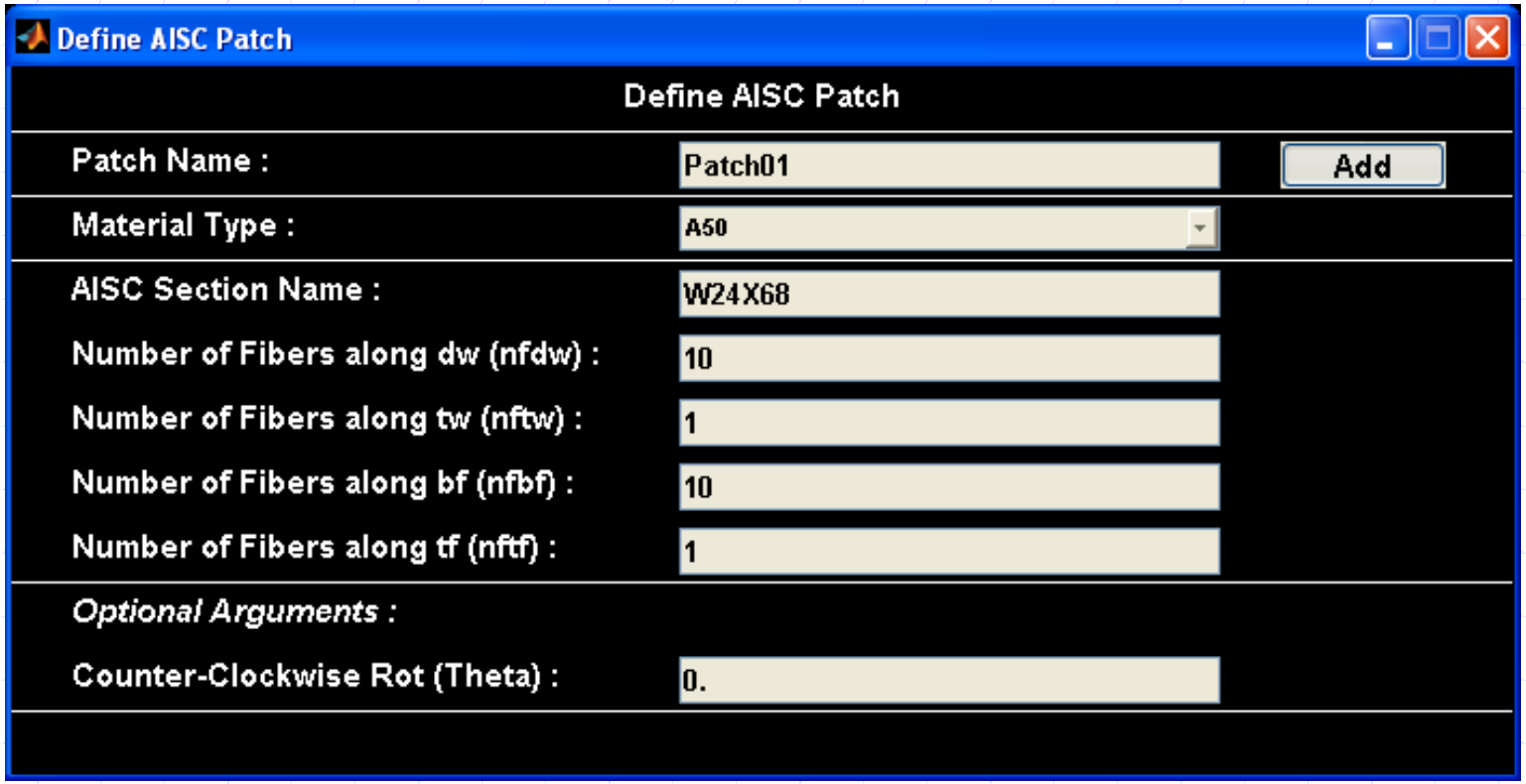
Define line section: fiber section



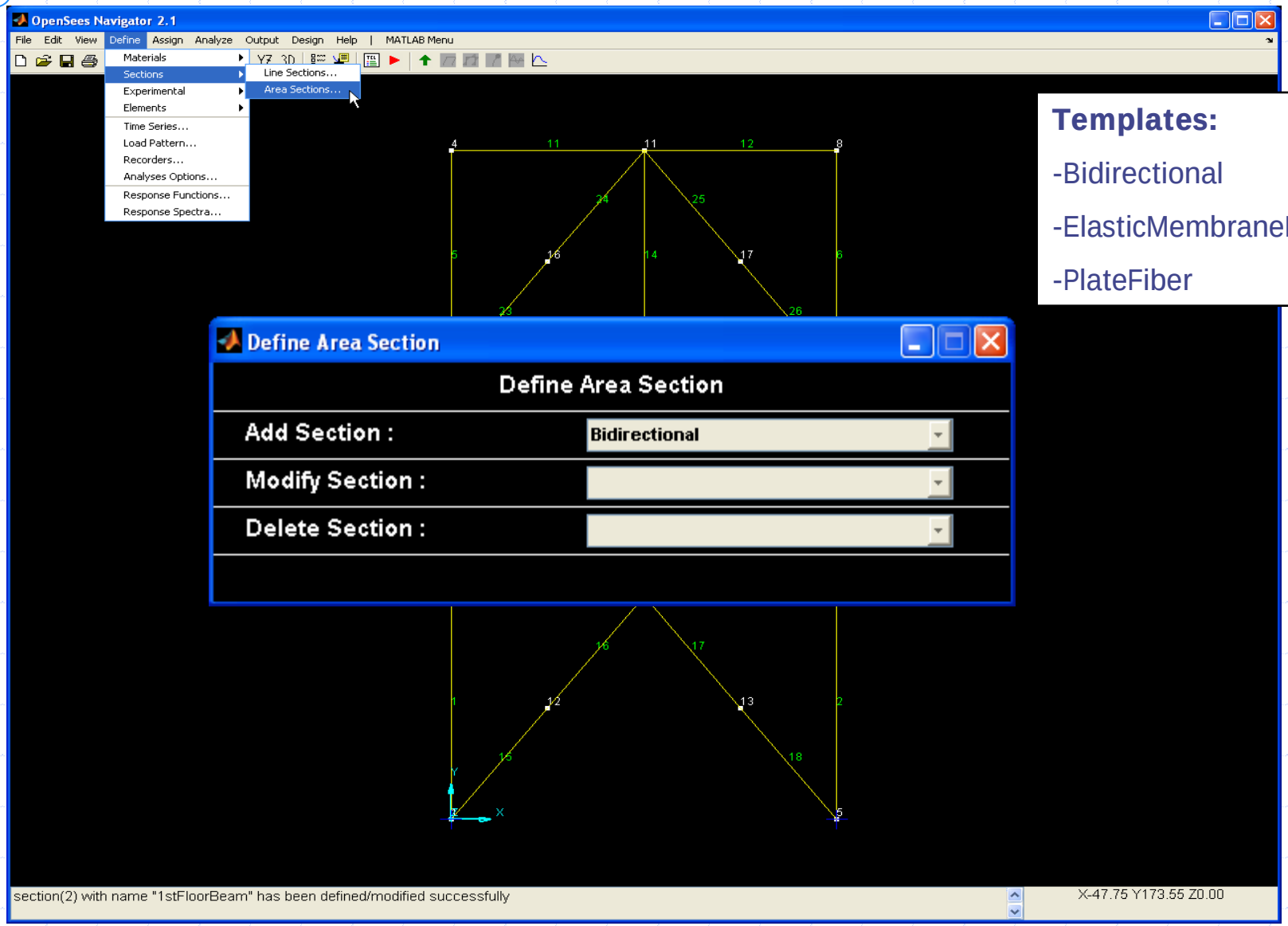
Define line section: quadrilateral patch



Define fiber section: AISC patch



Define section: area section



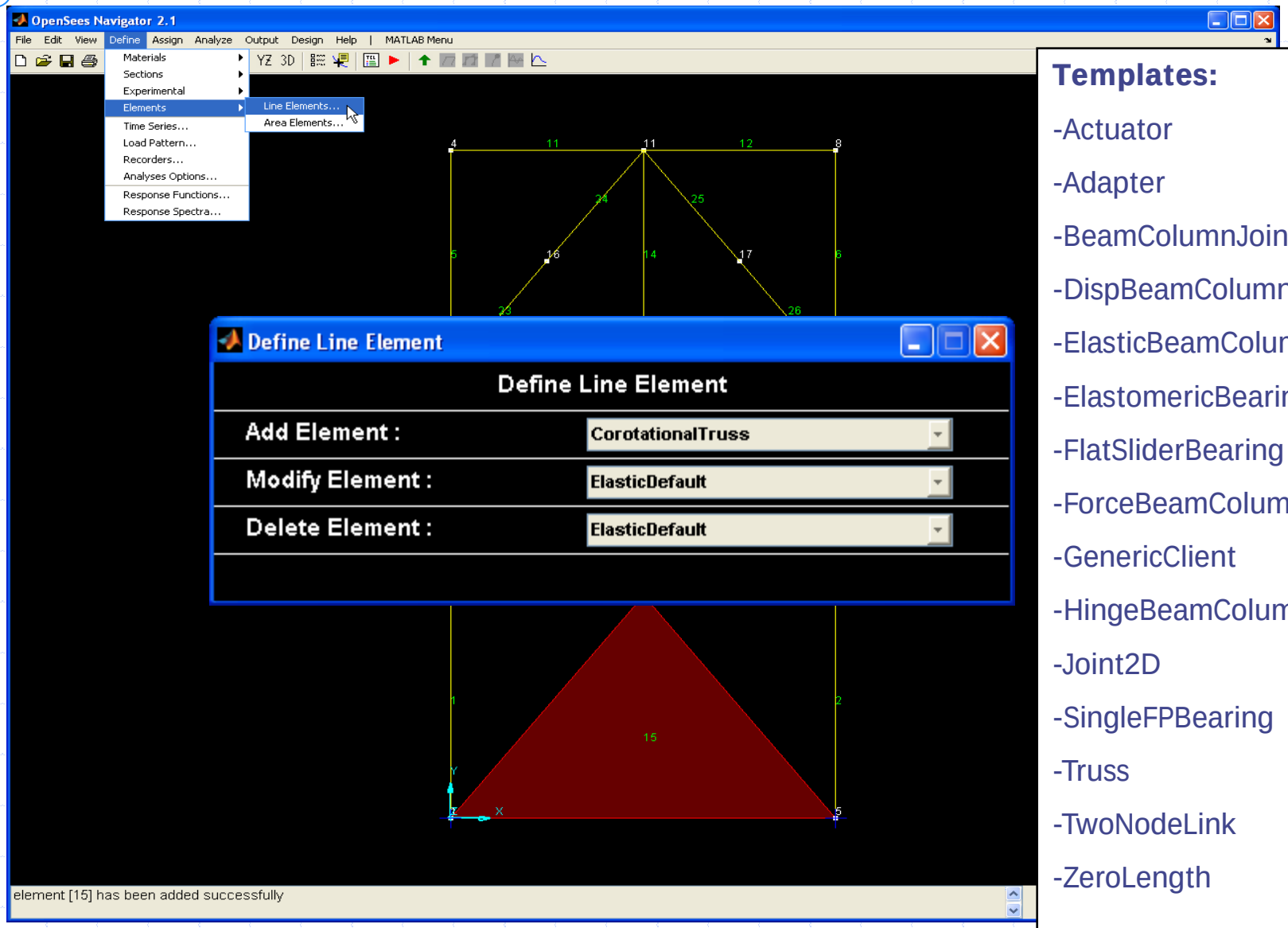
Templates:

-Bidirectional

-ElasticMembranePlate

-PlateFiber

Define element: line element



Templates:

-Actuator

-Adapter

-BeamColumnJoint

-DispBeamColumn

-ElasticBeamColumn

-ElastomericBearing

-FlatSliderBearing

-ForceBeamColumn

-GenericClient

-HingeBeamColumn

-Joint2D

-SingleFPBearing

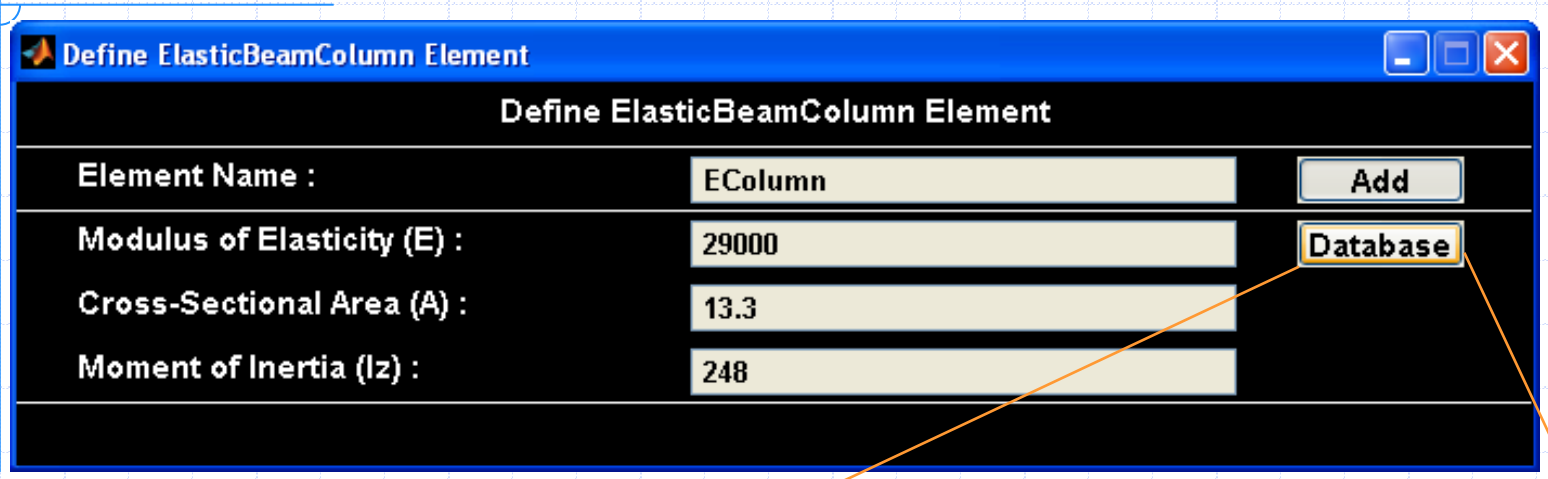
-Truss

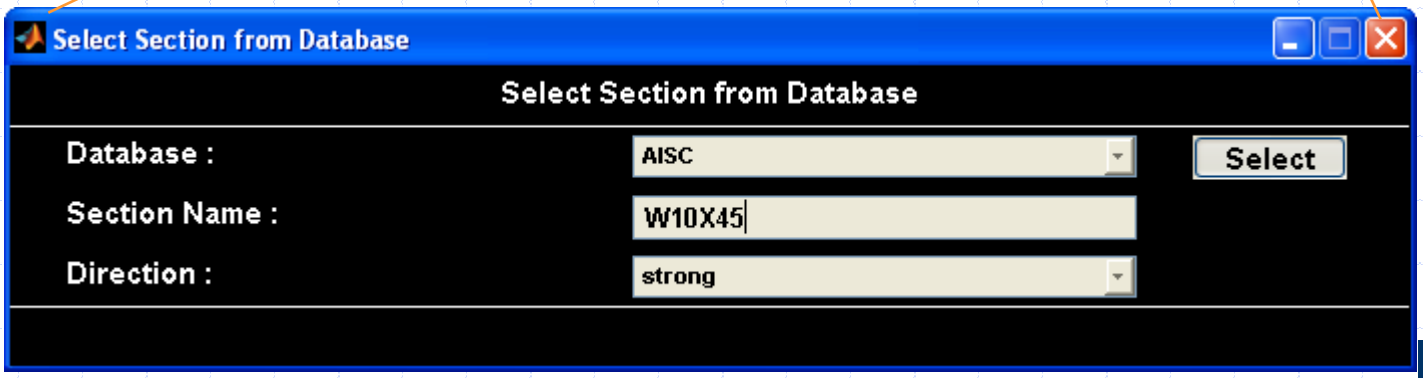
-TwoNodeLink

-ZeroLength

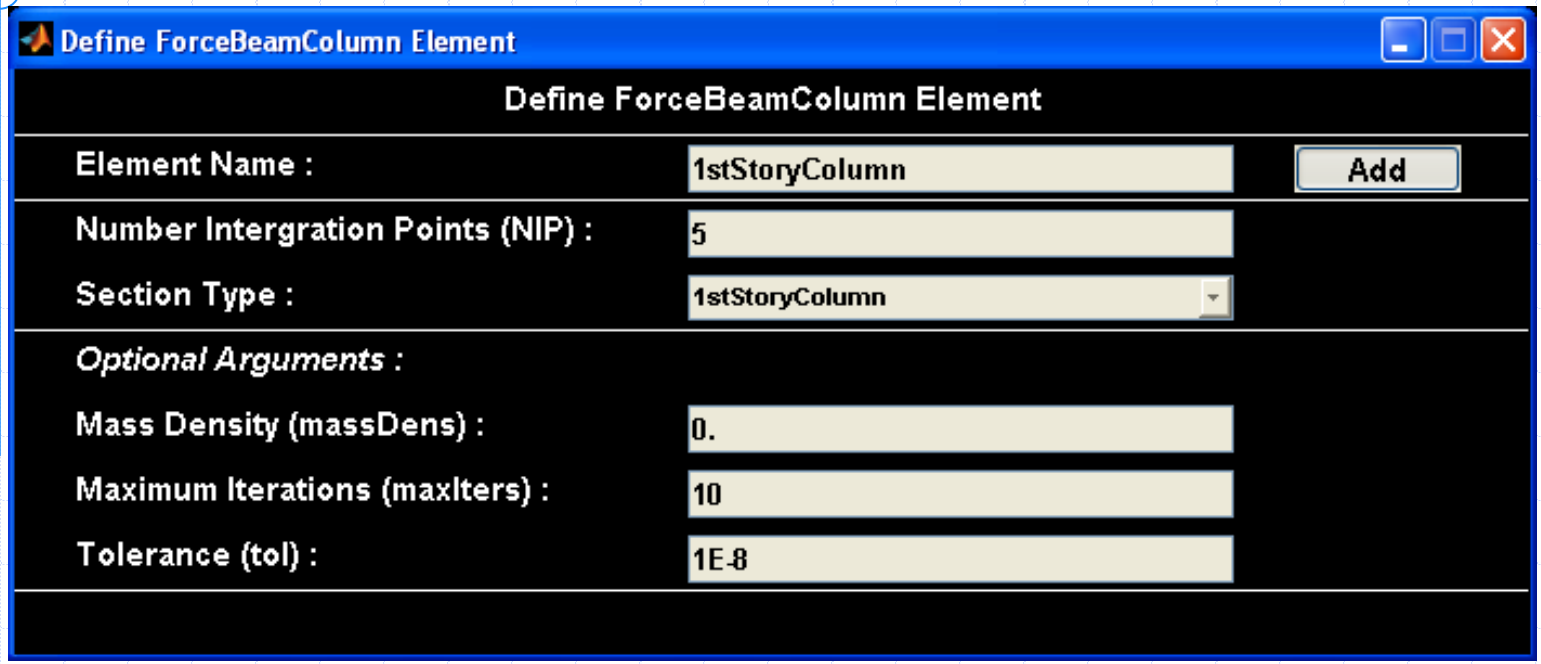
-...

Define line element: ElasticBeamColumn

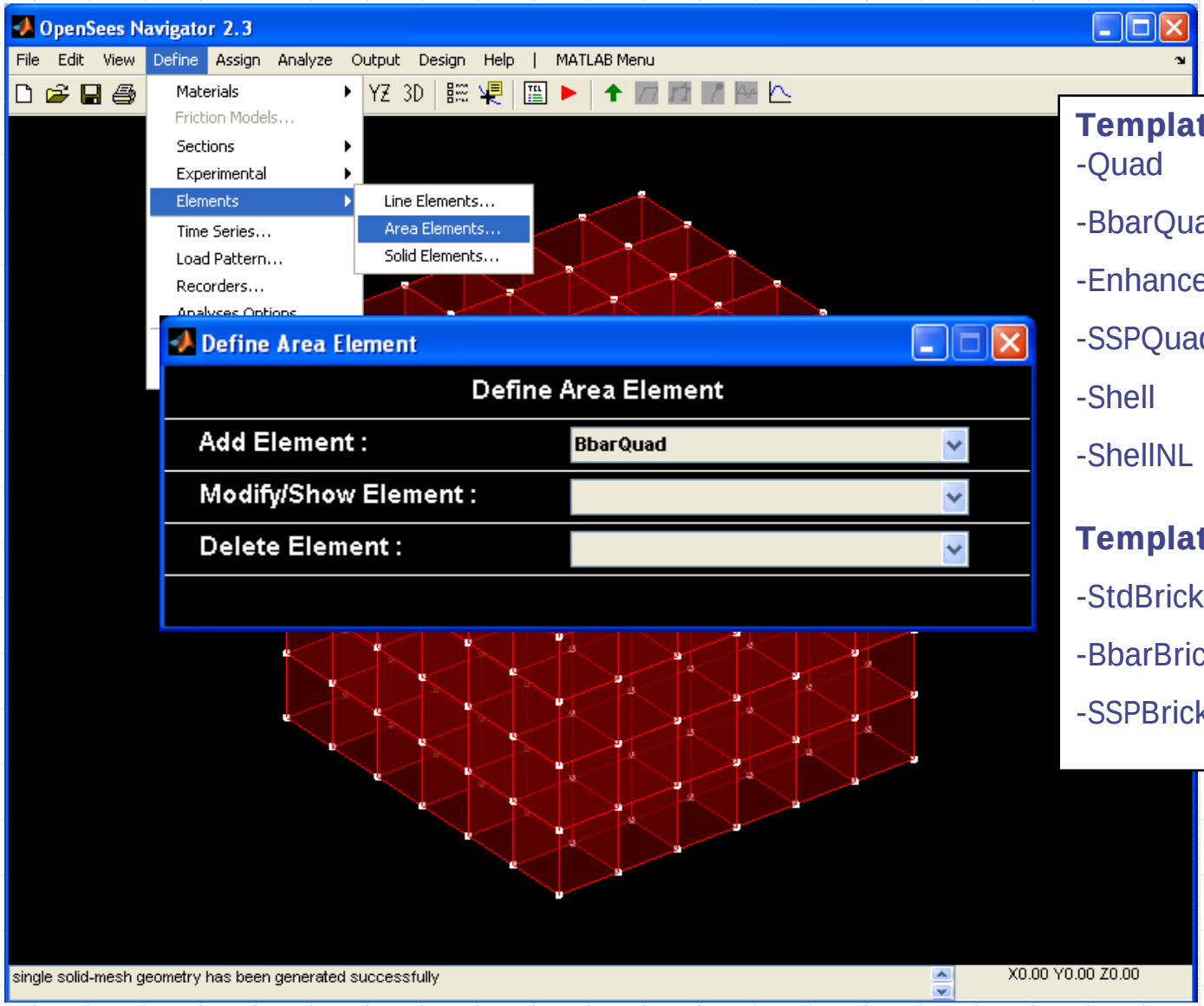




Define line element: ForceBeamColumn



Define element: area and solid elements



Templates (area):

-Quad

-BbarQuad

-EnhancedQuad

-SSPQuad

-Shell

-ShellNL

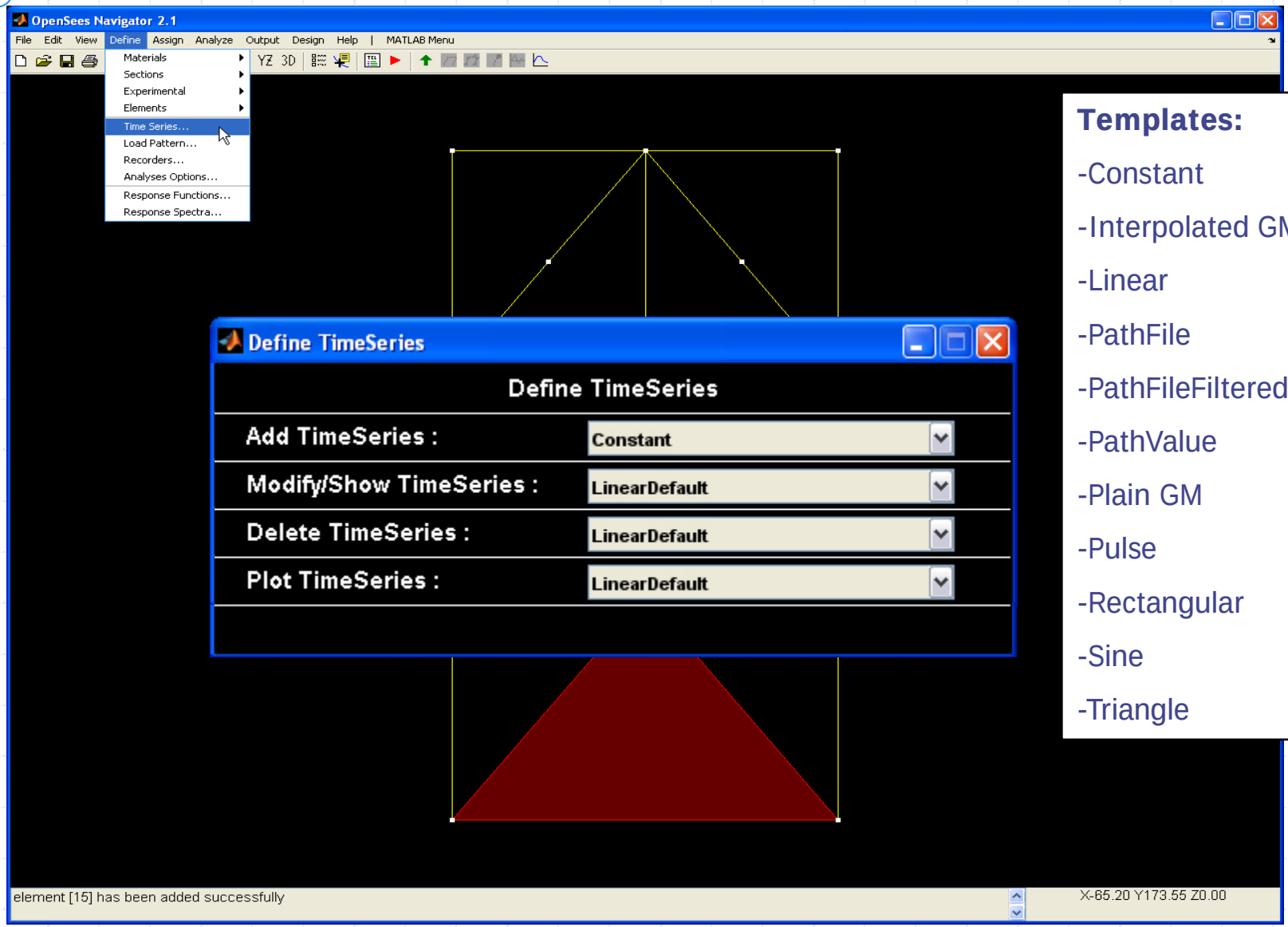
Templates (solid):

-StdBrick

-BbarBrick

-SSPBrick

Define TimeSeries:



Templates:

-Constant

-Interpolated GM

-Linear

-PathFile

-PathFileFiltered

-PathValue

-Plain GM

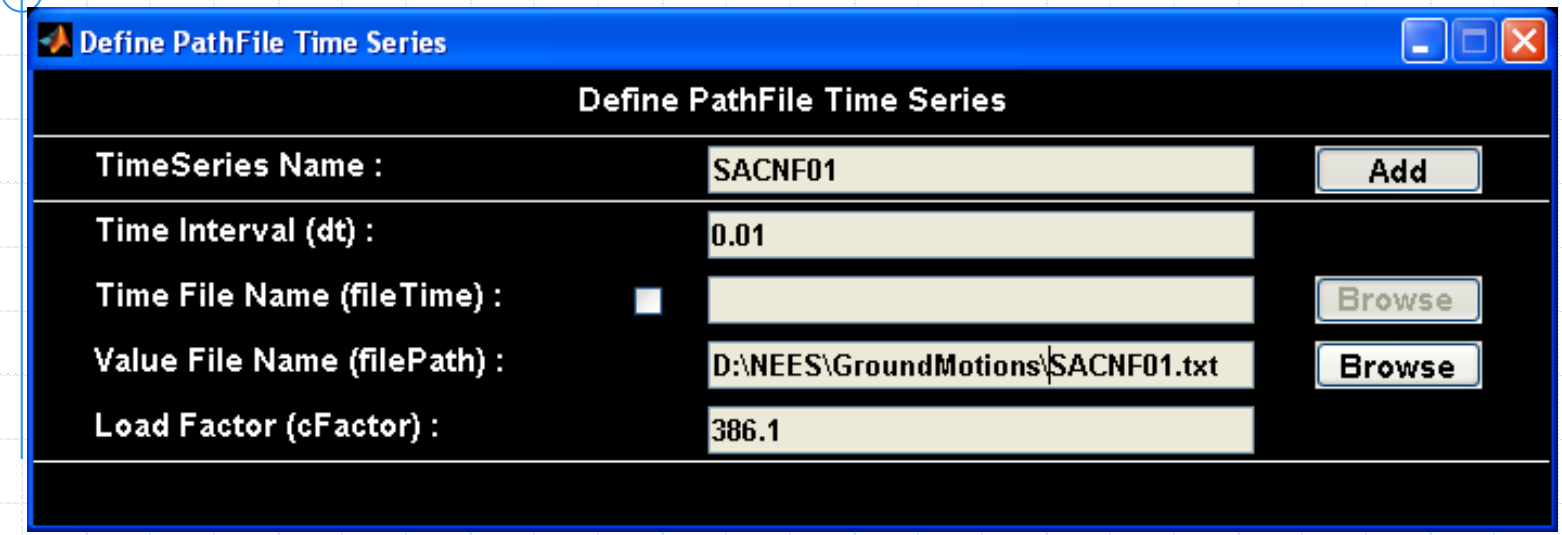
-Pulse

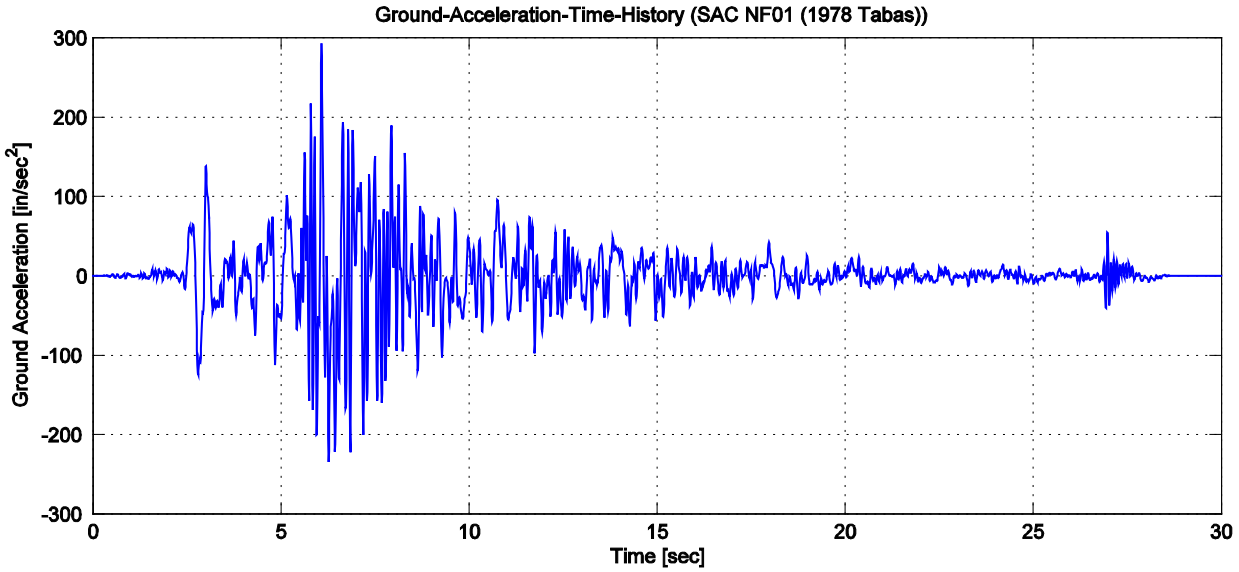
-Rectangular

-Sine

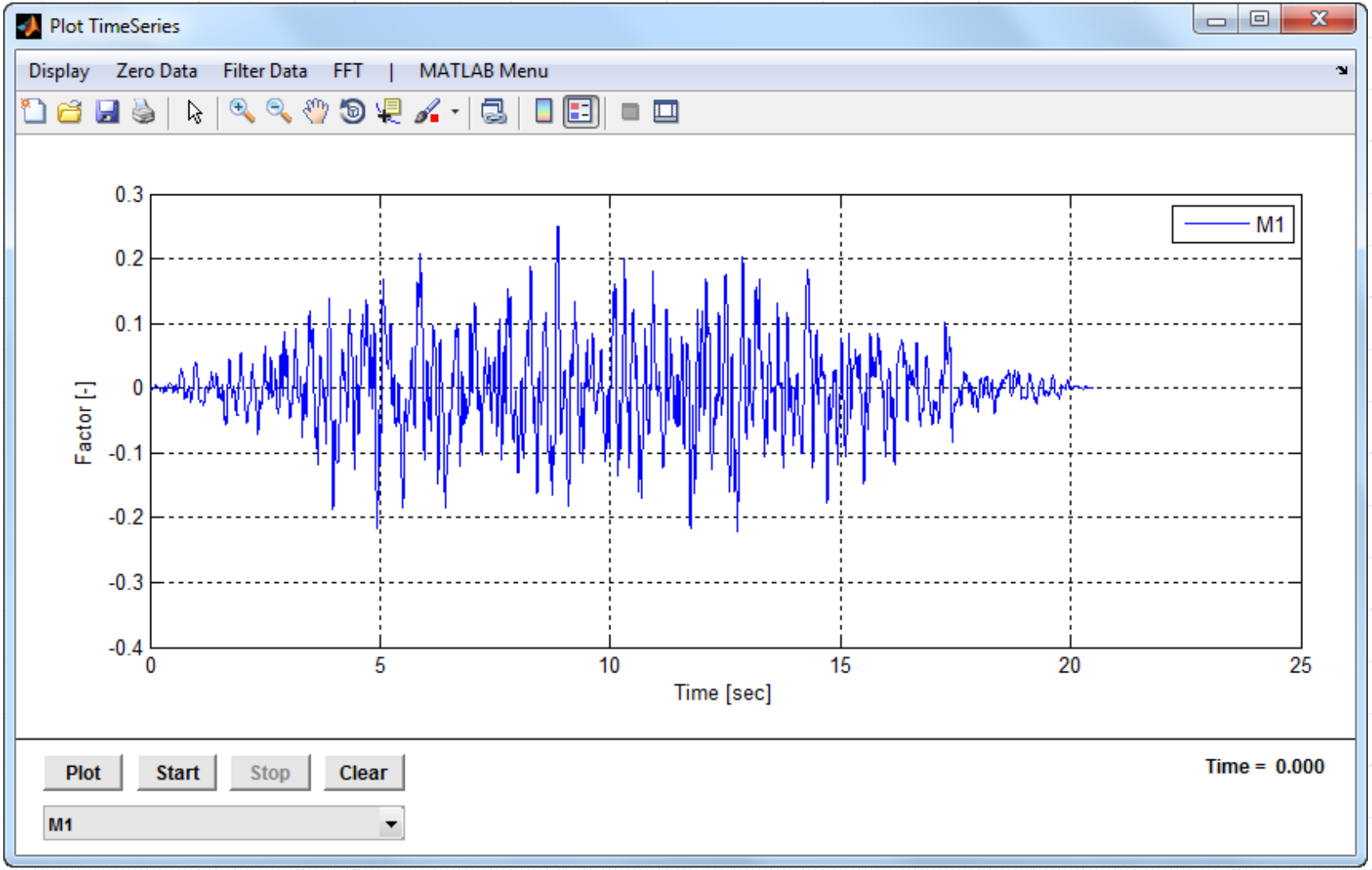
-Triangle

Define TimeSeries: PathFile

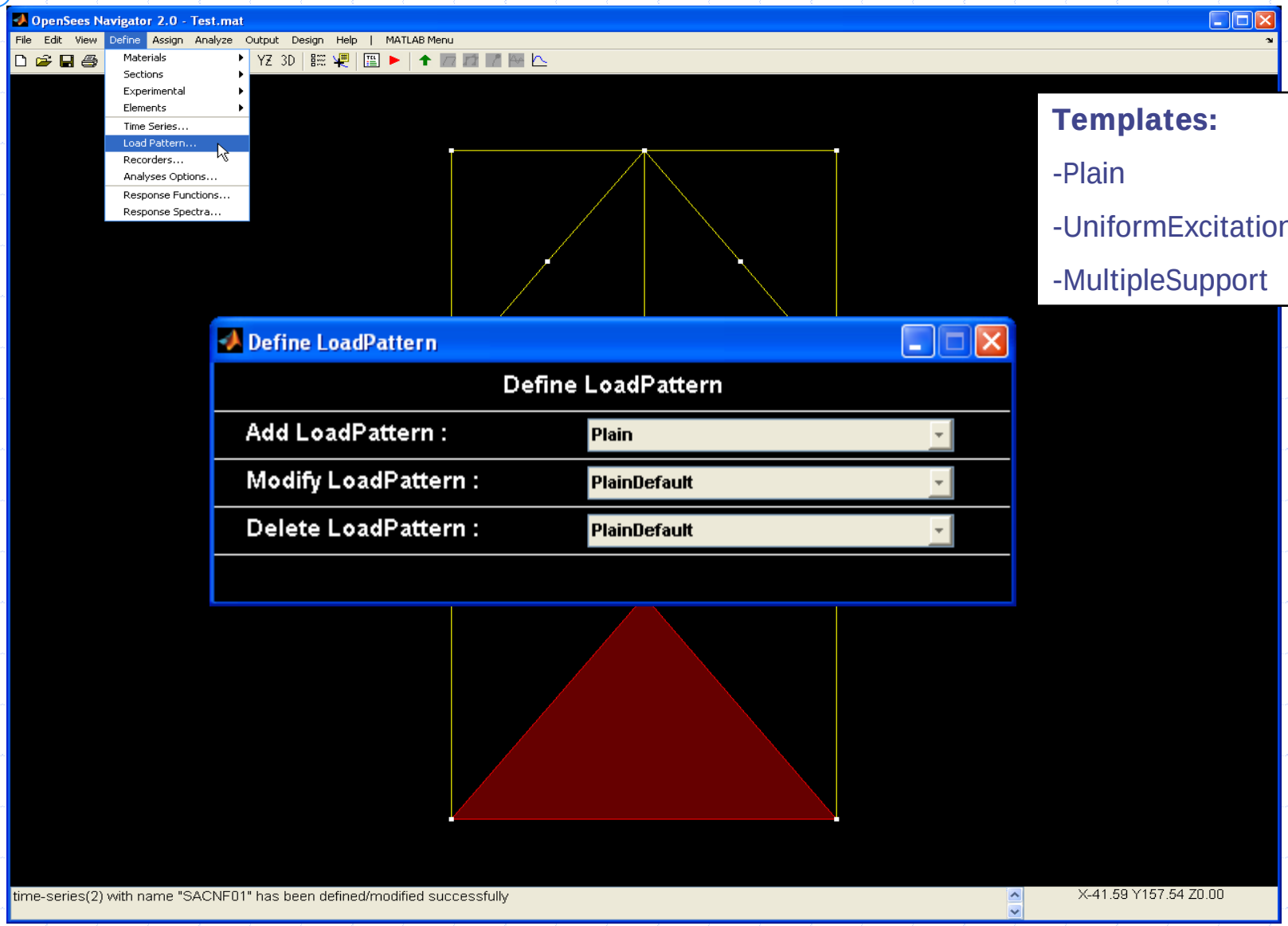




Plot TimeSeries



Define LoadPattern:



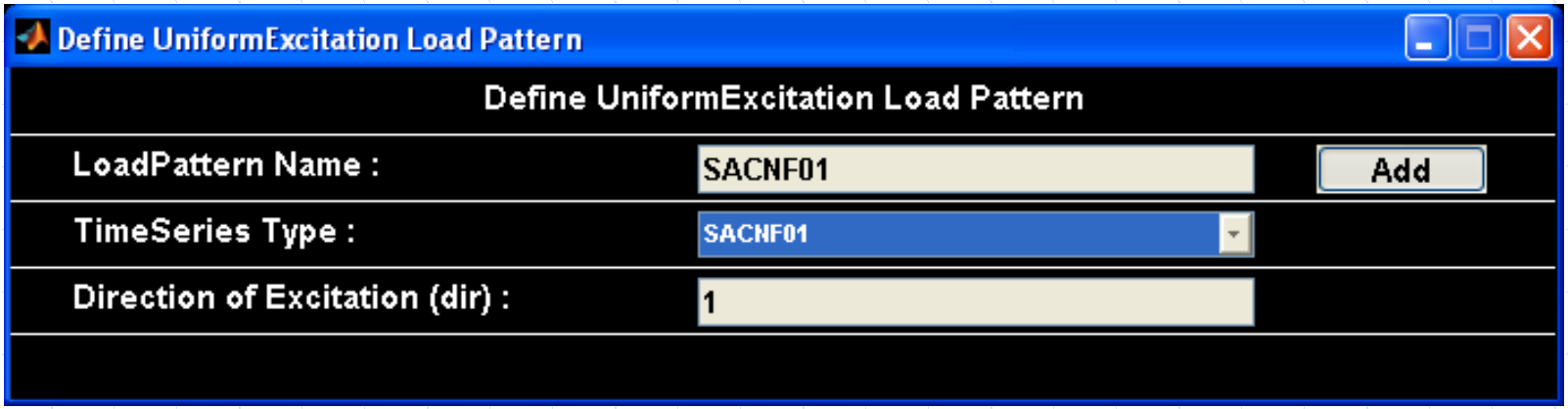
Templates:

-Plain

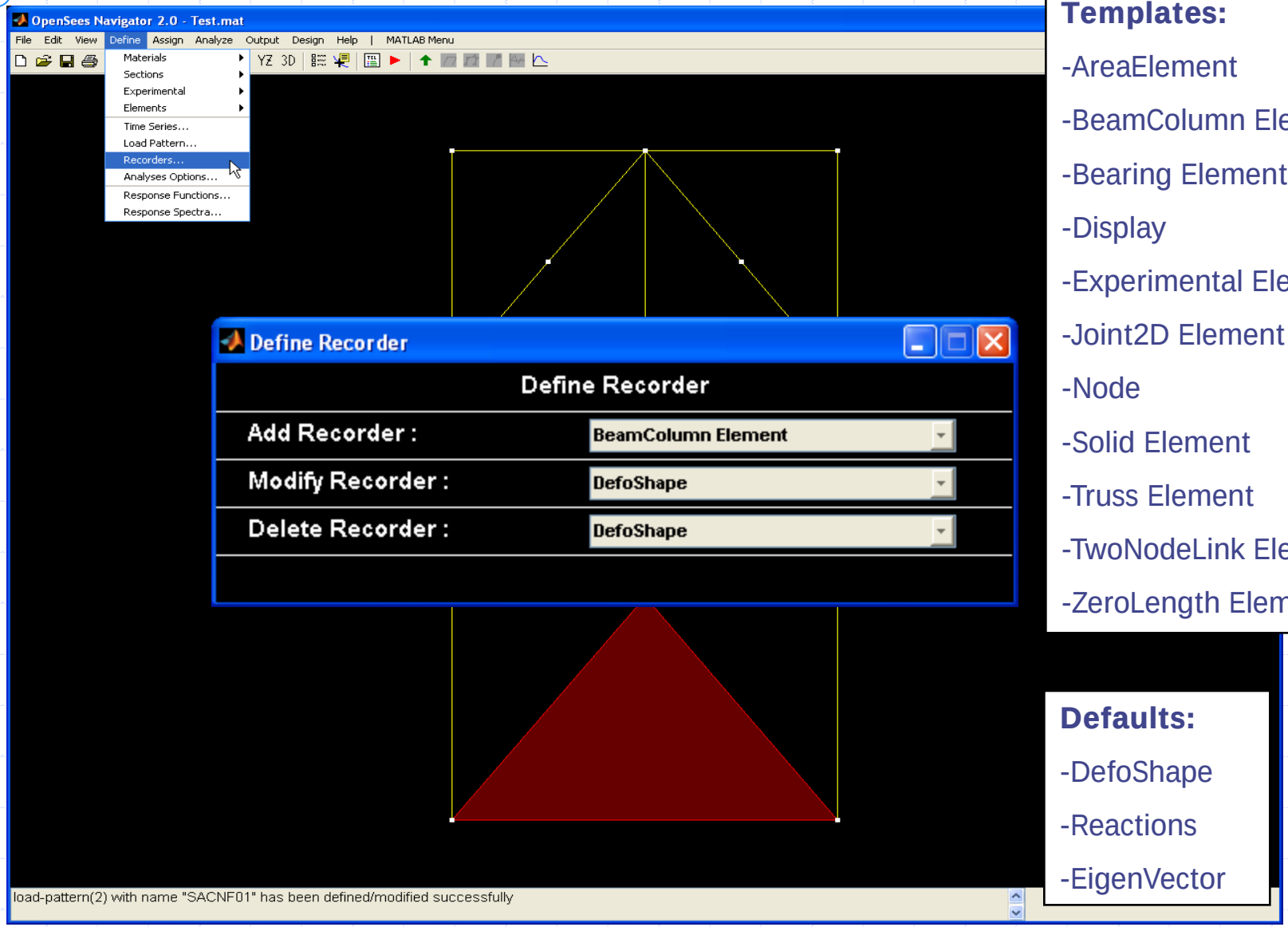
-UniformExcitation

-MultipleSupport

Define LoadPattern: UniformExcitation



Define recorder



Templates:

-AreaElement

-BeamColumn Element

-Bearing Element

-Display

-Experimental Element

-Joint2D Element

-Node

-Solid Element

-Truss Element

-TwoNodeLink Element

-ZeroLength Element

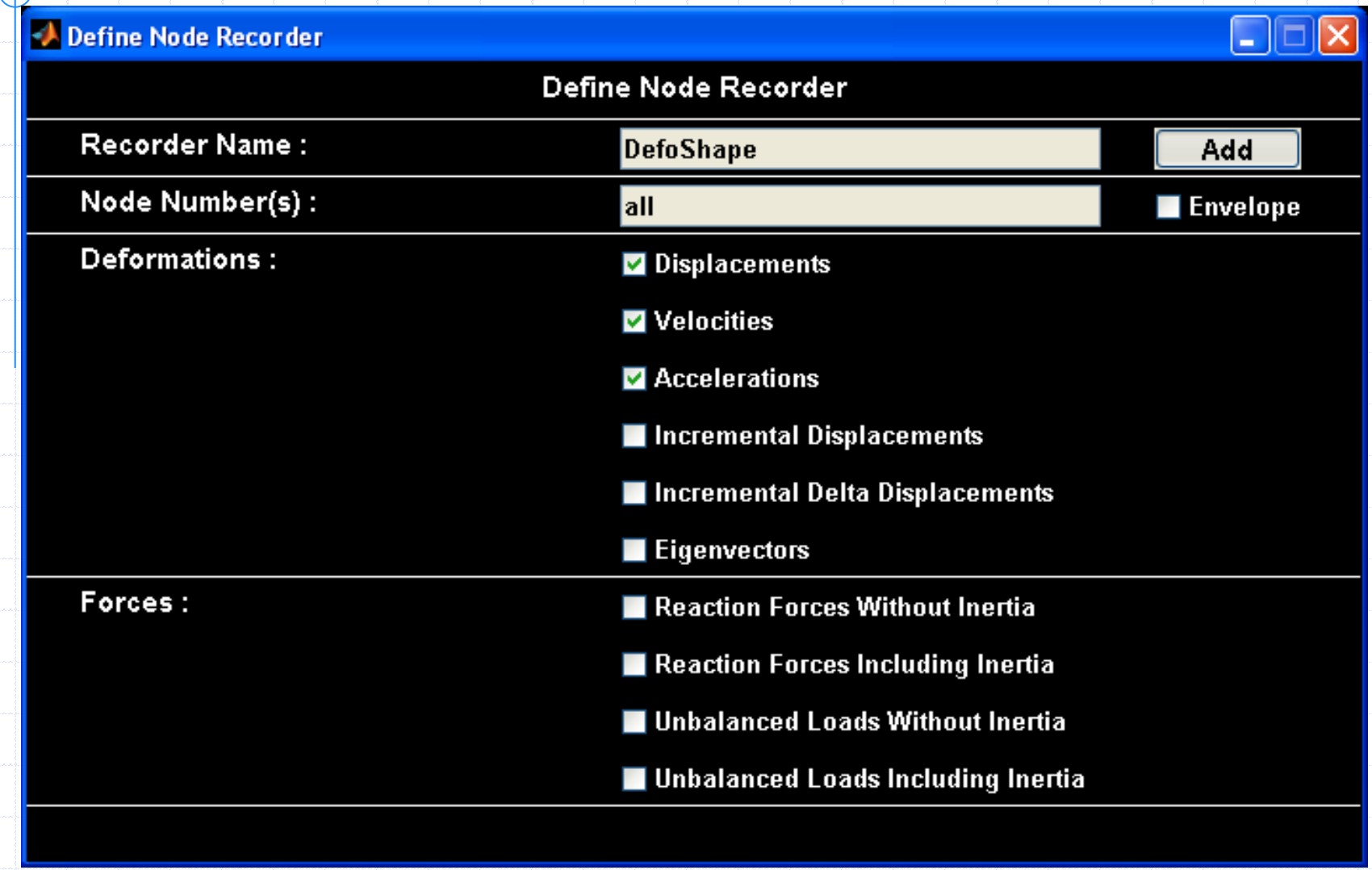
Defaults:

-DefoShape

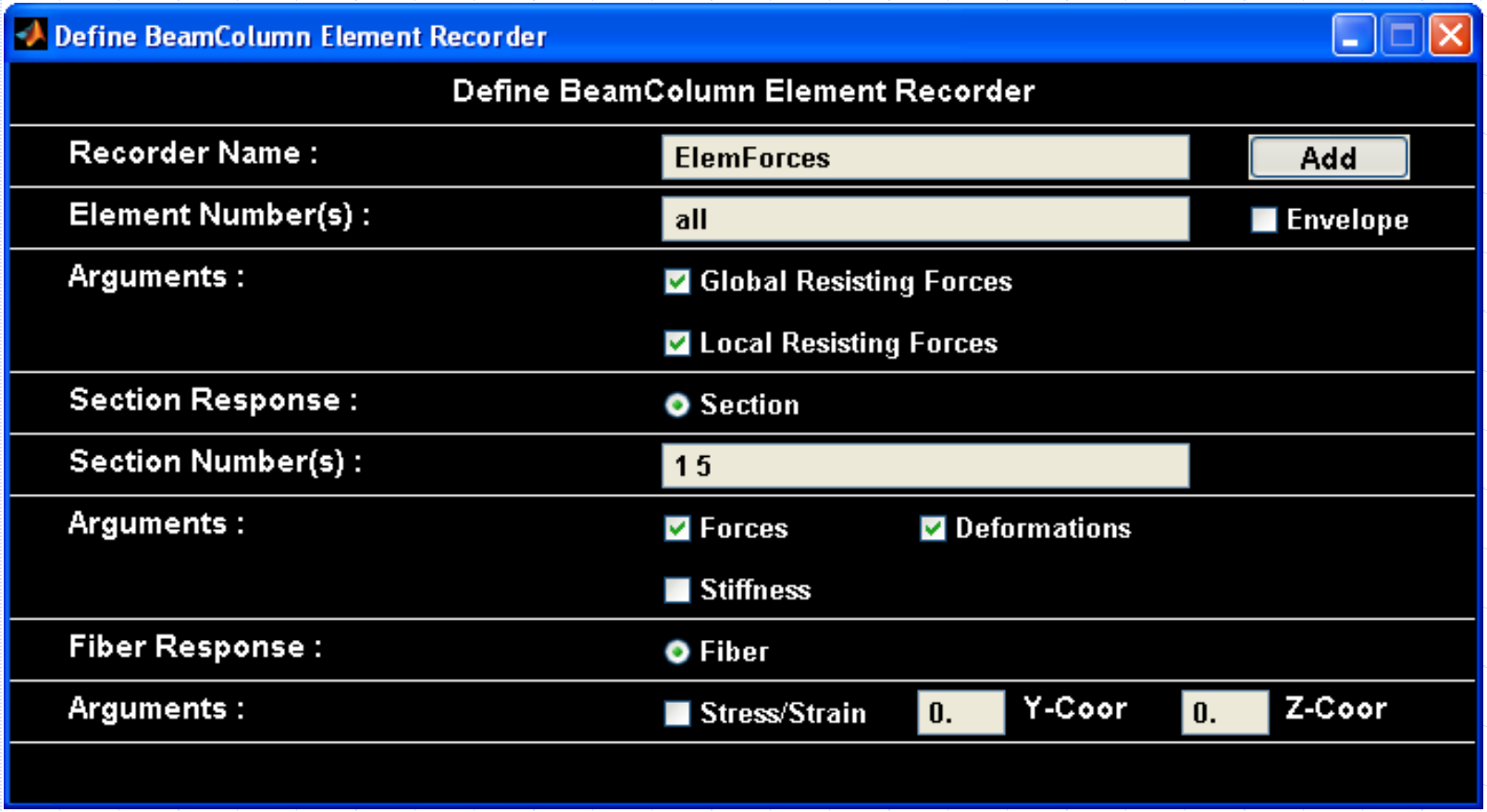
-Reactions

-EigenVector

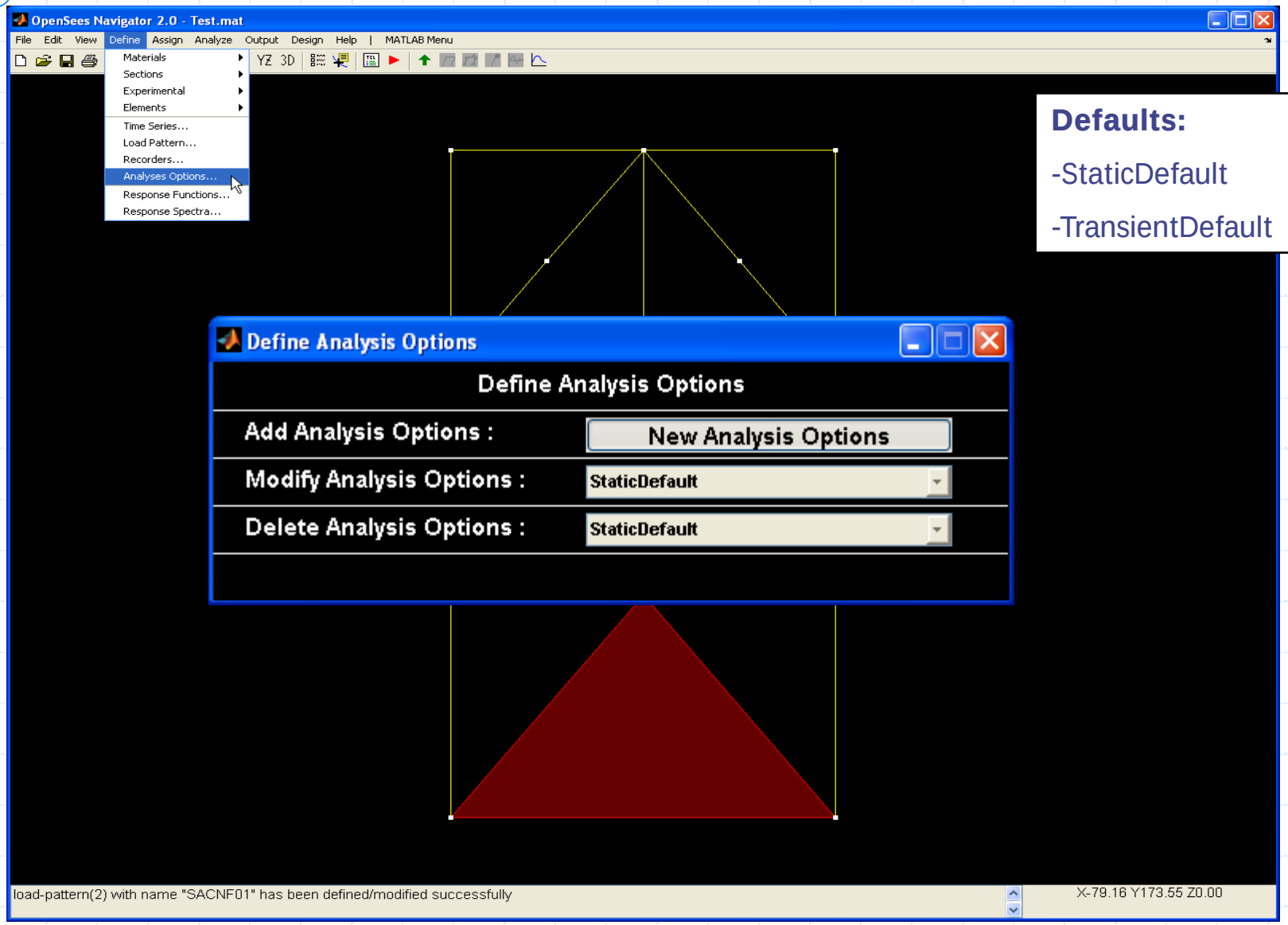
Define recorder: node recorder



Define recorder: BeamColumn recorder



Define analysis options

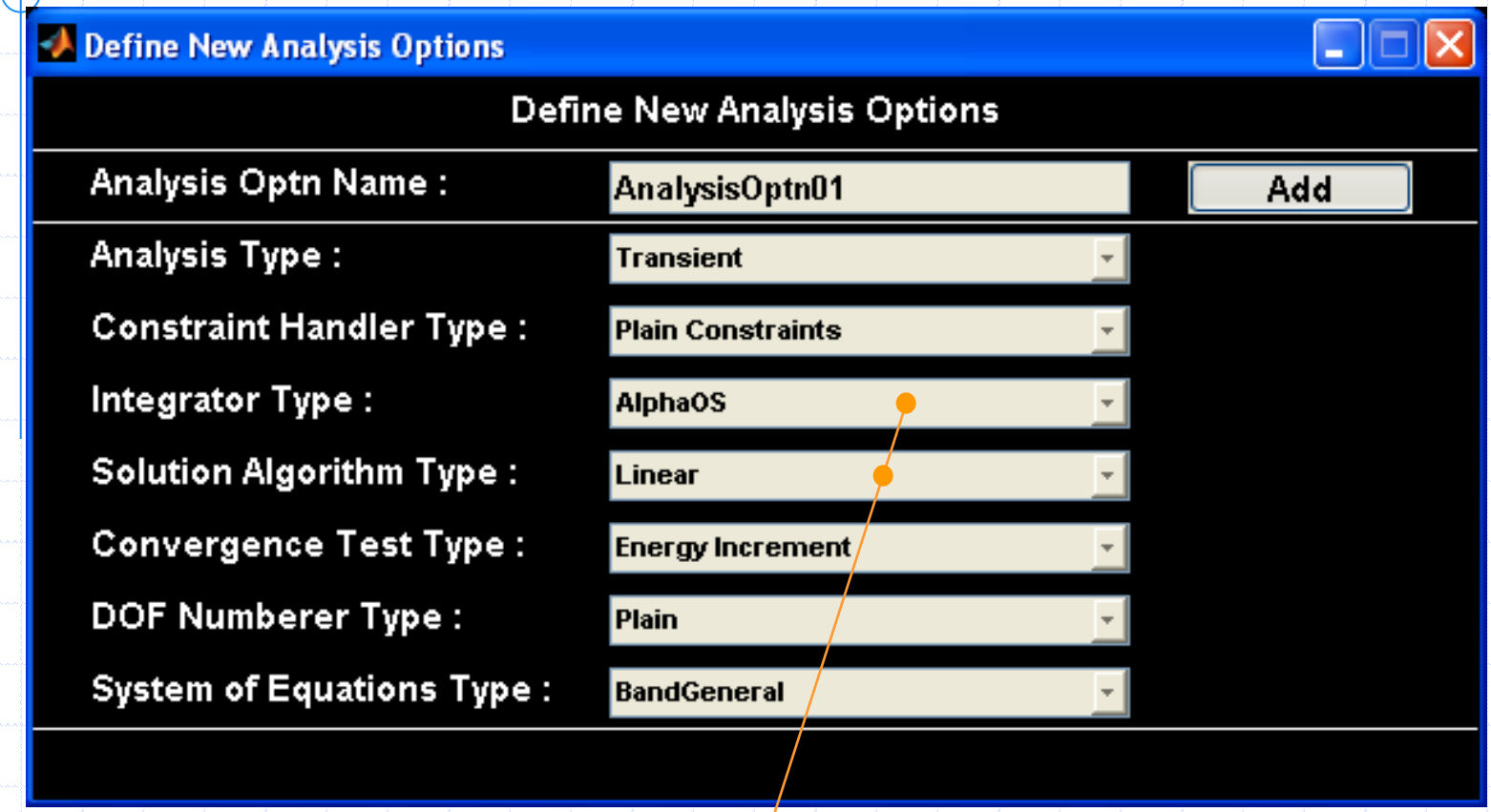


Defaults:

-StaticDefault

-TransientDefault

Define analysis options: new analysis



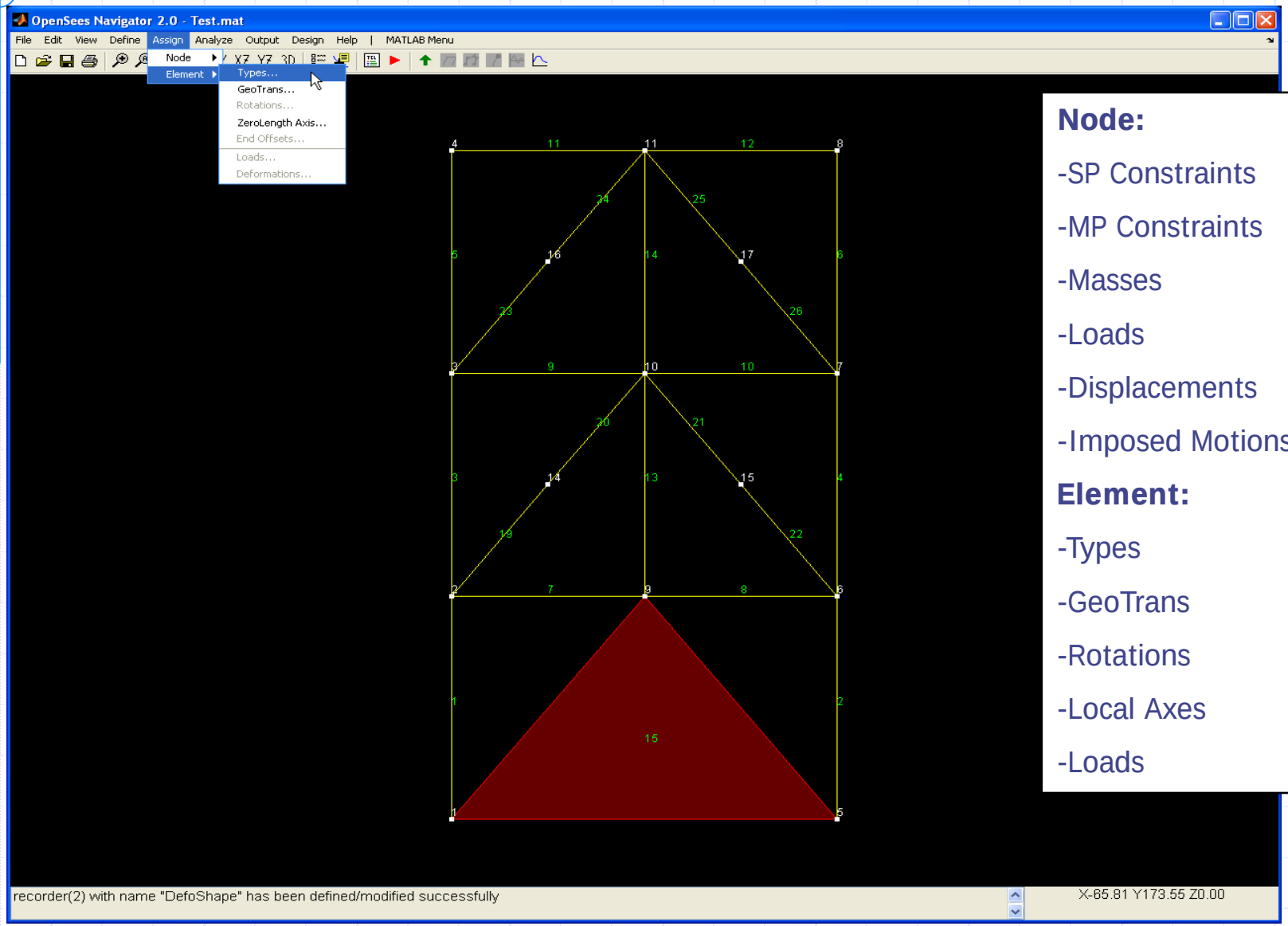
Integrator Type:

For example use AlphaOS Method for Hybrid Simulation

Solution Algorithm:

The AlphaOS Method requires a Linear solution algorithm

Assign menu



Node:

-SP Constraints

-MP Constraints

-Masses

-Loads

-Displacements

-Imposed Motions

Element:

-Types

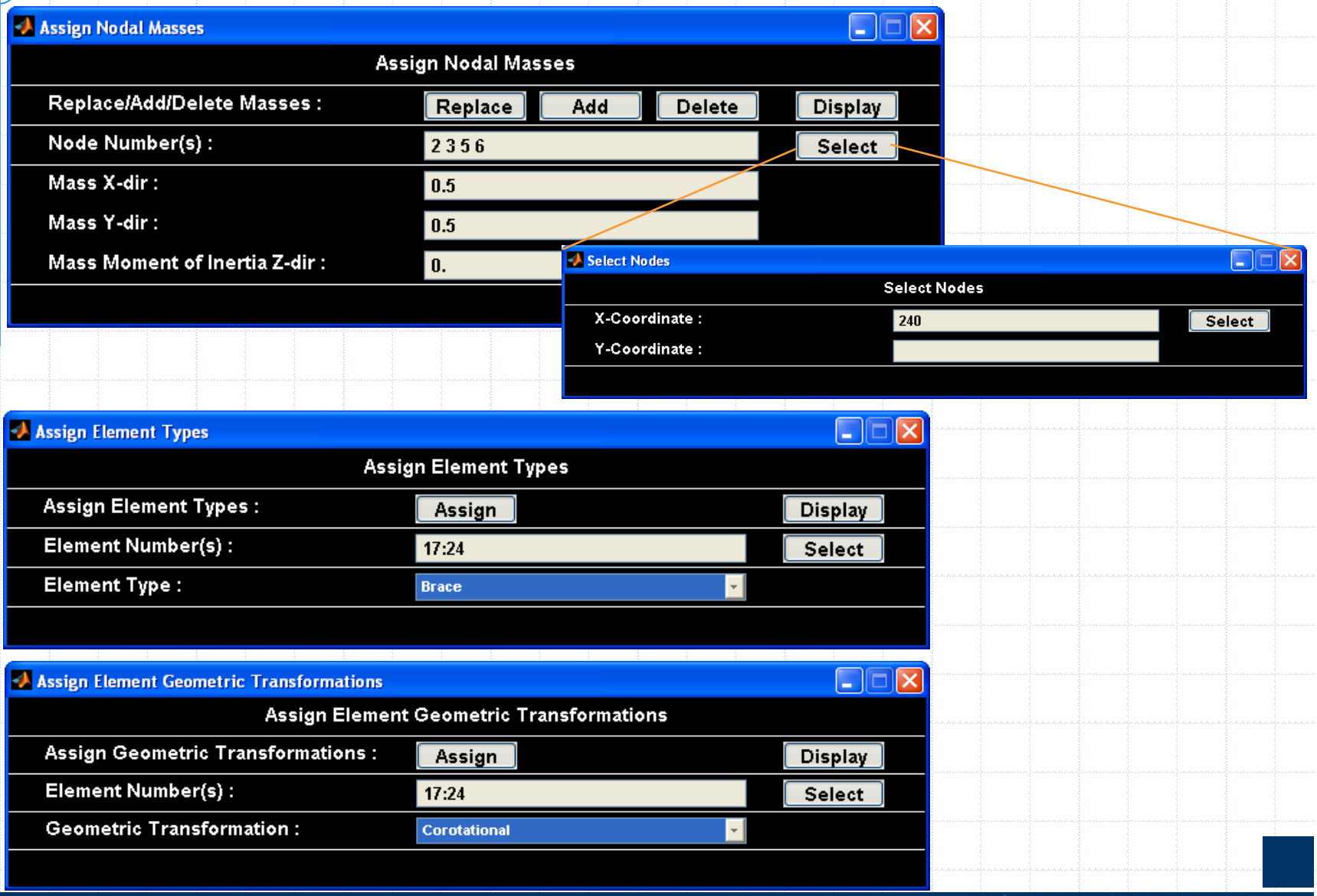
-GeoTrans

-Rotations

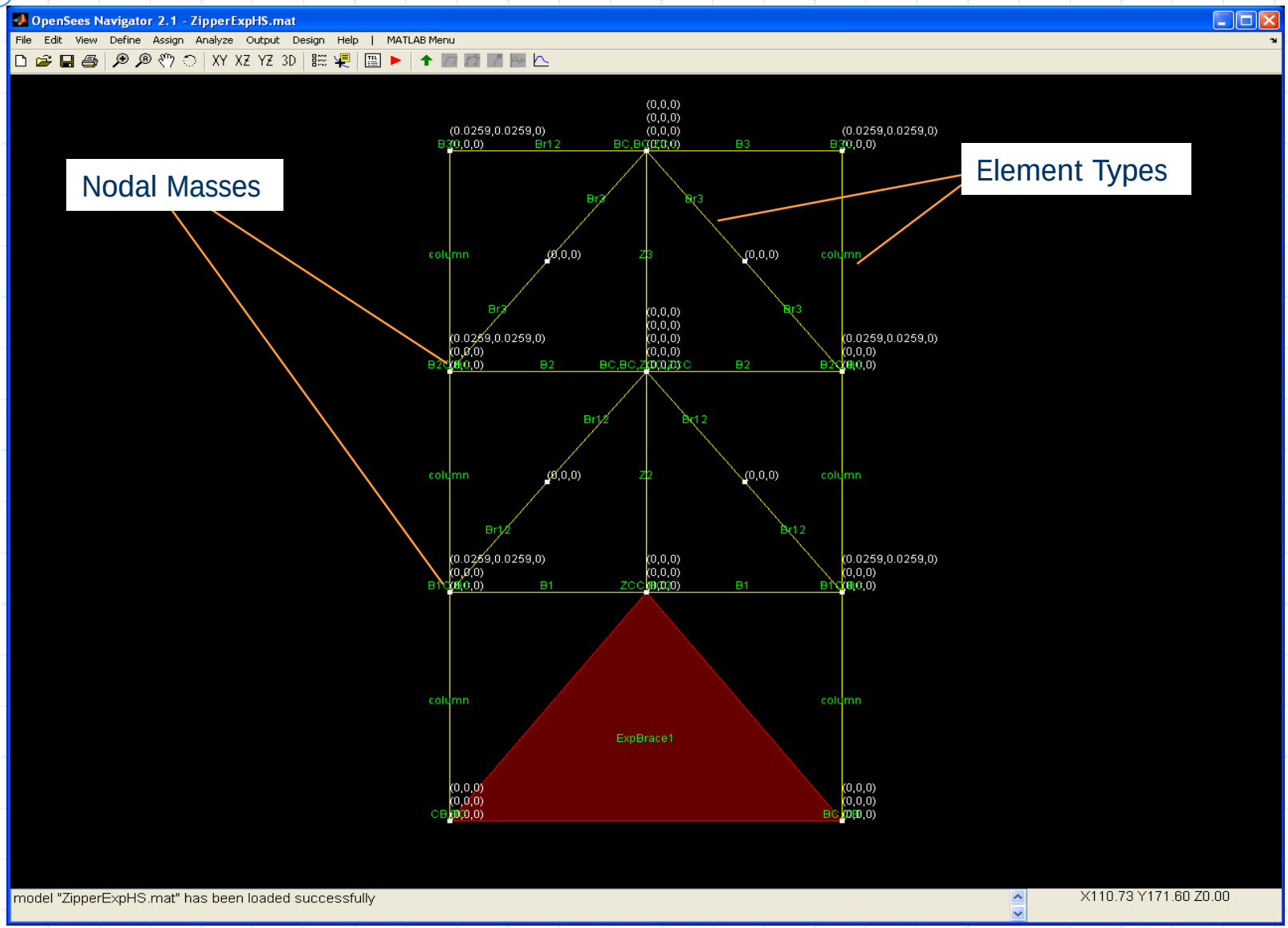
-Local Axes

-Loads

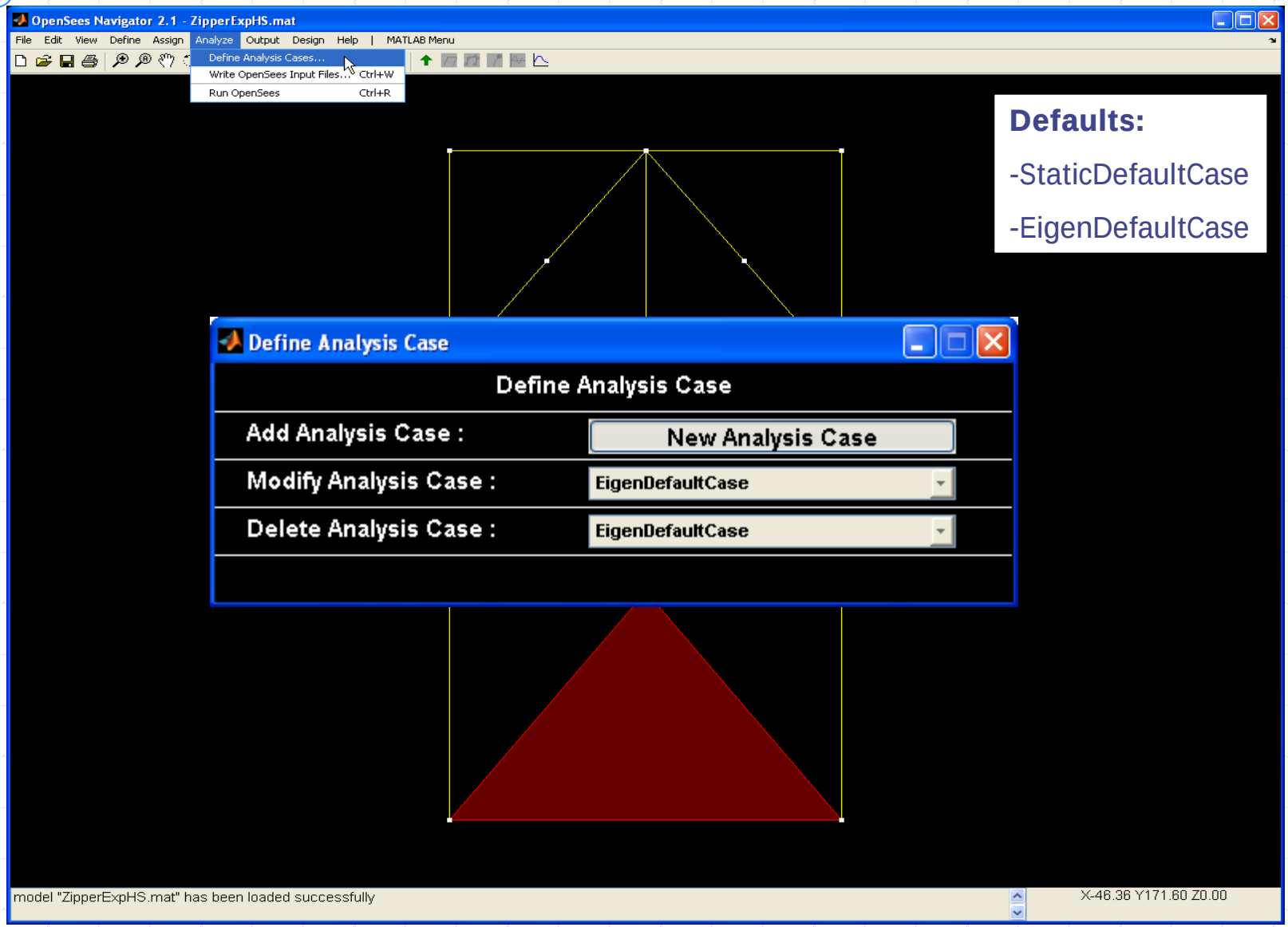
Assign menu


Assign Nodal Masses
Assign Nodal Masses
Replace/Add/Delete Masses :
Replace
Add
Delete
Display
Node Number(s) :
2 3 5 6
Select
Mass X-dir :
0.5
Mass Y-dir :
0.5
Mass Moment of Inertia Z-dir :
0.
Select Nodes
Select Nodes
X-Coordinate :
240
Select
Y-Coordinate :
Assign Element Types
Assign Element Types
Assign Element Types :
Assign
Display
Element Number(s) :
17:24
Select
Element Type :
Brace
Assign Element Geometric Transformations
Assign Element Geometric Transformations
Assign Geometric Transformations :
Assign
Display
Element Number(s) :
17:24
Select
Geometric Transformation :
Corotational

Display assigned properties



Define analysis case

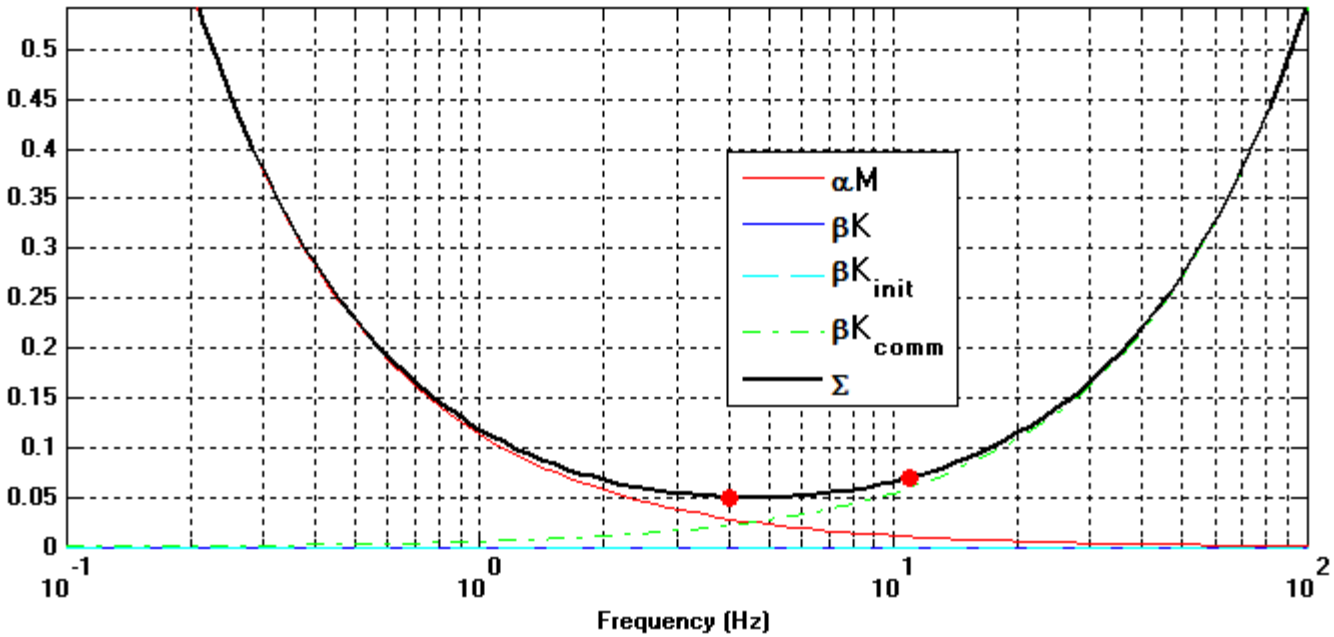


De

Define New Damping Parameters

Define New Damping Parameters

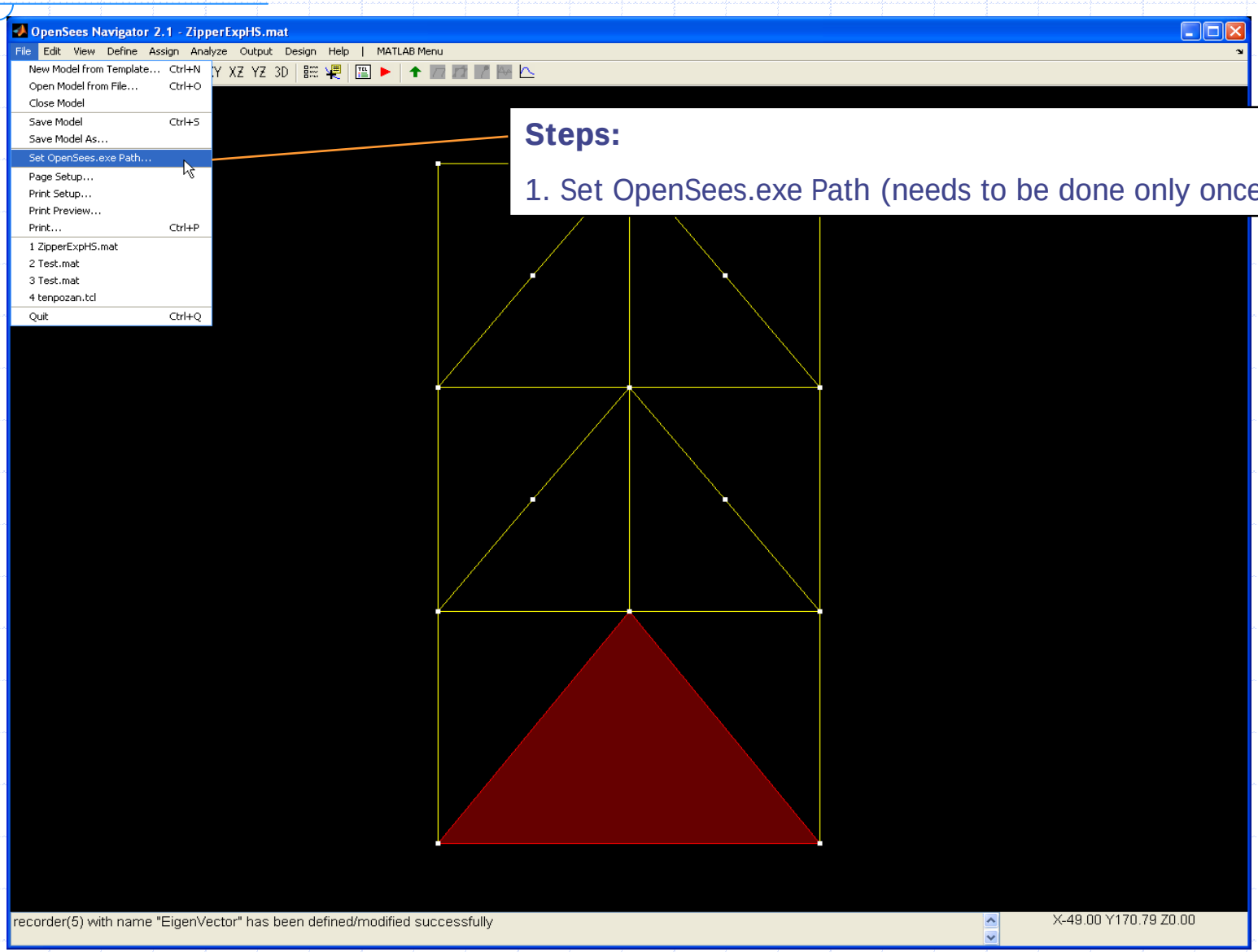
Damping Parameter Set Name :	DampingParam01	Add
Region Defined by :	Nodes	
Node or Element Number(s) :	all	
Mass Prop. Damping (alphaM) :	1.4218	Calculate
Kcurr Prop. Damping (betaK) :	0.	
Kinit Prop. Damping (betaKinit) :	0.	
Kcom Prop. Damping (betaKcom) :	0.001728	



User Defined Analysis Script

```
# set the test parameters
set testType NormDispIncr
set testTol 1.0e-8;
set testIter 25;
test  $testType $testTol $testIter

# set the algorithm parameters
set algoType KrylovNewton
algorithm  $algoType

set ok 0;
set tFinal [expr $numSteps * $dt]
set tCurrent [getTime]

while {$ok == 0 && $tCurrent < $tFinal} {
   if {fmod($tCurrent,1) < 1.0E-16} {
          puts "$i $tCurrent"
       }
       set ok [analyze 1 $dt]

   if {$ok != 0} {
      puts " "
      puts [format "KrylovNewton failed (time = %1.3e), try Newton" $tCurrent]
            algorithm Newton
      test $testType $testTol $testIter 0
      set ok [analyze 1 $dt]
      algorithm $algoType
   }

   if {$ok != 0} {
      puts " "
            puts [format "Newton failed (time = %1.3e), try Newton  w/ iniCurrent" $tCurrent]
            algorithm Newton -initialCurrent
      test $testType $testTol $testIter 0
      set ok [analyze 1 $dt]
      algorithm $algoType
   }

     if {$ok != 0} {
      puts " "
            puts [format "Newton  w/ iniCurrent failed (time = %1.3e), try Newton  w/ ini" $tCurrent]
            algorithm Newton -initial
      test $testType $testTol [expr 500 * $testIter] 0
      set ok [analyze 1 $dt]
      algorithm $algoType
      test $testType $testTol $testIter 0
```

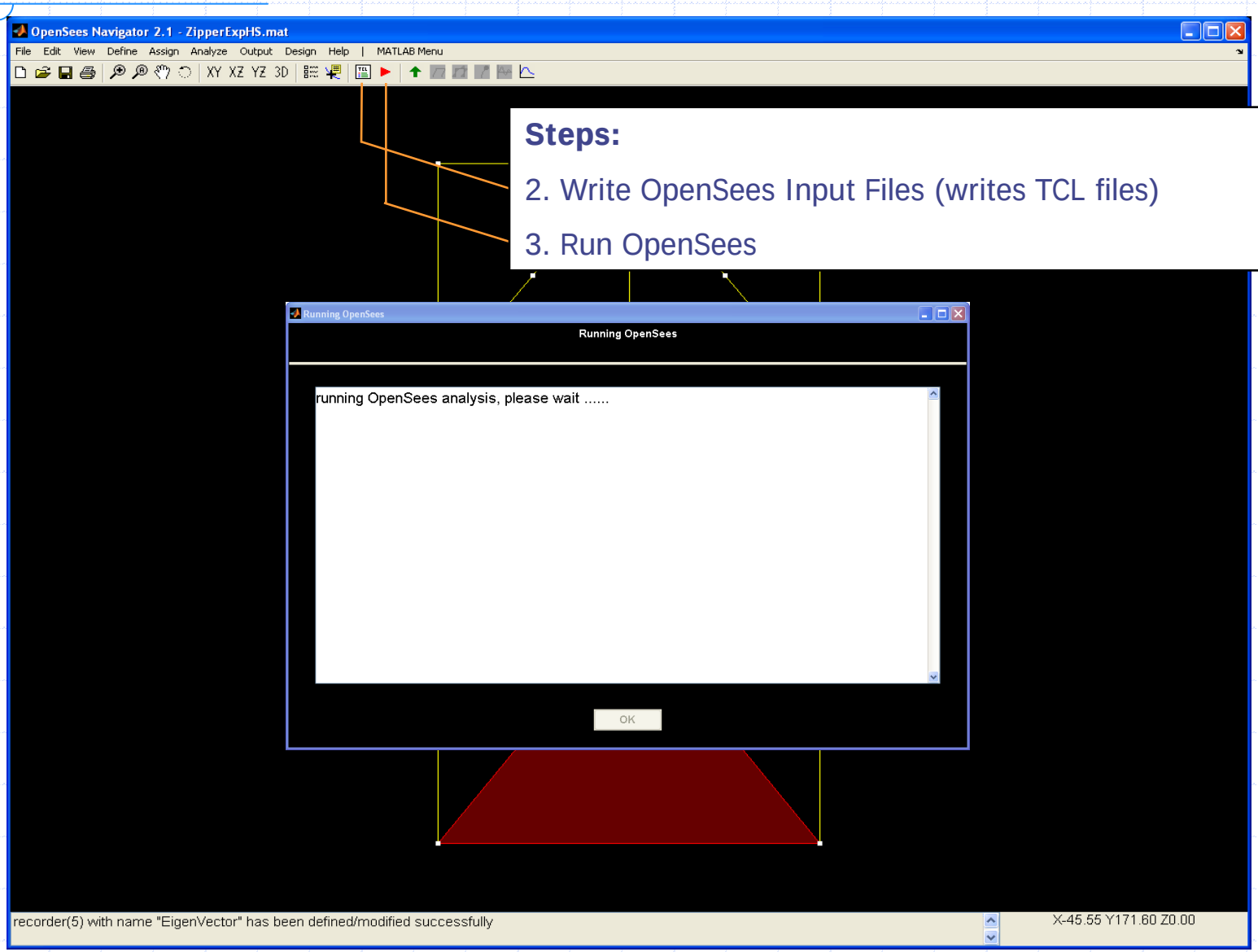
Run OpenSees: set OpenSees.exe path



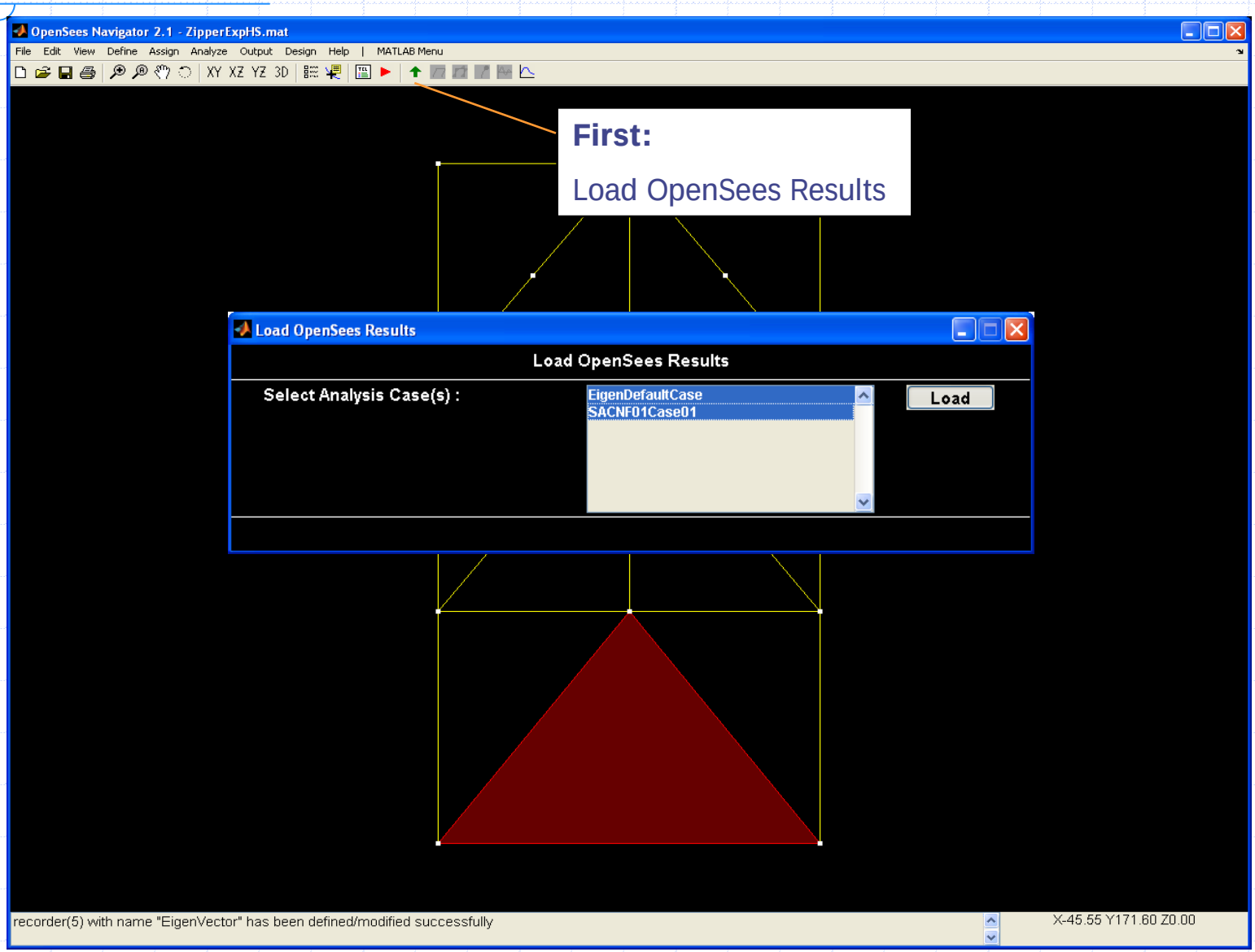
Steps:

1. Set OpenSees.exe Path (needs to be done only once)

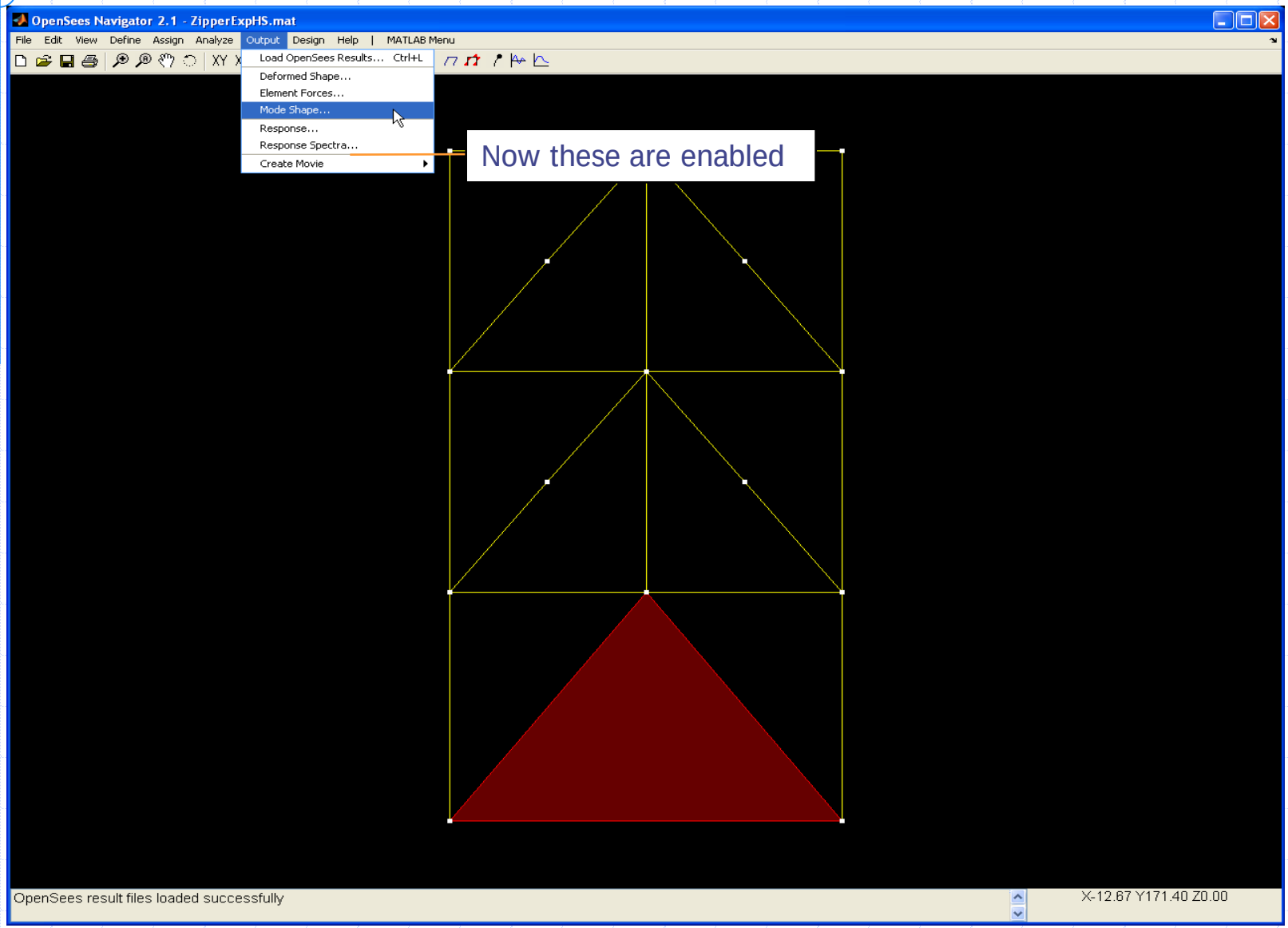
Run OpenSees: write TCL files



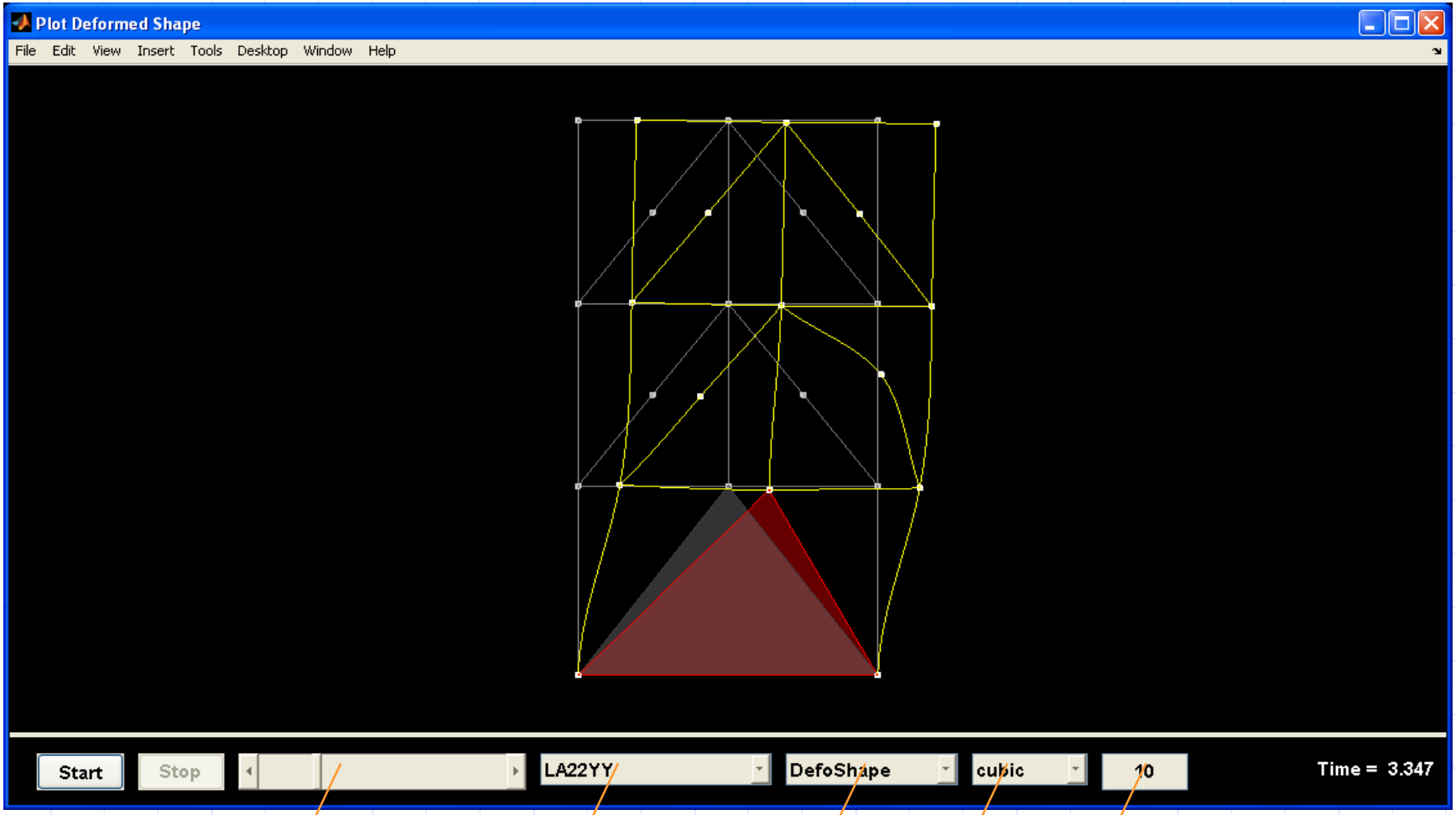
Post processing: load results



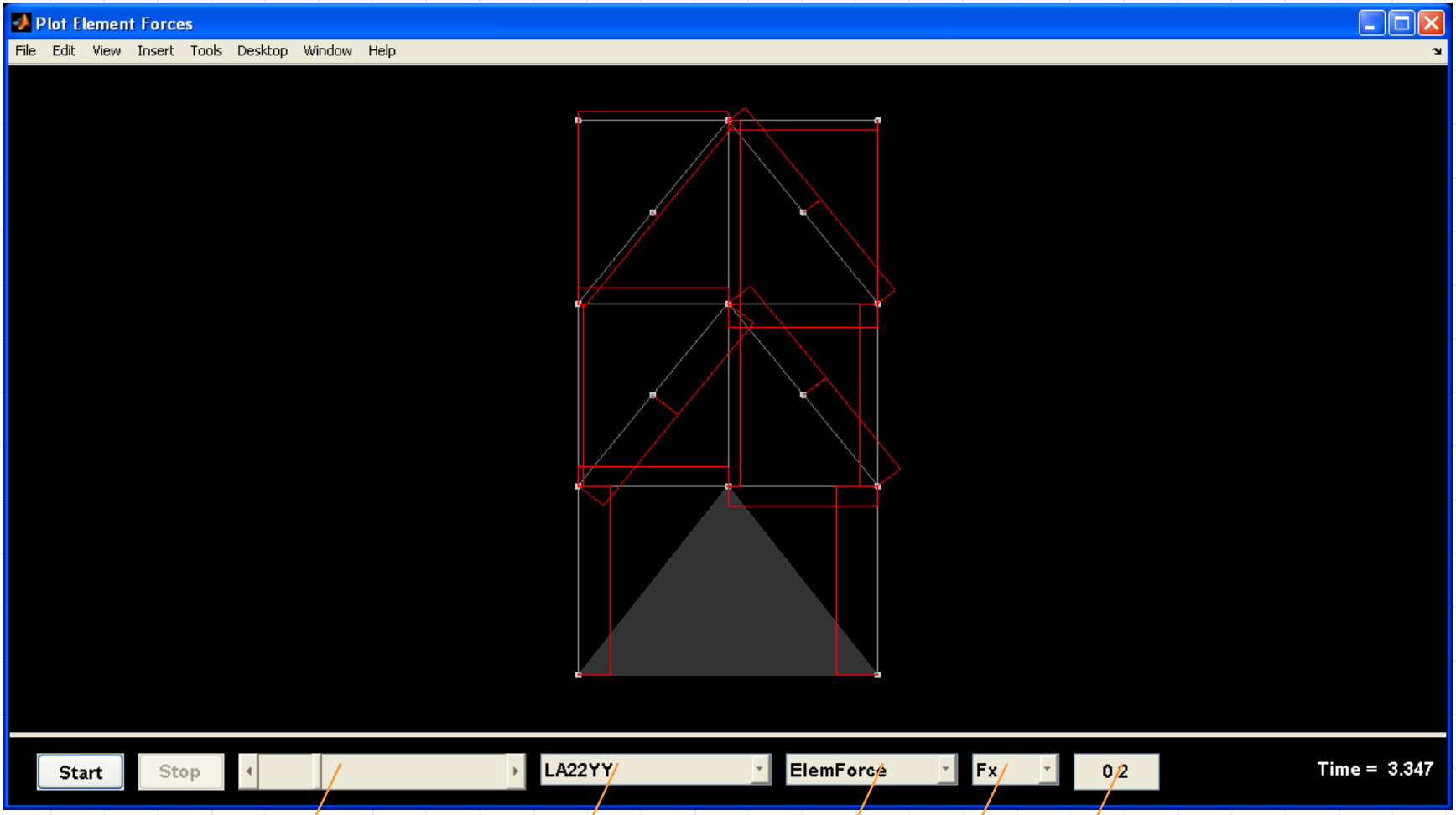
Post processing: output



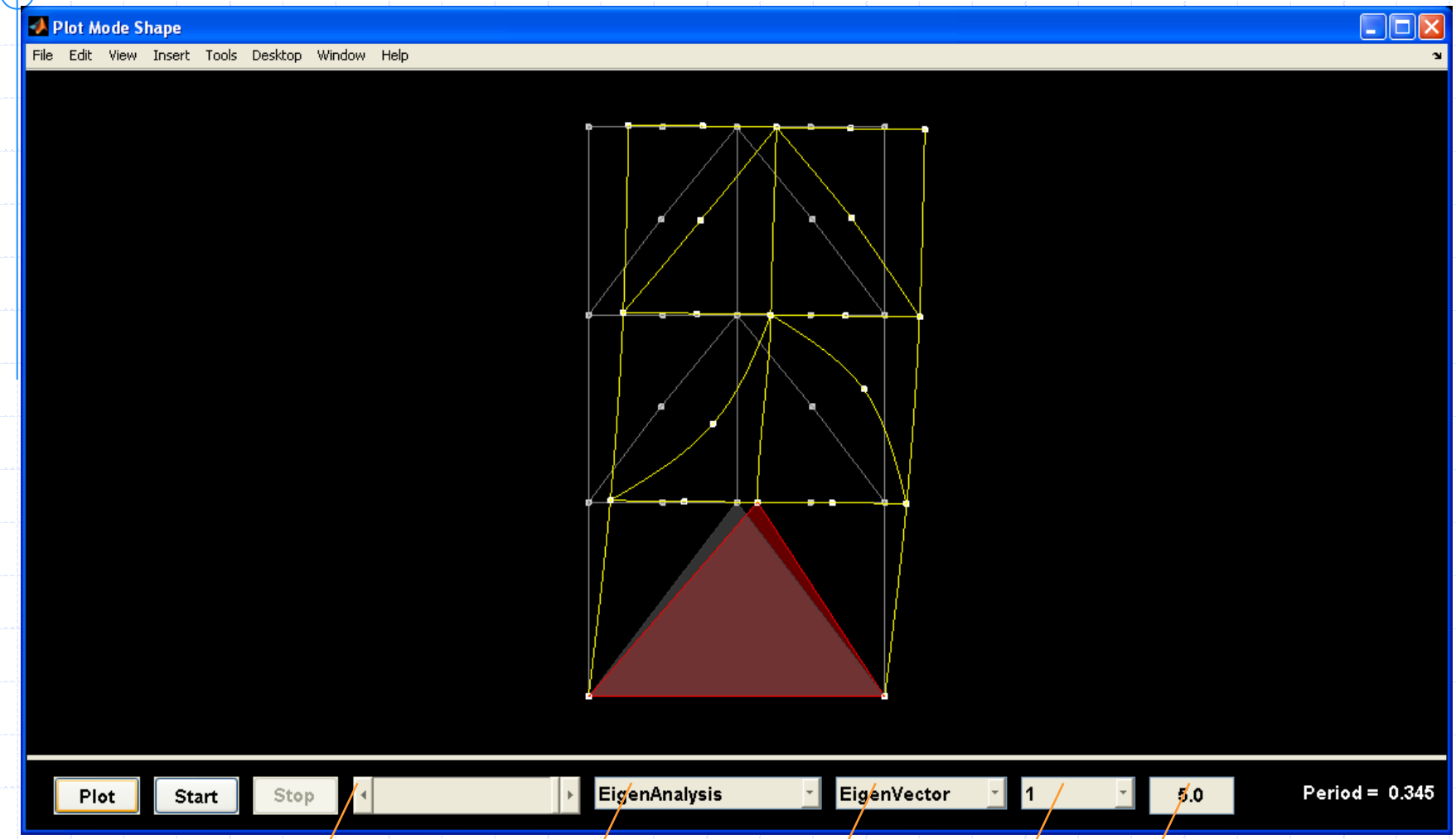
Post processing: plot deformed shape



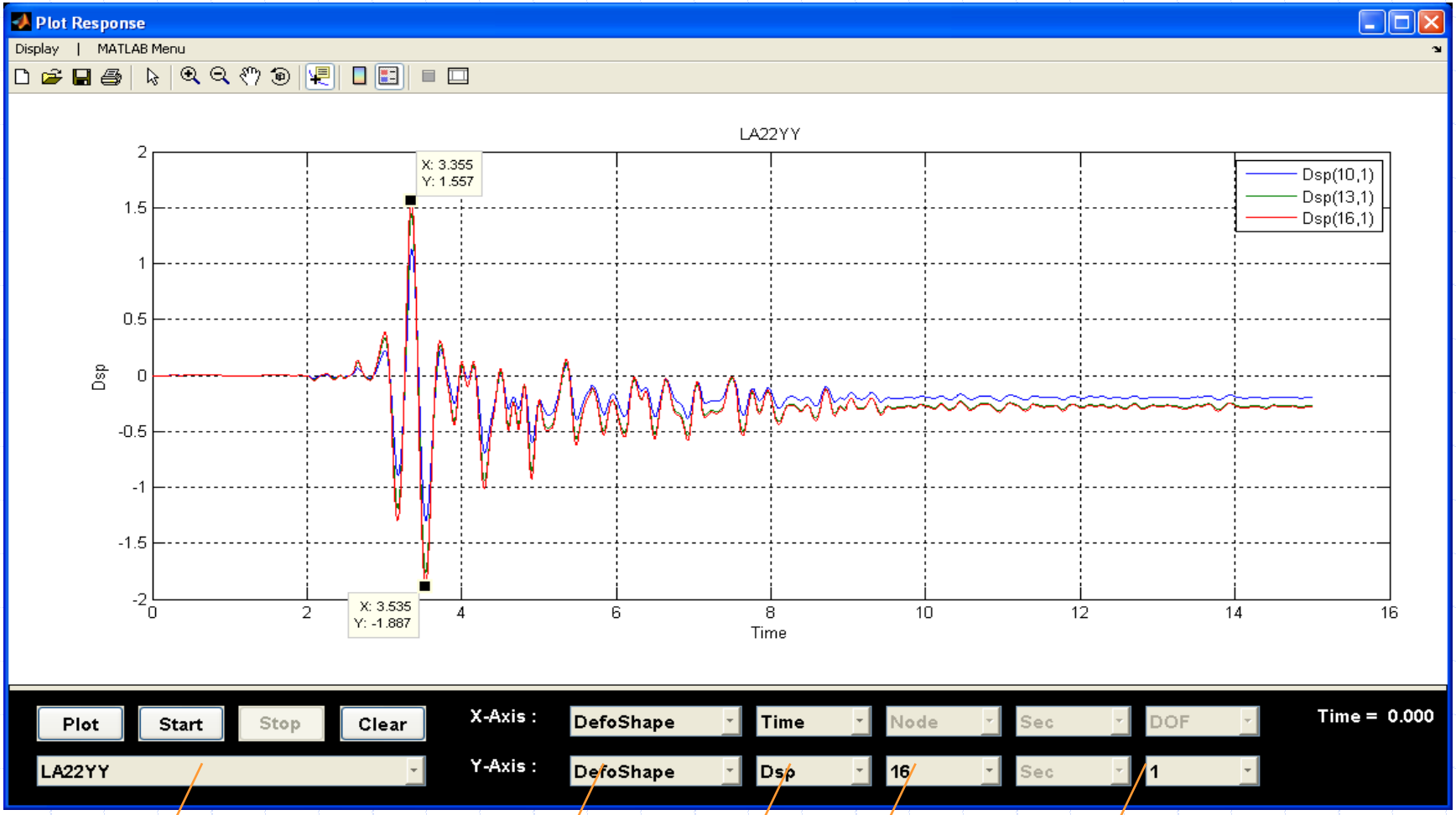
Post processing: plot element forces



Post processing: plot mode shape



Post processing: plot response histories



AnalysisCase

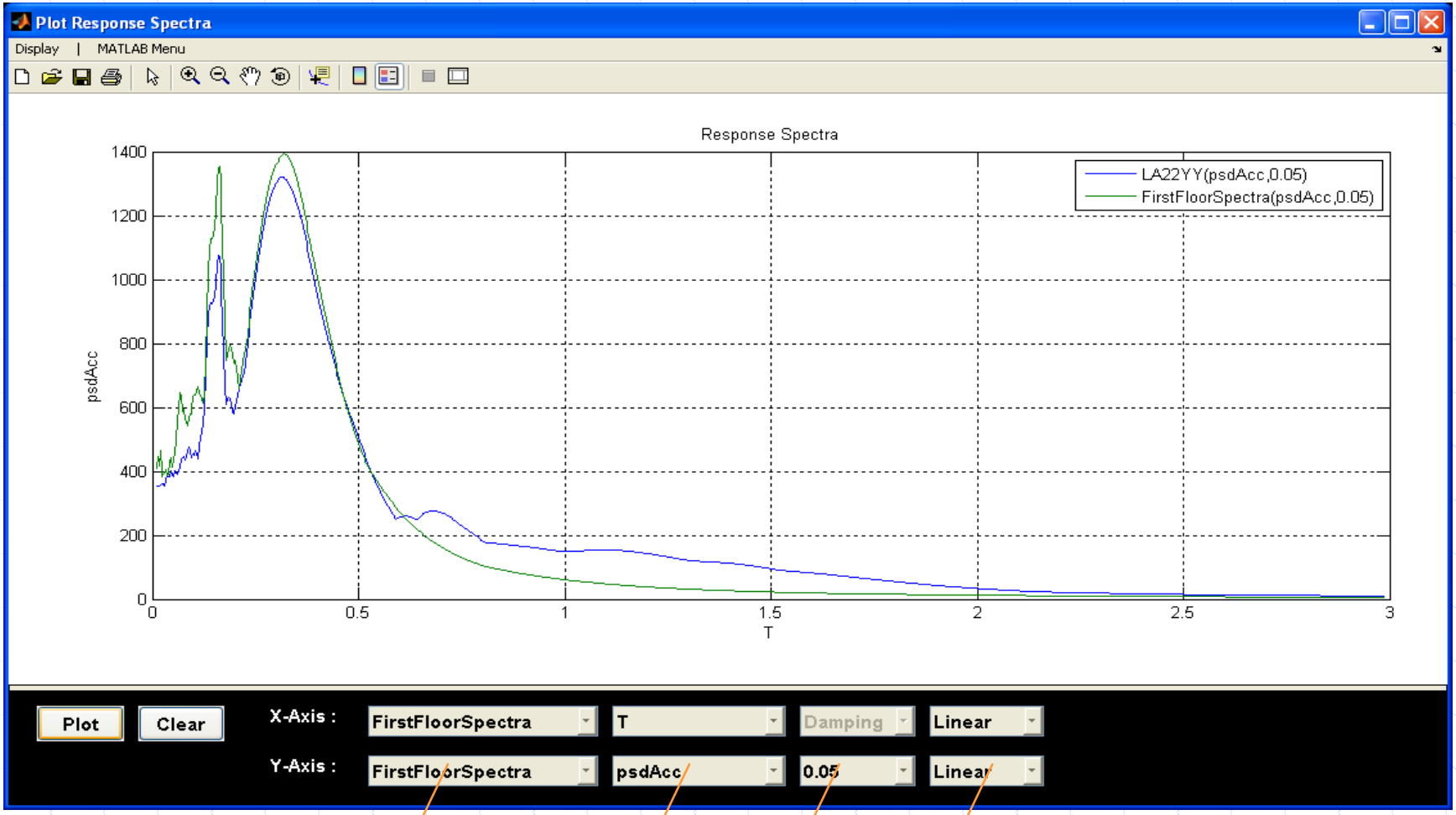
Recorder

Parameter

Node/Element

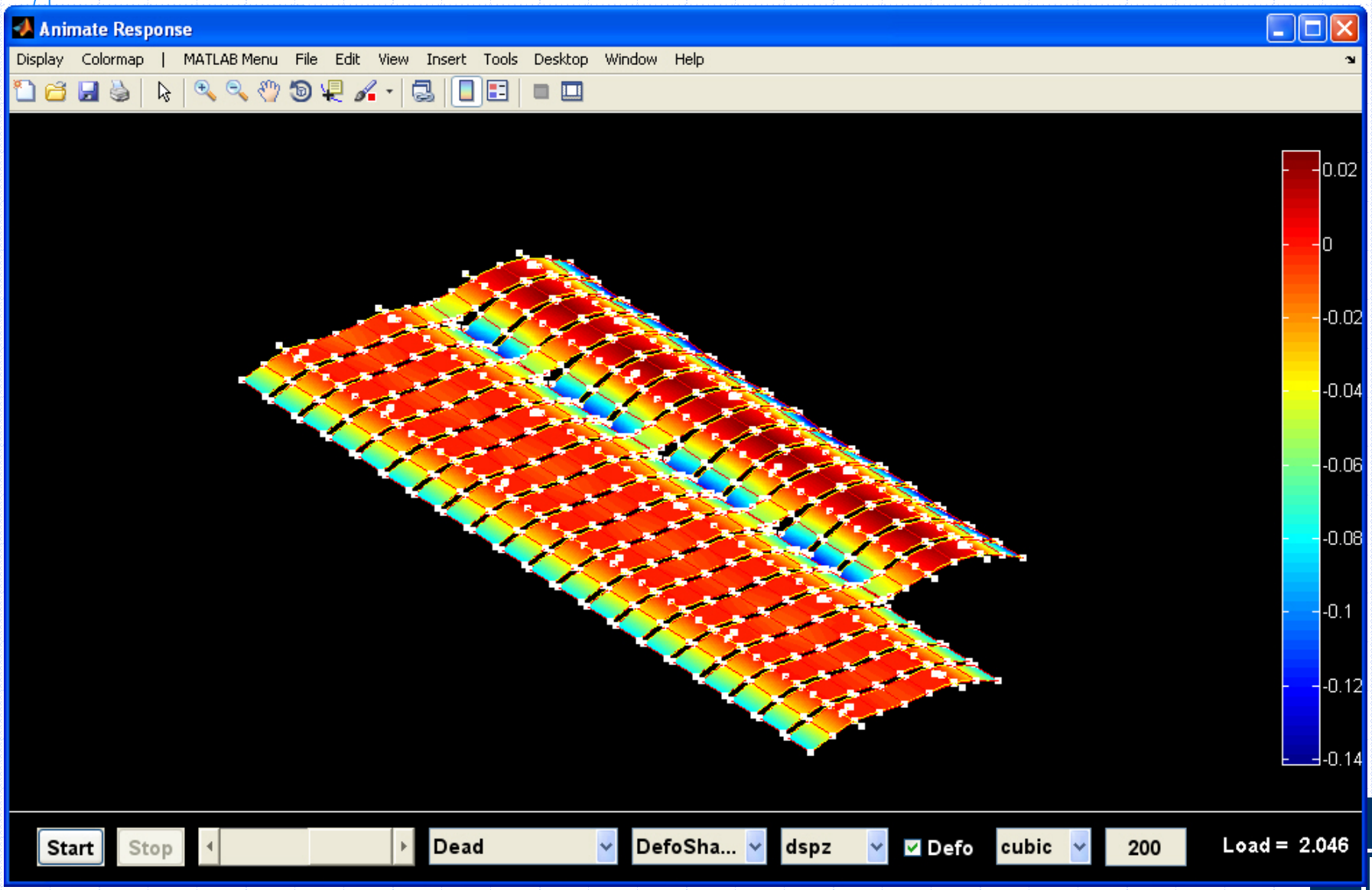
DOF

Post processing: plot response spectra

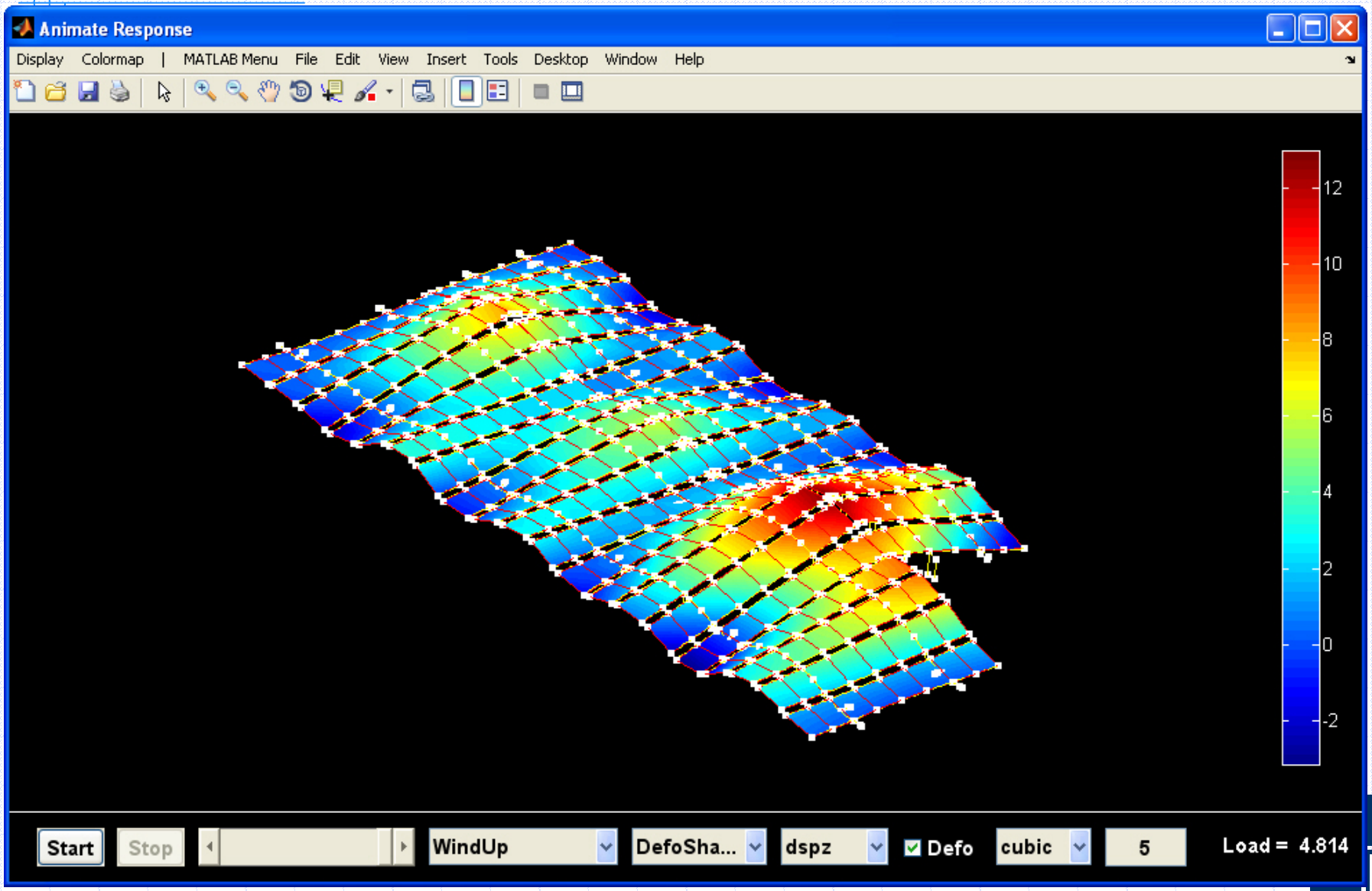


Spectra Name Response Damping Axis Scale

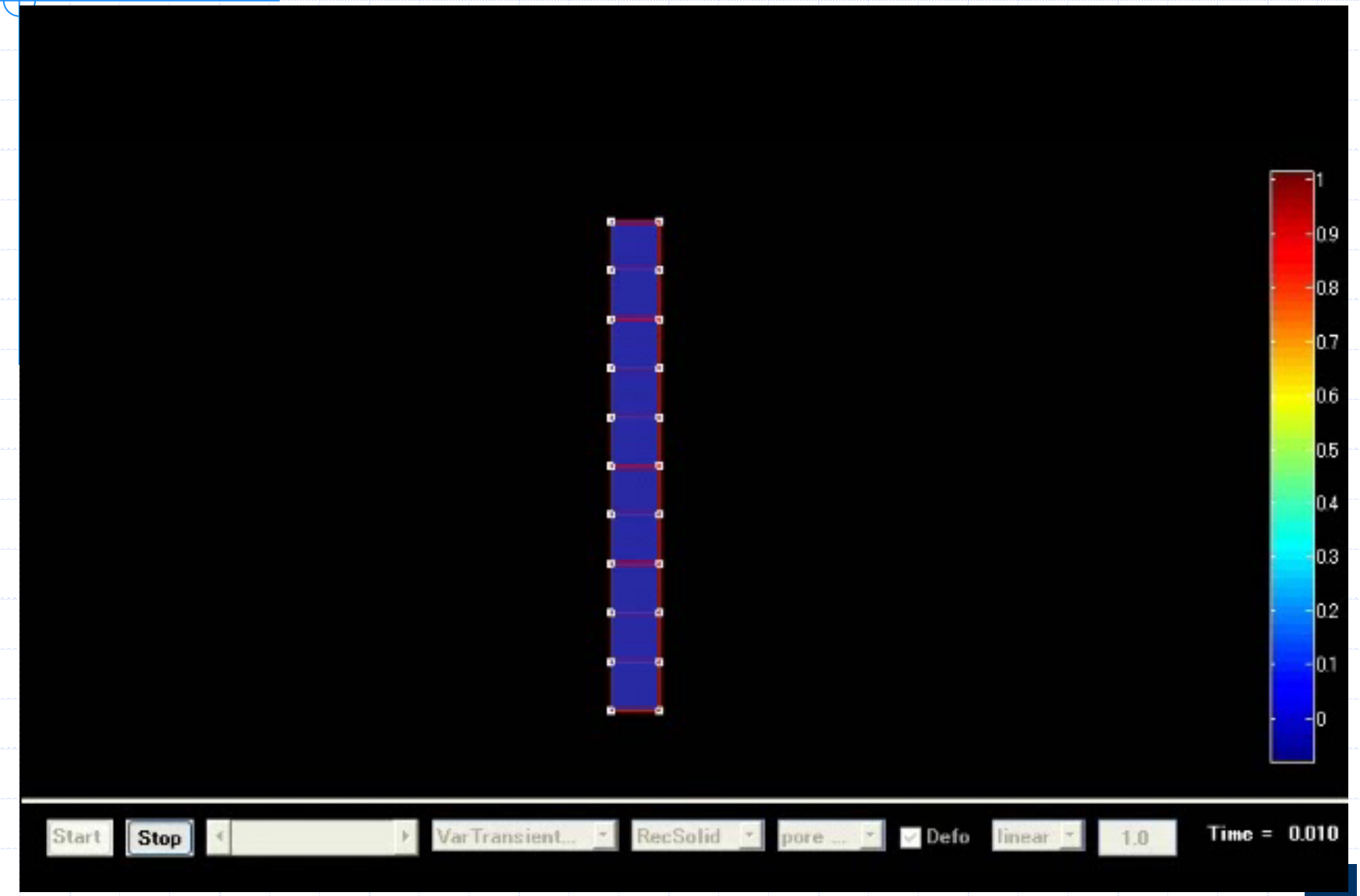
Post processing: animate response



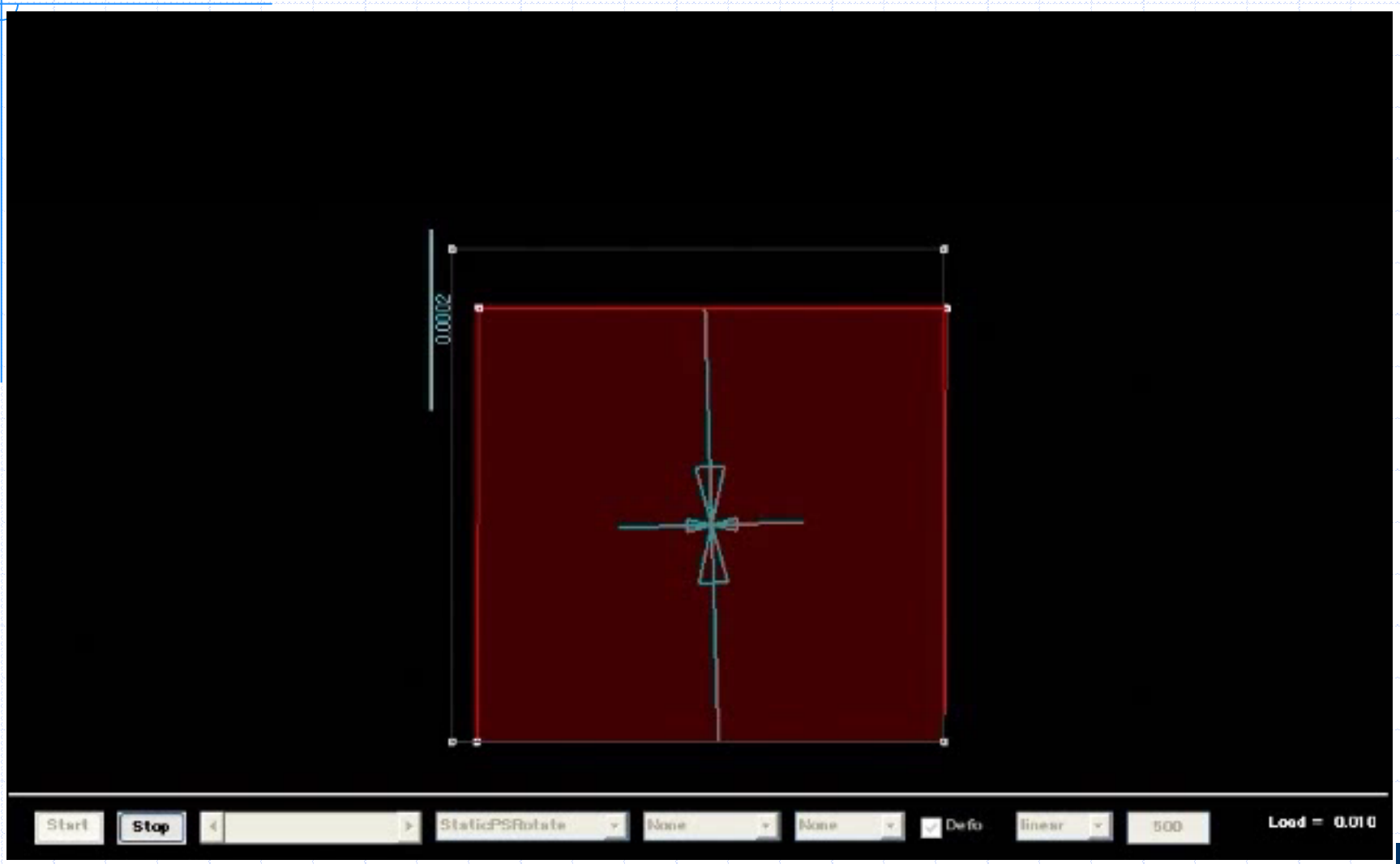
Post processing: animate response



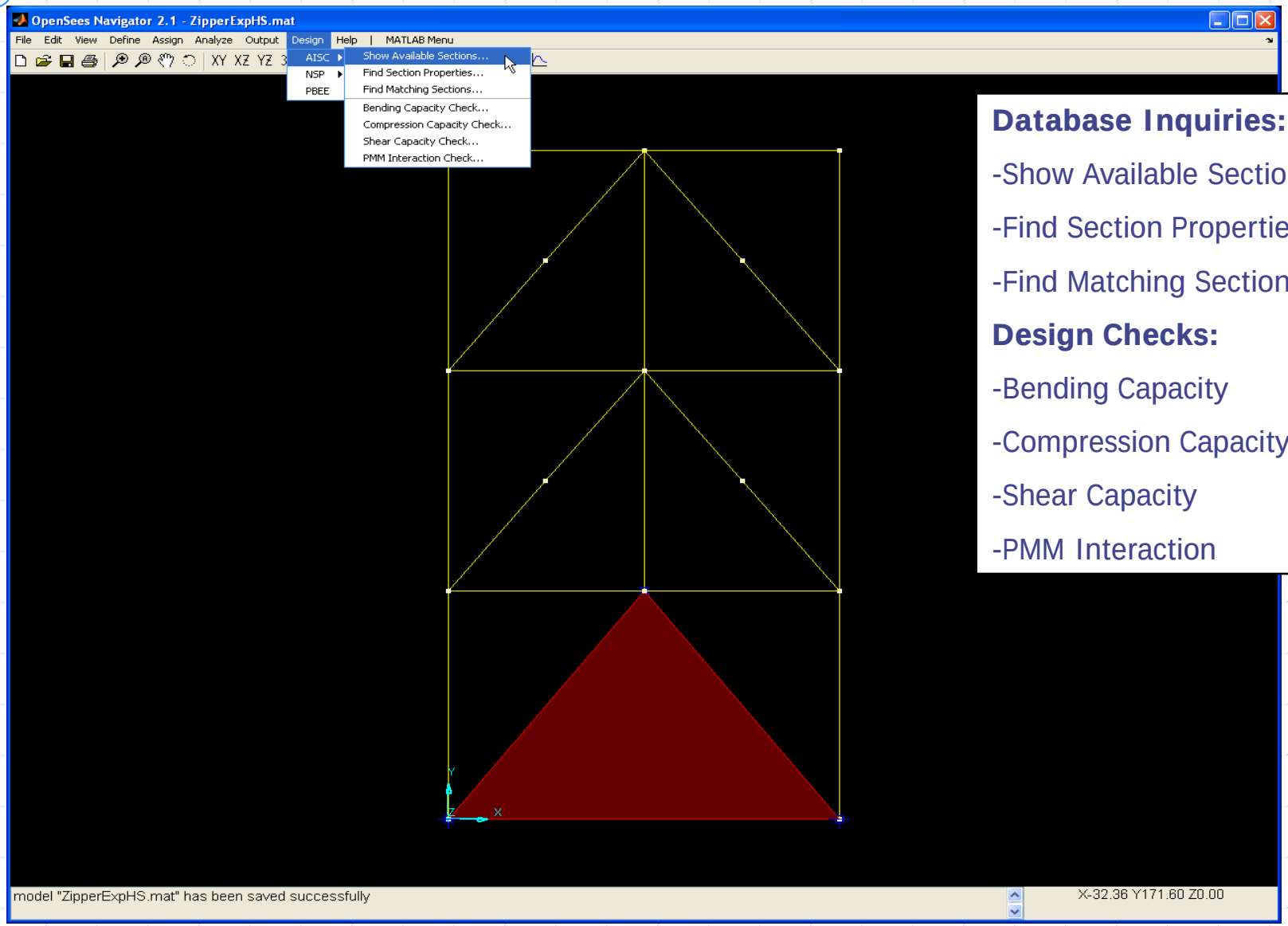
Post processing: animate response



Post processing: principal stress and strain



Design: AISC design toolbox



Database Inquiries:

-Show Available Sections

-Find Section Properties

-Find Matching Sections

Design Checks:

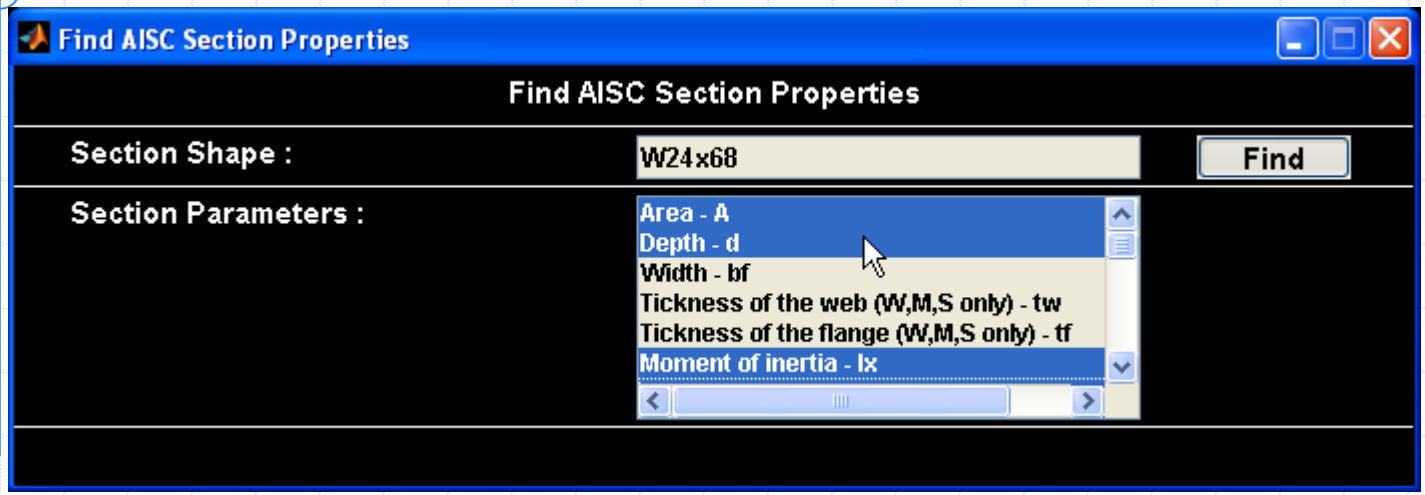
-Bending Capacity

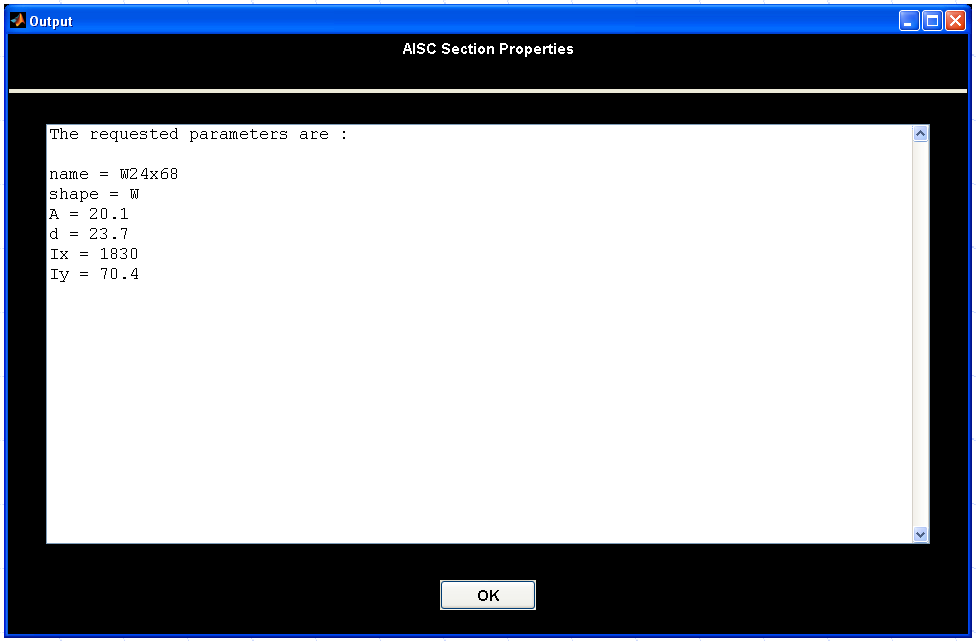
-Compression Capacity

-Shear Capacity

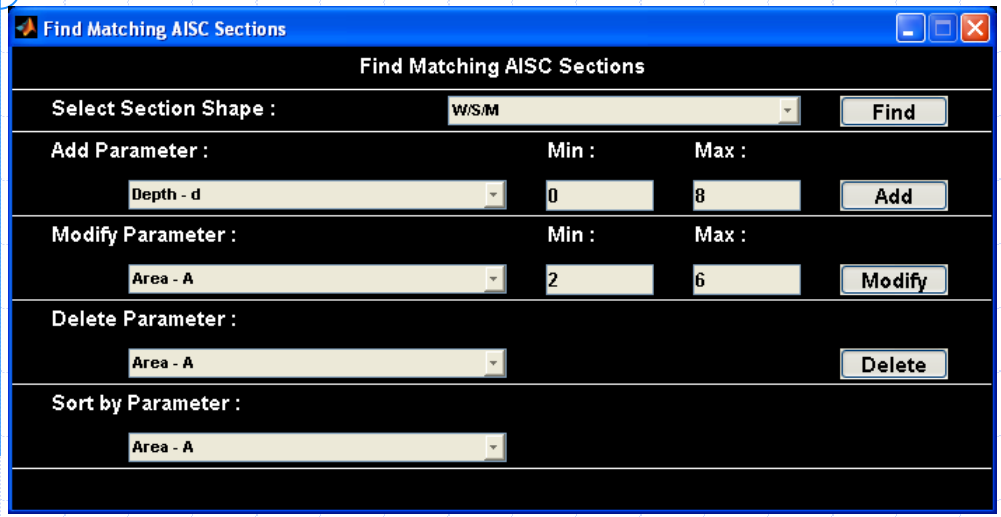
-PMM Interaction

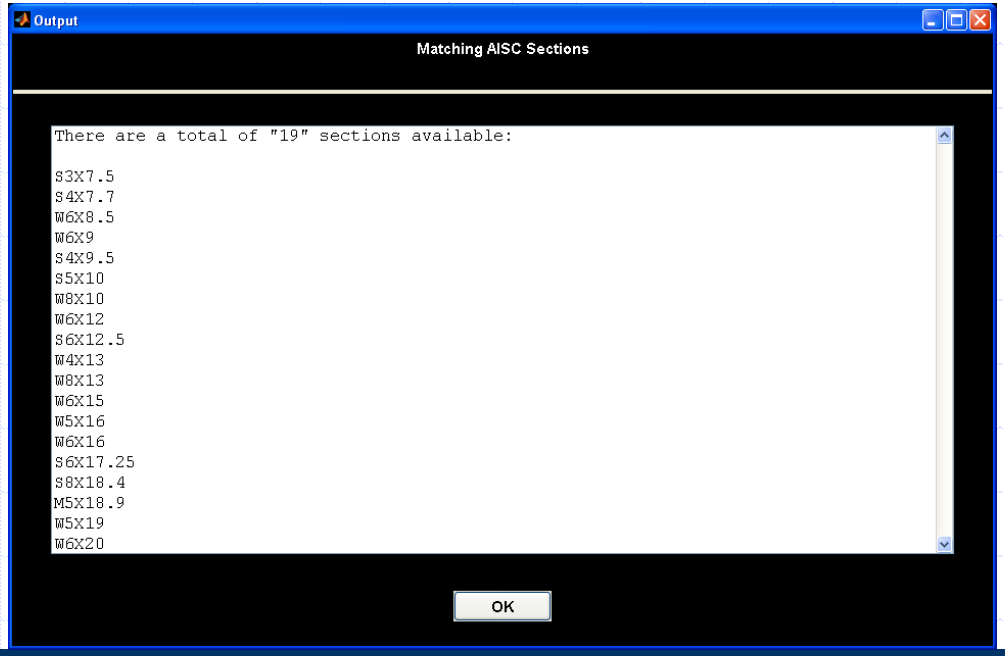
AISC design toolbox: section properties



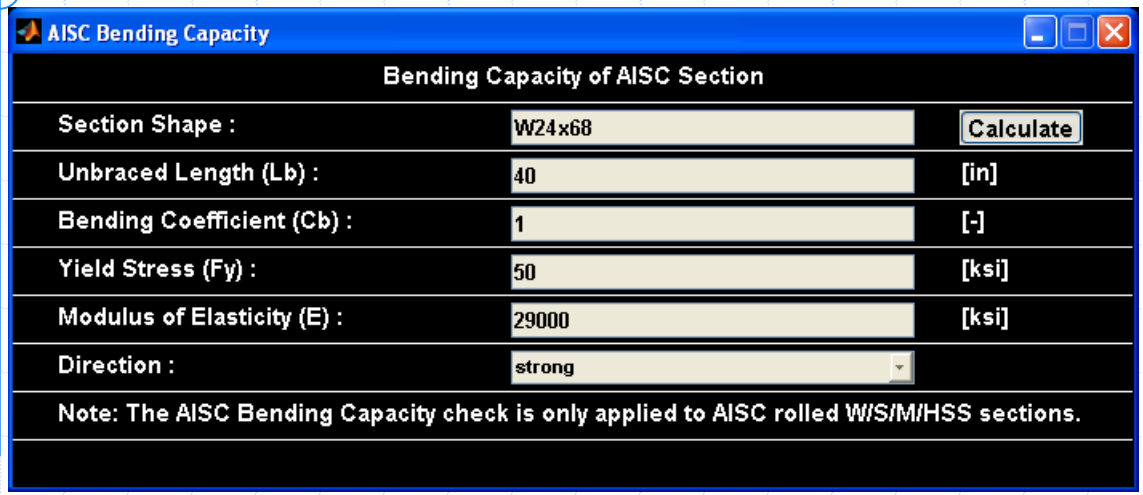


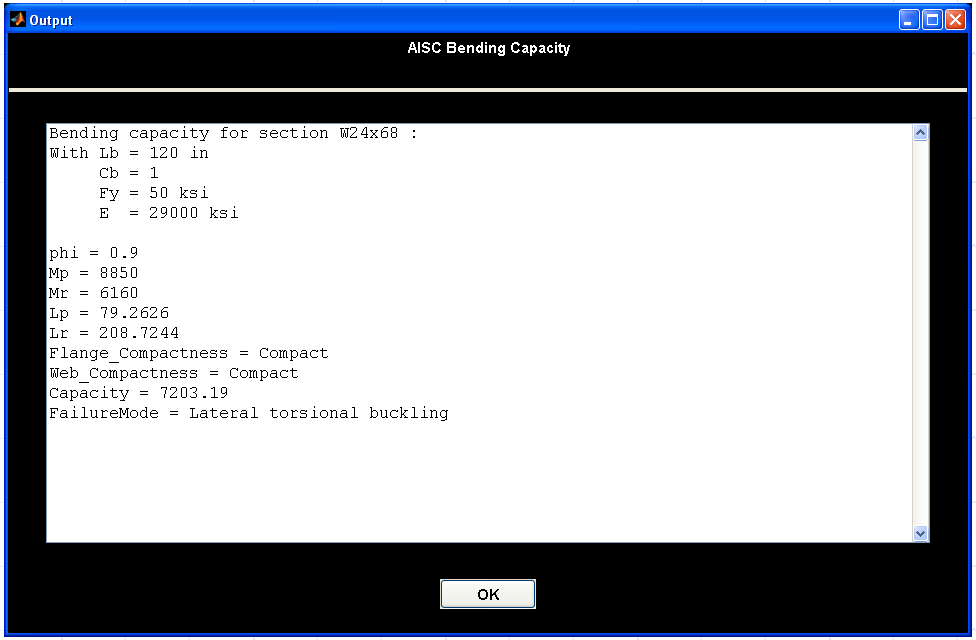
AISC design toolbox: matching sections



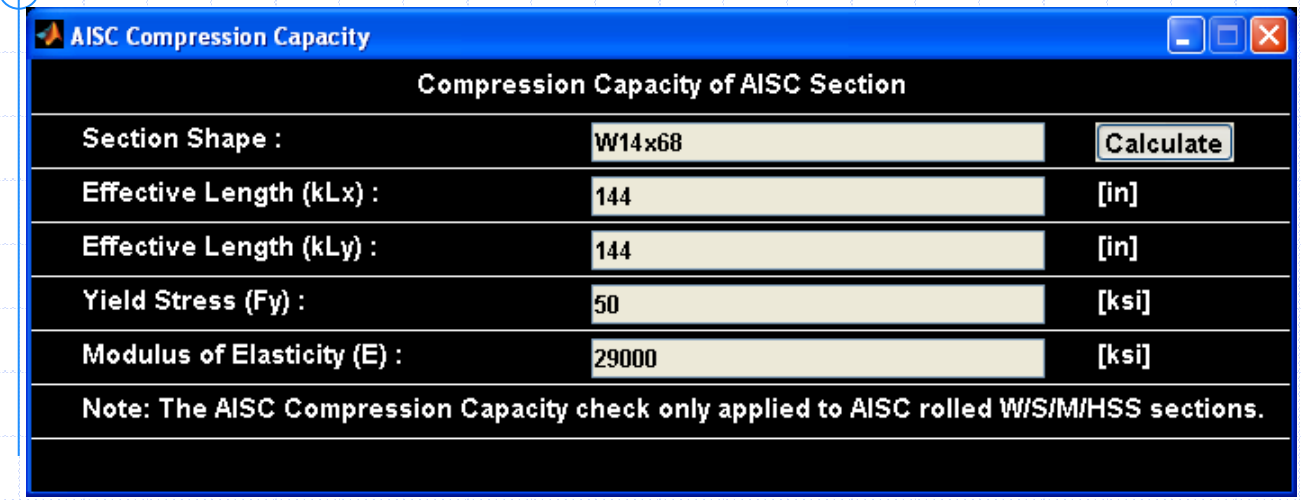


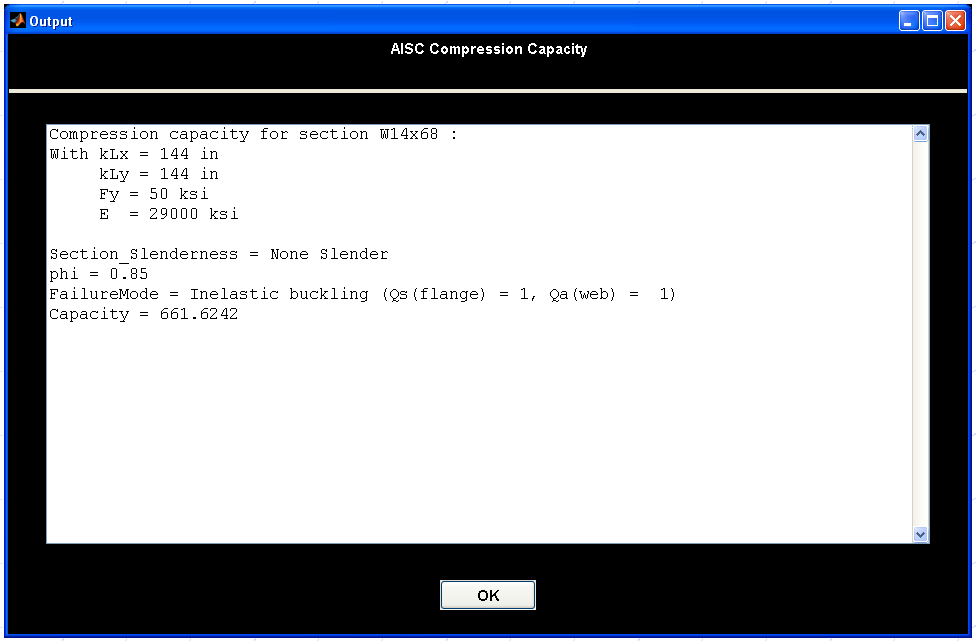
AISC design toolbox: bending capacity



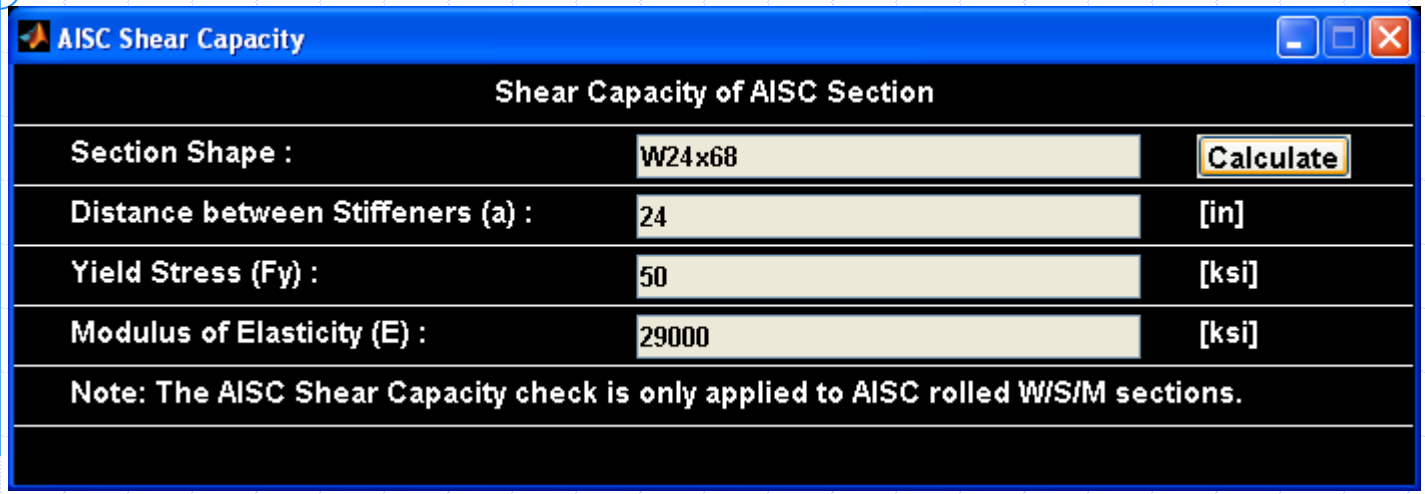


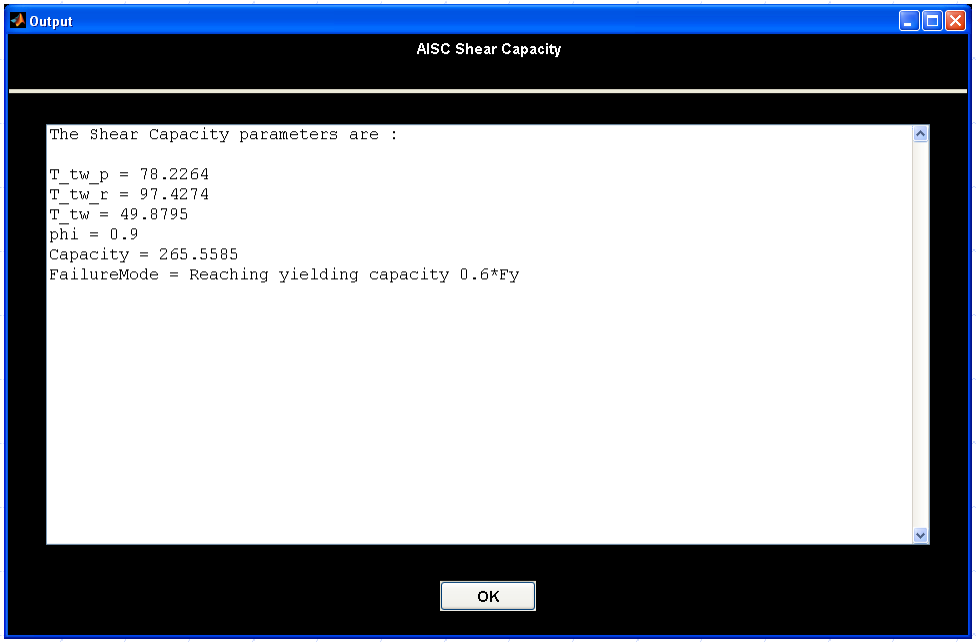
AISC design toolbox: compression cap.



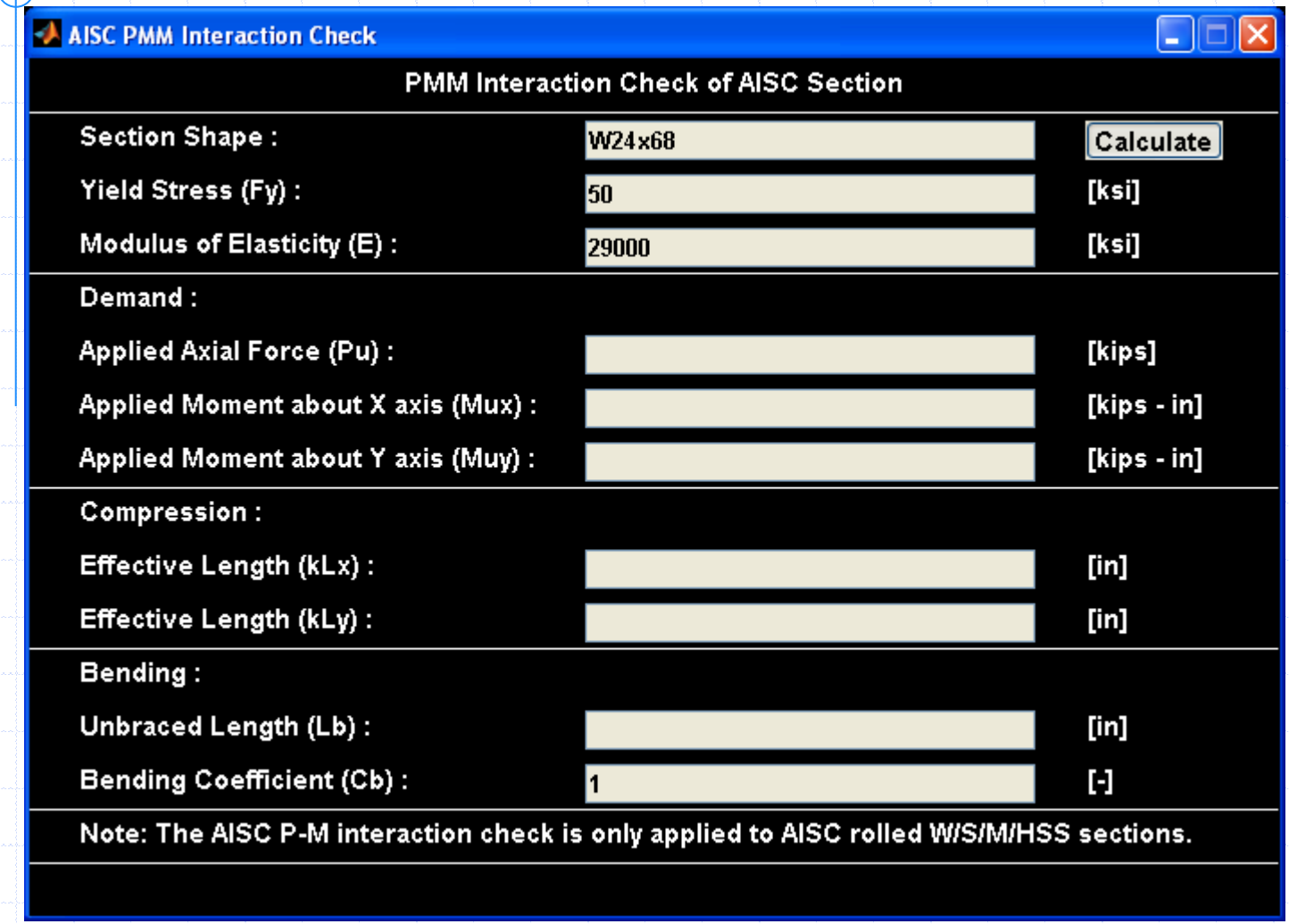


AISC design toolbox: shear capacity





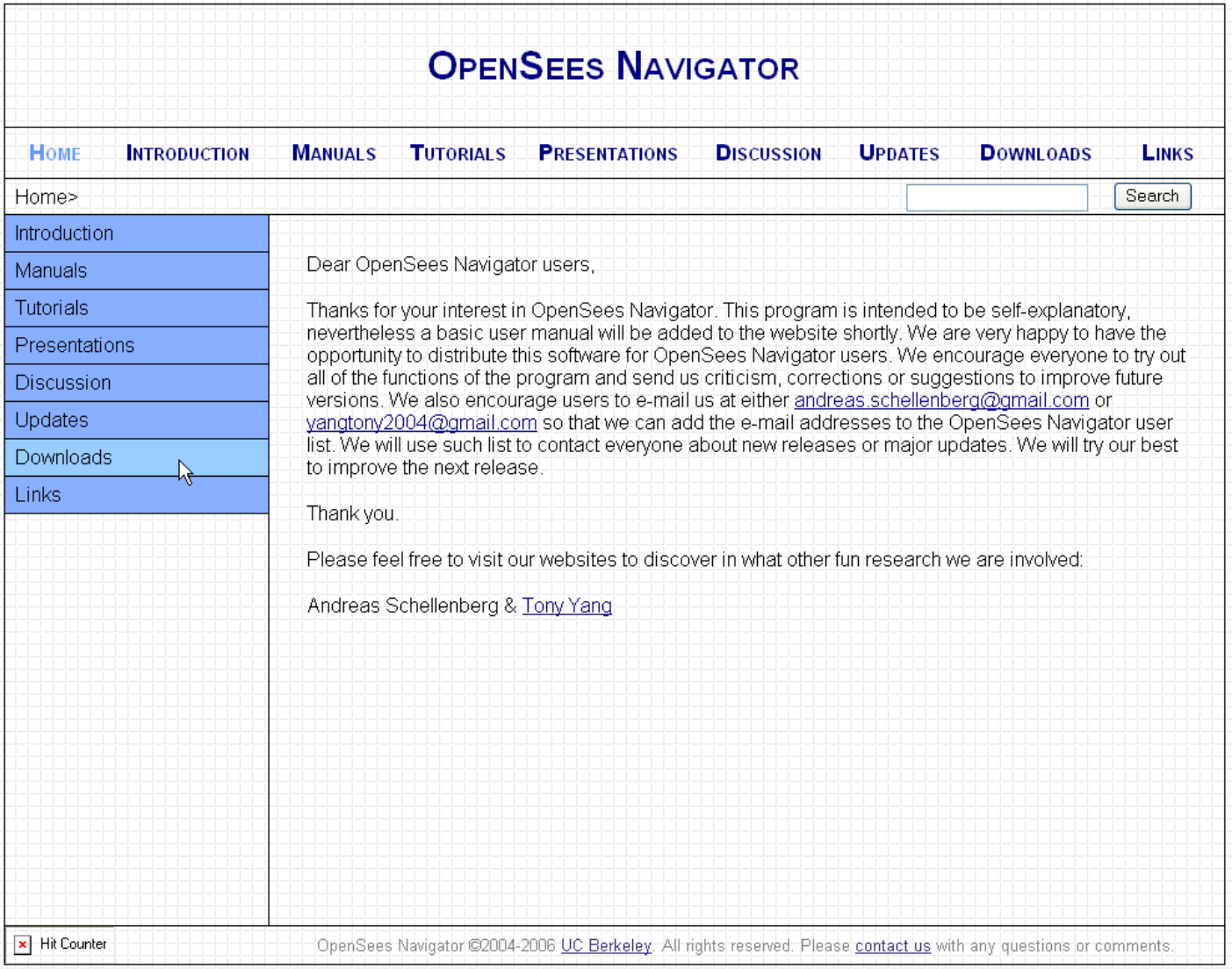
AISC design toolbox: PMM interaction



Summary

- OpenSees Navigator provides
  - Flexible and user friendly graphical user interface.
  - Great tool to visualize structural behavior.
  - Easy way to study material, section, element or system behavior.
- Hybrid simulation interface (OpenFresco).
- Many design toolboxes: NSP, PBEE, AISC design checks, AISC database, response spectra for linear and bilinear systems and signal filtering.
- Both MATLAB Pcode (32 bit and 64 bit) and self-executable versions for Windows & Mac are available.

Website: home



Website: download