

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



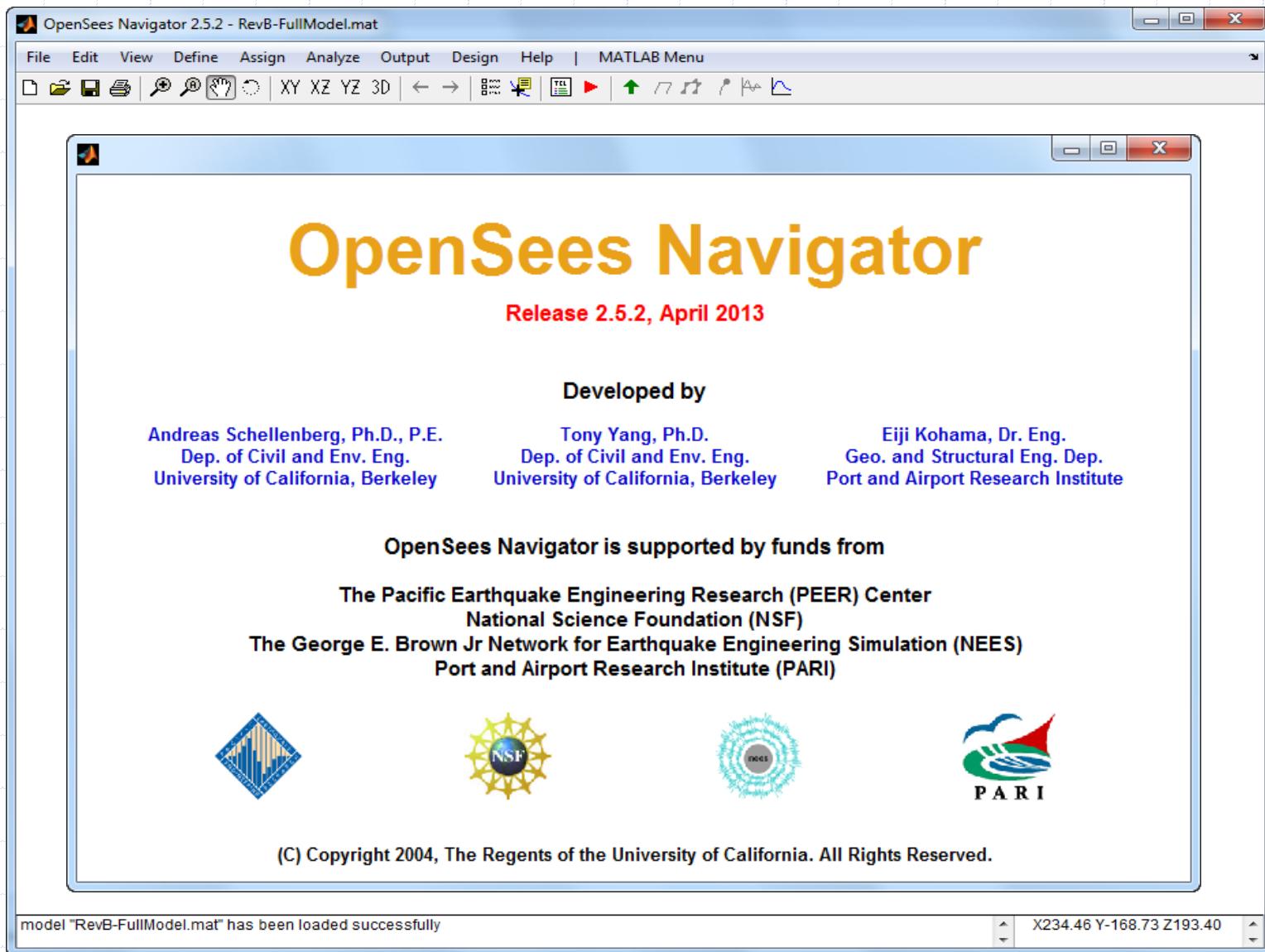
Introduction

- ◆ 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.

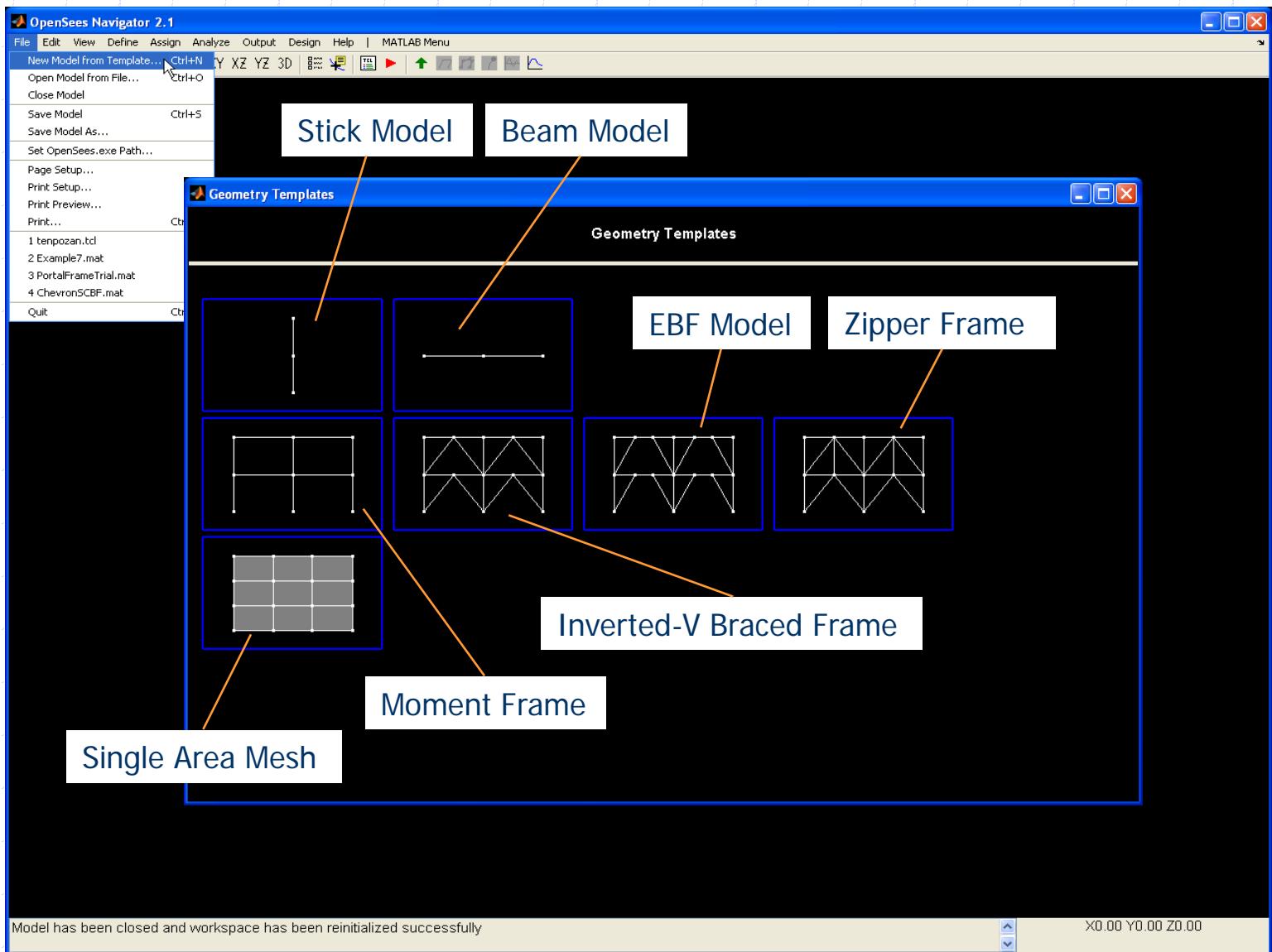
Motivations

- ◆ 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

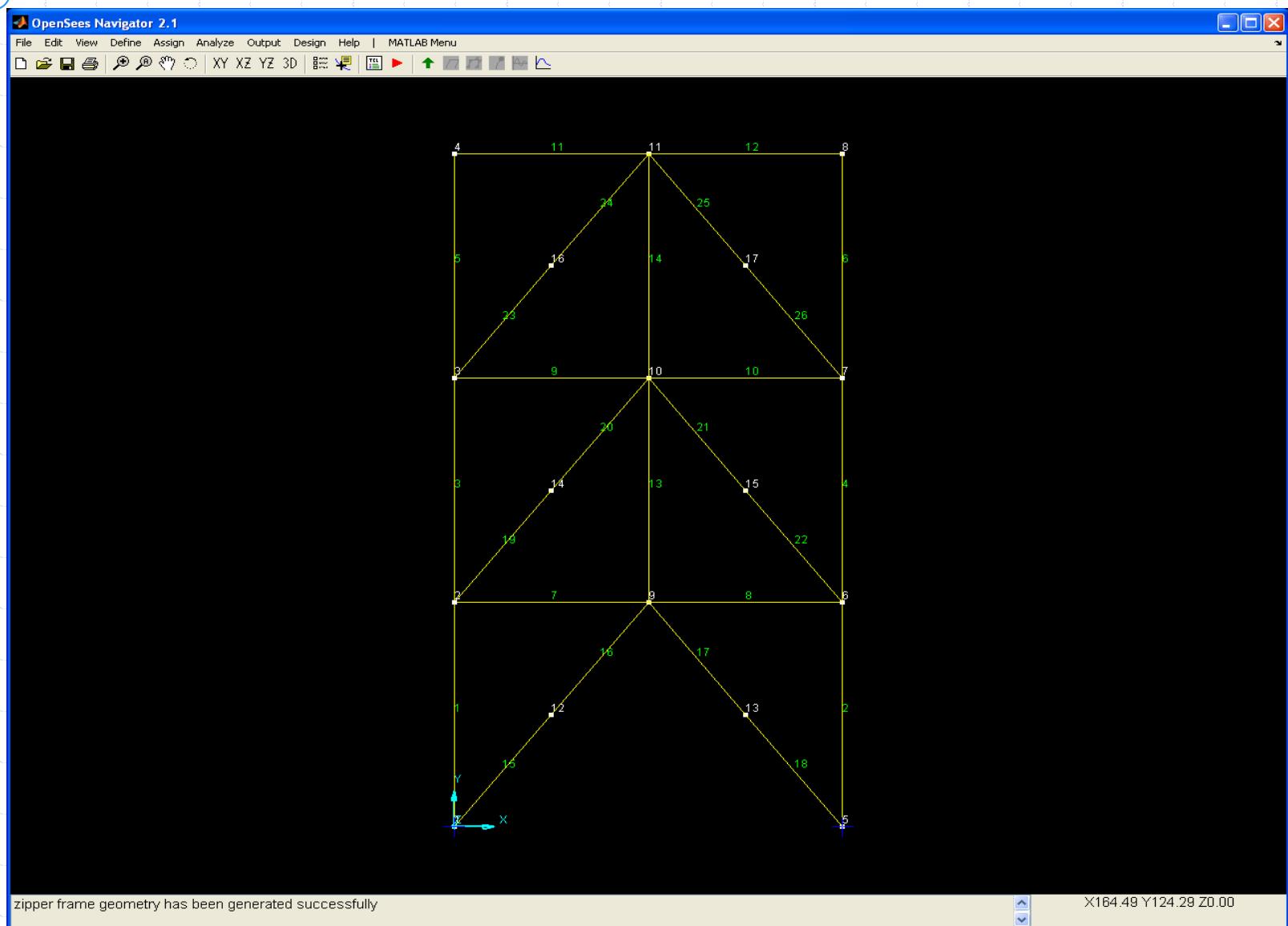


Define geometry: Zipper braced frame

Define Zipper Frame Geometry

Dimension (ndm) :	2d	Generate
Number of Stories (NOS) :	3	
Number of Bays (NOB) :	1	
Story Height (SH) :	52	
Bay Width (BW) :	80	
Boundary Condition (BC) :	pinned	
Brace Bay Config (BraceBay) :	BraceBay	
Num Segments in Col (NSC) :	1	
Num Segments in Beam (NSB) :	1	
Num Segments in Brace (NSBR) :	2	
Num Segments in Z-Col (NSZC) :	1	
Brace Offset (BraceOffset) :	None	

View geometry: display

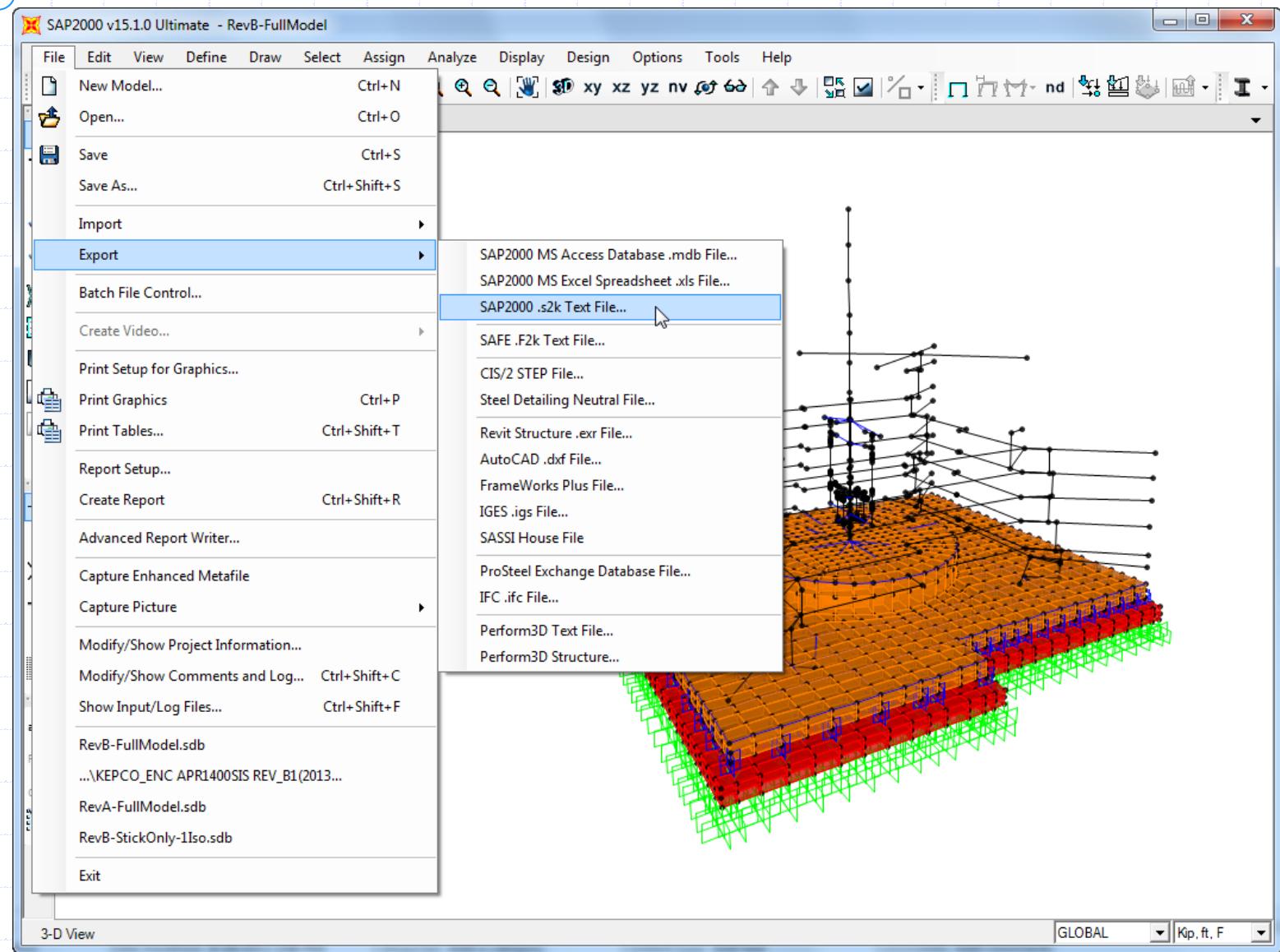


Conversion: SAP2000 to OpenSees

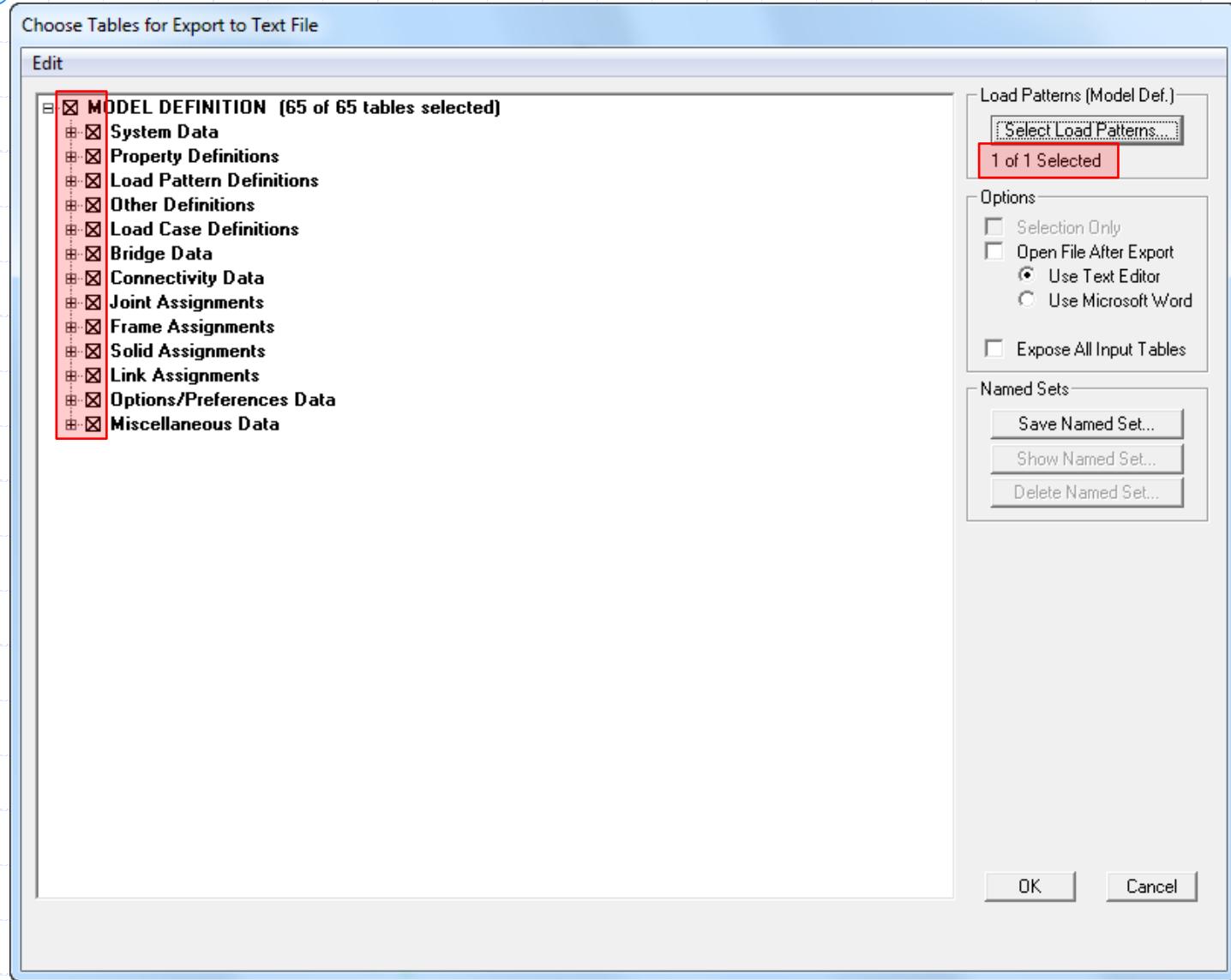
- ◆ Export SAP2000 model to .s2k file
- ◆ Start OpenSees Navigator program
- ◆ Select “Open Model from File...” and choose the earlier exported .s2k file
- ◆ There is no one-to-one relationship between SAP2000 and OpenSees objects, therefore
 - ◆ Carefully check the converted model
 - ◆ Element loads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



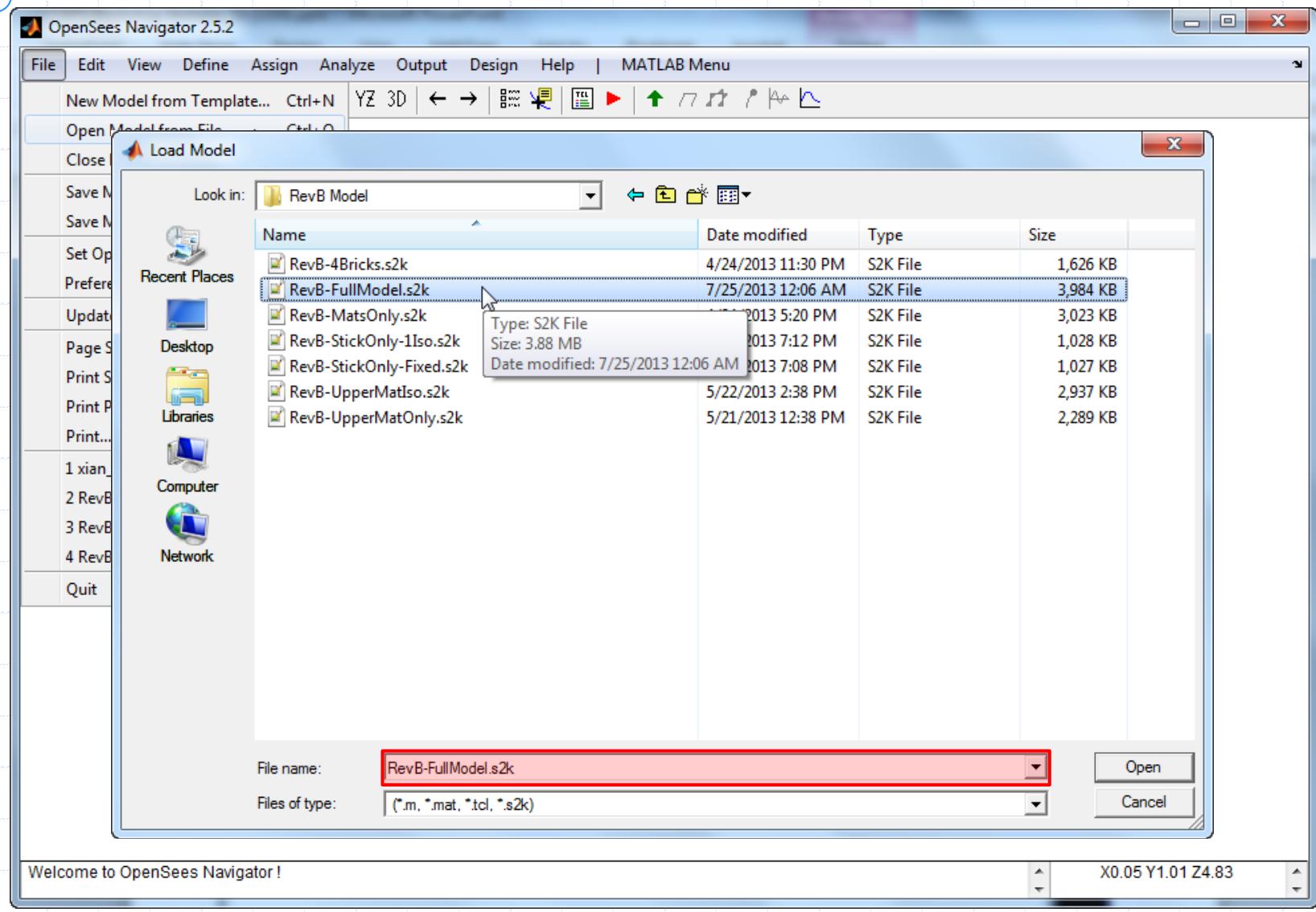
Export to s2k file



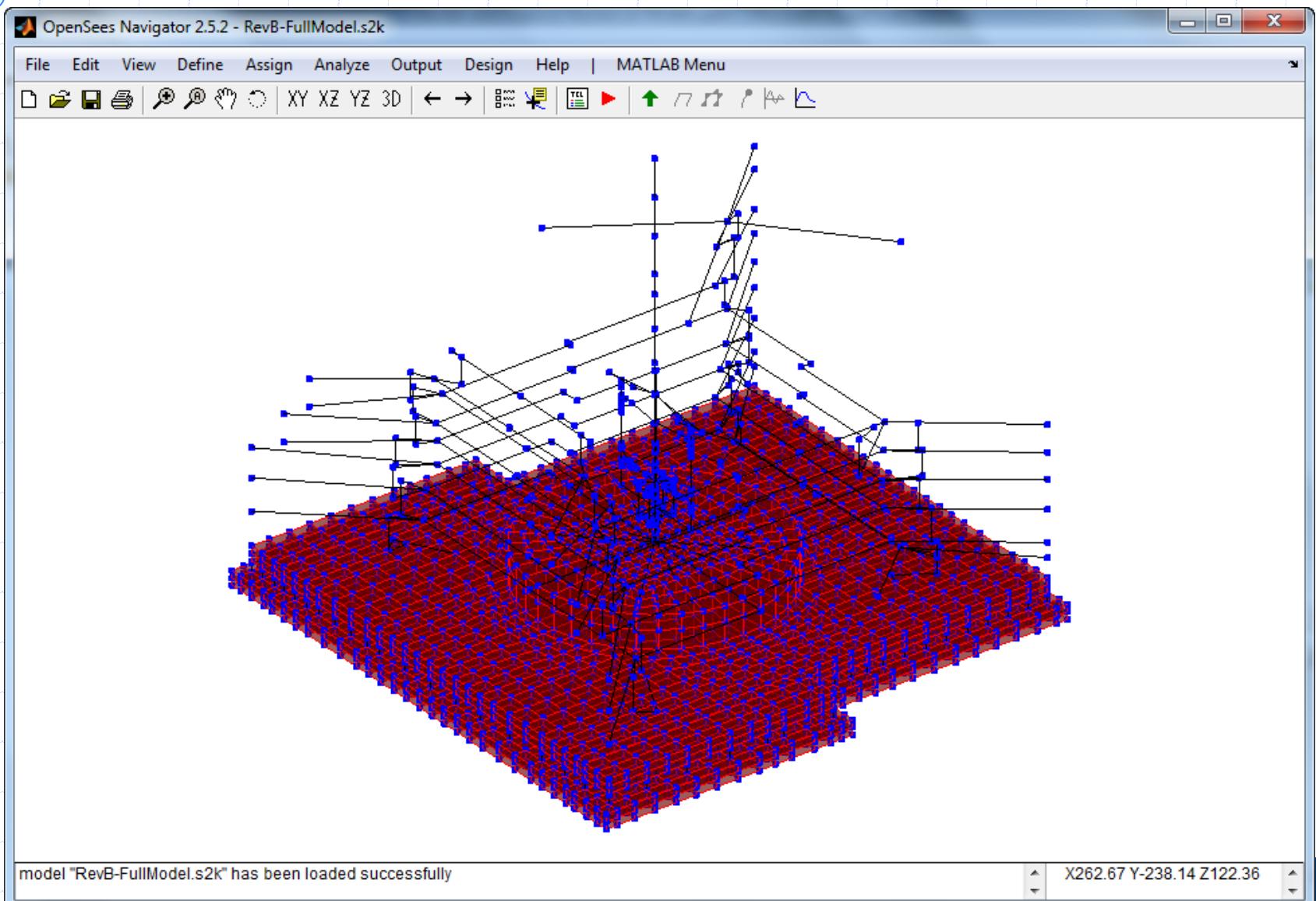
Export to s2k file



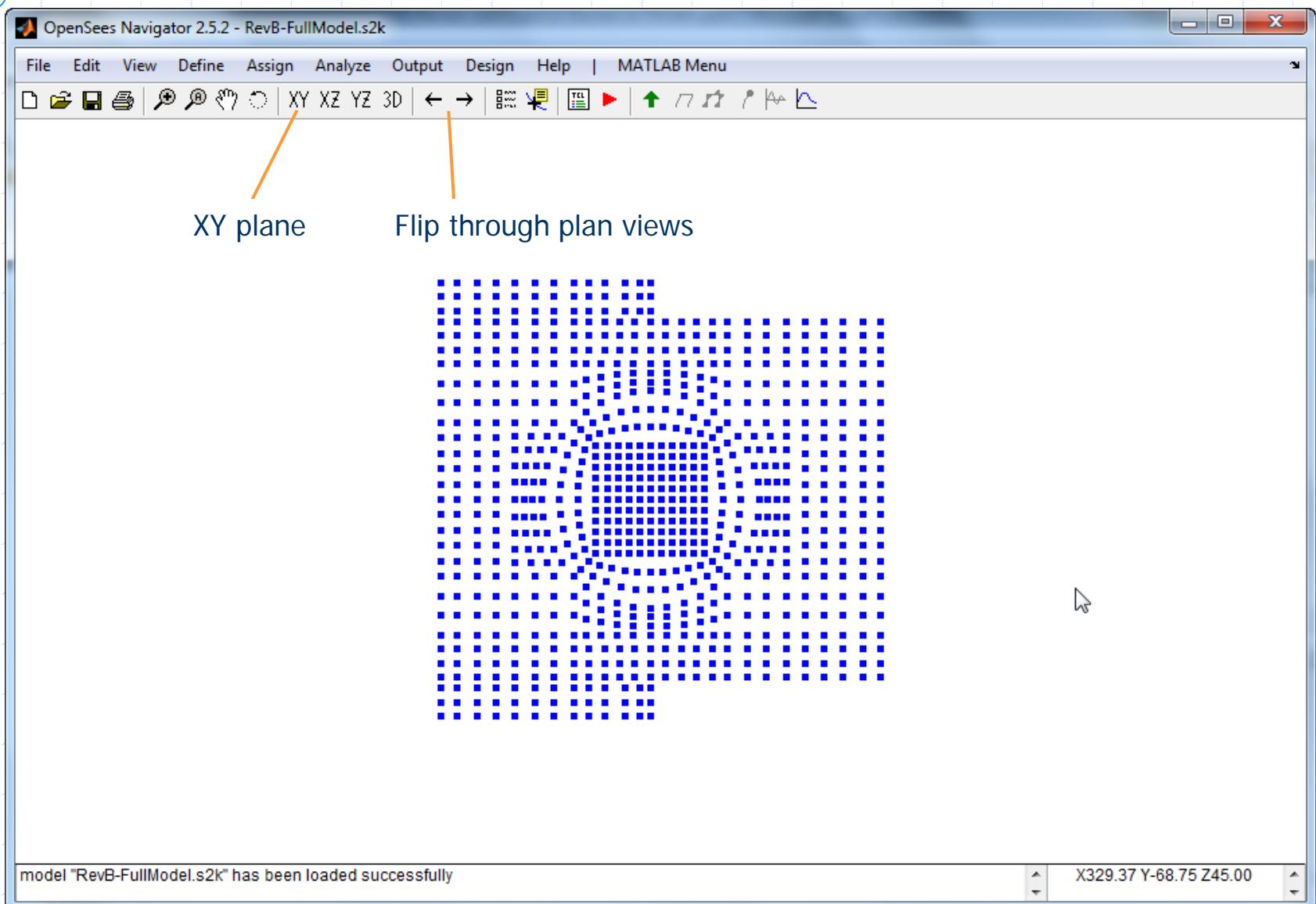
Define geometry: import from SAP2000



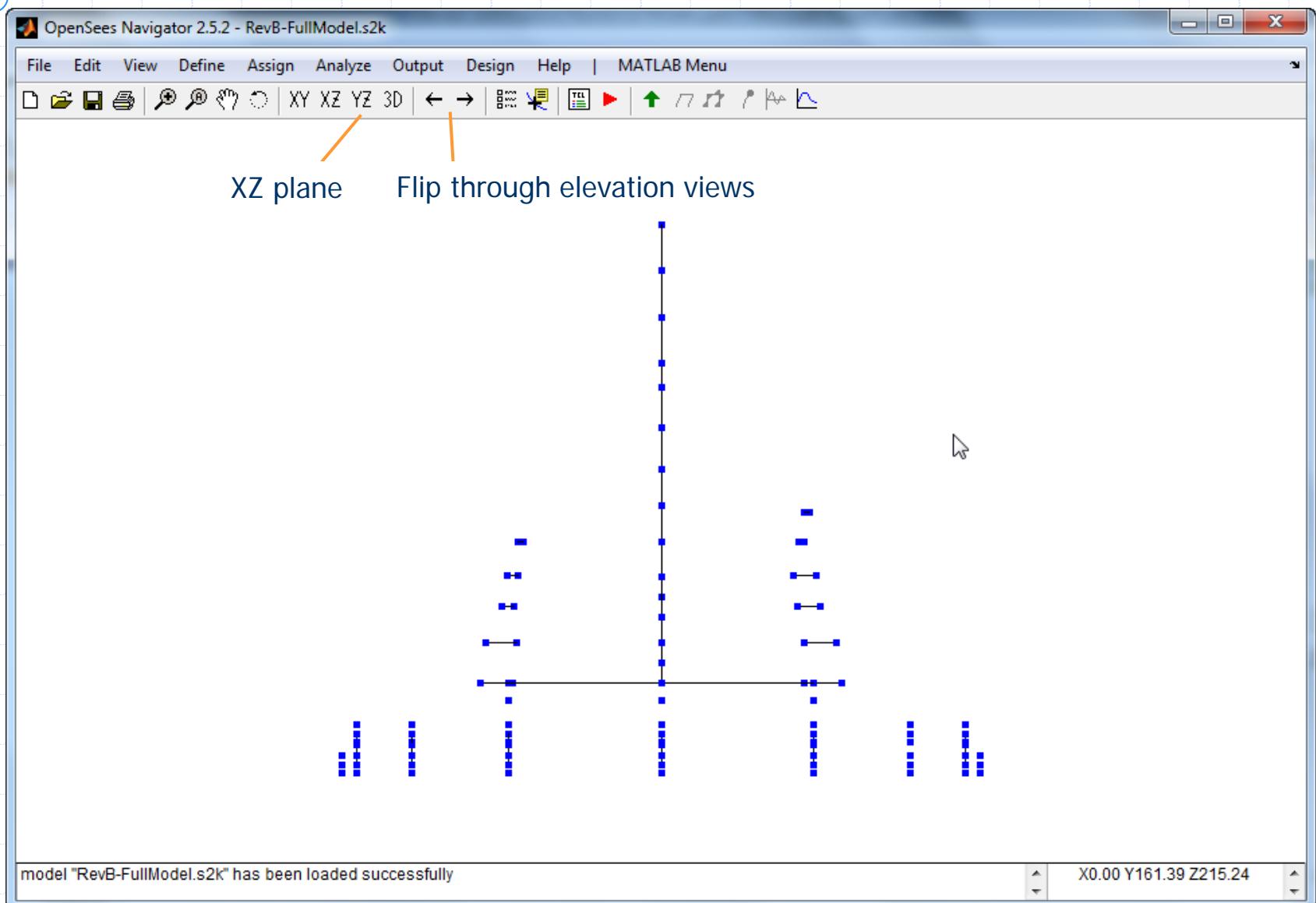
Define geometry: import from SAP2000



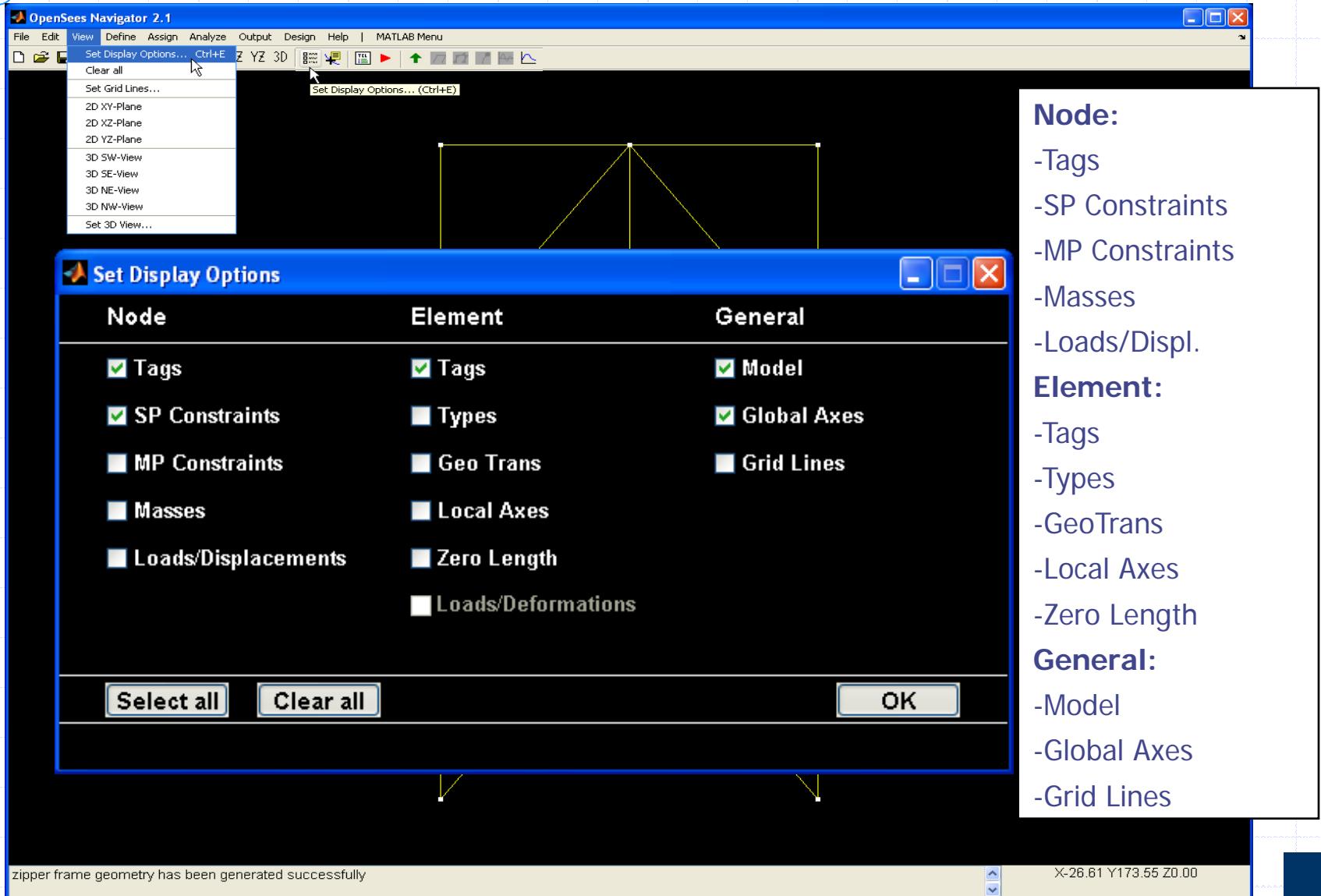
Define geometry: import from SAP2000



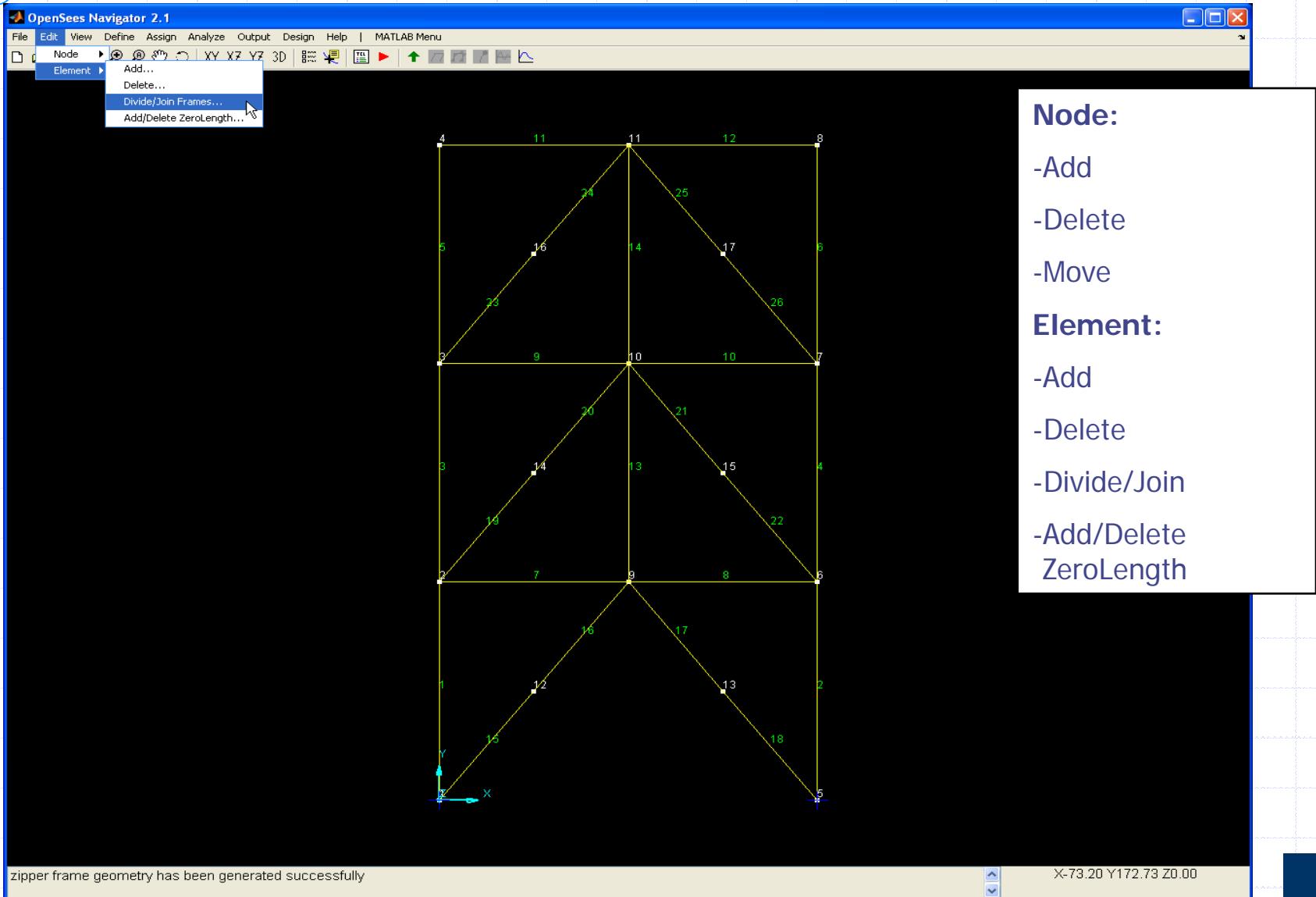
Define geometry: import from SAP2000



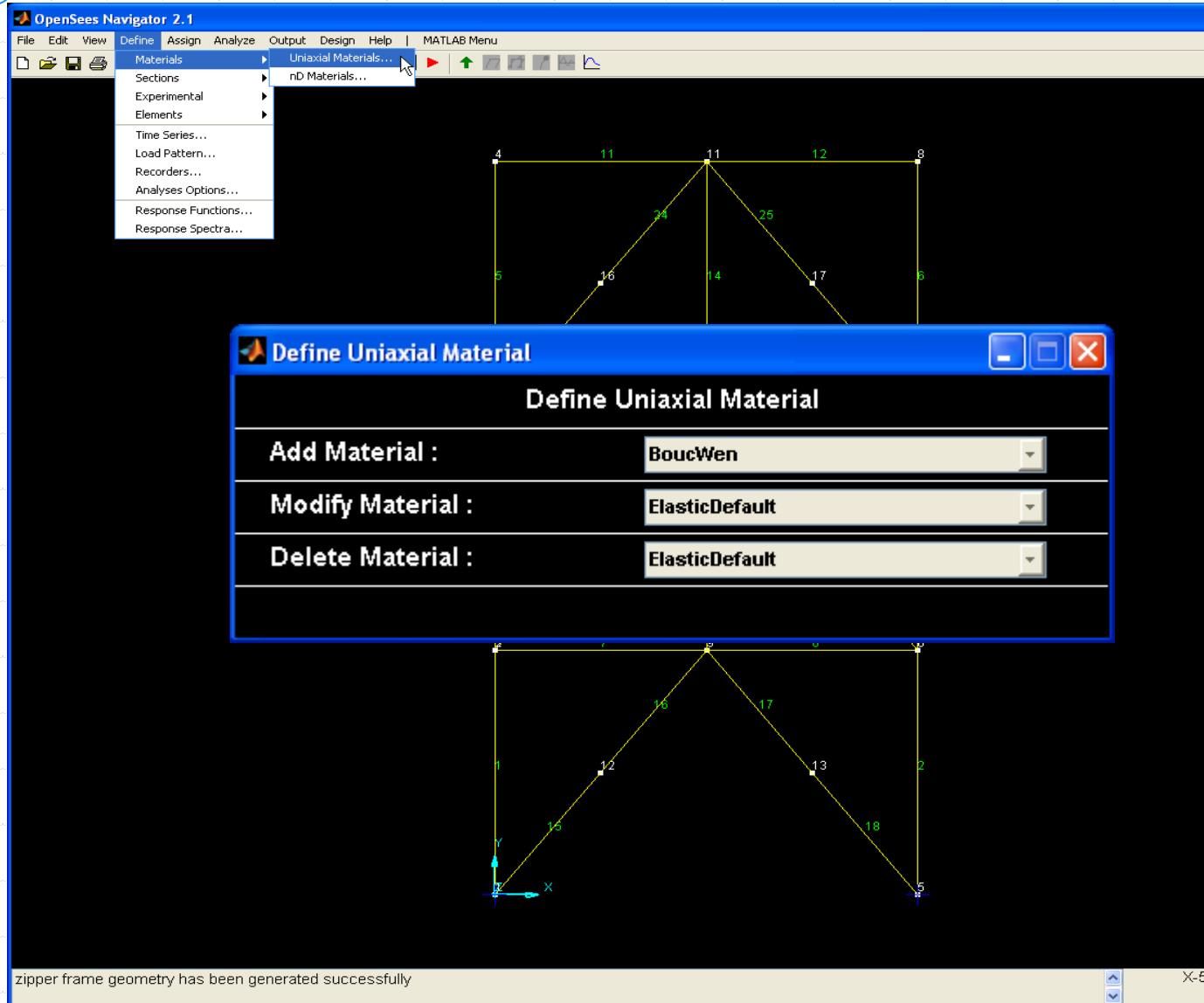
View geometry: set display options



Edit geometry



Define material: uniaxial materials



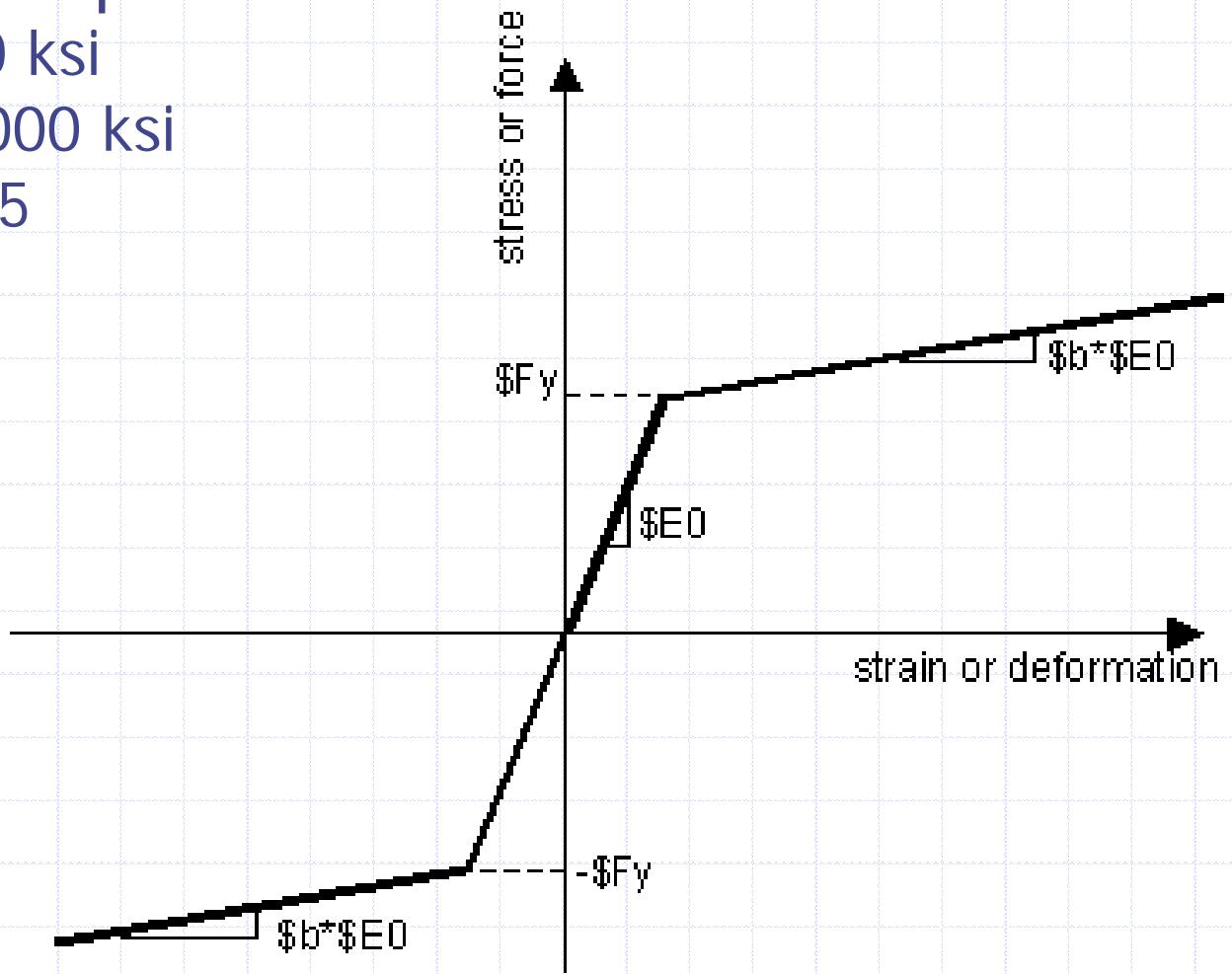
Templates:

- BoucWen
- Concrete01
- Concrete02
- Concrete04
- Elastic
- ElasticNoTension
- ElasticPP
- ElasticPPGap
- Fatigue
- Hardening
- Hysteretic
- MinMax
- Parallel
- ReinforcingSteel
- Series
- Steel01
- Steel02
- Viscous
-

Define uniaxial material: Steel01

Material properties

- $F_y = 50 \text{ ksi}$
- $E = 29000 \text{ ksi}$
- $b = 0.05$

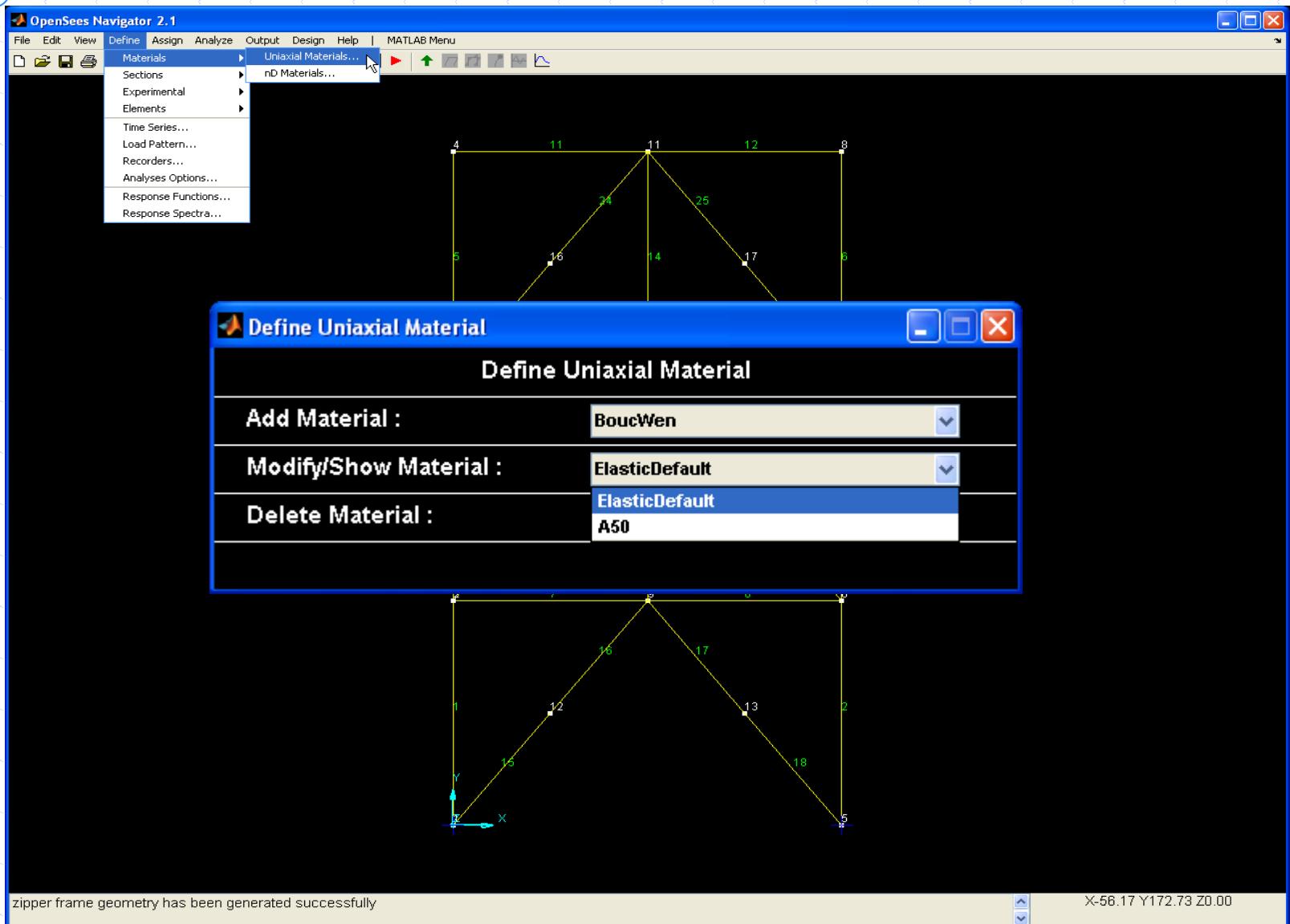


Define uniaxial material: Steel01

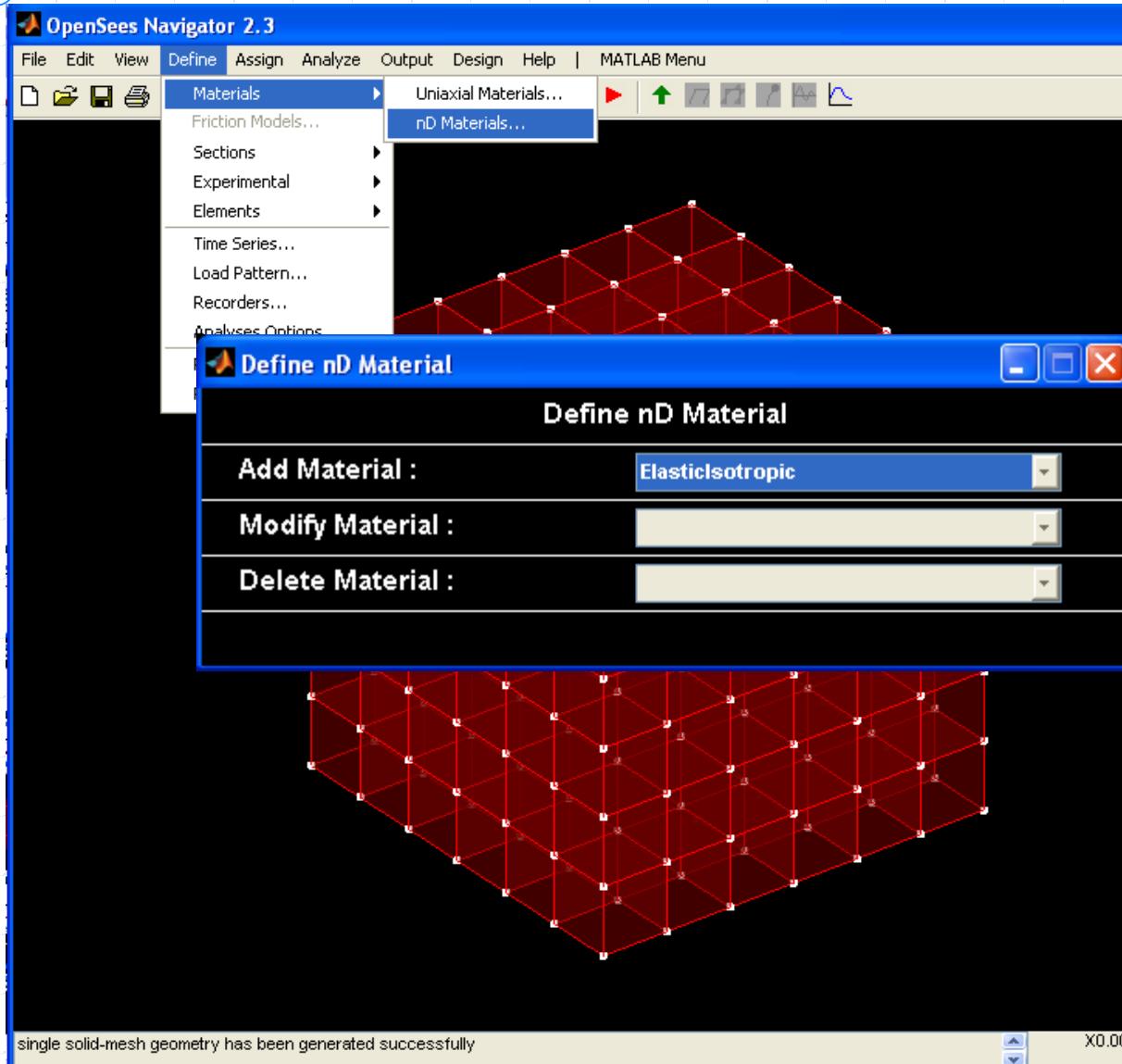
Define Steel01 Material

Material Name :	A50	Add
Yield Stress (Fy) :	50	
Modulus of Elasticity (E) :	29000	
Hardening Ratio (b) :	0.05	
<i>Optional Parameters :</i>		
Iso Hardening Parameter (a1) :	0.0	
Iso Hardening Parameter (a2) :	1.0	
Iso Hardening Parameter (a3) :	0.0	
Iso Hardening Parameter (a4) :	1.0	

Define material: uniaxial materials



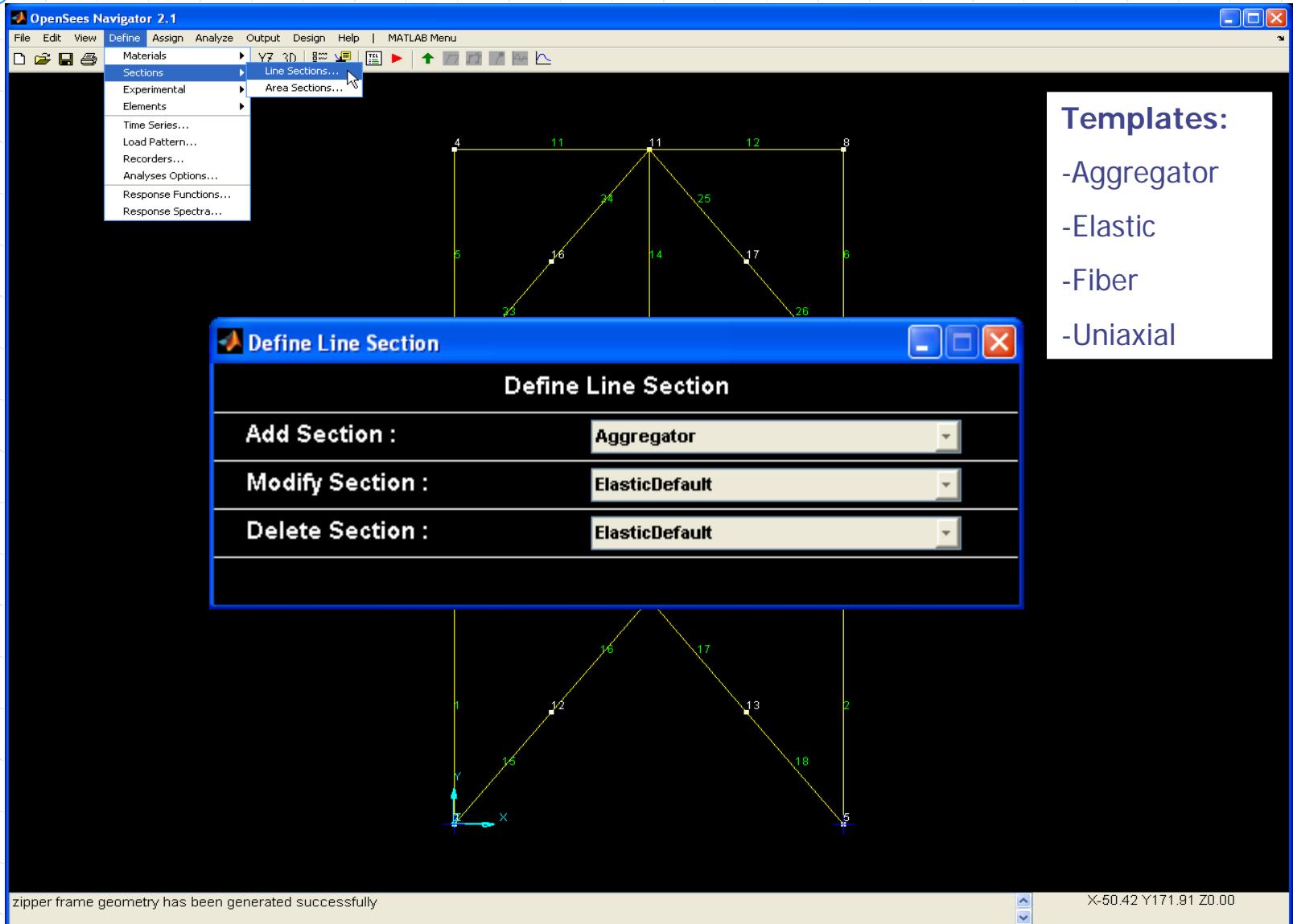
Define material: nD materials



Templates:

- ElasticCrossAnisotropic3D
- ElasticIsotropic
- FluidSolidPorous
- J2Plasticity
- MultiaxialCyclicPlasticity
- PlaneStress
- PlateFiber
- PressureDependMultiYield
- PressureDependMultiYield02
- PressureDependentElastic3D
- PressureIndependMultiYield
- Template3DElastoPlastic
- ...

Define section: line sections



Define line section: elastic section

Define Elastic Section

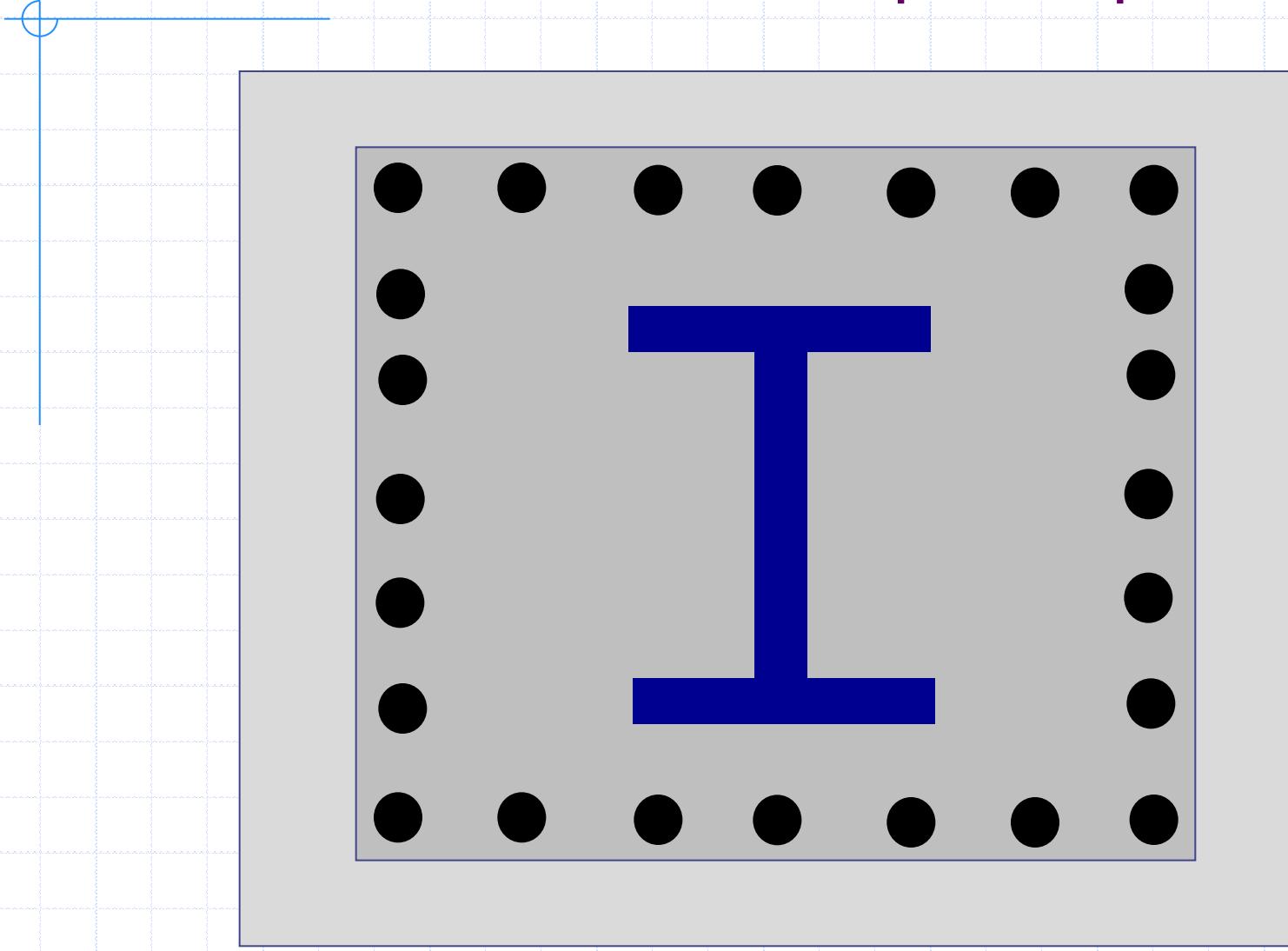
Section Name :	Section01	Add
Modulus of Elasticity (E) :	29000	Database
Cross-Sectional Area (A) :	20	
Moment of Inertia (Iz) :	2000	

If the model is 3D

Define Elastic Section

Section Name :	Section01	Add
Modulus of Elasticity (E) :	29000	Database
Shear Modulus (G) :	29000	
Cross-Sectional Area (A) :	20	
Torsional Moment of Inertia (J) :	1.87	
Moment of Inertia (Iy) :	2000	
Moment of Inertia (Iz) :	2000	

Define fiber section: Composite patch



Define line section: fiber section

Define Fiber Section

Section Name :	<input type="text" value="1stStoryColumn"/>	<input type="button" value="Add"/>
Add Fiber :	<input type="button" value="Fiber"/>	<input type="button"/>
Modify Fiber :	<input type="button"/>	<input type="button"/>
Delete Fiber :	<input type="button"/>	<input type="button"/>
Add Patch :	<input type="button" value="Quadrilateral"/>	<input type="button"/>
Modify Patch :	<input type="button"/>	<input type="button"/>
Delete Patch :	<input type="button"/>	<input type="button"/>
Add Layer :	<input type="button" value="Straight"/>	<input type="button"/>
Modify Layer :	<input type="button"/>	<input type="button"/>
Delete Layer :	<input type="button"/>	<input type="button"/>

Define line section: quadrilateral patch

Define Quadrilateral Patch

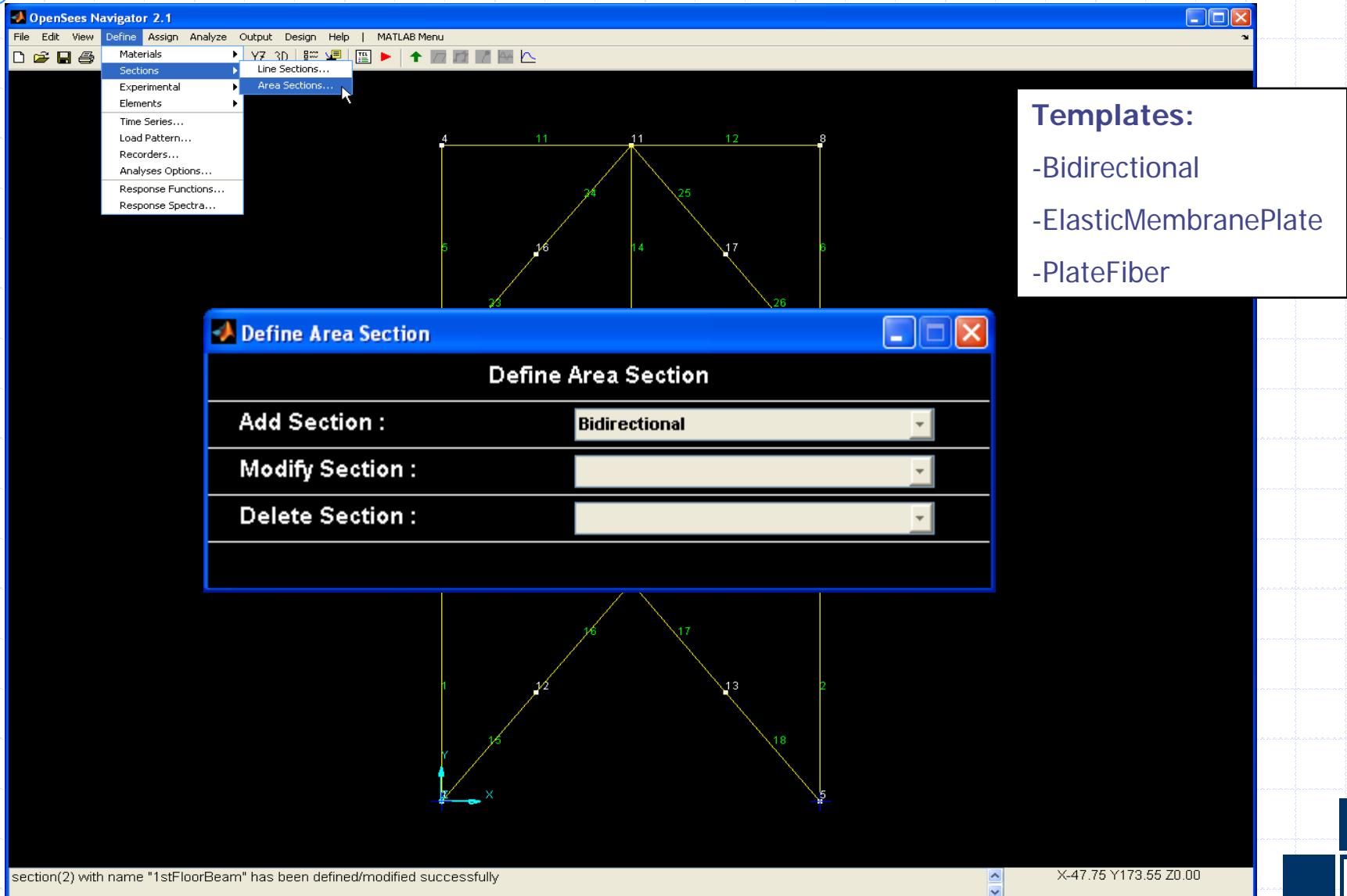
Patch Name :	CoreConcrete	Add
Material Type :	ConfinedConcrete	
Lower Left Corner (yL,zL) :	A50	
	ConfinedConcrete	
	UnconfinedConcrete	
Lower Right Corner (yJ,zJ) :	[0 0]	
Upper Right Corner (yK,zK) :	[0 0]	
Upper Left Corner (yL,zL) :	[0 0]	
Number of Fibers in I-J dir (nfIJ) :	1	
Number of Fibers in J-K dir (nfJK) :	1	
<i>Optional Arguments :</i>		
Counter-Clockwise Rot (Theta) :	0.	

Define fiber section: AISC patch

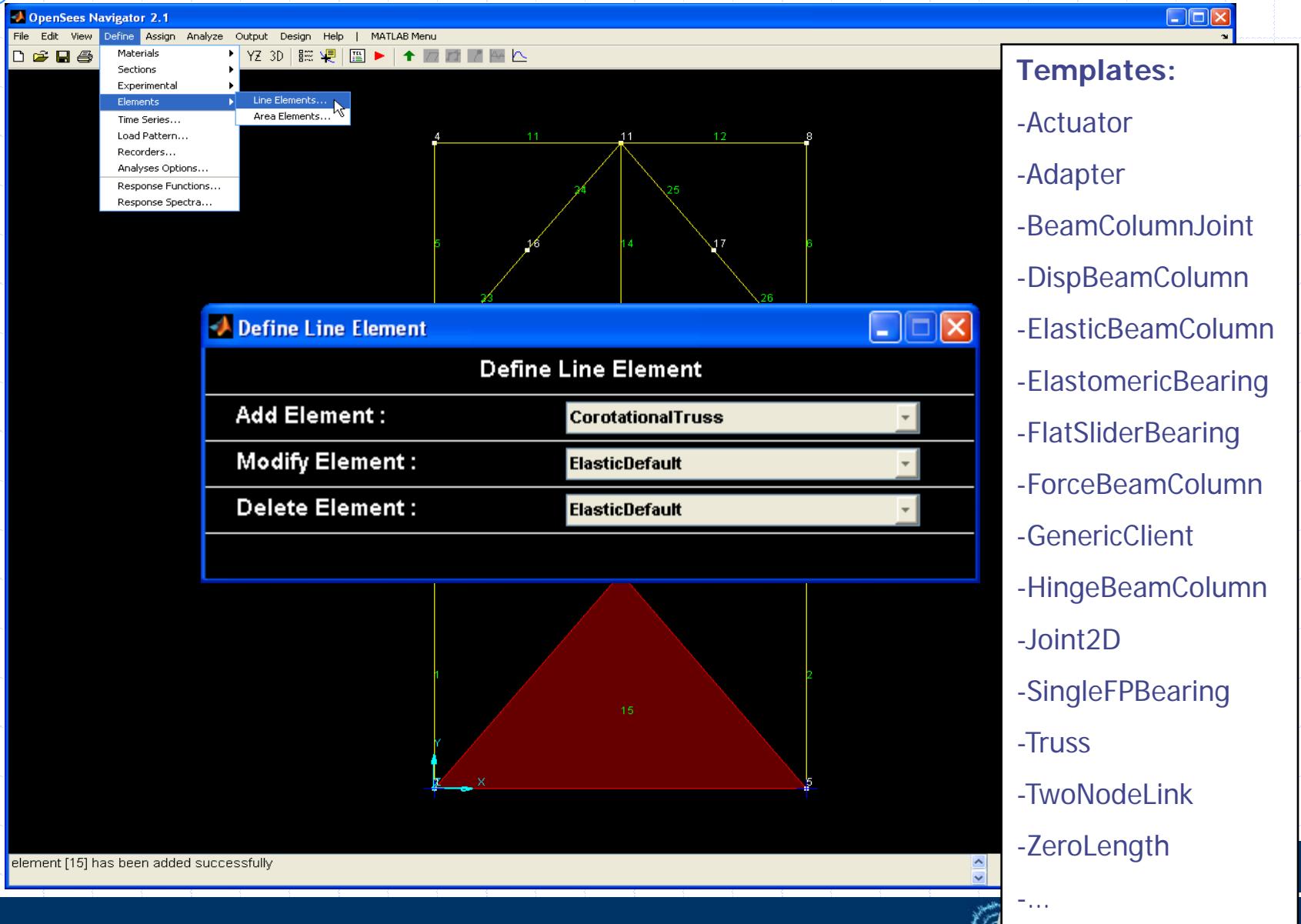
Define AISC Patch

Patch Name :	Patch01	Add
Material Type :	A50	
AISC Section Name :	W24X68	
Number of Fibers along dw (nfdw) :	10	
Number of Fibers along tw (nftw) :	1	
Number of Fibers along bf (nfbf) :	10	
Number of Fibers along tf (nftf) :	1	
<i>Optional Arguments :</i>		
Counter-Clockwise Rot (Theta) :	0.	

Define section: area section



Define element: line element



Define line element: ElasticBeamColumn

Define ElasticBeamColumn Element

Element Name :	EColumn	Add
Modulus of Elasticity (E) :	29000	Database
Cross-Sectional Area (A) :	13.3	
Moment of Inertia (Iz) :	248	

Select Section from Database

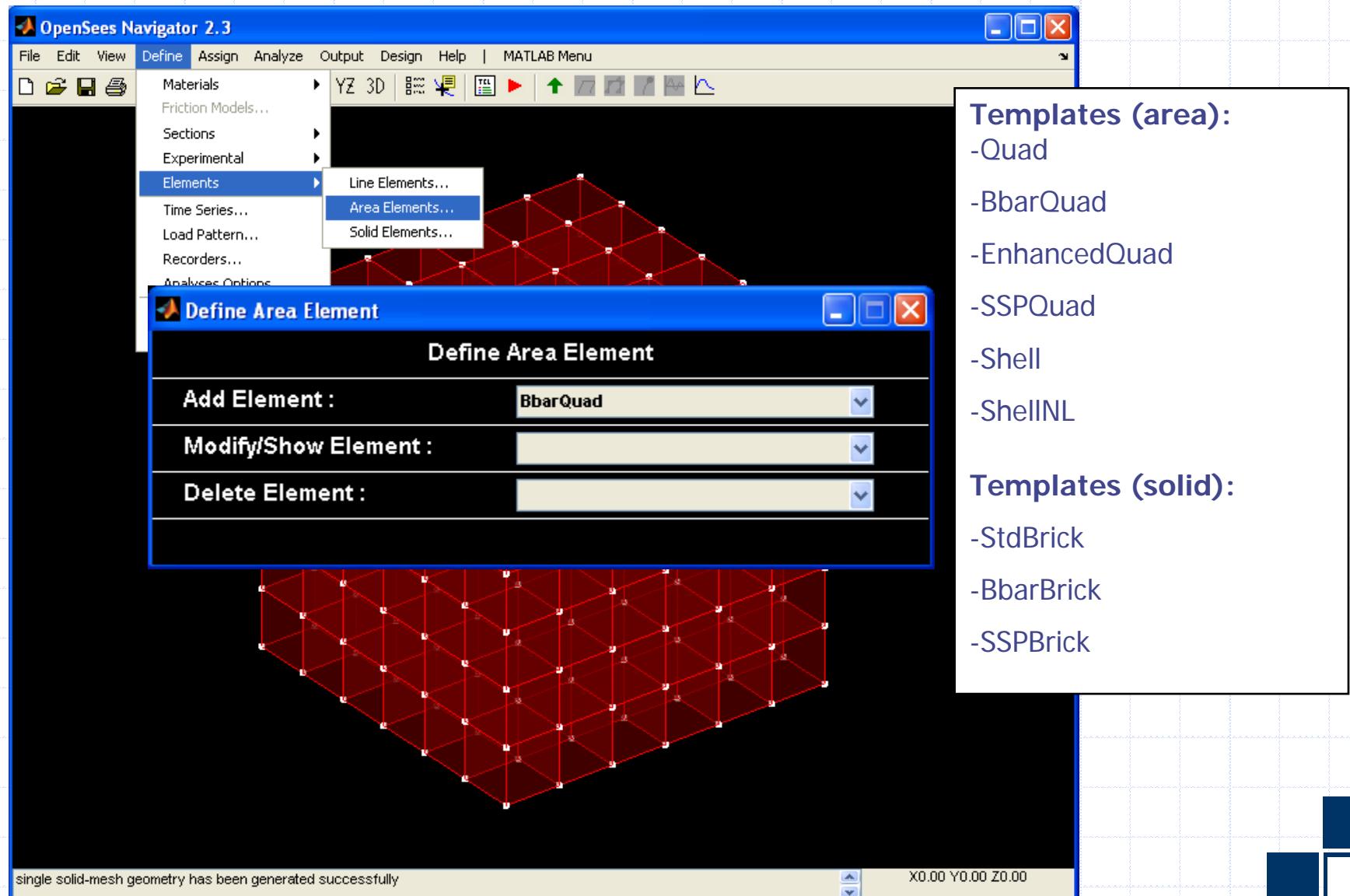
Database :	AISC	Select
Section Name :	W10X45	
Direction :	strong	

Define line element: ForceBeamColumn

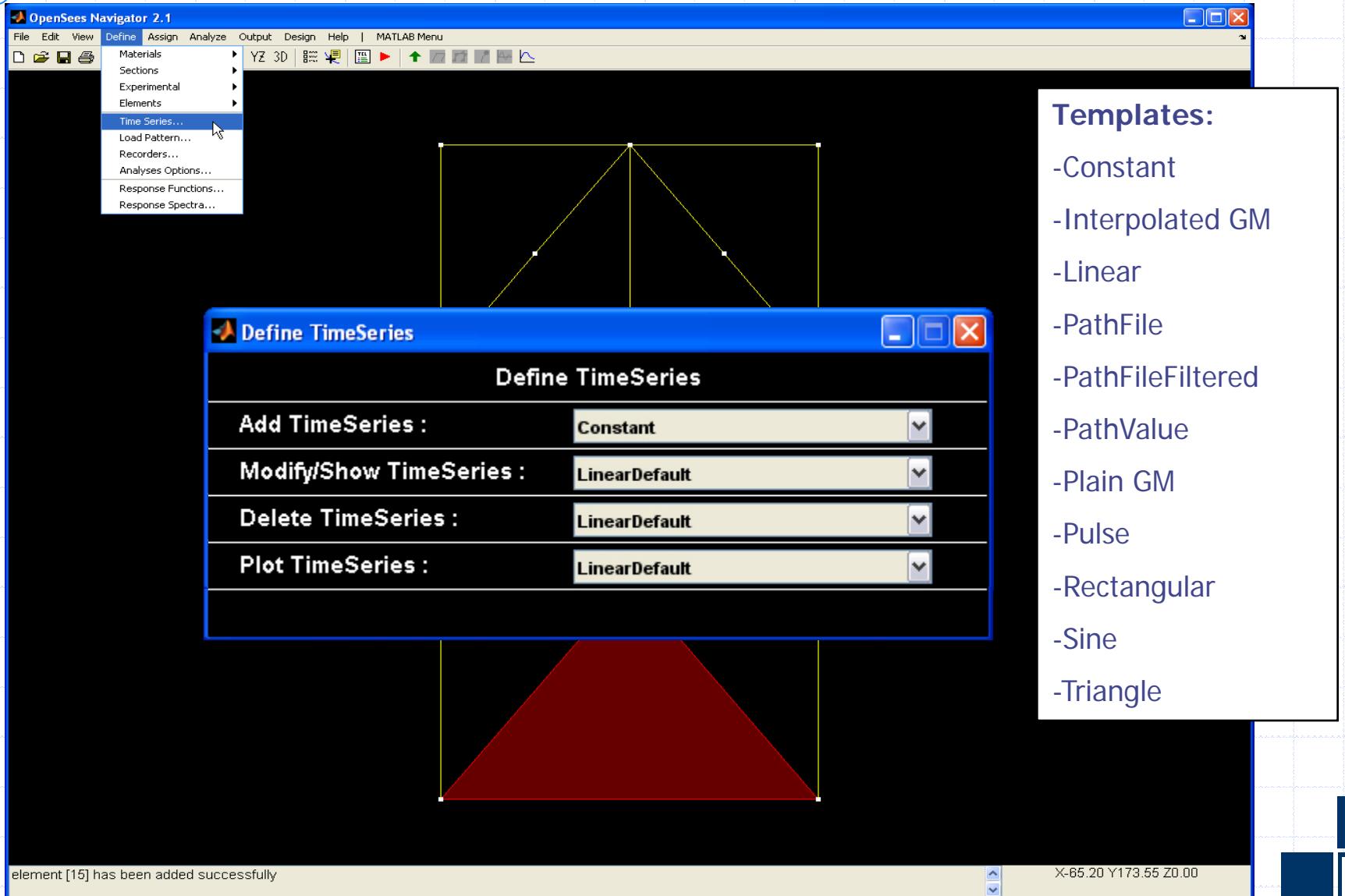
Define ForceBeamColumn Element

Element Name :	1stStoryColumn	Add
Number Intergration Points (NIP) :	5	
Section Type :	1stStoryColumn	
<i>Optional Arguments :</i>		
Mass Density (massDens) :	0.	
Maximum Iterations (maxIters) :	10	
Tolerance (tol) :	1E-8	

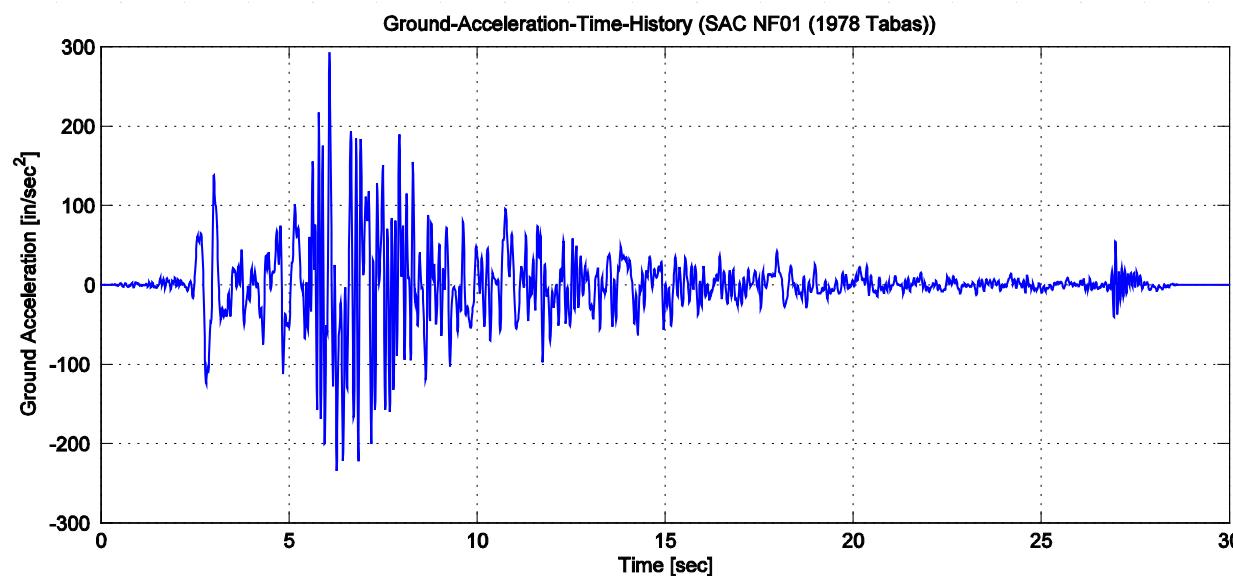
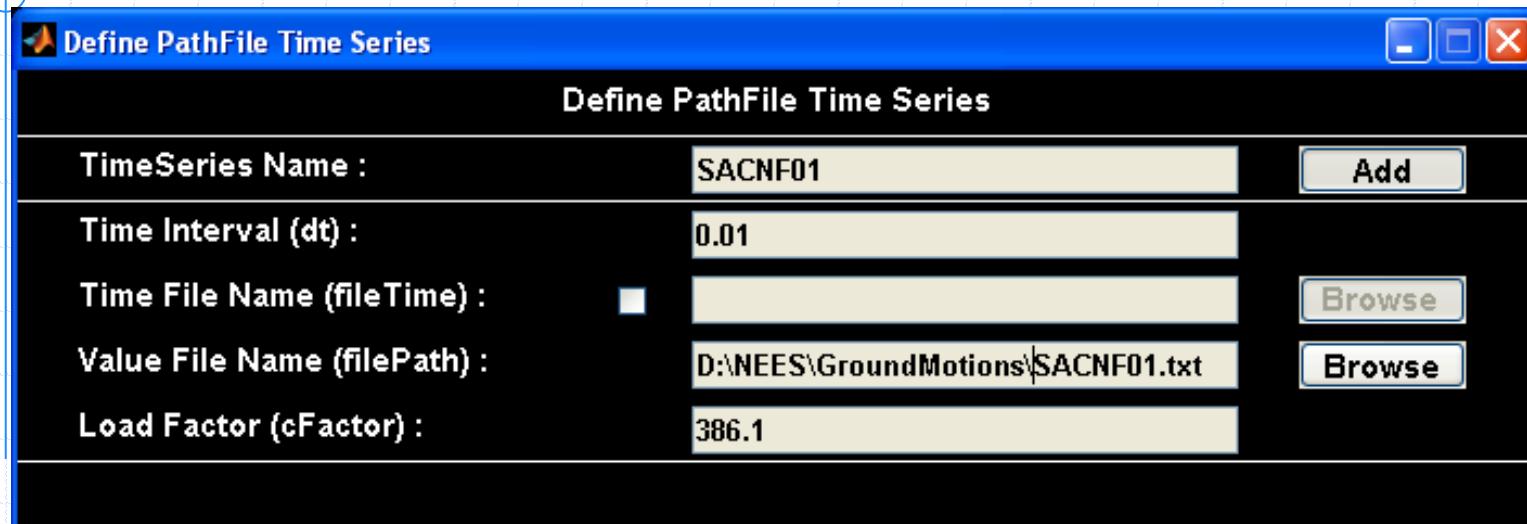
Define element: area and solid elements



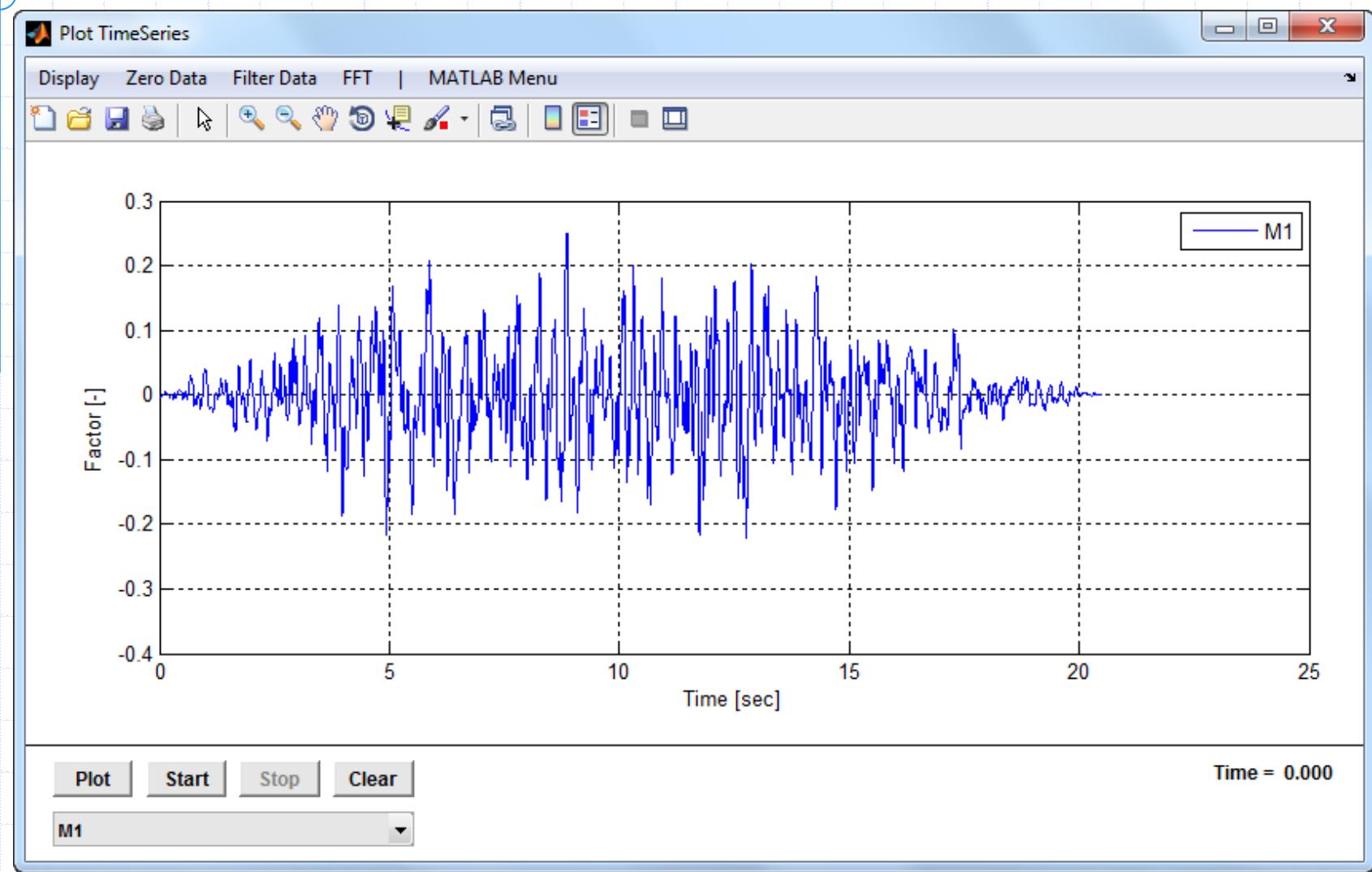
Define TimeSeries:



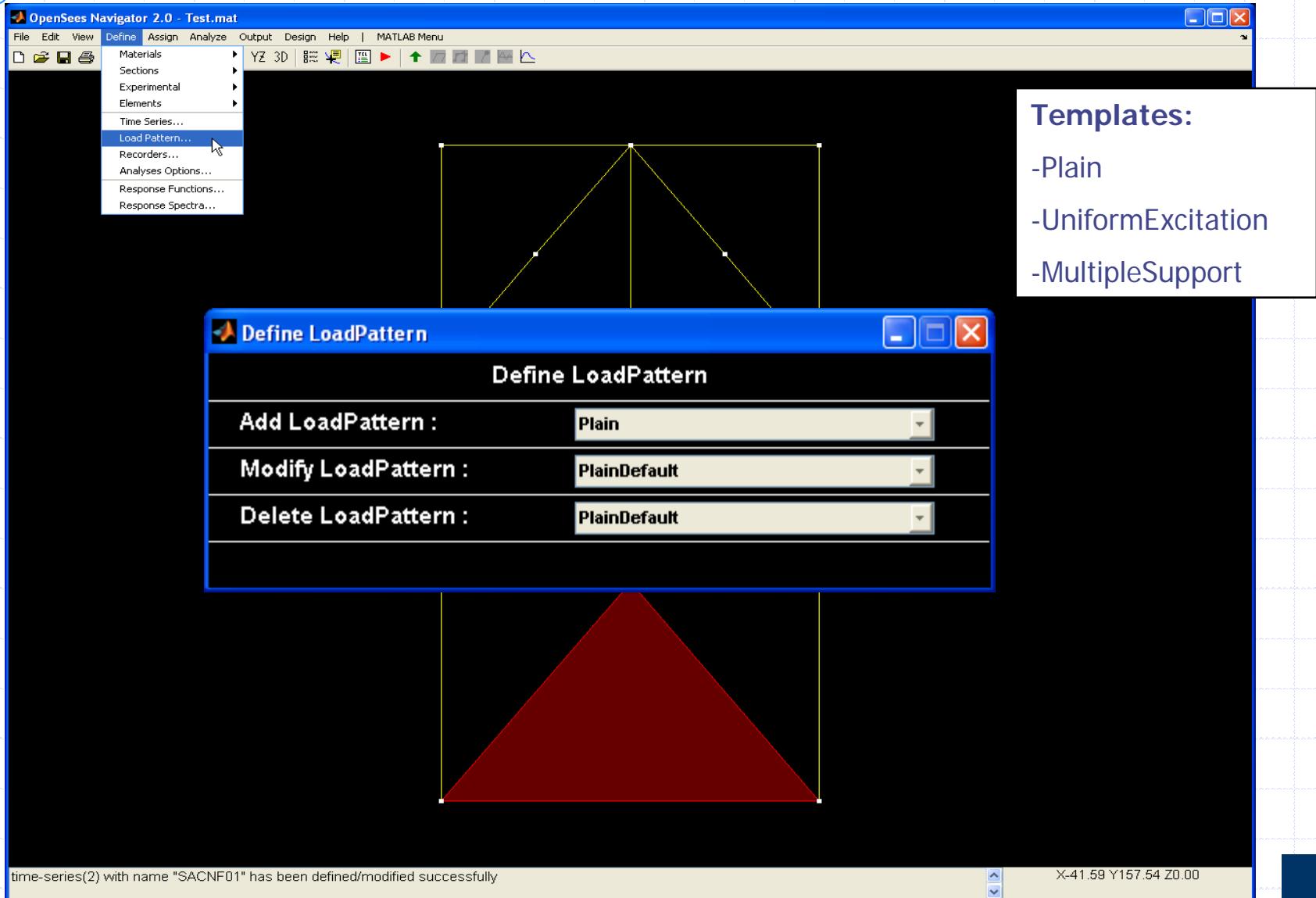
Define TimeSeries: PathFile



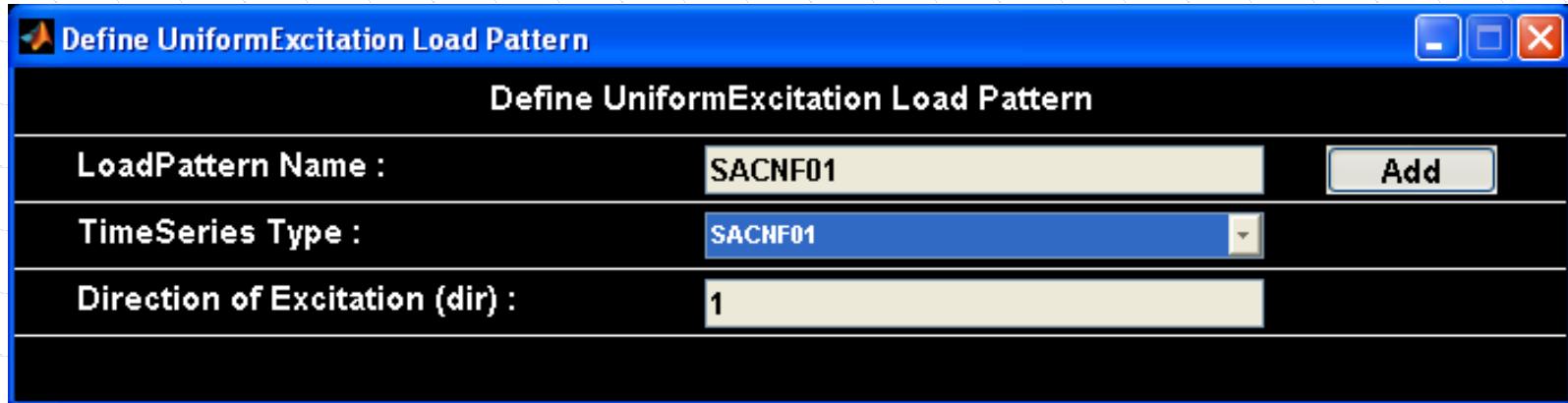
Plot TimeSeries



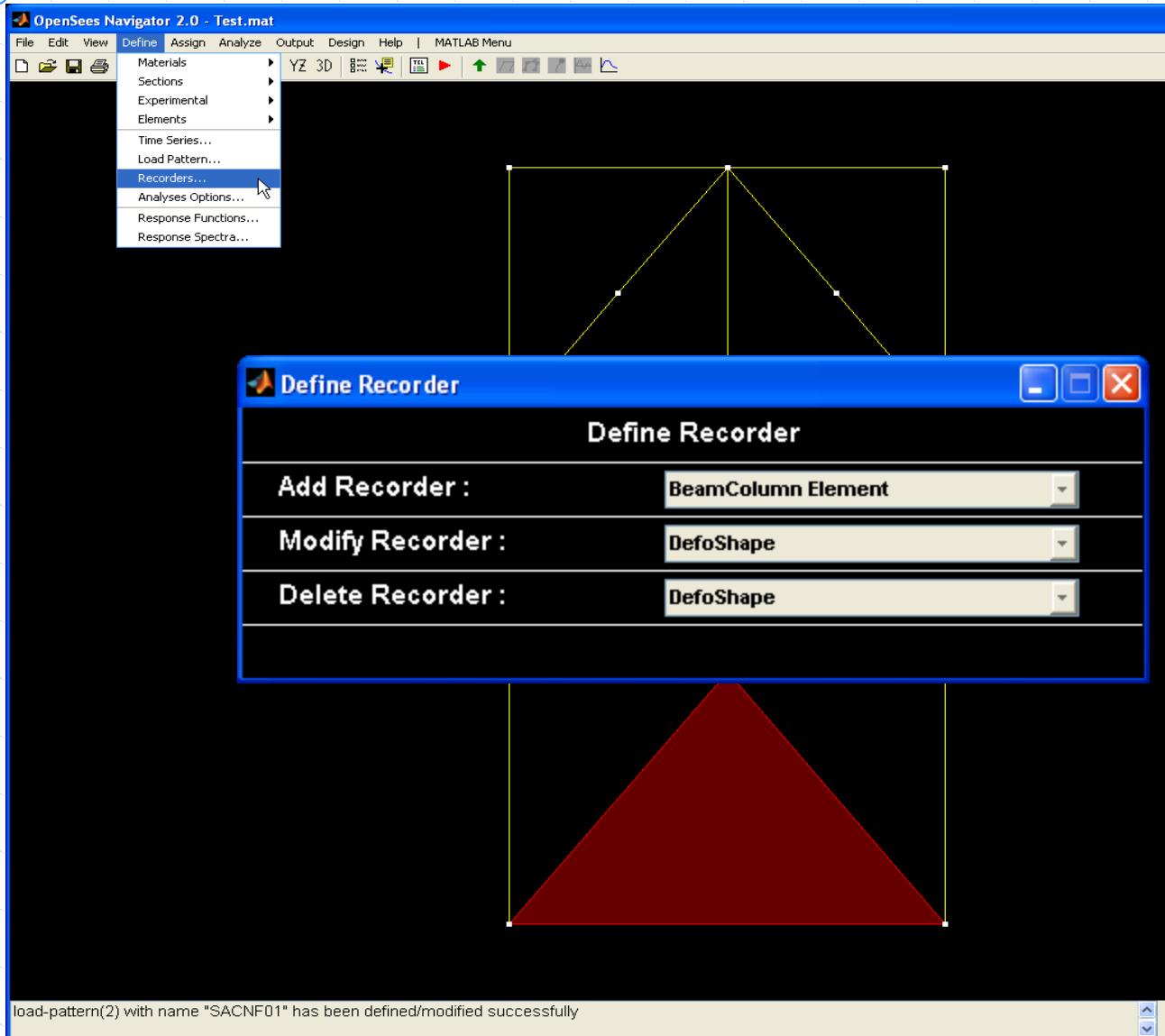
Define LoadPattern:



Define LoadPattern: UniformExcitation



Define recorder



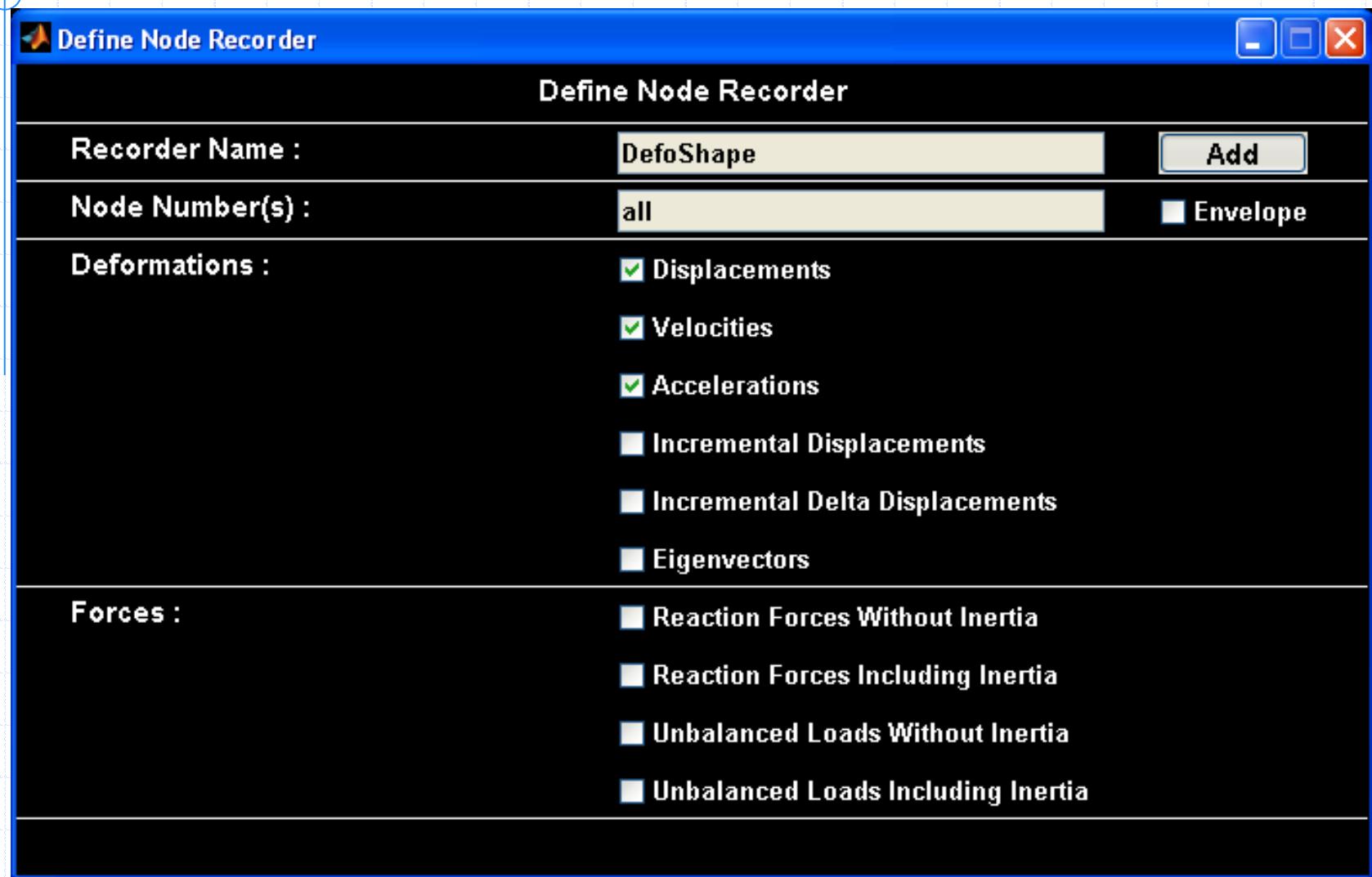
Templates:

- 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

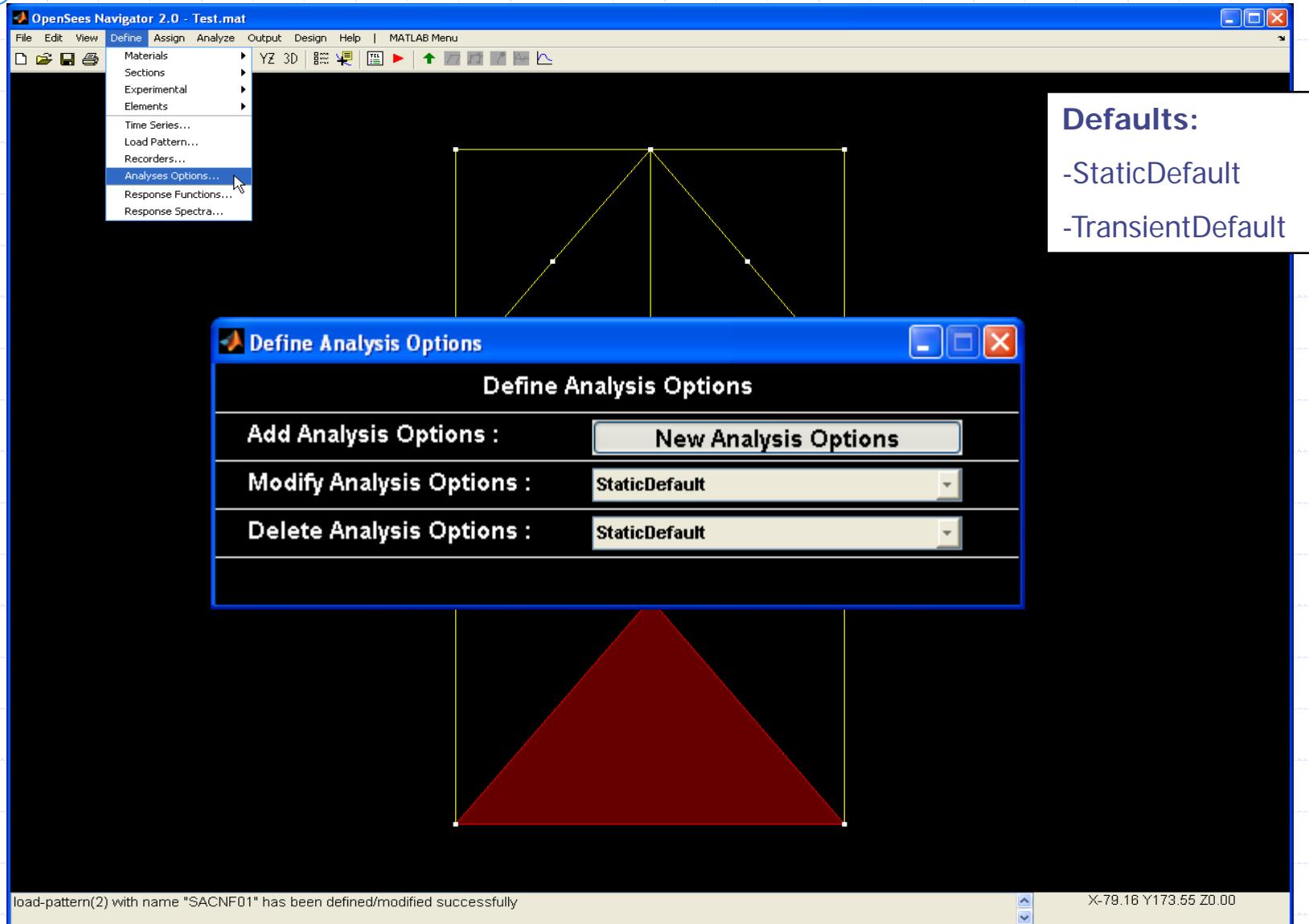


Define recorder: BeamColumn recorder

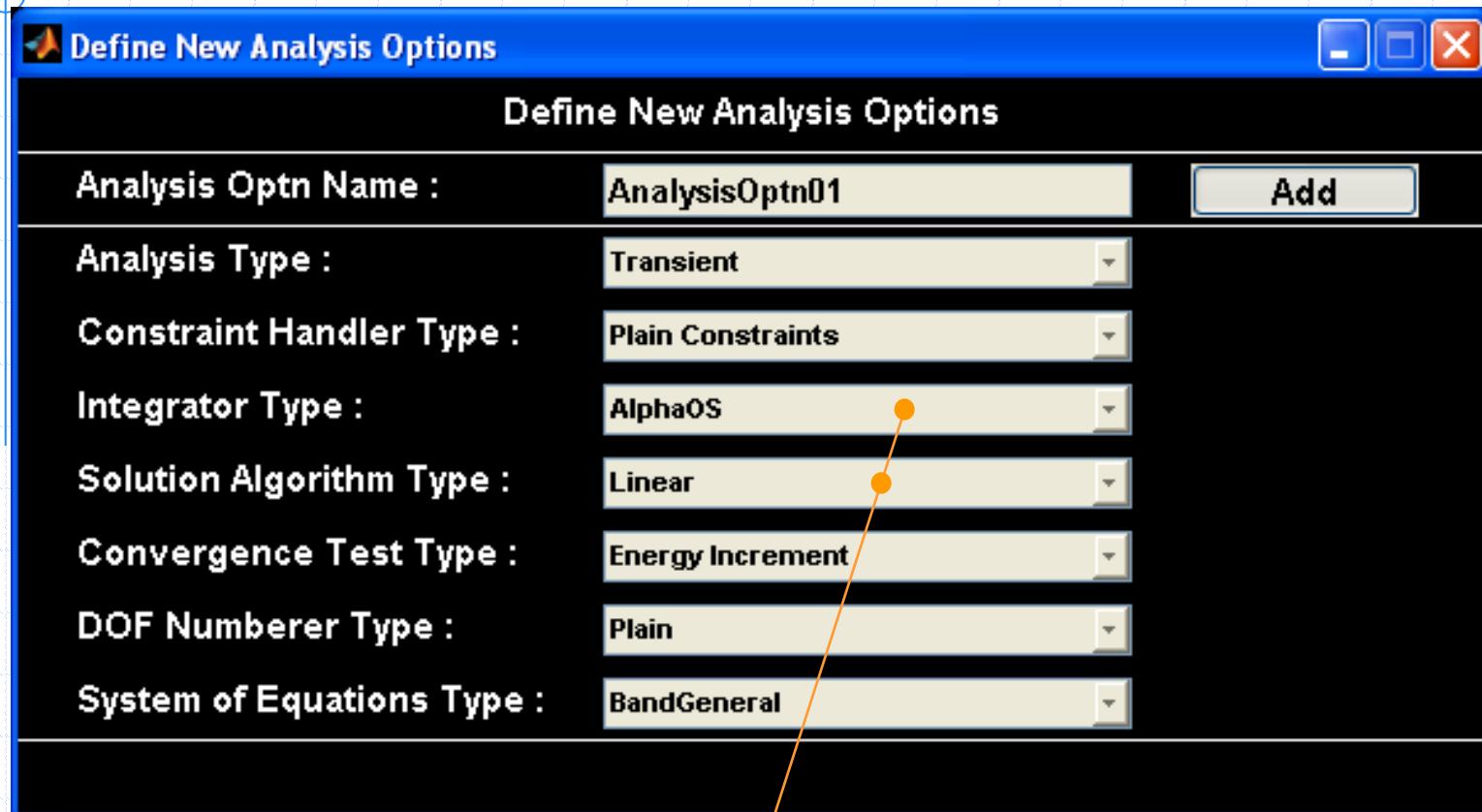
Define BeamColumn Element Recorder

Recorder Name :	ElemForces	Add			
Element Number(s) :	all	<input type="checkbox"/> Envelope			
Arguments :	<input checked="" type="checkbox"/> Global Resisting Forces <input checked="" type="checkbox"/> Local Resisting Forces				
Section Response :	<input checked="" type="radio"/> Section				
Section Number(s) :	1 5				
Arguments :	<input checked="" type="checkbox"/> Forces <input type="checkbox"/> Stiffness	<input checked="" type="checkbox"/> Deformations			
Fiber Response :	<input checked="" type="radio"/> Fiber				
Arguments :	<input type="checkbox"/> Stress/Strain	0.	Y-Coor	0.	Z-Coor

Define analysis options



Define analysis options: new analysis



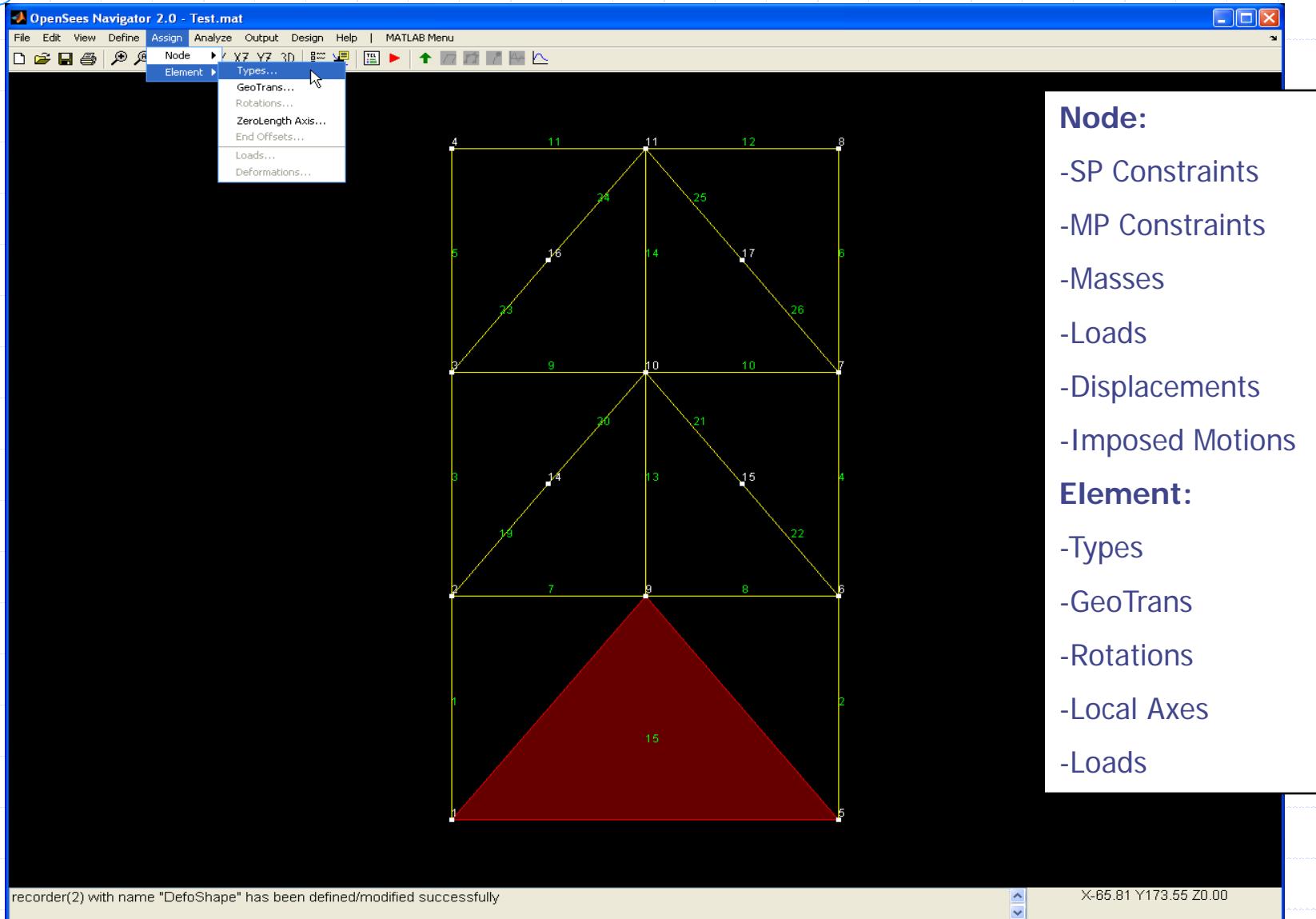
Integrator Type:

For example use AlphaOS Method for Hybrid Simulation

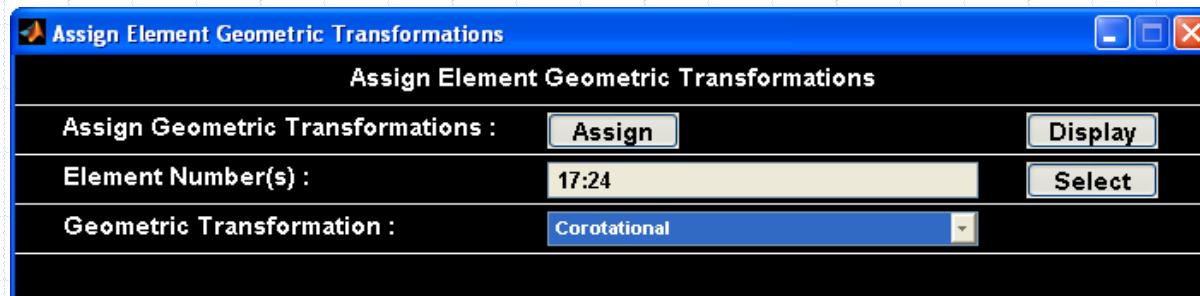
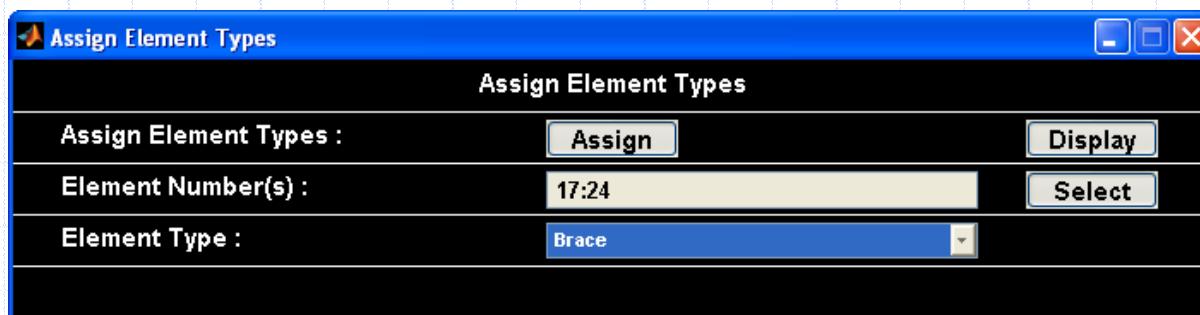
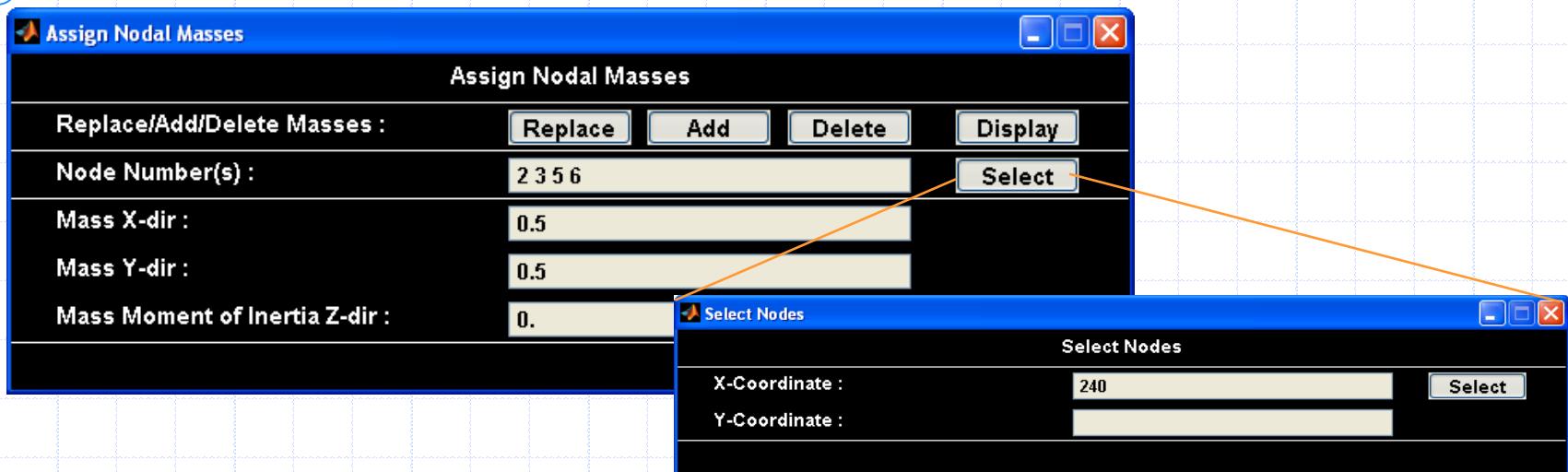
Solution Algorithm:

The AlphaOS Method requires a Linear solution algorithm

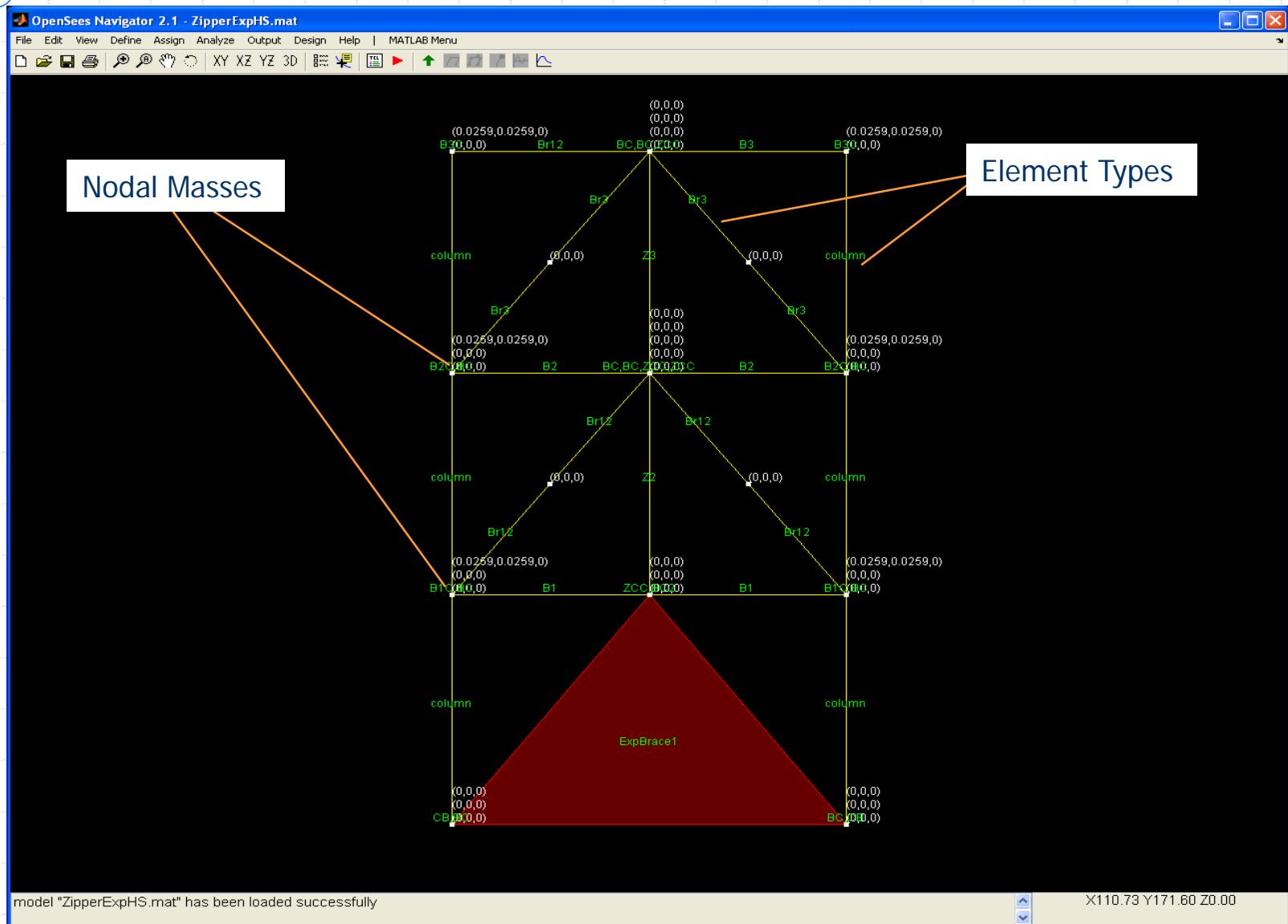
Assign menu



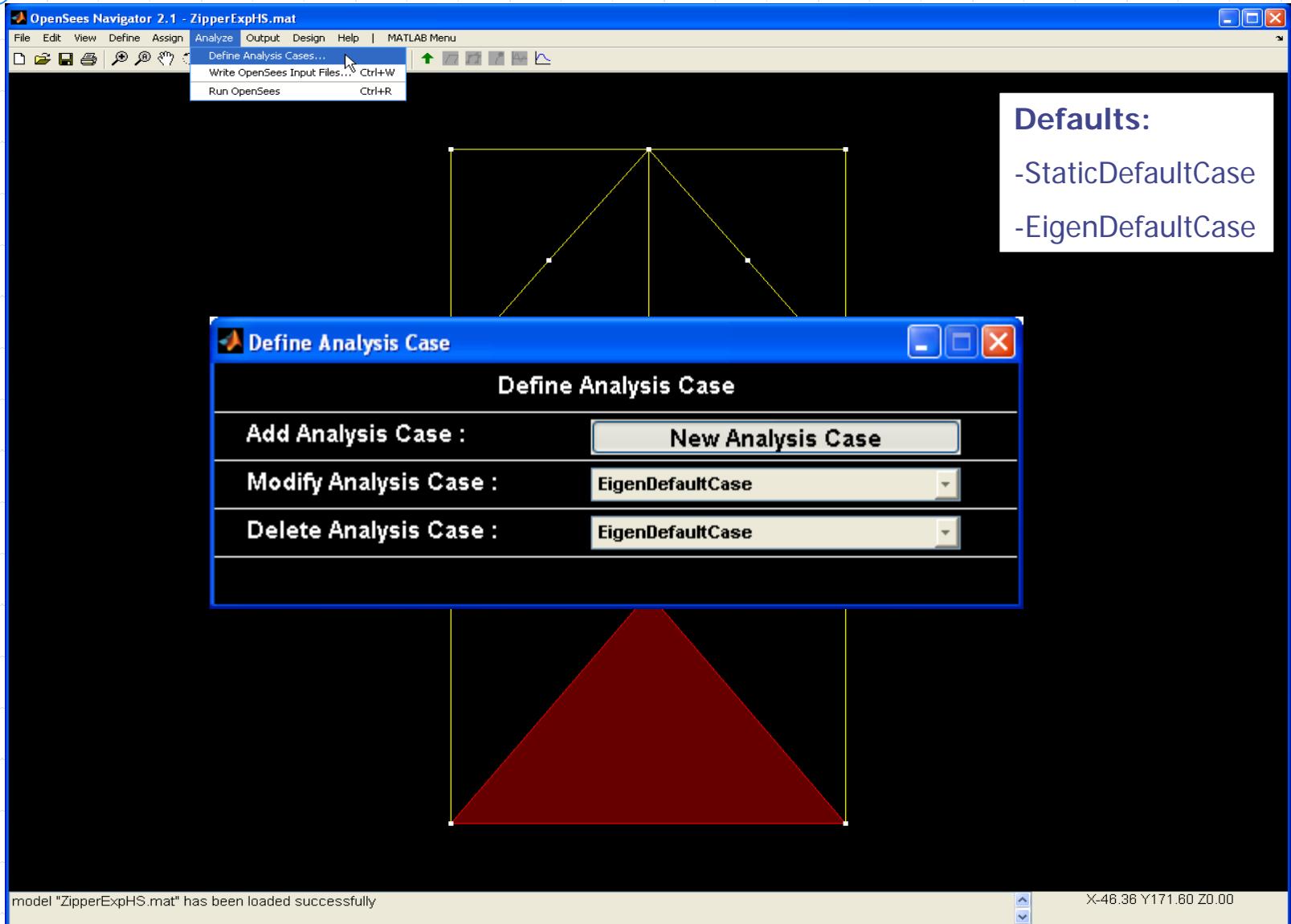
Assign menu



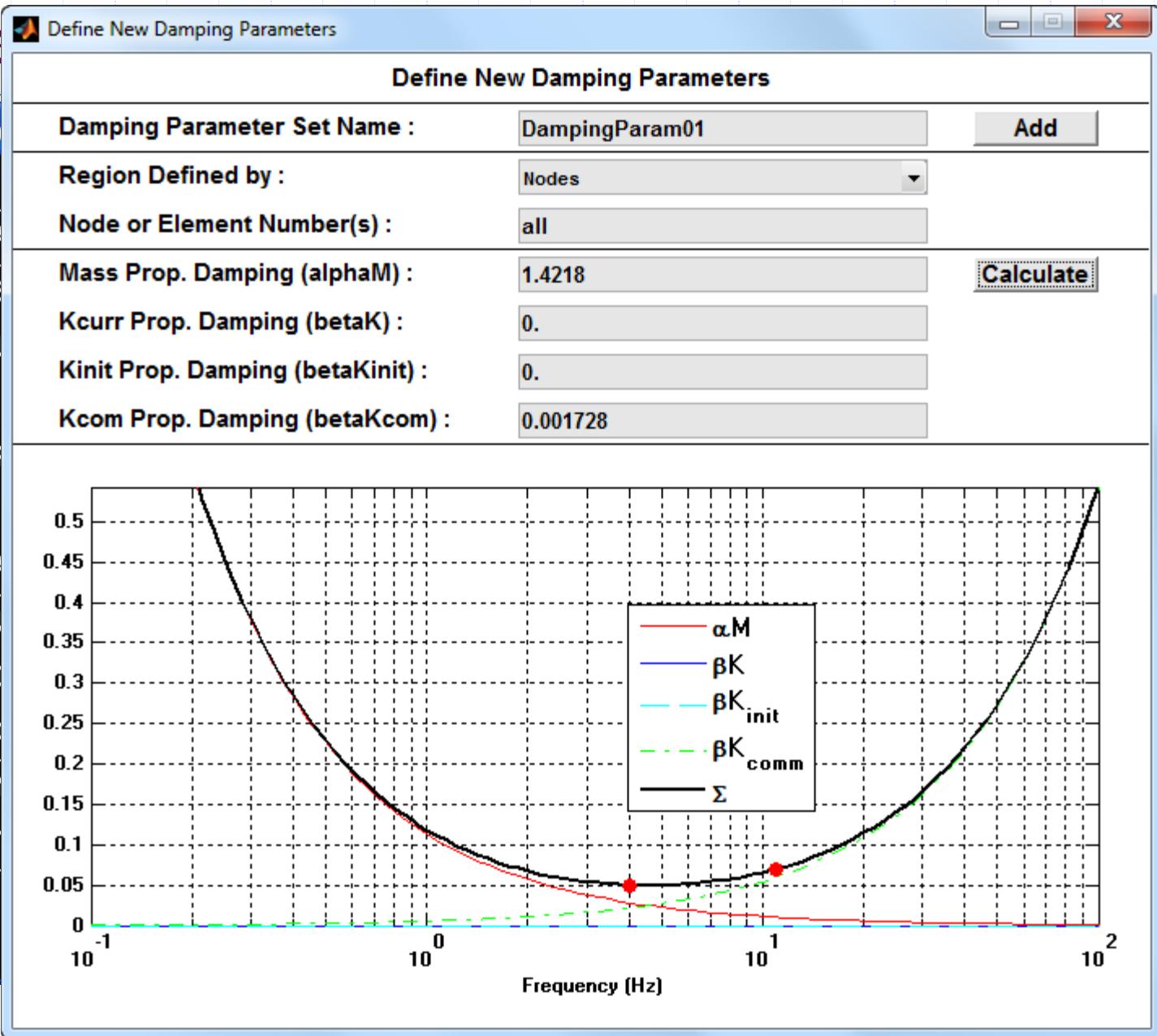
Display assigned properties



Define analysis case



De



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]

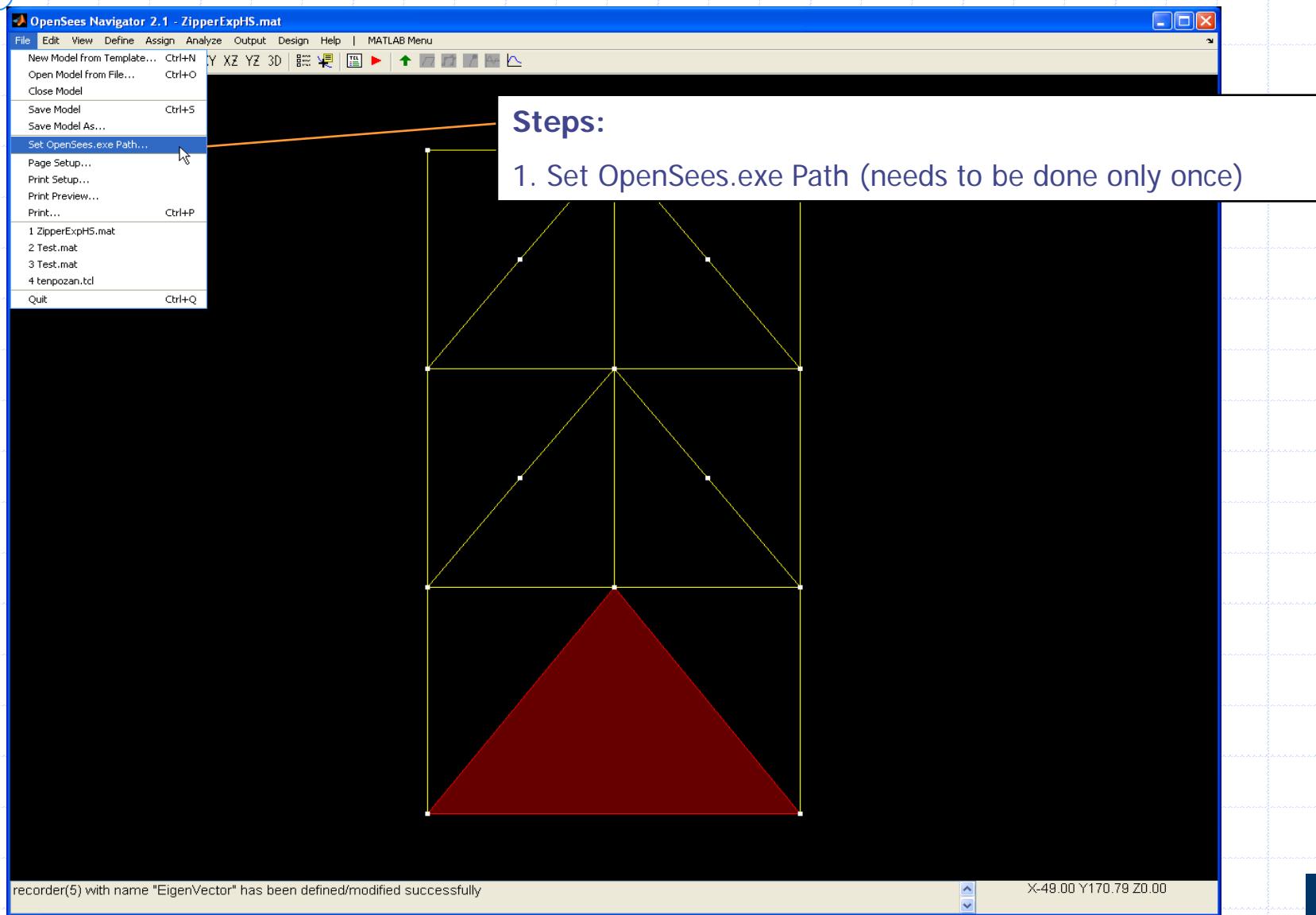
while ($ok == 0 && $tCurrent < $tFinal) {
    if (fmod($tCurrent,1) < 1.0E-16) {
        puts "$i $tCurrent"
    }
    set ok [analyze 1 $dt]

    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
    }

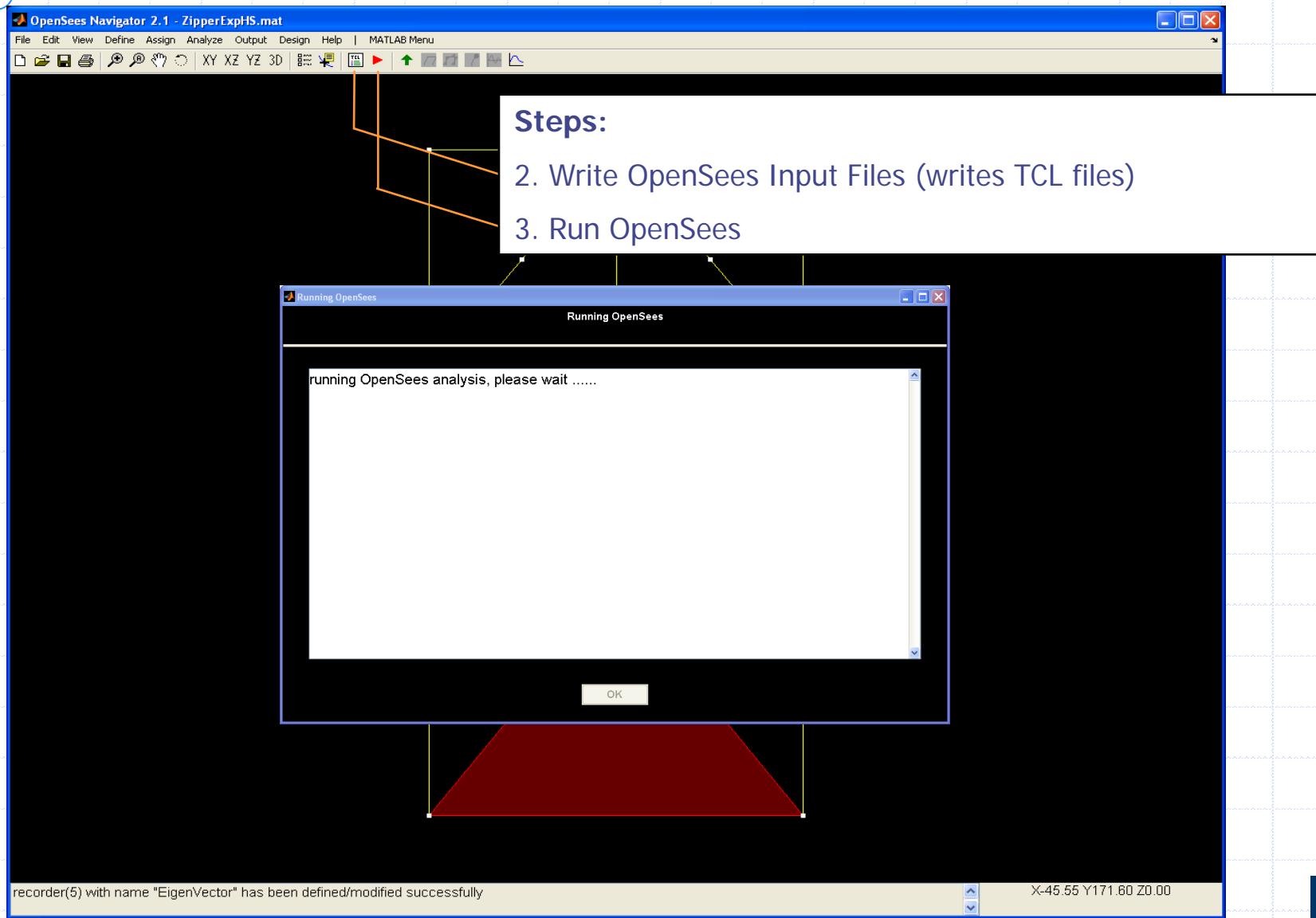
    if ($ok != 0) {
        puts ""
        puts [format "Newton failed (time = %1.3e), try Newton w/ iniCurrent" $tCurrent]
        algorithm Newton -initialCurrent
        test $testType $testTol $testIter 0
        set ok [analyze 1 $dt]
        algorithm $algoType
    }

    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
    }
}
```

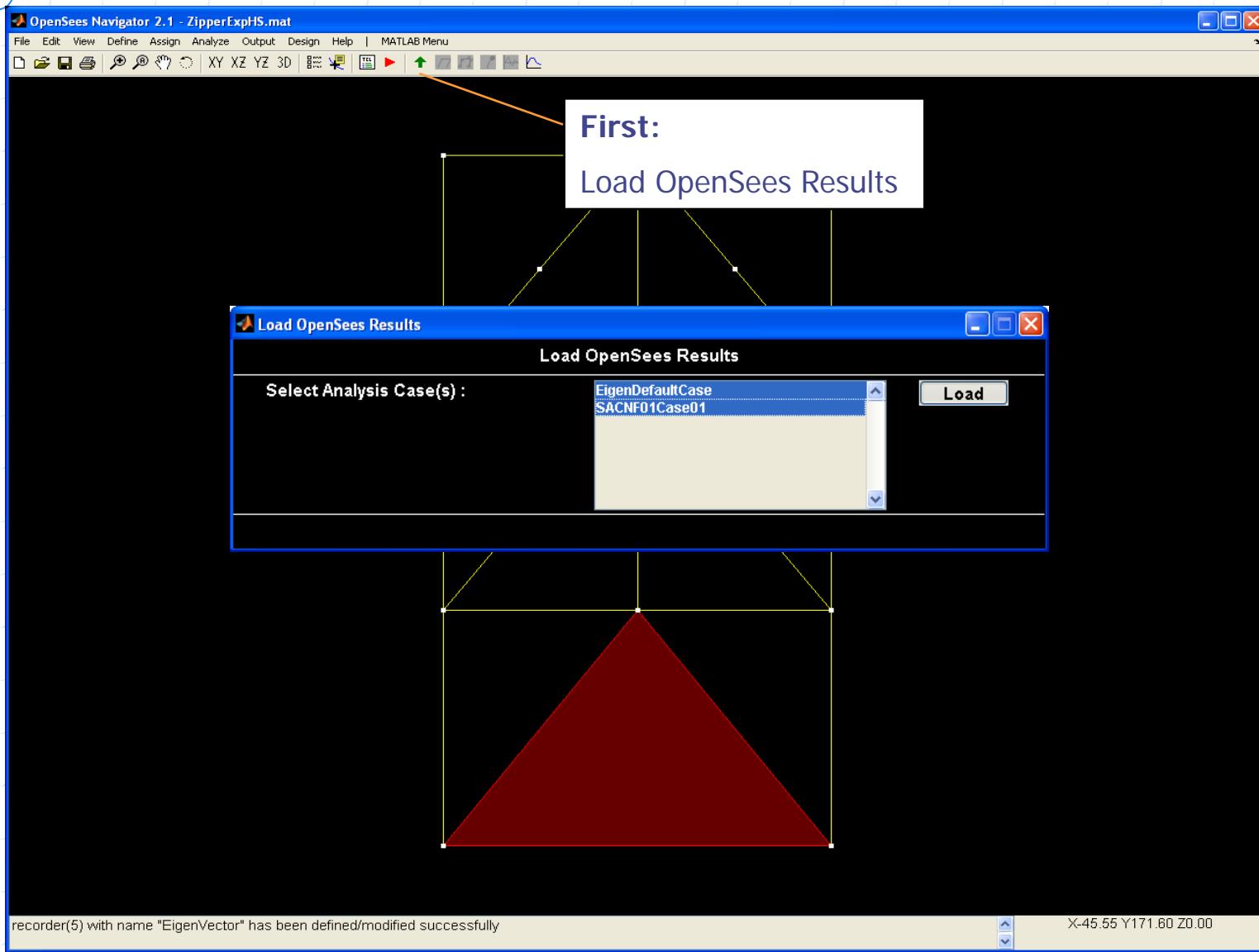
Run OpenSees: set OpenSees.exe path



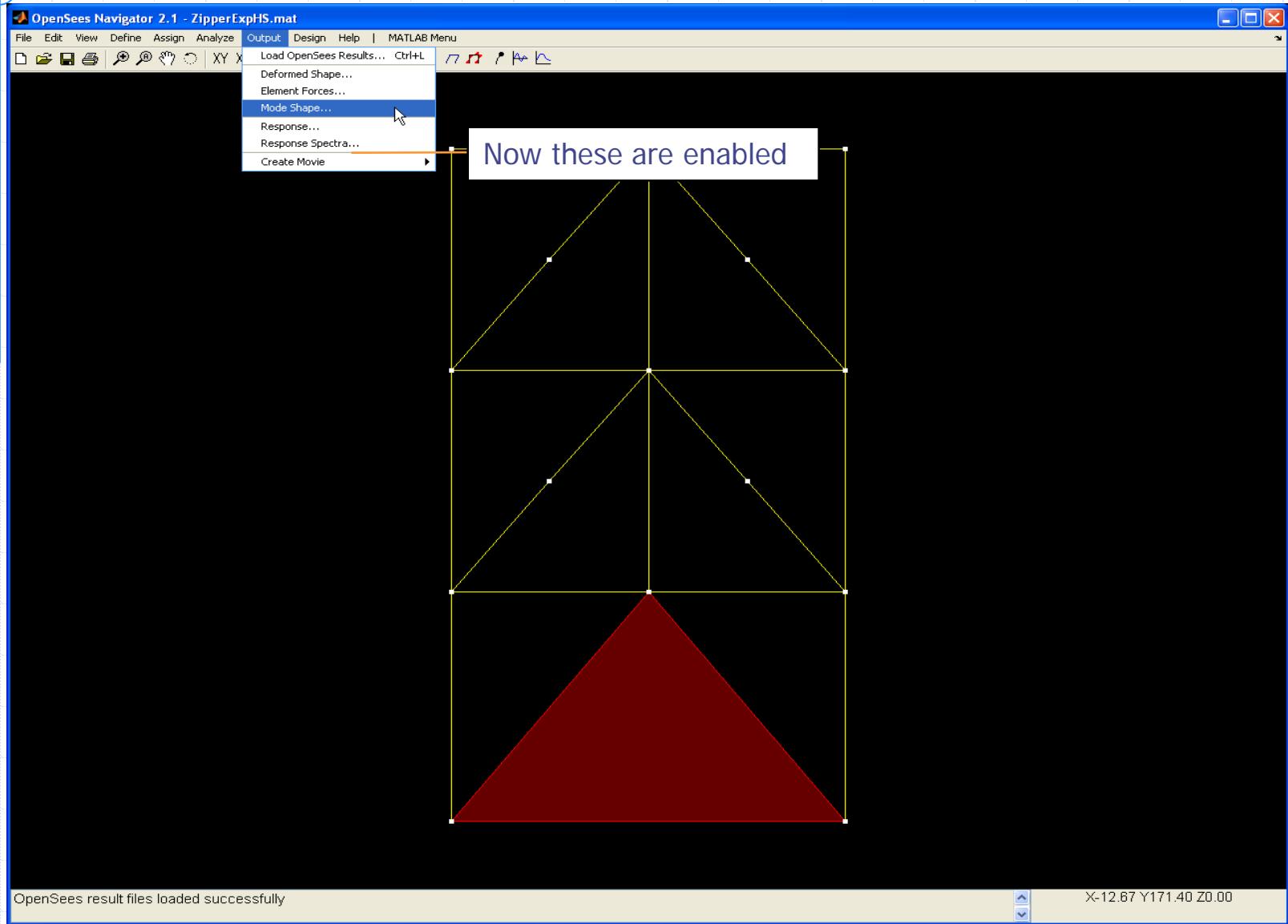
Run OpenSees: write TCL files



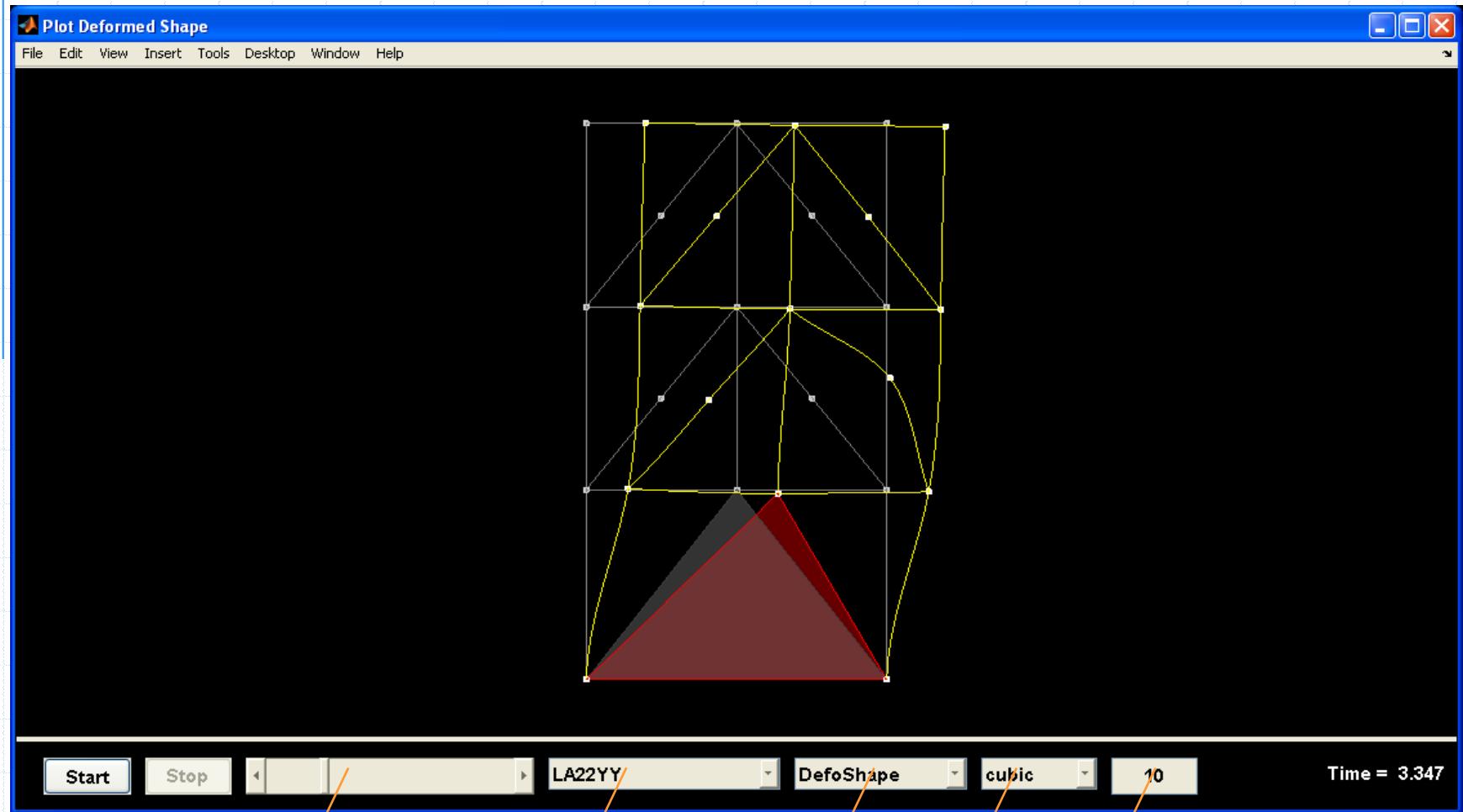
Post processing: load results



Post processing: output



Post processing: plot deformed shape



Time Step

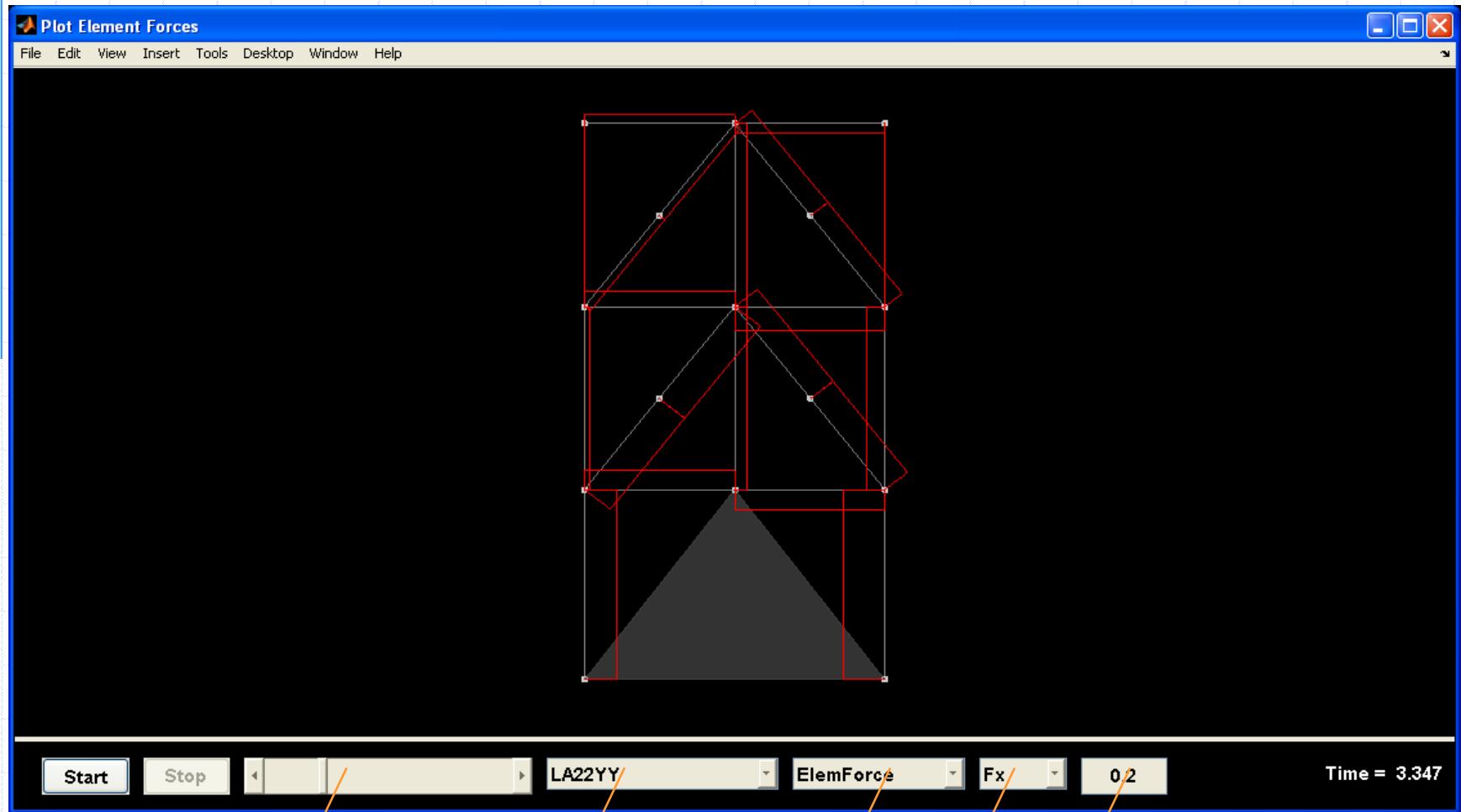
AnalysisCase

Recorder

Order

Magnification

Post processing: plot element forces



Time Step

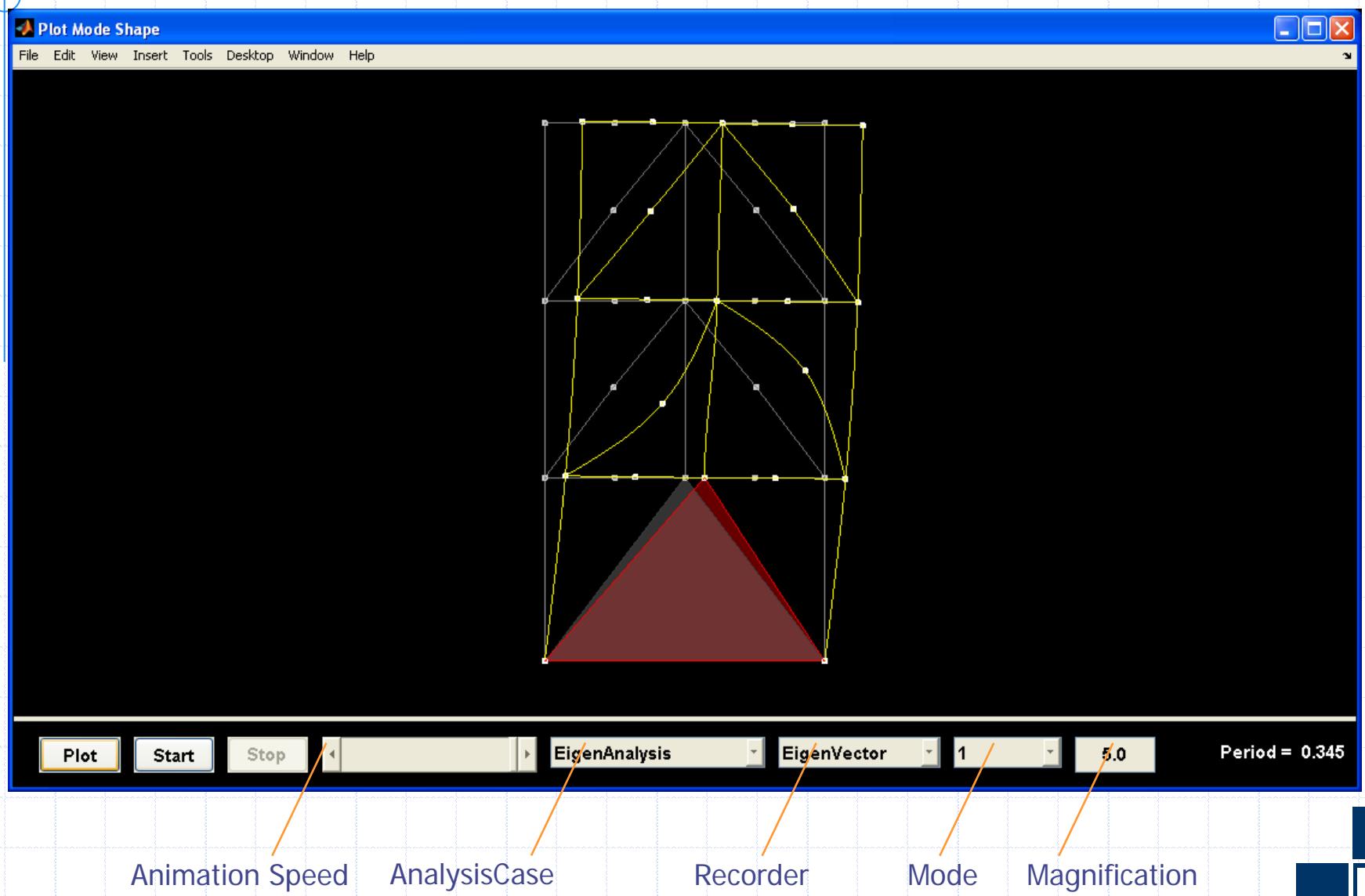
AnalysisCase

Recorder

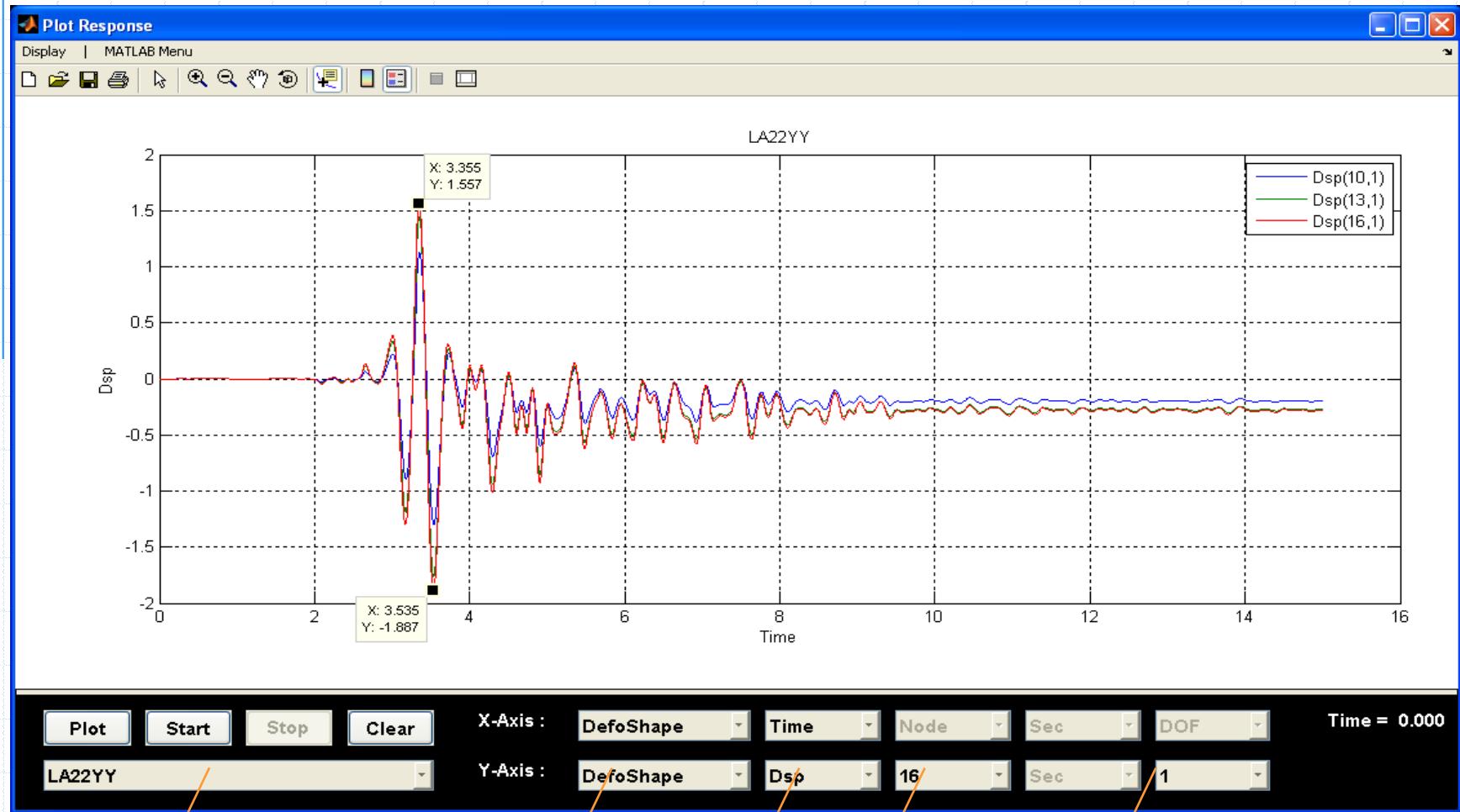
Response

Magnification

Post processing: plot mode shape



Post processing: plot response histories



AnalysisCase

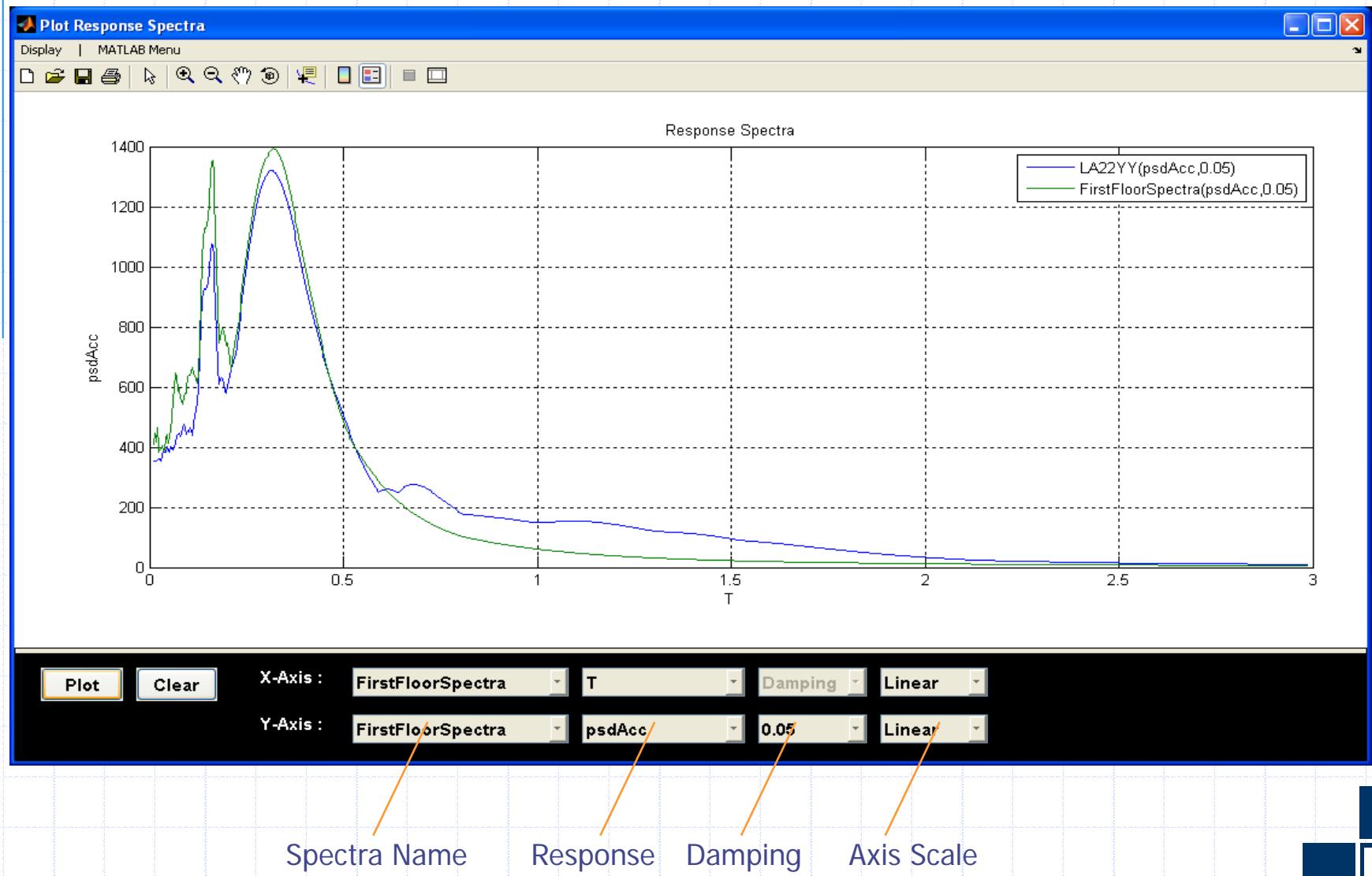
Recorder

Parameter

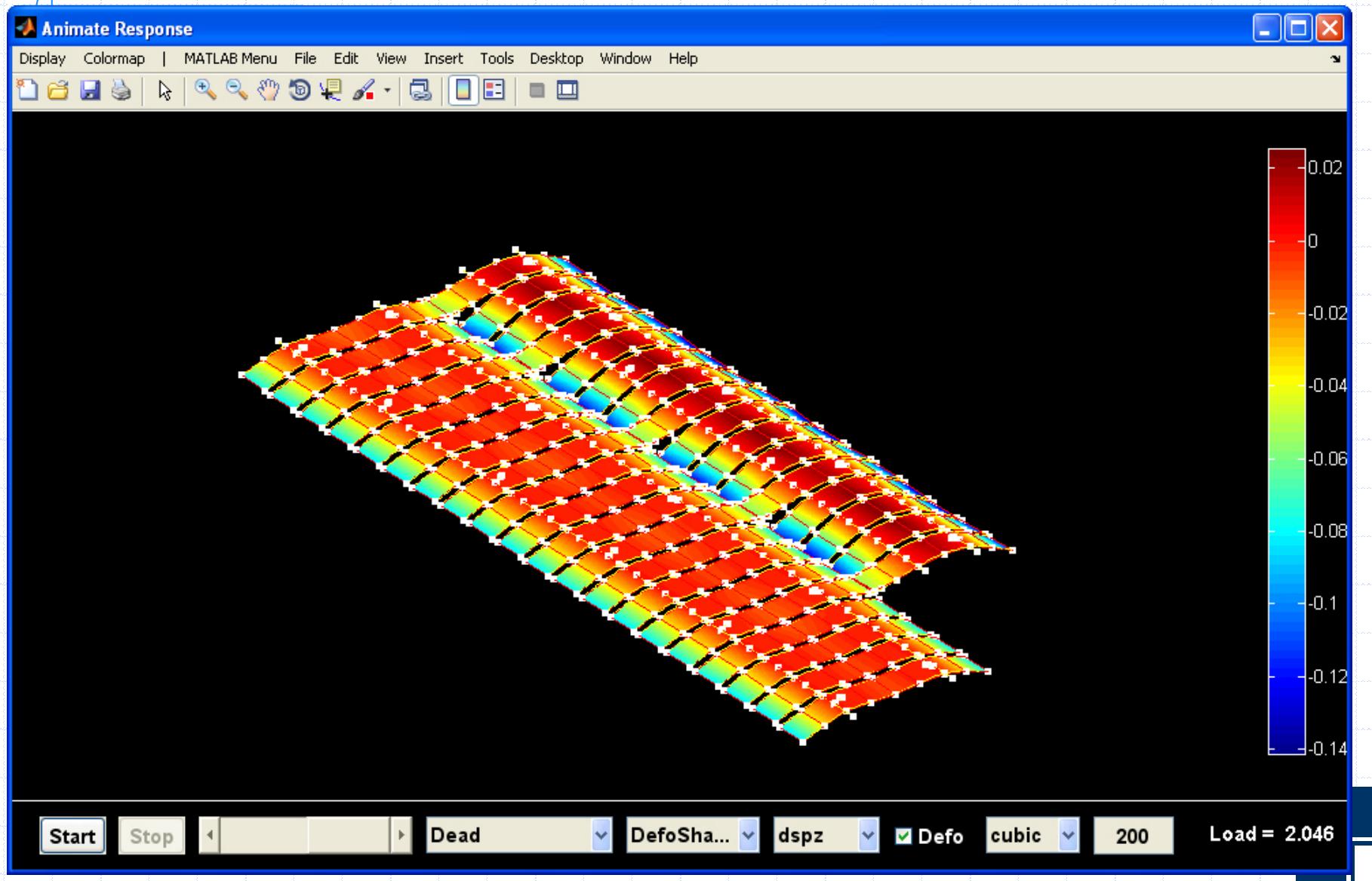
Node/Element

DOF

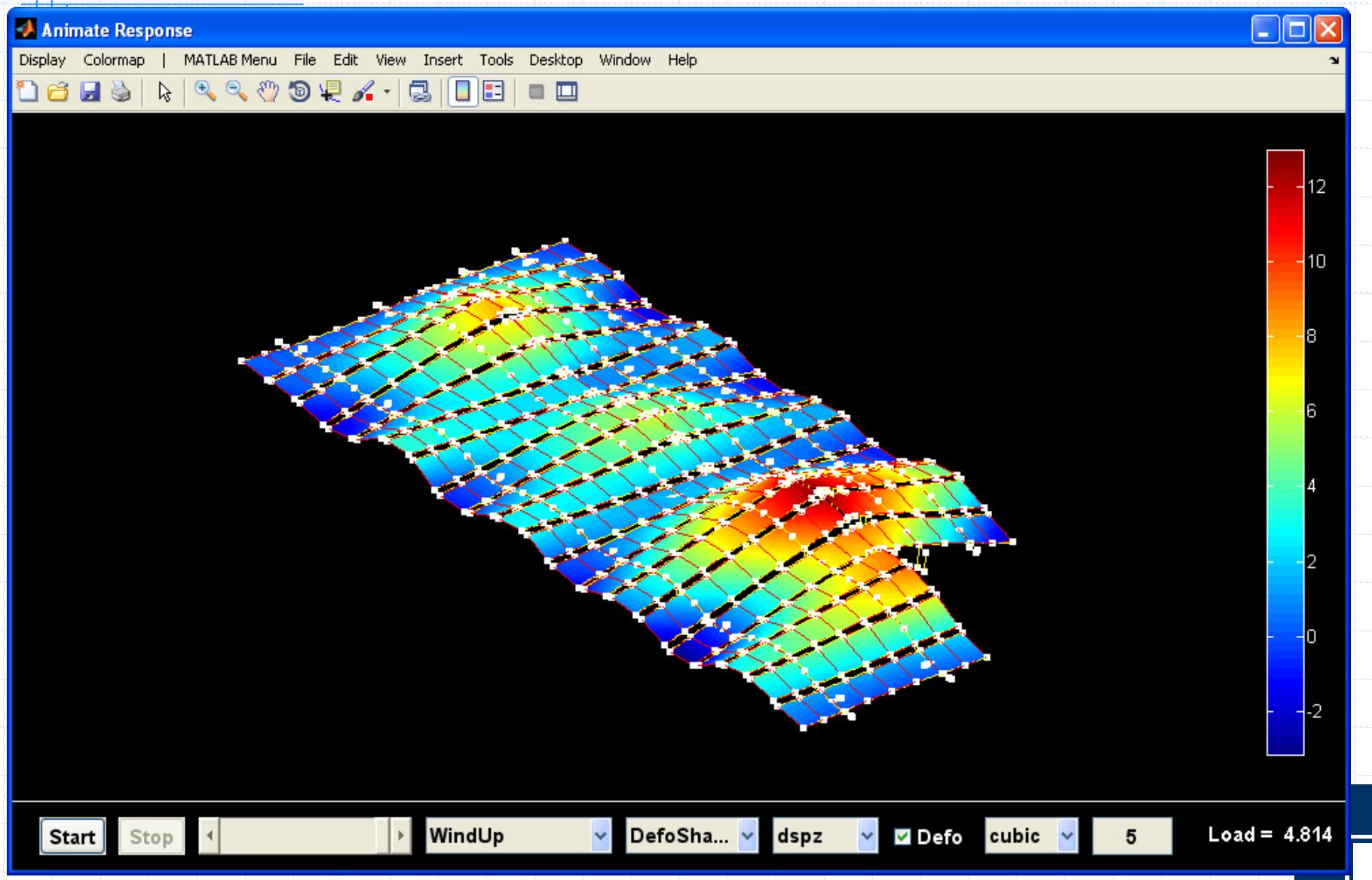
Post processing: plot response spectra



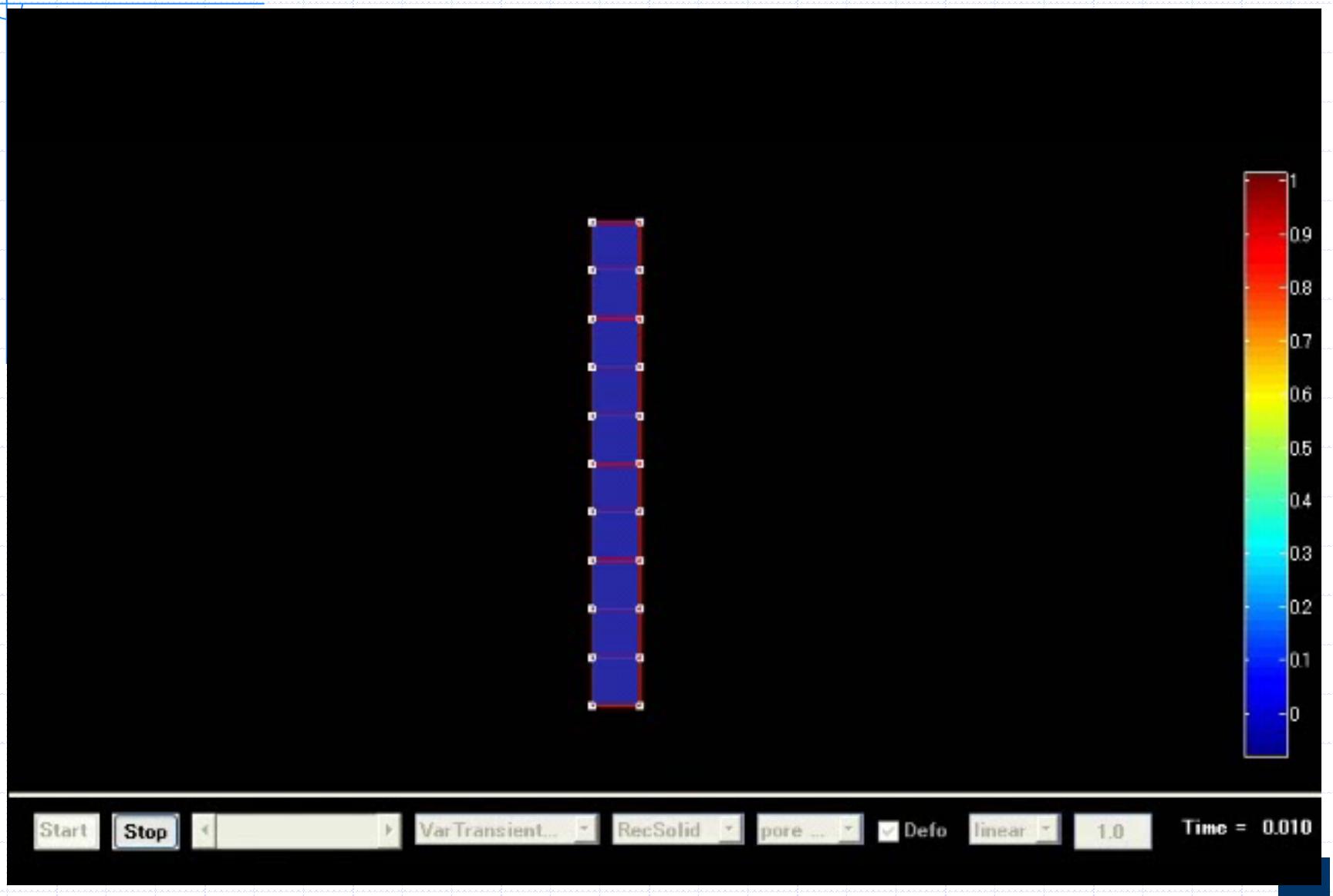
Post processing: animate response



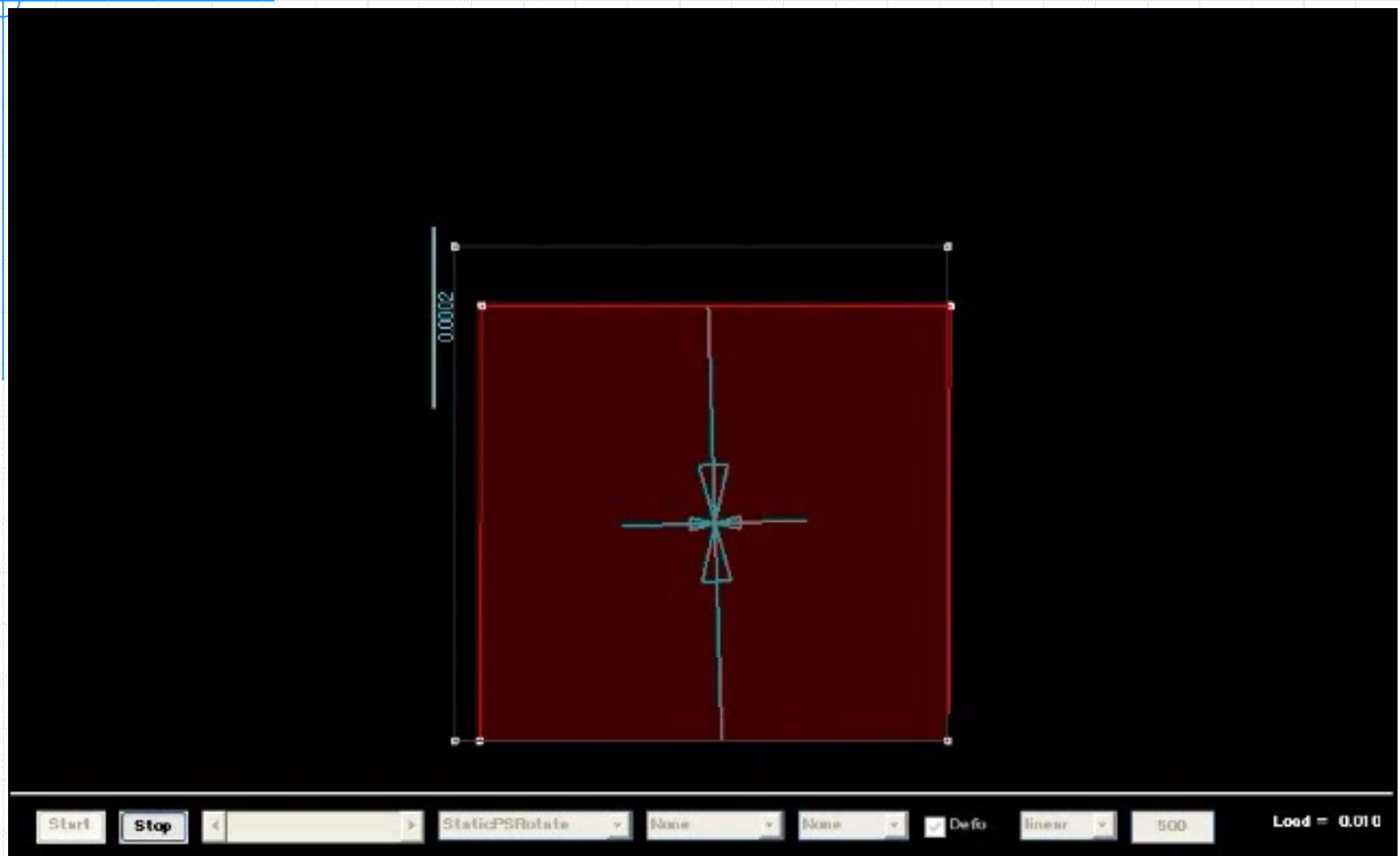
Post processing: animate response



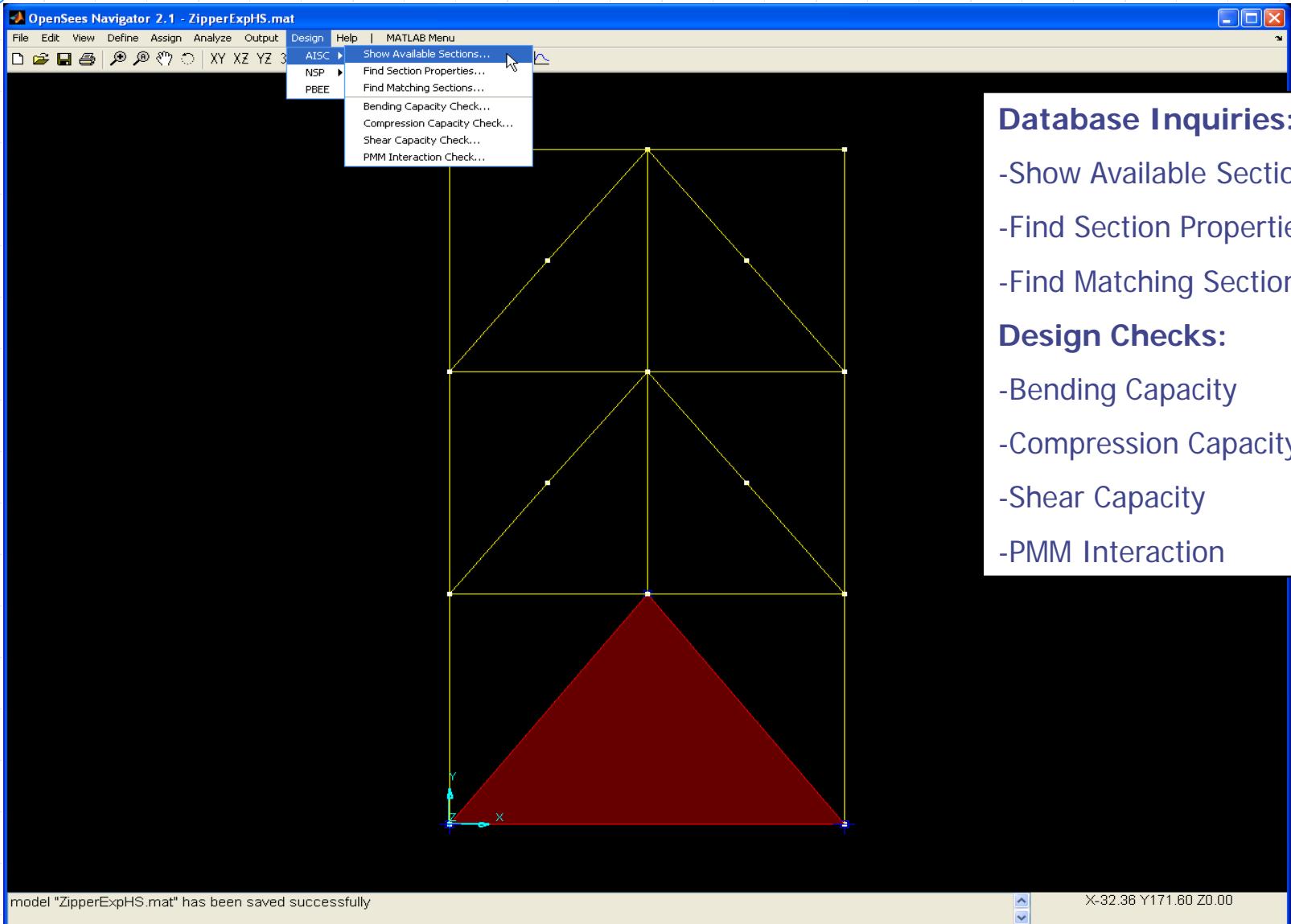
Post processing: animate response



Post processing: principal stress and strain



Design: AISC design toolbox



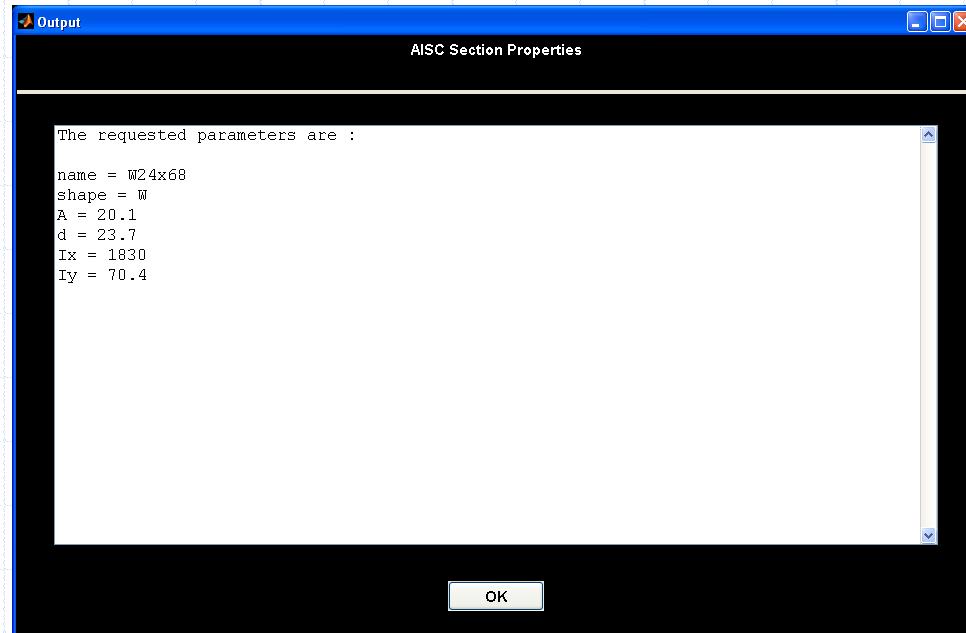
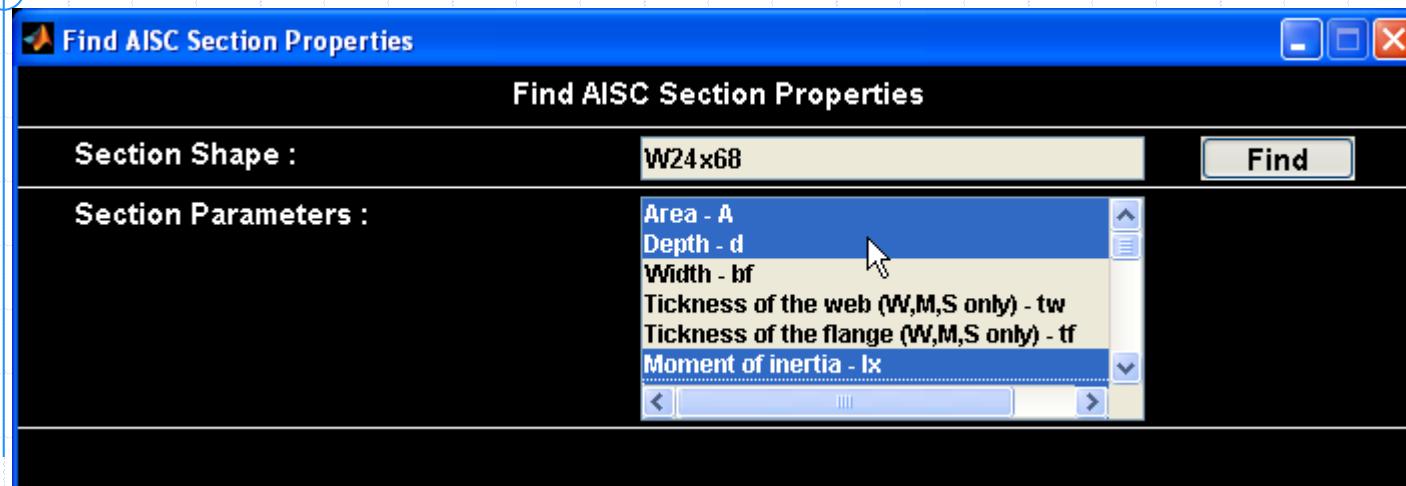
Database Inquiries:

- 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

Find Matching AISC Sections

Find Matching AISC Sections

Select Section Shape : W/S/M Find

Add Parameter : Depth - d Min : 0 Max : 8 Add

Modify Parameter : Area - A Min : 2 Max : 6 Modify

Delete Parameter : Area - A Delete

Sort by Parameter : Area - A

Output

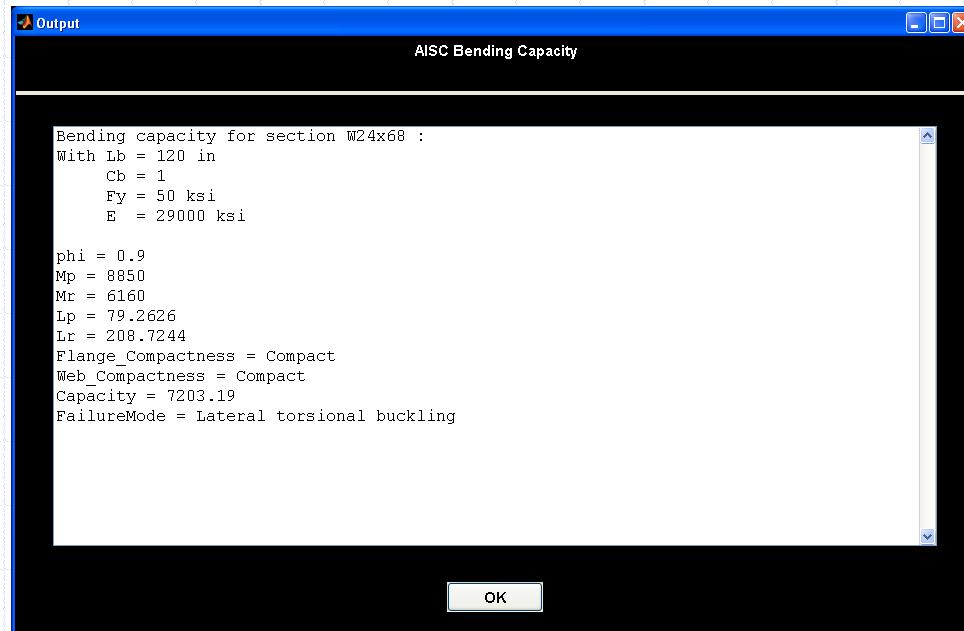
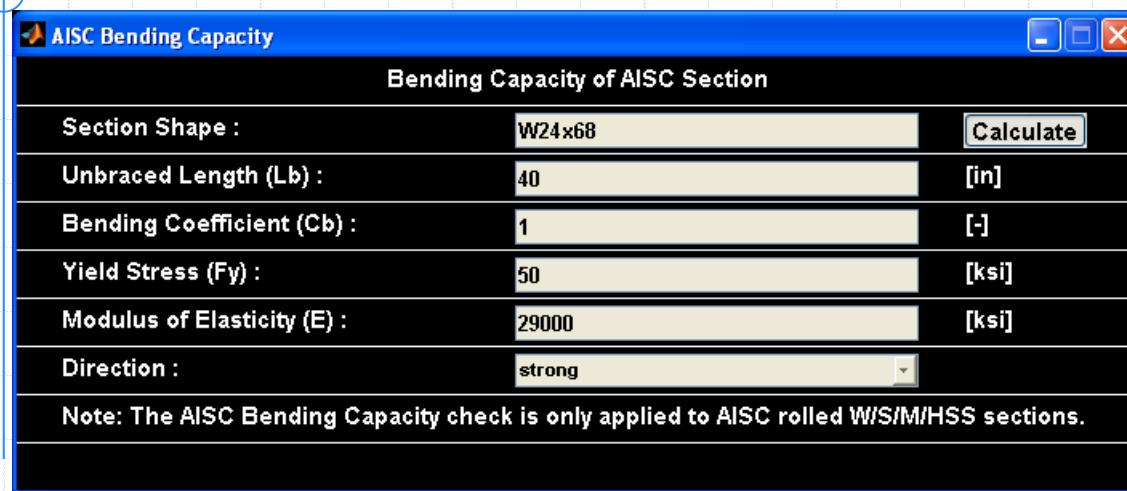
Matching AISC Sections

There are a total of "19" sections available:

- S3X7.5
- S4X7.7
- W6X8.5
- W6X9
- S4X9.5
- S5X10
- W8X10
- W6X12
- S6X12.5
- W4X13
- W8X13
- W6X15
- W5X16
- W6X16
- S6X17.25
- S8X18.4
- M5X18.9
- W5X19
- W6X20

OK

AISC design toolbox: bending capacity



AISC design toolbox: compression cap.

AISC Compression Capacity

Compression Capacity of AISC Section

Section Shape :	W14x68	Calculate
Effective Length (kLx) :	144	[in]
Effective Length (kLy) :	144	[in]
Yield Stress (Fy) :	50	[ksi]
Modulus of Elasticity (E) :	29000	[ksi]

Note: The AISC Compression Capacity check only applied to AISC rolled W/S/M/HSS sections.

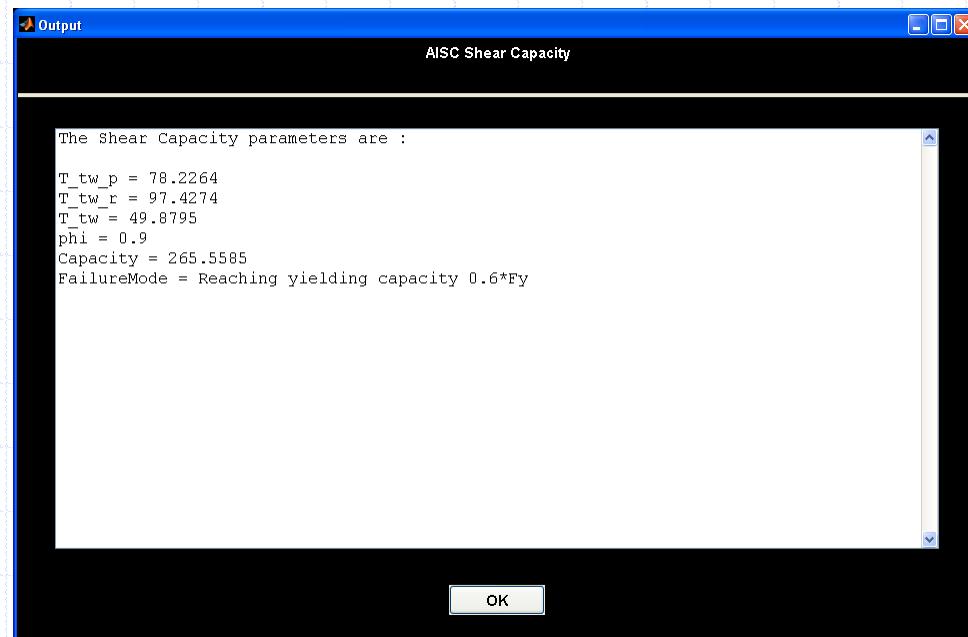
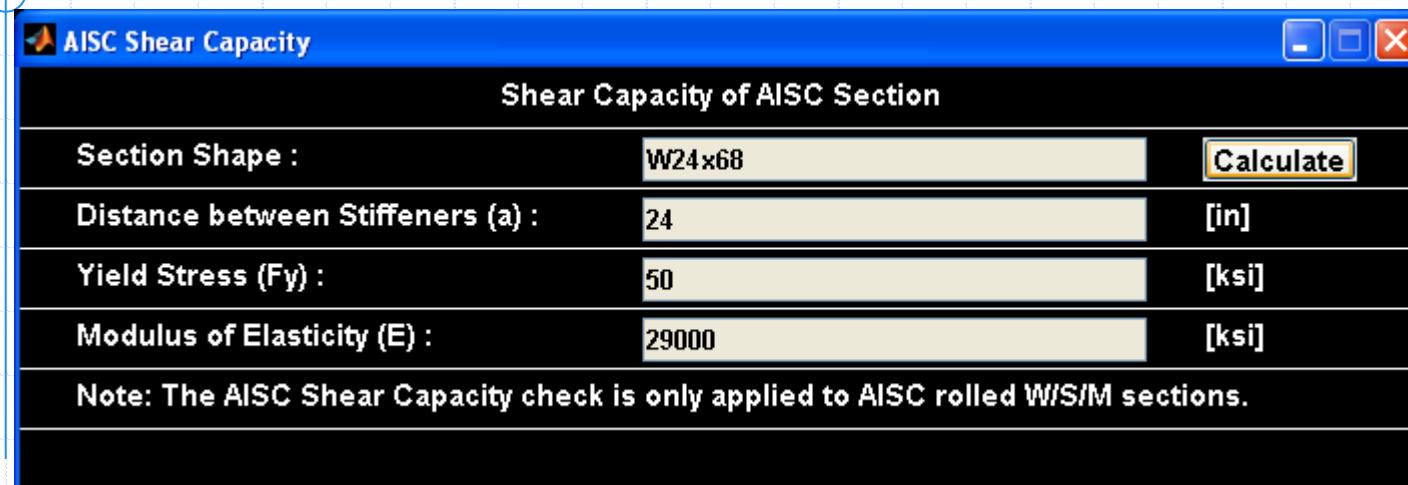
Output

AISC Compression Capacity

```
Compression capacity for section W14x68 :  
With kLx = 144 in  
    kLy = 144 in  
    Fy = 50 ksi  
    E = 29000 ksi  
  
Section_Slenderness = None Slender  
phi = 0.85  
FailureMode = Inelastic buckling (Qs(flange) = 1, Qs(web) = 1)  
Capacity = 661.6242
```

OK

AISC design toolbox: shear capacity



AISC design toolbox: PMM interaction

AISC PMM Interaction Check

PMM Interaction Check of AISC Section

Section Shape :	W24x68	<input type="button" value="Calculate"/>
Yield Stress (Fy) :	50	[ksi]
Modulus of Elasticity (E) :	29000	[ksi]
Demand :		
Applied Axial Force (Pu) :		[kips]
Applied Moment about X axis (Mux) :		[kips - in]
Applied Moment about Y axis (Muy) :		[kips - in]
Compression :		
Effective Length (kLx) :		[in]
Effective Length (kLy) :		[in]
Bending :		
Unbraced Length (Lb) :		[in]
Bending Coefficient (Cb) :	1	[\cdot]
Note: The AISC P-M interaction check is only applied to AISC rolled W/S/M/HSS sections.		

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

OPENSEES NAVIGATOR

HOME **INTRODUCTION** **MANUALS** **TUTORIALS** **PRESENTATIONS** **DISCUSSION** **UPDATES** **DOWNLOADS** **LINKS**

Home>

Introduction
Manuals
Tutorials
Presentations
Discussion
Updates
Downloads 
Links

Dear OpenSees Navigator users,

Thanks for your interest in OpenSees Navigator. This program is intended to be self-explanatory, nevertheless a basic user manual will be added to the website shortly. We are very happy to have the opportunity to distribute this software for OpenSees Navigator users. We encourage everyone to try out all of the functions of the program and send us criticism, corrections or suggestions to improve future versions. We also encourage users to e-mail us at either andreas.schellenberg@gmail.com or yangtony2004@gmail.com so that we can add the e-mail addresses to the OpenSees Navigator user list. We will use such list to contact everyone about new releases or major updates. We will try our best to improve the next release.

Thank you.

Please feel free to visit our websites to discover in what other fun research we are involved:

Andreas Schellenberg & [Tony Yang](#)

Hit Counter OpenSees Navigator ©2004-2006 UC Berkeley. All rights reserved. Please [contact us](#) with any questions or comments.

Website: download

The screenshot shows the OpenSees Navigator website's 'Downloads' page for Stand-Alone Windows. The top navigation bar includes links for Home, Introduction, Manuals, Tutorials, Presentations, Discussion, Updates, Downloads, and Links. Below the navigation is a breadcrumb trail: Home > Downloads > Stand-Alone Windows. A search bar is also present. On the left, a sidebar lists two files: MCRInstaller.exe and OpenSeesNavigator.zip. The main content area contains the following text:

Installation Instructions:

1. Download the two files on the left.
2. Install the Matlab component runtime libraries by executing MCRInstaller.exe and following the on screen instructions (this has only to be done once).
3. Extract OpenSeesNavigator.zip in any folder of your choice and then execute OpenSeesNavigator.exe.
4. If you like you can create a shortcut to OpenSeesNavigator.exe on your Desktop.

At the bottom of the page, there is a hit counter link and a copyright notice: OpenSees Navigator ©2004-2006 UC Berkeley. All rights reserved. Please [contact us](#) with any questions or comments.