

OpenSees Navigator & Hybrid Simulation

Andreas Schellenberg & Tony Yang

Department of Civil and Environmental Engineering
University of California, Berkeley



nees@berkeley



The George E. Brown, Jr. Network for Earthquake Engineering Simulation

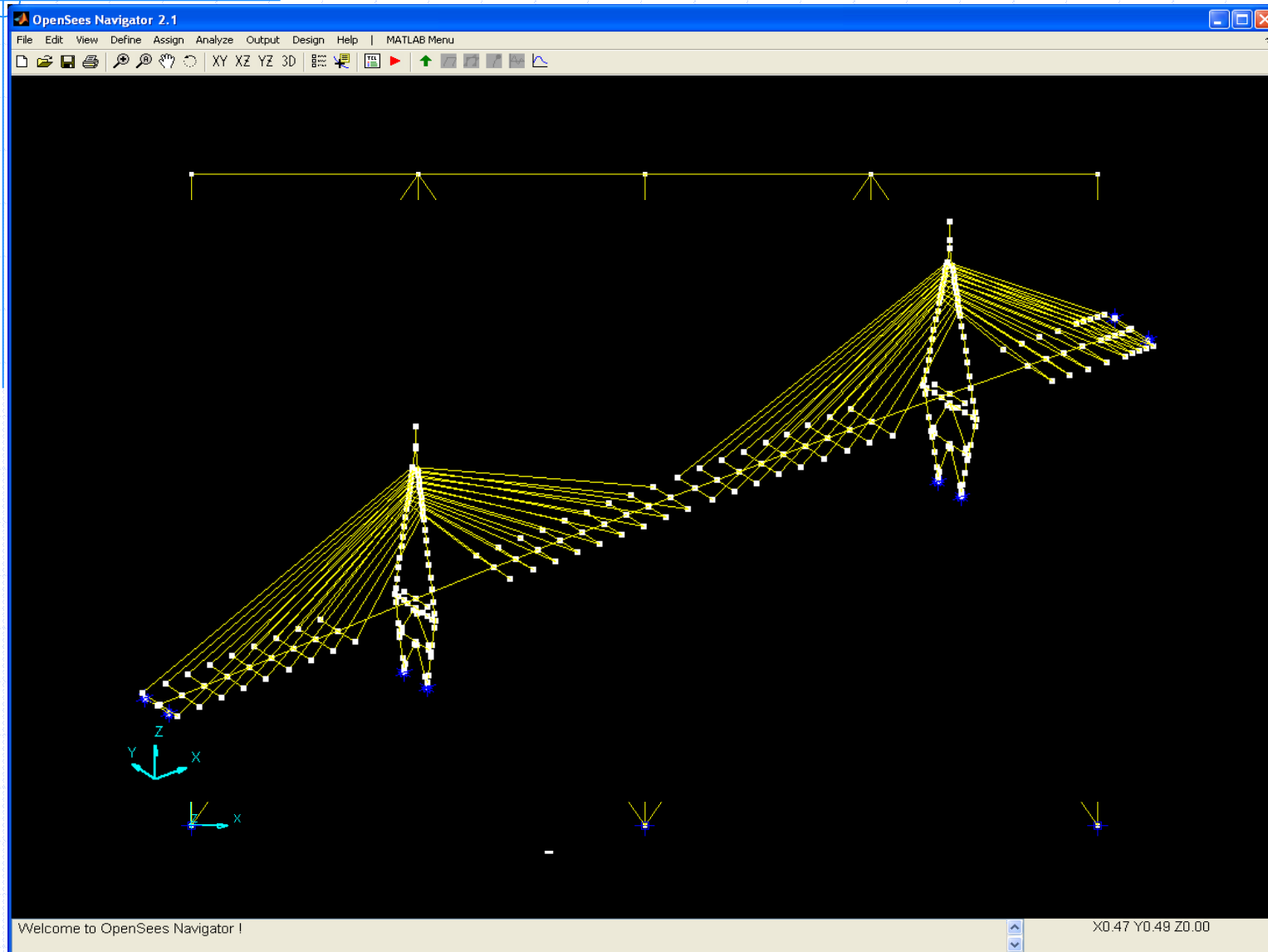
Introduction

- ◆ MATLAB based Graphical User Interface
- ◆ Pre- and post-processing for OpenSees
- ◆ Integration of Hybrid Simulation into the graphical user interface
- ◆ Response Spectra generation
- ◆ Integrated AISC data base
- ◆ Design toolboxes: NSP, AISC design checks, PBEE, ...
- ◆ Self-executable version available

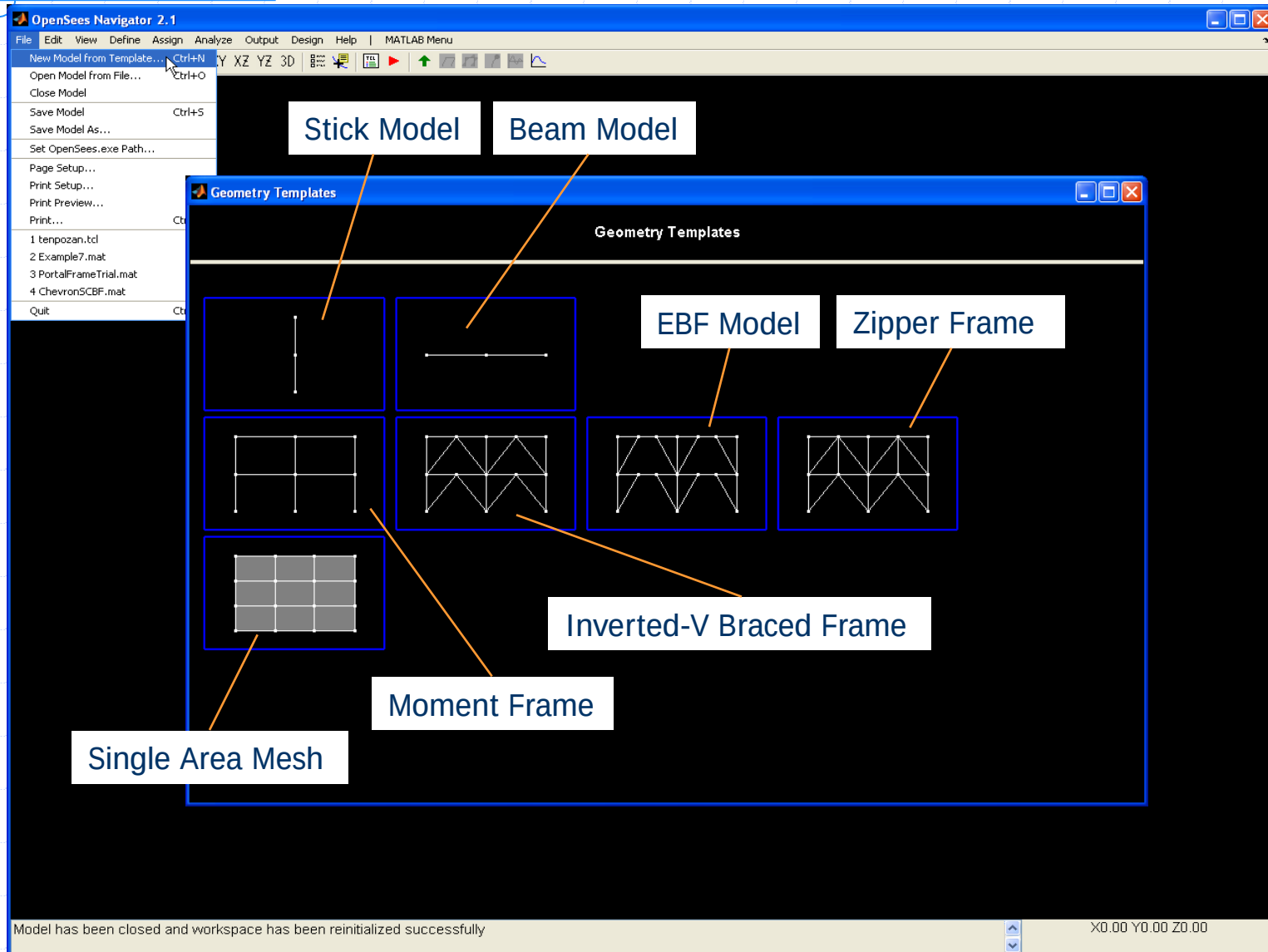
Motivation

- ◆ Graphical input is more user-friendly than TCL text input
- ◆ Most researchers use MATLAB to do the post-processing, and MATLAB/Simulink is the typical framework for implementing hybrid analyses
- ◆ OpenSees Navigator will create the OpenSees (hybrid) model and graphically display results before, during or after a test
- ◆ Flexible to use and requires no programming skills

OpenSees Navigator



Define Geometry



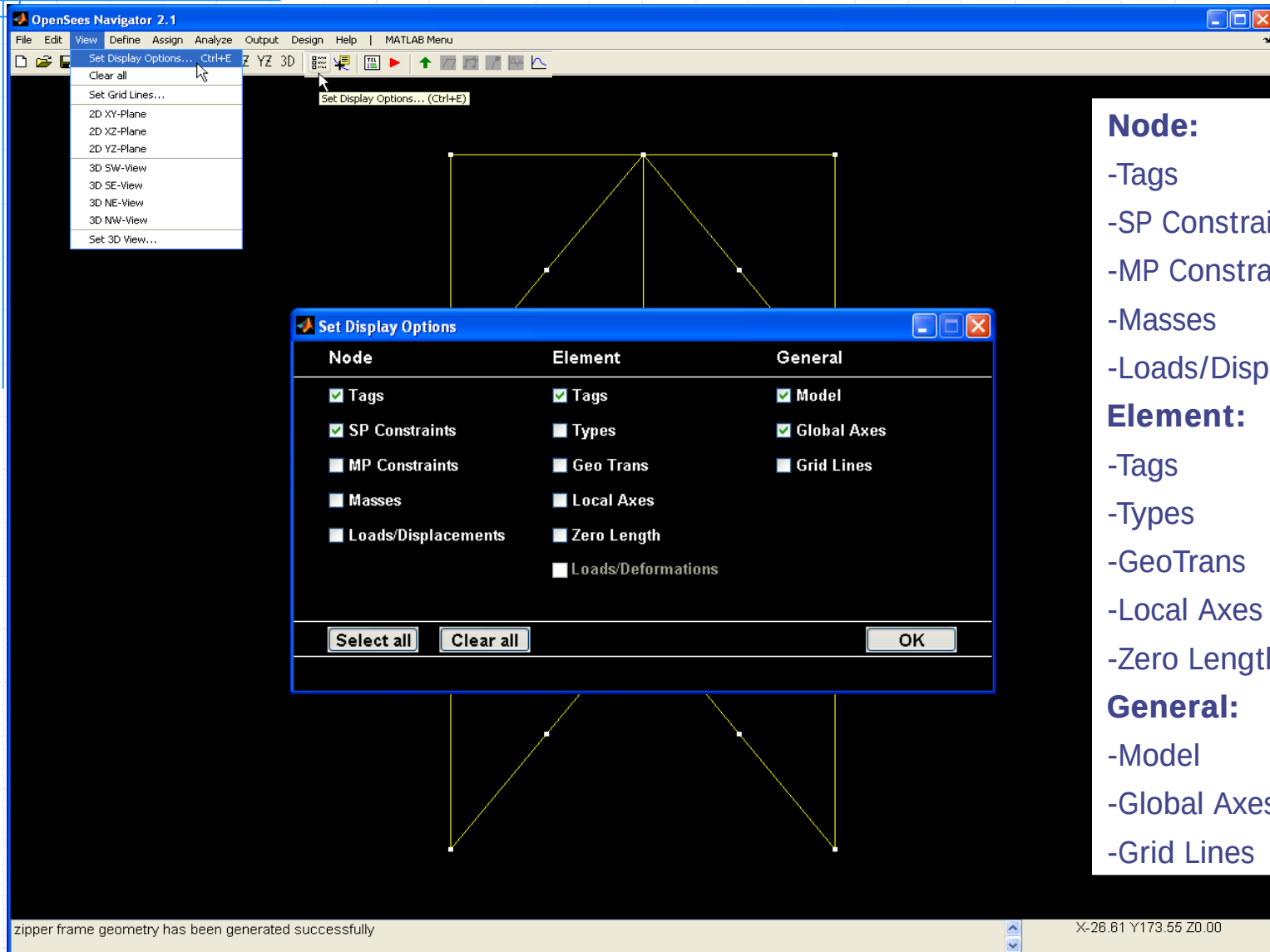
Define Geometry: Zipper Frame

Define Zipper Frame Geometry

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: Set Display Options



Node:

- Tags
- SP Constraints
- MP Constraints
- Masses
- Loads/Displ.

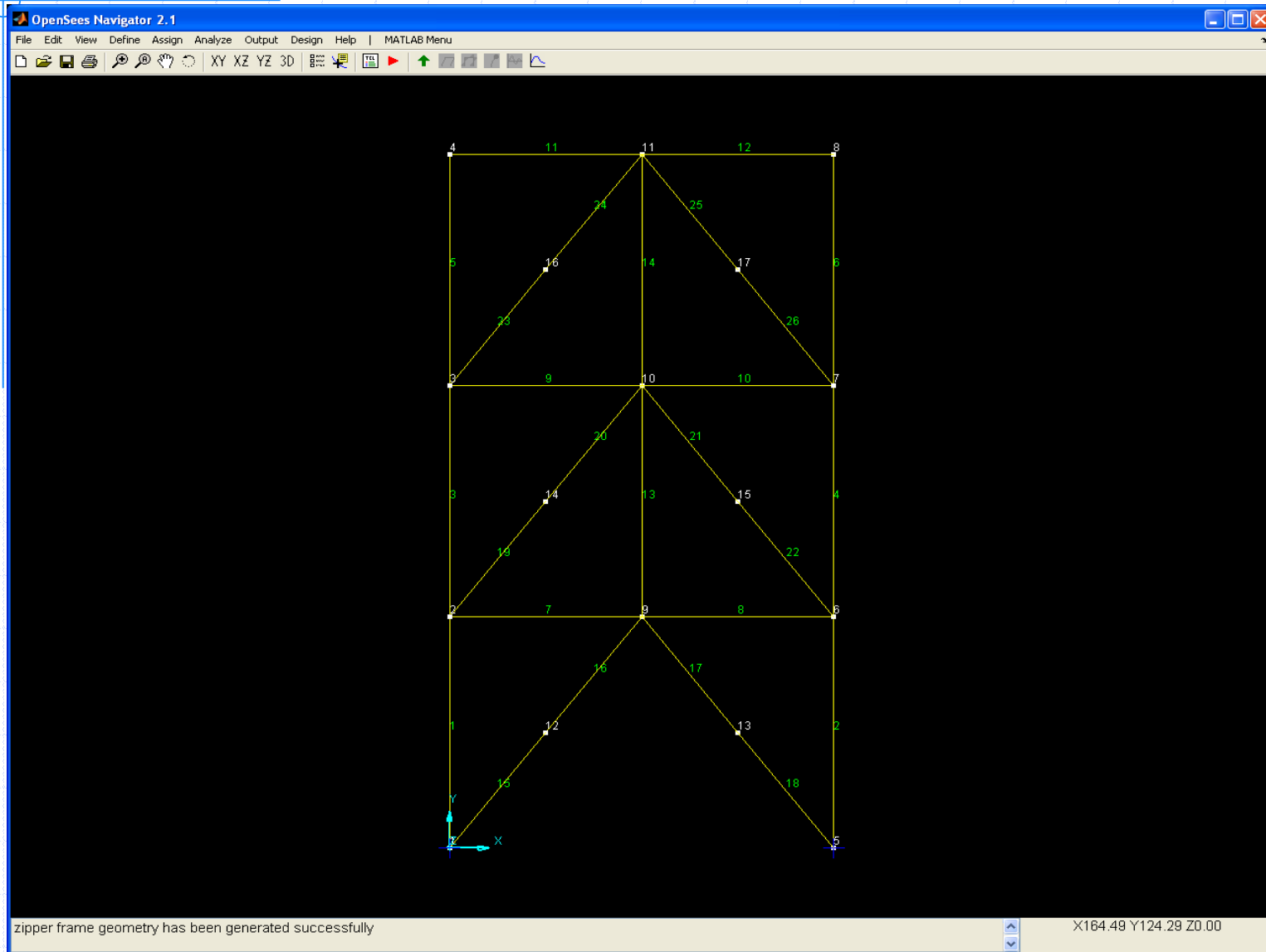
Element:

- Tags
- Types
- GeoTrans
- Local Axes
- Zero Length

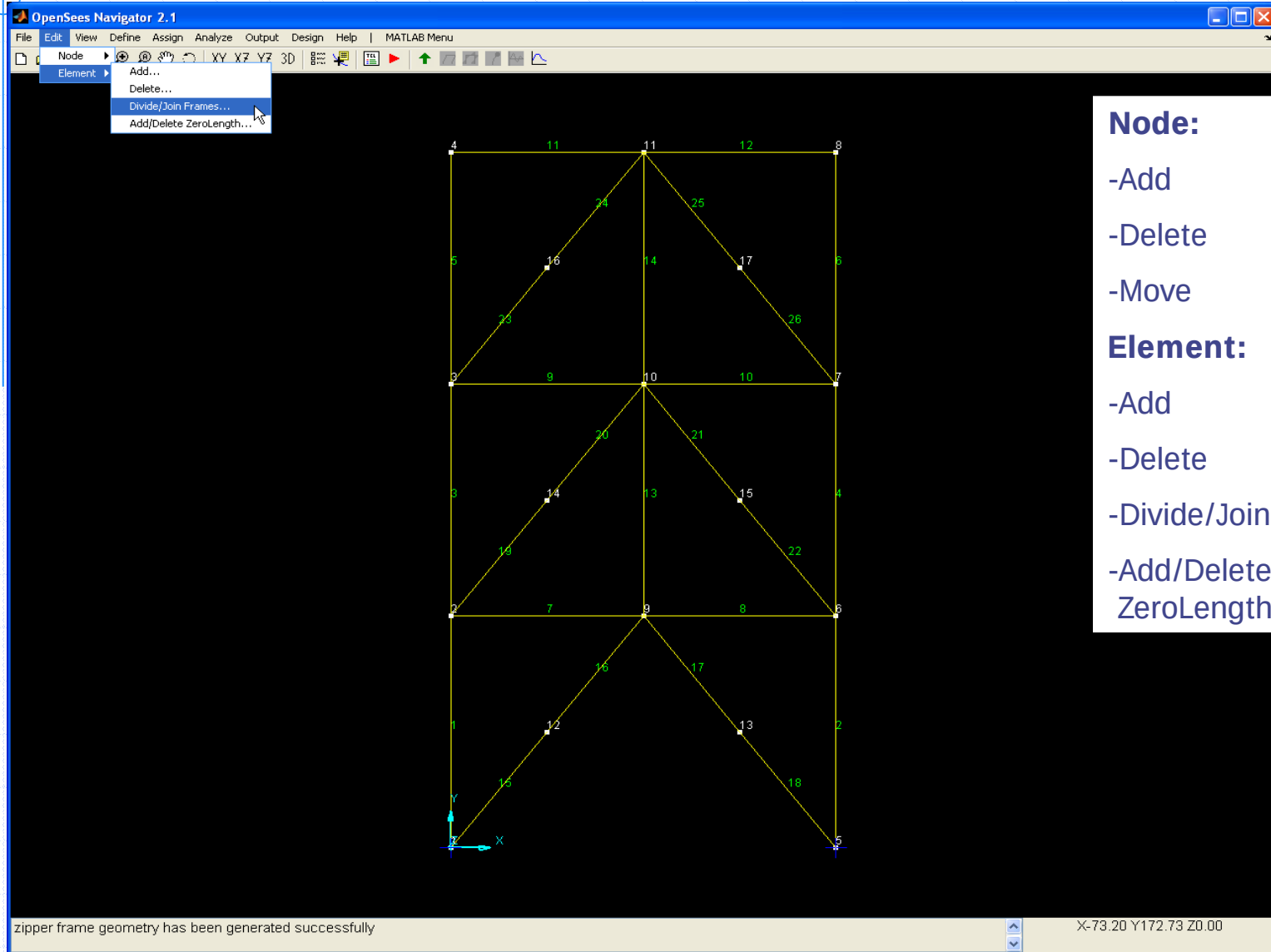
General:

- Model
- Global Axes
- Grid Lines

View Geometry: Display



Edit Geometry



Node:

- Add
- Delete
- Move

Element:

- Add
- Delete
- Divide/Join
- Add/Delete ZeroLength

Define Material: Uniaxial Materials

The screenshot shows the OpenSees Navigator 2.1 interface. The 'Define' menu is open, showing 'Uniaxial Materials...' and 'nD Materials...'. The 'Define Uniaxial Material' dialog box is open, showing the following options:

- Add Material : BoucWen
- Modify/Show Material : ElasticDefault
- Delete Material : ElasticDefault

The background shows a zipper frame geometry with nodes 1 through 18. A status bar at the bottom indicates 'zipper frame geometry has been generated successfully' and 'X=56.17 Y=172.73 Z=0.00'.

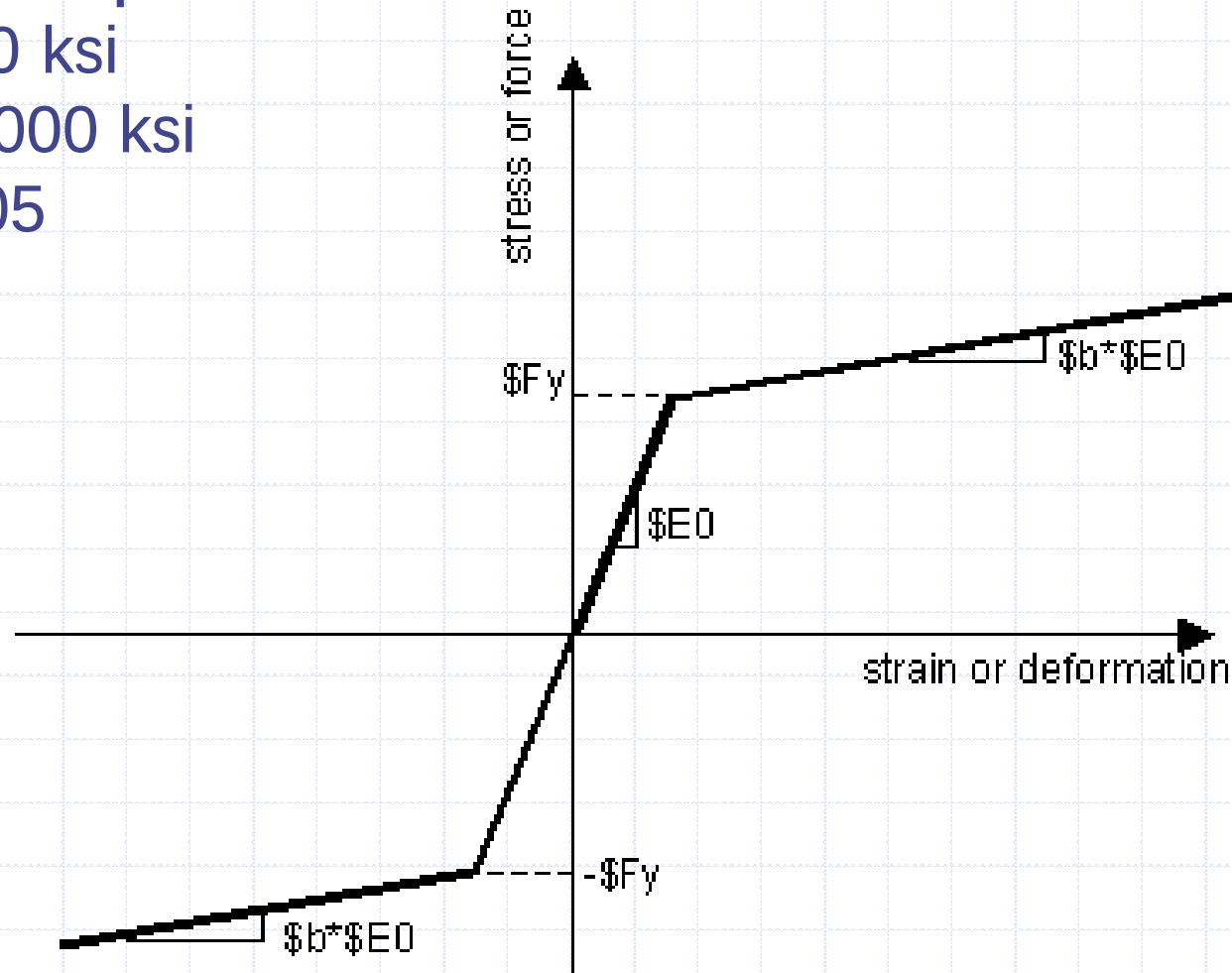
Templates:

- BoucWen
- Concrete01
- Concrete02
- Concrete03
- Elastic
- ElasticNoTension
- ElasticPP
- ElasticPPGap
- Fatigue
- Hardening
- Hysteretic
- MinMax
- Parallel
- Series
- Steel01
- Steel02
- Viscous

Define Steel01 Material: A50

Material Properties

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



Define Steel01 Material: A50

Define Steel01 Material

Material Name :

Yield Stress (Fy) :

Modulus of Elasticity (E) :

Hardening Ratio (b) :

Optional Parameters :

Iso Hardening Parameter (a1) :

Iso Hardening Parameter (a2) :

Iso Hardening Parameter (a3) :

Iso Hardening Parameter (a4) :

Define Material: nD Materials

The screenshot shows the OpenSees Navigator 2.1 interface. The 'Define' menu is open, and 'nD Materials...' is selected. The 'Define nD Material' dialog box is displayed, showing the 'Add Material' dropdown set to 'ElasticIsotropic'. The background shows a zipper frame geometry with nodes numbered 1 through 18. The status bar at the bottom indicates 'zipper frame geometry has been generated successfully' and 'X-61.09 Y173.14 Z0.00'.

OpenSees Navigator 2.1

File Edit View Define Assign Analyze Output Design Help MATLAB Menu

Materials

- Uniaxial Materials...
- nD Materials...

Sections

Experimental

Elements

Time Series...

Load Pattern...

Recorders...

Analyses Options...

Response Functions...

Response Spectra...

Define nD Material

Define nD Material

Add Material : ElasticIsotropic

Modify/Show Material :

Delete Material :

zipper frame geometry has been generated successfully

X-61.09 Y173.14 Z0.00

Templates:

- ElasticIsotropic
- J2Plasticity
- PlaneStress
- PlateFiber

Define Section: Line Sections

The screenshot shows the OpenSees Navigator 2.1 interface. The 'Define' menu is open, and 'Line Sections...' is selected. A 'Define Line Section' dialog box is displayed in the foreground. The dialog box has three rows for defining sections:

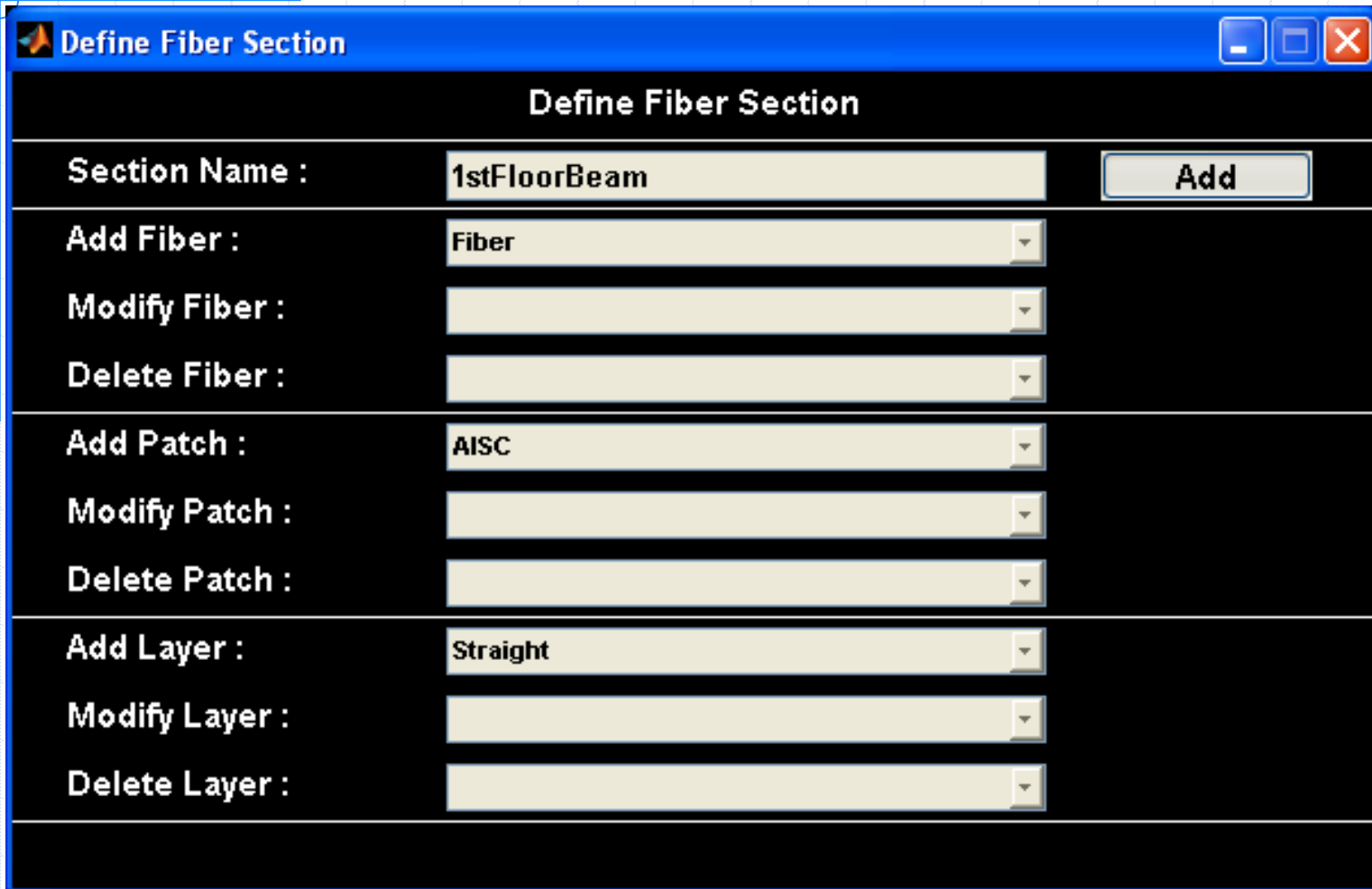
Define Line Section	
Add Section :	Aggregator
Modify/Show Section :	ElasticDefault
Delete Section :	ElasticDefault

The background shows a zipper frame geometry with nodes 1 through 26. A status bar at the bottom indicates 'zipper frame geometry has been generated successfully' and 'X:50.42 Y:171.91 Z:0.00'.

Templates:

- Aggregator
- Elastic
- Fiber
- Uniaxial

Define Fiber Section: 1stFloorBeam



Define Fiber Section

Section Name :

Add Fiber :

Modify Fiber :

Delete Fiber :

Add Patch :

Modify Patch :

Delete Patch :

Add Layer :

Modify Layer :

Delete Layer :

Define Fiber Section: AISC Patch

Define AISC Patch

Patch Name : Patch01

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

Optional Arguments :

Counter-Clockwise Rot (Theta) : 0.

Define Section: Area Sections

The screenshot shows the OpenSees Navigator 2.1 interface. The 'Define' menu is open, and 'Area Sections...' is selected. A 'Define Area Section' dialog box is displayed in the foreground. The dialog box has three sections: 'Add Section :', 'Modify/Show Section :', and 'Delete Section :'. The 'Add Section :' dropdown is set to 'Bidirectional'. Below the dialog box, a structural model is visible, showing a truss-like structure with nodes and elements. The status bar at the bottom indicates that a section named '1stFloorBeam' has been defined successfully.

OpenSees Navigator 2.1

File Edit View Define Assign Analyze Output Design Help MATLAB Menu

Materials
Sections
Experimental
Elements
Time Series...
Load Pattern...
Recorders...
Analyses Options...
Response Functions...
Response Spectra...

YZ 3D

Define Area Section

Define Area Section

Add Section : Bidirectional

Modify/Show Section :

Delete Section :

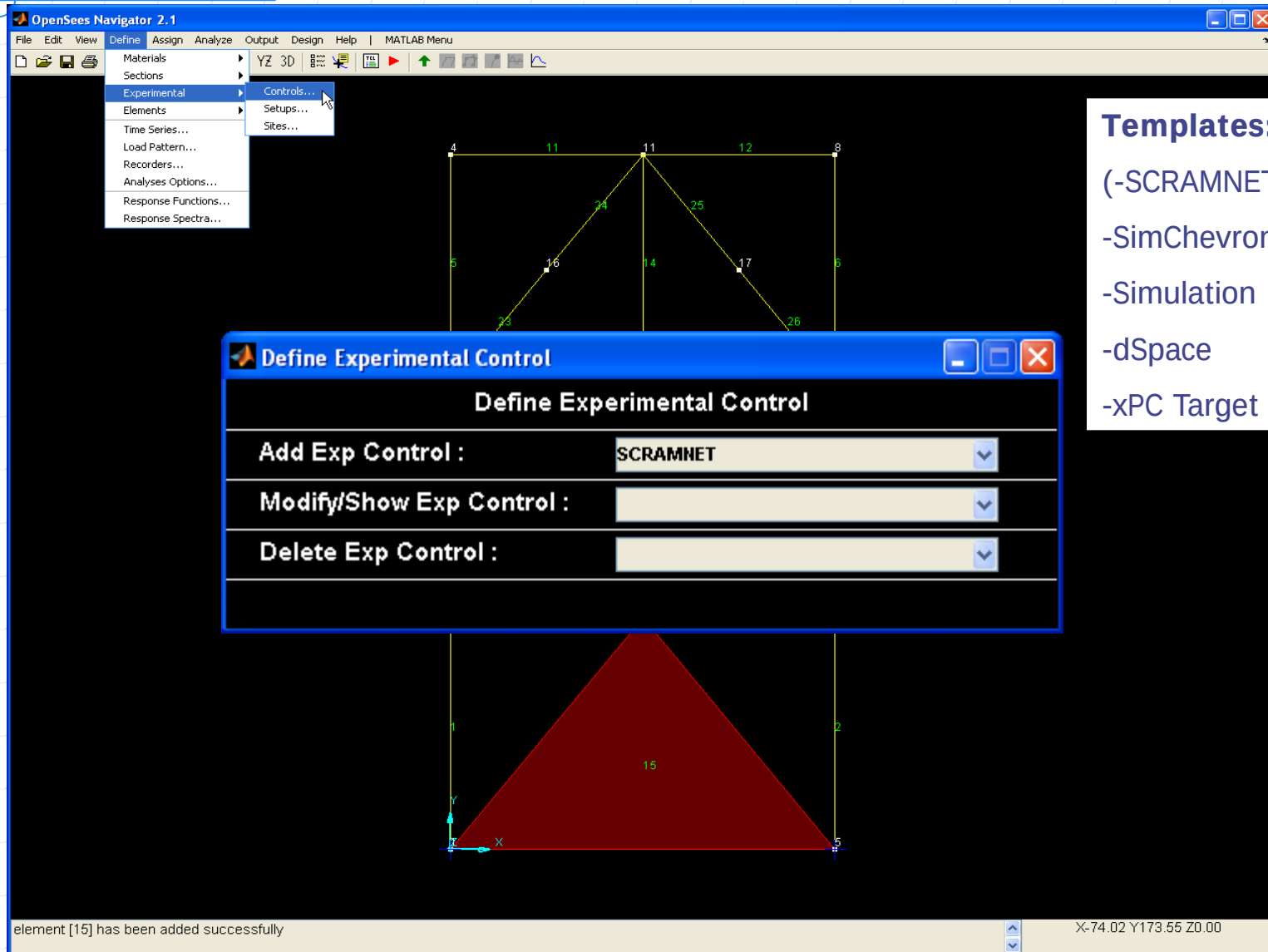
section(2) with name "1stFloorBeam" has been defined/modified successfully

X-47.75 Y173.55 Z0.00

Templates:

- Bidirectional
- ElasticMembranePlate
- PlateFiber

Define Experimental Control



Templates:

(-SCRAMNET)

-SimChevronBrace

-Simulation

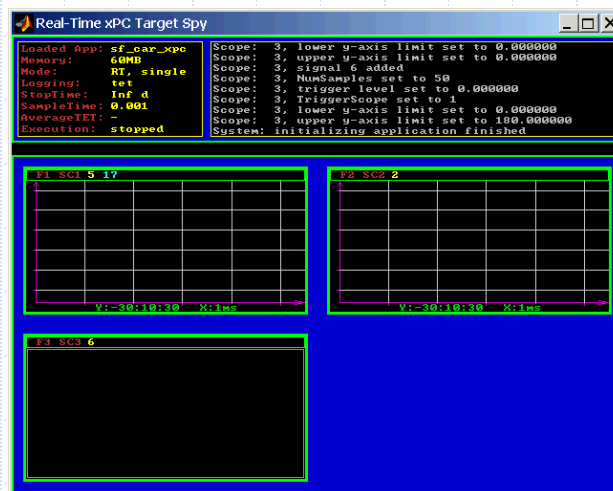
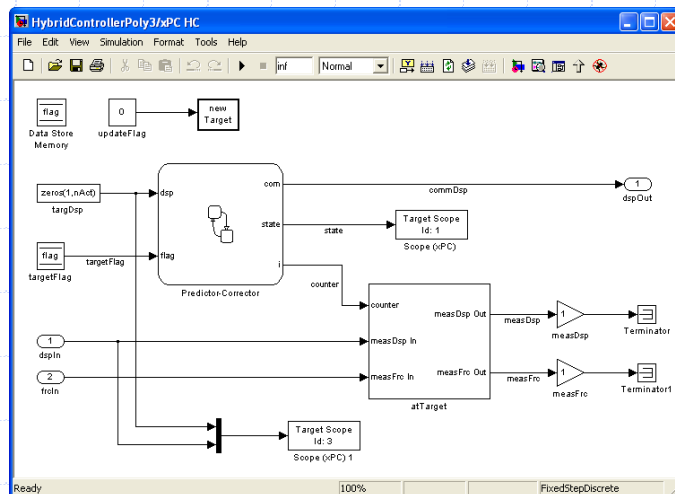
-dSpace

-xPC Target

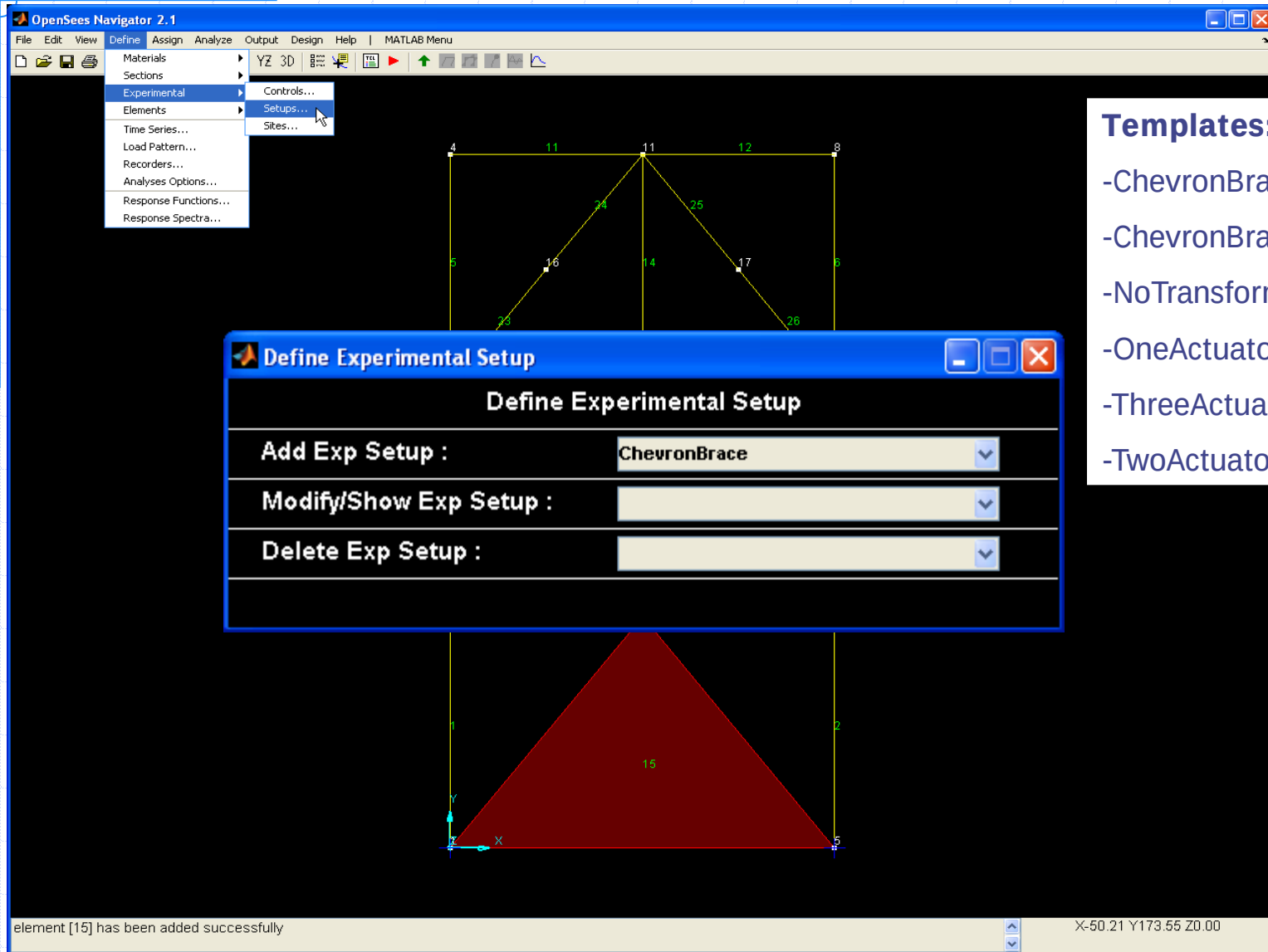
Define ExpControl: xPC Target

Define xPC Target Control

Control Name :	BraceExpCtrlXPC	Add
Number of Setups (numSetups) :	1	
Predictor-Corrector Type (type) :	Dsp	
xPC Target IP Address (ipAddr) :	192.168.2.20	
xPC Target IP Port (ipPort) :	22222	
Application Name (appName) :	HybridControllerPoly3	Browse
Application Path (appPath) :	TestModels\\c&mCode-xPCTarget-STS\\	



Define Experimental Setup



Templates:

- ChevronBrace
- ChevronBraceIntOff
- NoTransformation
- OneActuator
- ThreeActuators
- TwoActuators

Define ExpSetup: ChevronBrace

Define ChevronBraceJntOff Setup

Setup Name :

Experimental Control Type :

Geometry Type (nlGeomFlag) :

Actuator Length 1 (La1) :

Actuator Length 2 (La2) :

Actuator Length 3 (La3) :

Rigid Link Length 1 (L1) :

Rigid Link Length 2 (L2) :

Rigid Link Length 3 (L3) :

Rigid Link Length 4 (L4) :

Rigid Link Length 5 (L5) :

Rigid Link Length 6 (L6) :

Optional Parameters :

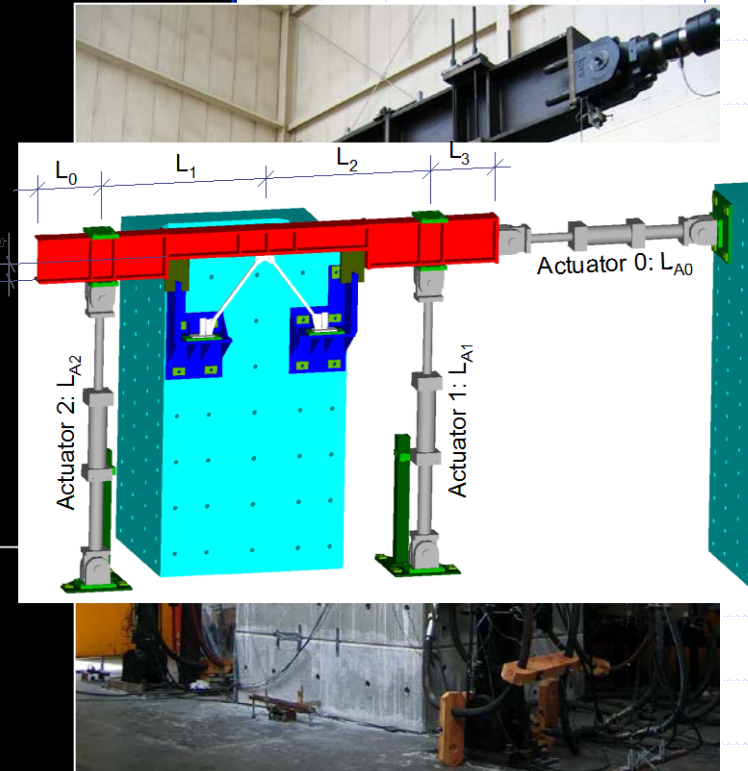
Dsp Control Factor (dspCtrlFact) :

Vel Control Factor (velCtrlFact) :

Acc Control Factor (accCtrlFact) :

Dsp Daq Factor (dspDaqFact) :

Force Daq Factor (frcDaqFact) :



Define Experimental Site

The screenshot shows the OpenSees Navigator 2.1 interface. The 'Define' menu is open, and the 'Sites...' option is selected. A dialog box titled 'Define Experimental Site' is displayed in the foreground. The dialog box contains three rows of controls:

Define Experimental Site	
Add Exp Site :	LocalSite
Modify/Show Exp Site :	RFS
Delete Exp Site :	RFS

Below the dialog box, a 3D model of a structure is visible. The structure is a triangular frame with nodes labeled 4, 11, 8, 5, 16, 14, 17, and 26. The element 15 is highlighted in red. The status bar at the bottom indicates 'element [15] has been added successfully' and 'X-72.79 Y173.14 Z0.00'.

Templates:

- LocalSite
- RemoteSite

Define ExpSite: RFS

Define Local Site

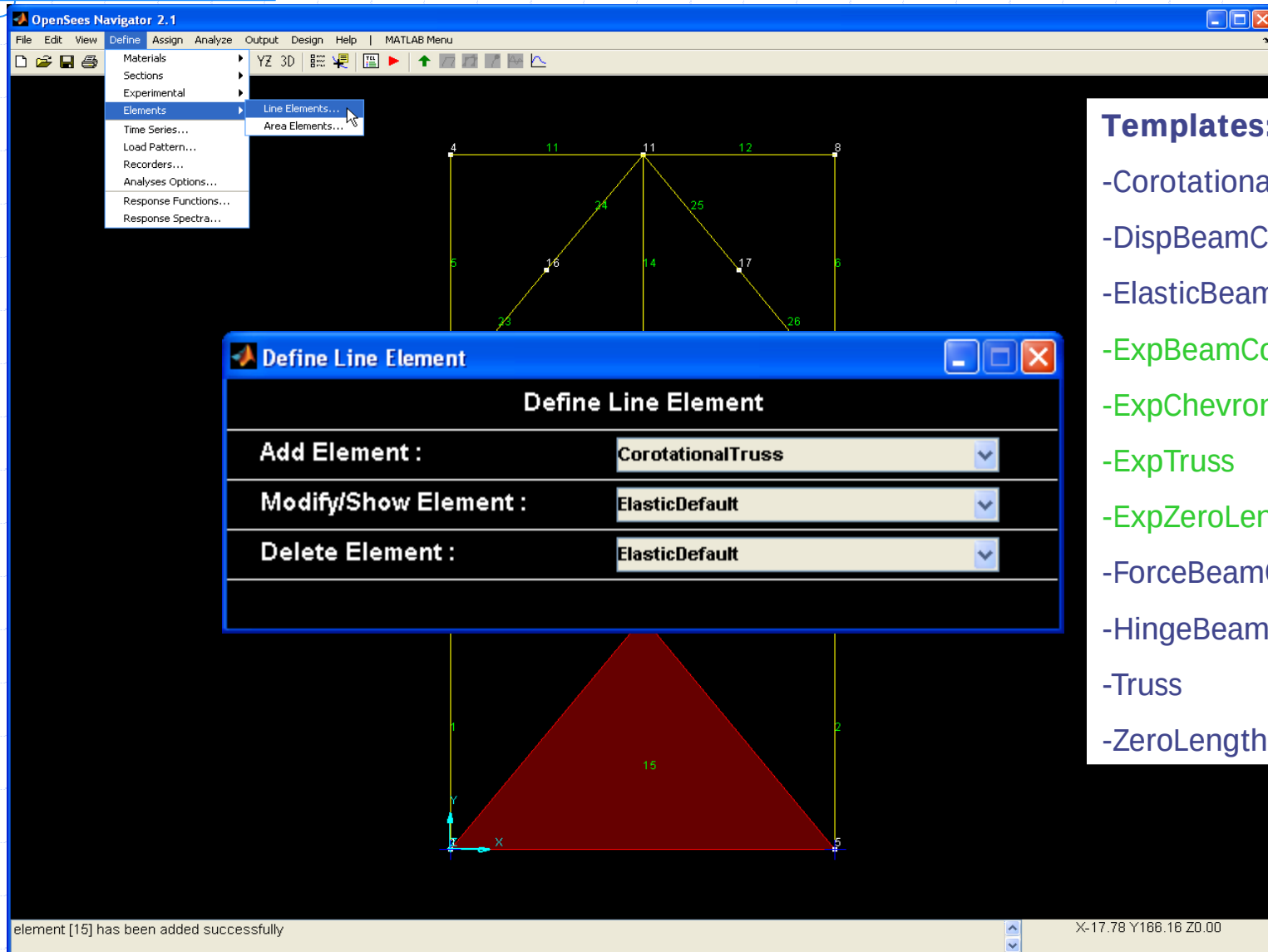
Define Local Site

Site Name :

Experimental Setup Type :



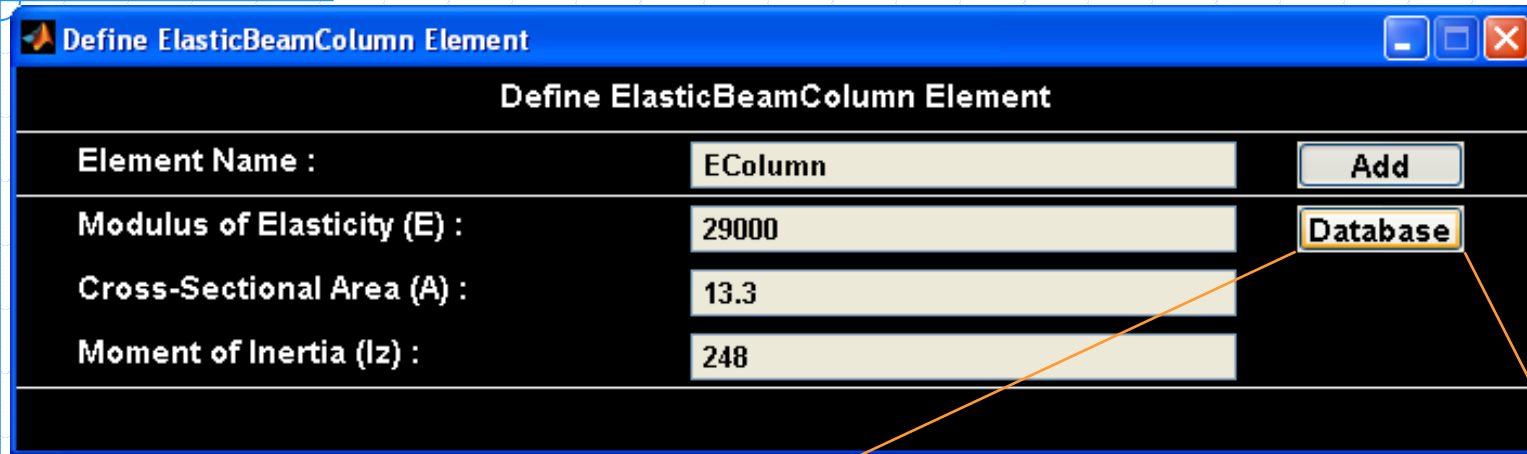
Define Element: Line Element



Templates:

- CorotationalTruss
- DispBeamColumn
- ElasticBeamColumn
- ExpBeamColumn
- ExpChevronBrace
- ExpTruss
- ExpZeroLength
- ForceBeamColumn
- HingeBeamColumn
- Truss
- ZeroLength

Define ElasticBeamColumn Element



Define ElasticBeamColumn Element

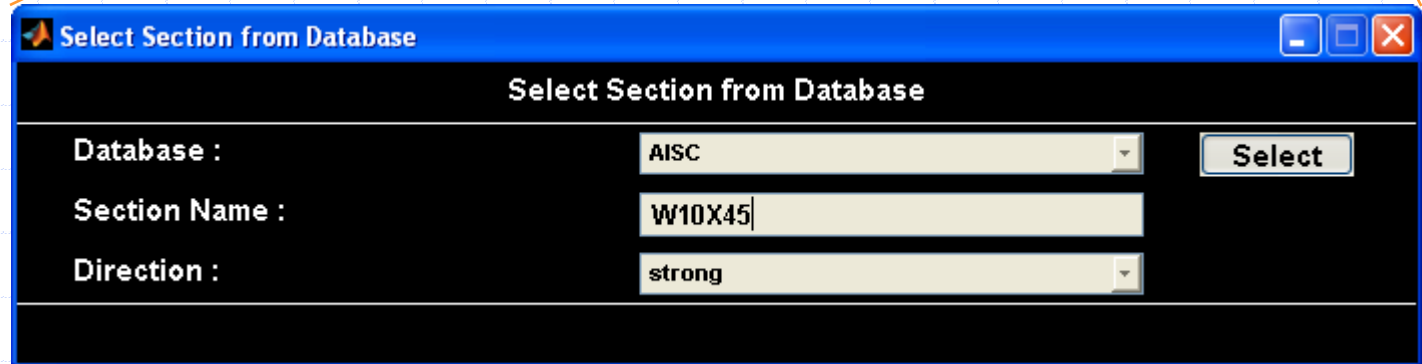
Element Name : EColumn

Modulus of Elasticity (E) : 29000

Cross-Sectional Area (A) : 13.3

Moment of Inertia (Iz) : 248

The dialog box has a blue title bar with the text "Define ElasticBeamColumn Element" and standard window controls. The main area has a black background with white text. It contains four input fields for "Element Name", "Modulus of Elasticity (E)", "Cross-Sectional Area (A)", and "Moment of Inertia (Iz)". The "Element Name" field contains "EColumn" and has an "Add" button to its right. The "Modulus of Elasticity (E)" field contains "29000" and has a "Database" button to its right. The "Cross-Sectional Area (A)" field contains "13.3". The "Moment of Inertia (Iz)" field contains "248". An orange line connects the "Database" button in this dialog to the "Select Section from Database" dialog below.



Select Section from Database

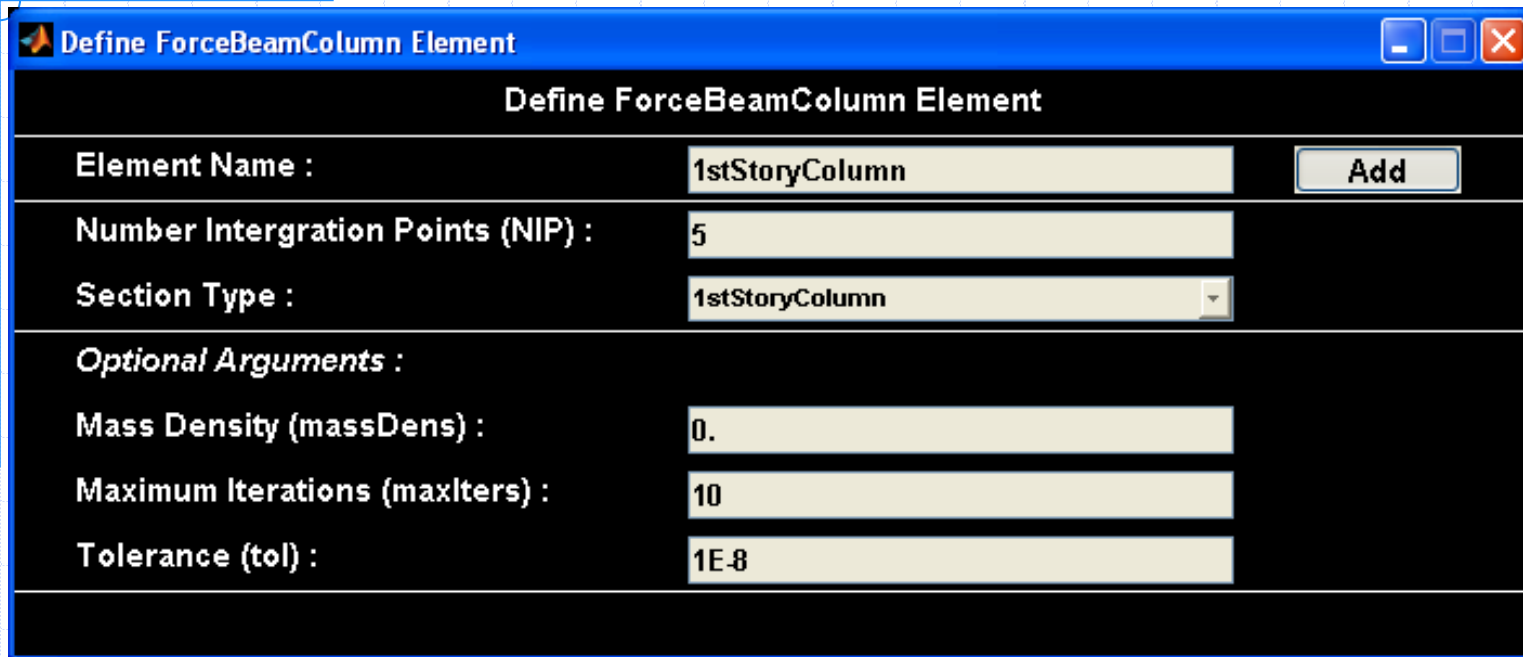
Database : AISC

Section Name : W10X45

Direction : strong

The dialog box has a blue title bar with the text "Select Section from Database" and standard window controls. The main area has a black background with white text. It contains three input fields: "Database" with a dropdown menu showing "AISC", "Section Name" with a text field containing "W10X45", and "Direction" with a dropdown menu showing "strong". A "Select" button is to the right of the "Database" dropdown. An orange line connects the "Database" button from the "Define ElasticBeamColumn Element" dialog to this "Select" button.

Define ForceBeamColumn Element



Define ForceBeamColumn Element

Define ForceBeamColumn Element

Element Name : 1stStoryColumn

Number Intergration Points (NIP) : 5

Section Type : 1stStoryColumn

Optional Arguments :

Mass Density (massDens) : 0.

Maximum Iterations (maxlters) : 10

Tolerance (tol) : 1E-8

Define ExpElement: ChevronBrace

Define ExpChevronBrace Element

Element Name :

Experimental Site Type :

Initial Stiffness (initStif) :

0	0	0
0	0	0
0	0	0

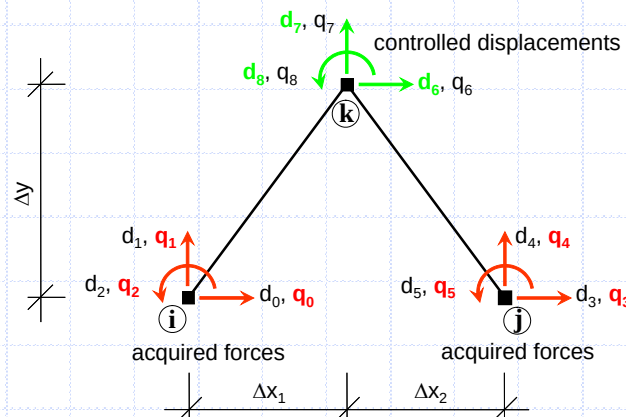
Optional Arguments :

I-Modification (iMod) :

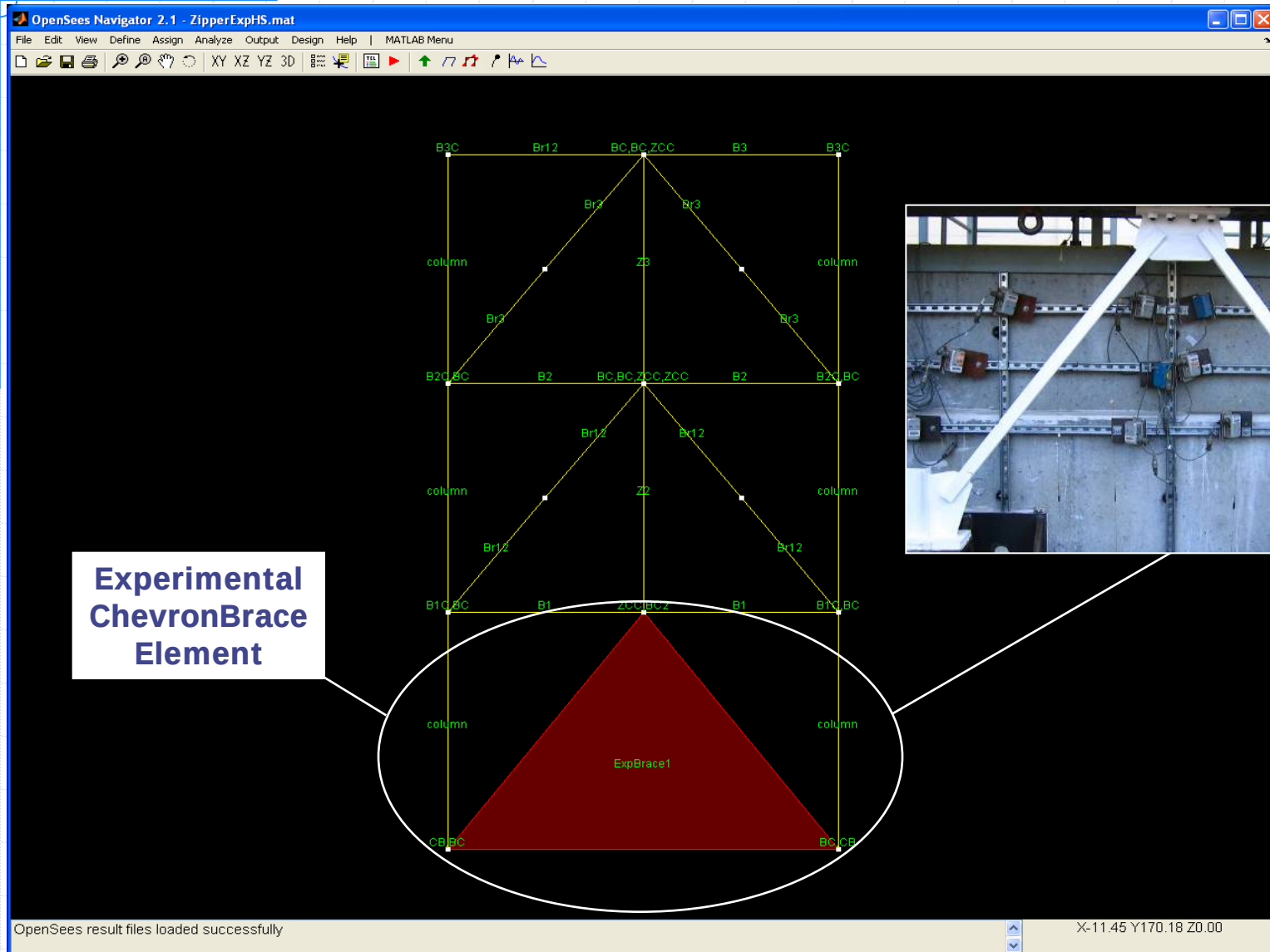
Is Copy (isCopy) :

Mass Density 1 (massDens1) :

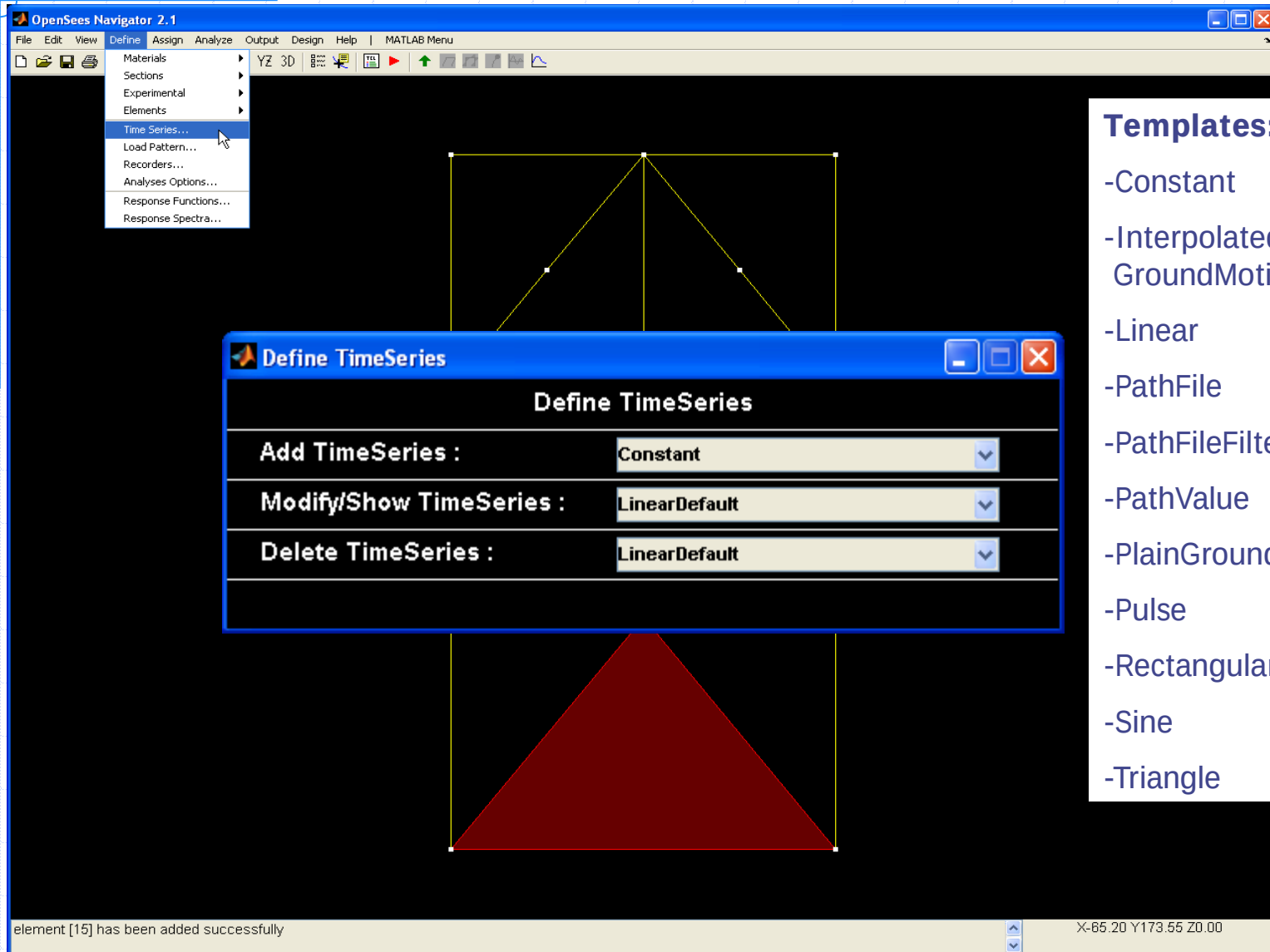
Mass Density 2 (massDens2) :



Define ExpElement: ChevronBrace



Define TimeSeries



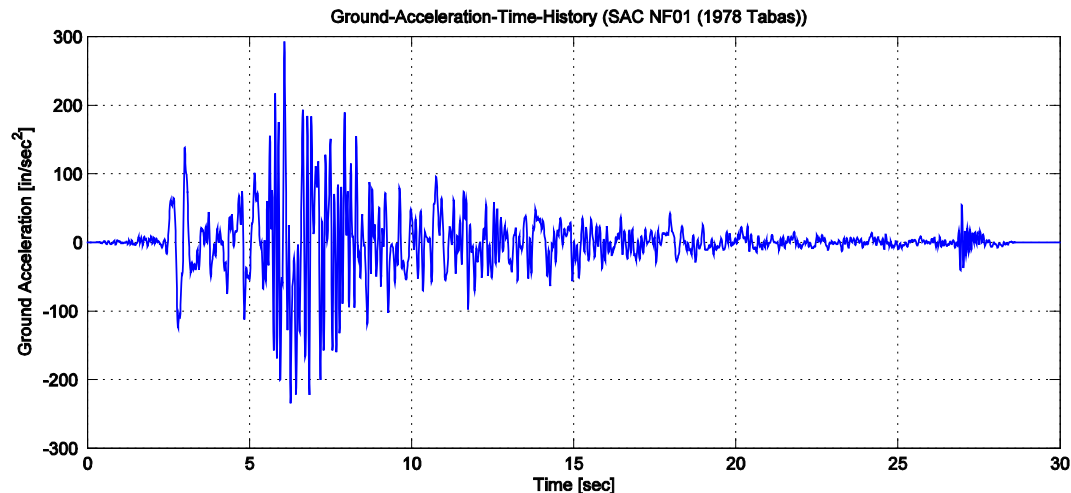
Templates:

- Constant
- Interpolated GroundMotion
- Linear
- PathFile
- PathFileFiltered
- PathValue
- PlainGroundMotion
- Pulse
- Rectangular
- Sine
- Triangle

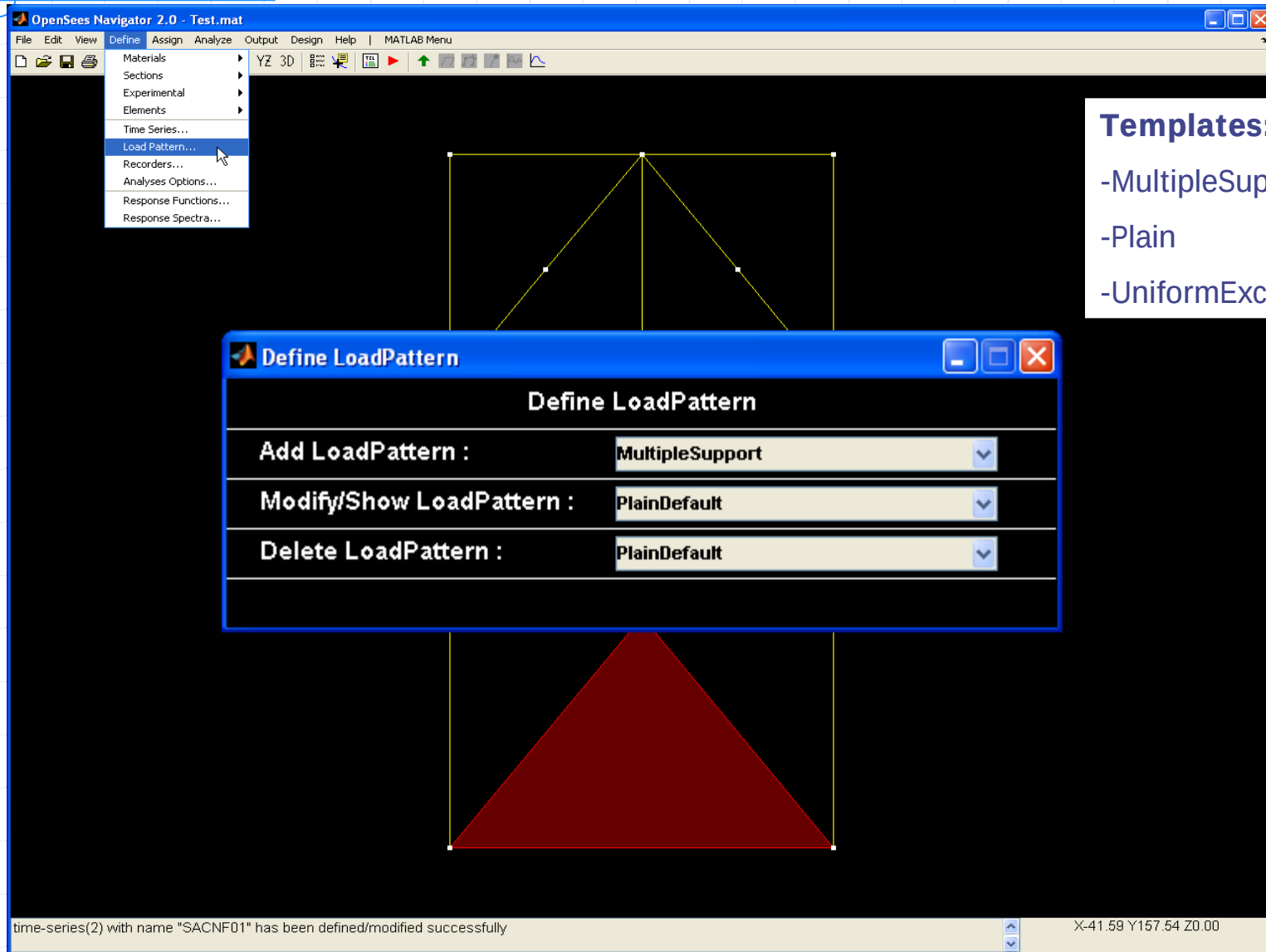
Define PathFile TimeSeries: SACNF01

Define PathFile Time Series

TimeSeries Name :	SACNF01	Add
Time Interval (dt) :	0.01	
Time File Name (fileTime) :	☐	Browse
Value File Name (filePath) :	D:\NEES\GroundMotions\SACNF01.txt	Browse
Load Factor (cFactor) :	386.1	



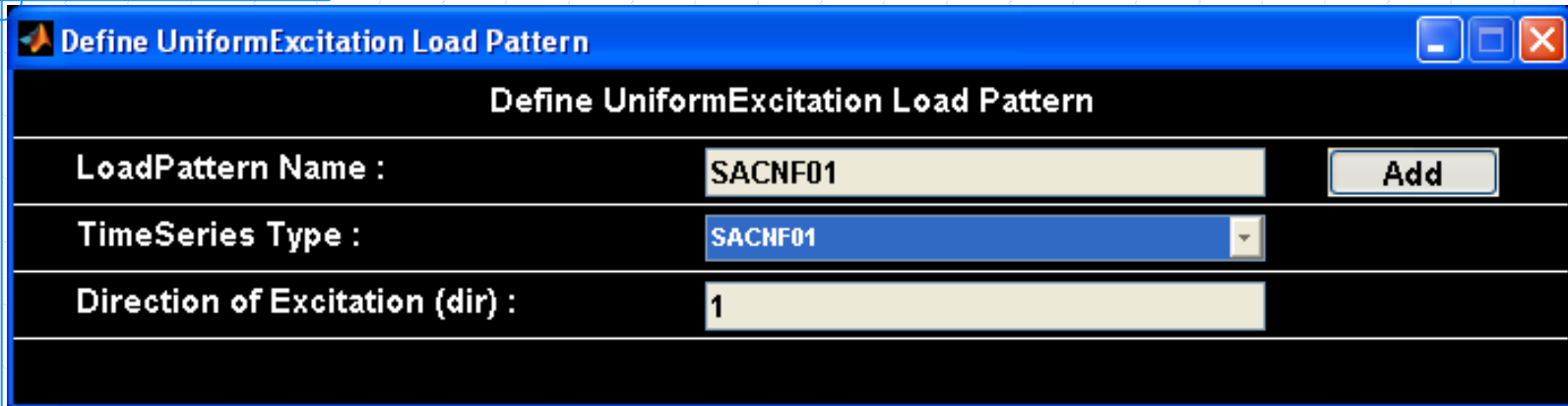
Define LoadPattern



Templates:

- MultipleSupport
- Plain
- UniformExcitation

Define UniformExcitation LoadPattern



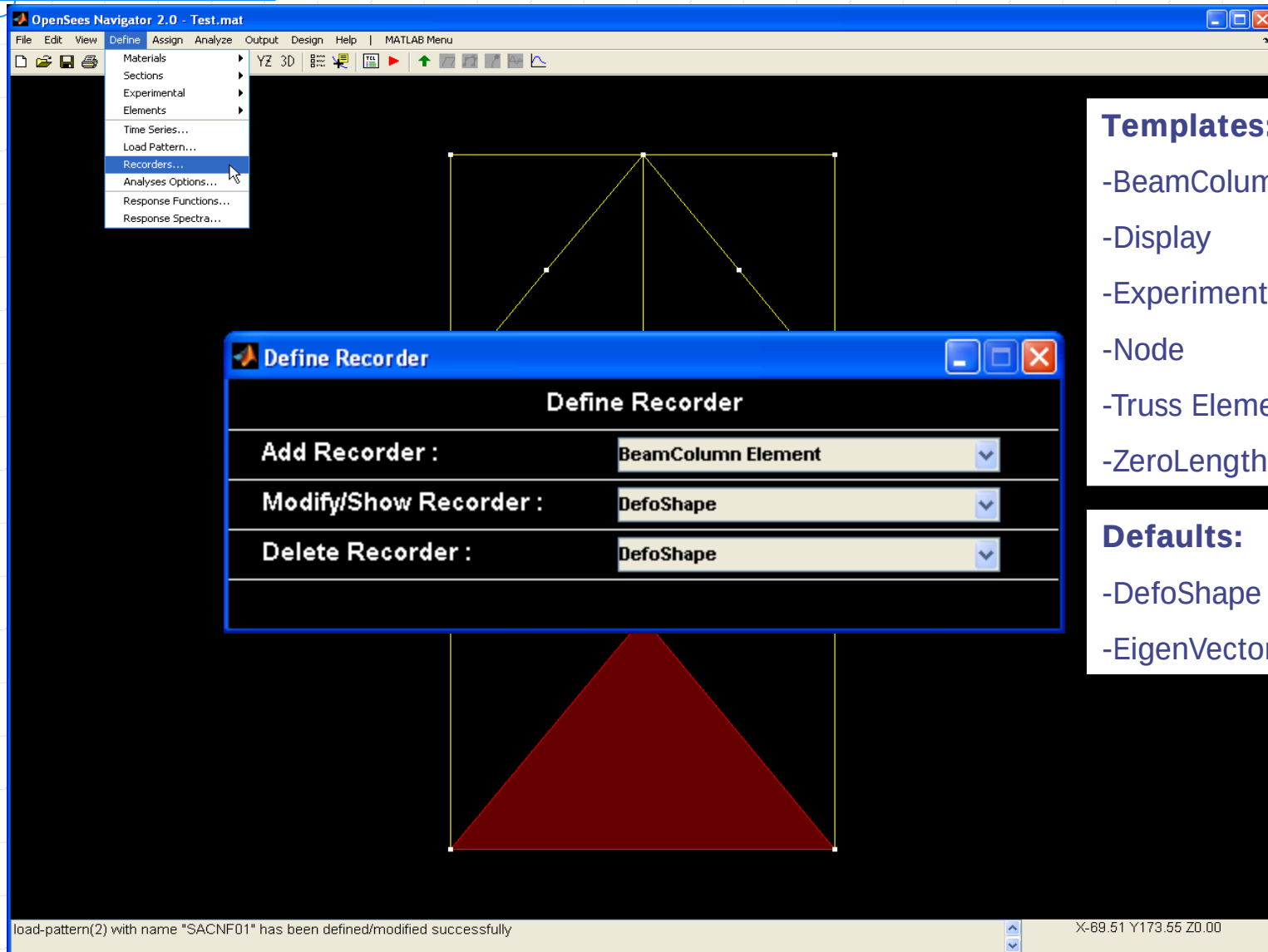
Define UniformExcitation Load Pattern

LoadPattern Name :

TimeSeries Type :

Direction of Excitation (dir) :

Define Recorder



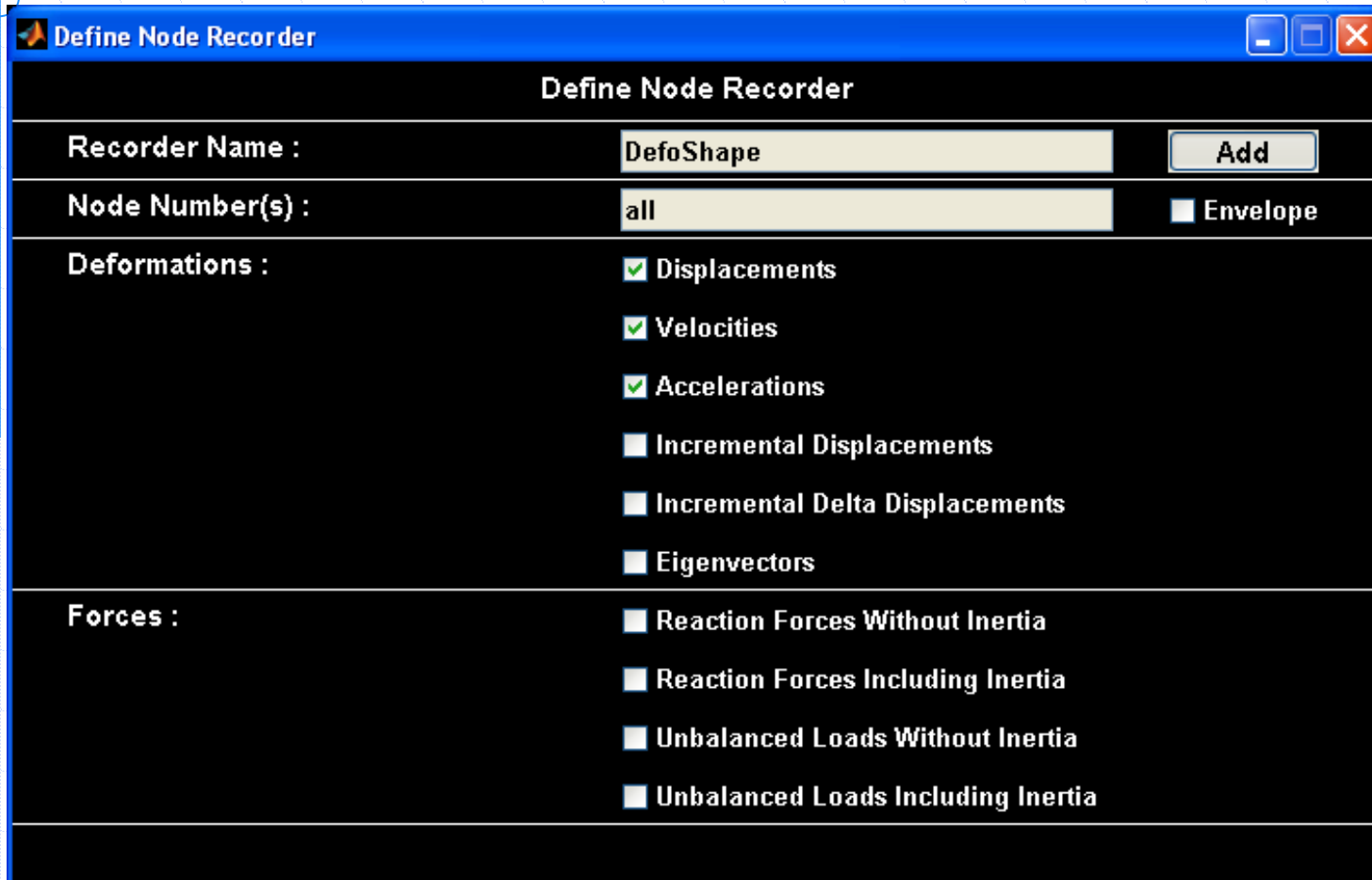
Templates:

- BeamColumn Element
- Display
- Experimental Element
- Node
- Truss Element
- ZeroLength Element

Defaults:

- DefoShape
- EigenVector

Define Node Recorder



The image shows a software dialog box titled "Define Node Recorder". It has a blue title bar with standard window controls. The dialog is divided into several sections. The first section has a label "Recorder Name :" followed by a text input field containing "DefoShape" and an "Add" button. The second section has a label "Node Number(s) :" followed by a text input field containing "all" and an "Envelope" checkbox. The third section, labeled "Deformations :", contains a list of options with checkboxes: "Displacements" (checked), "Velocities" (checked), "Accelerations" (checked), "Incremental Displacements" (unchecked), "Incremental Delta Displacements" (unchecked), and "Eigenvectors" (unchecked). The fourth section, labeled "Forces :", contains a list of options with checkboxes: "Reaction Forces Without Inertia" (unchecked), "Reaction Forces Including Inertia" (unchecked), "Unbalanced Loads Without Inertia" (unchecked), and "Unbalanced Loads Including Inertia" (unchecked).

Define Node Recorder

Recorder Name : DefoShape **Add**

Node Number(s) : all ☐ Envelope

Deformations :

- ☒ Displacements
- ☒ Velocities
- ☒ Accelerations
- ☐ Incremental Displacements
- ☐ Incremental Delta Displacements
- ☐ Eigenvectors

Forces :

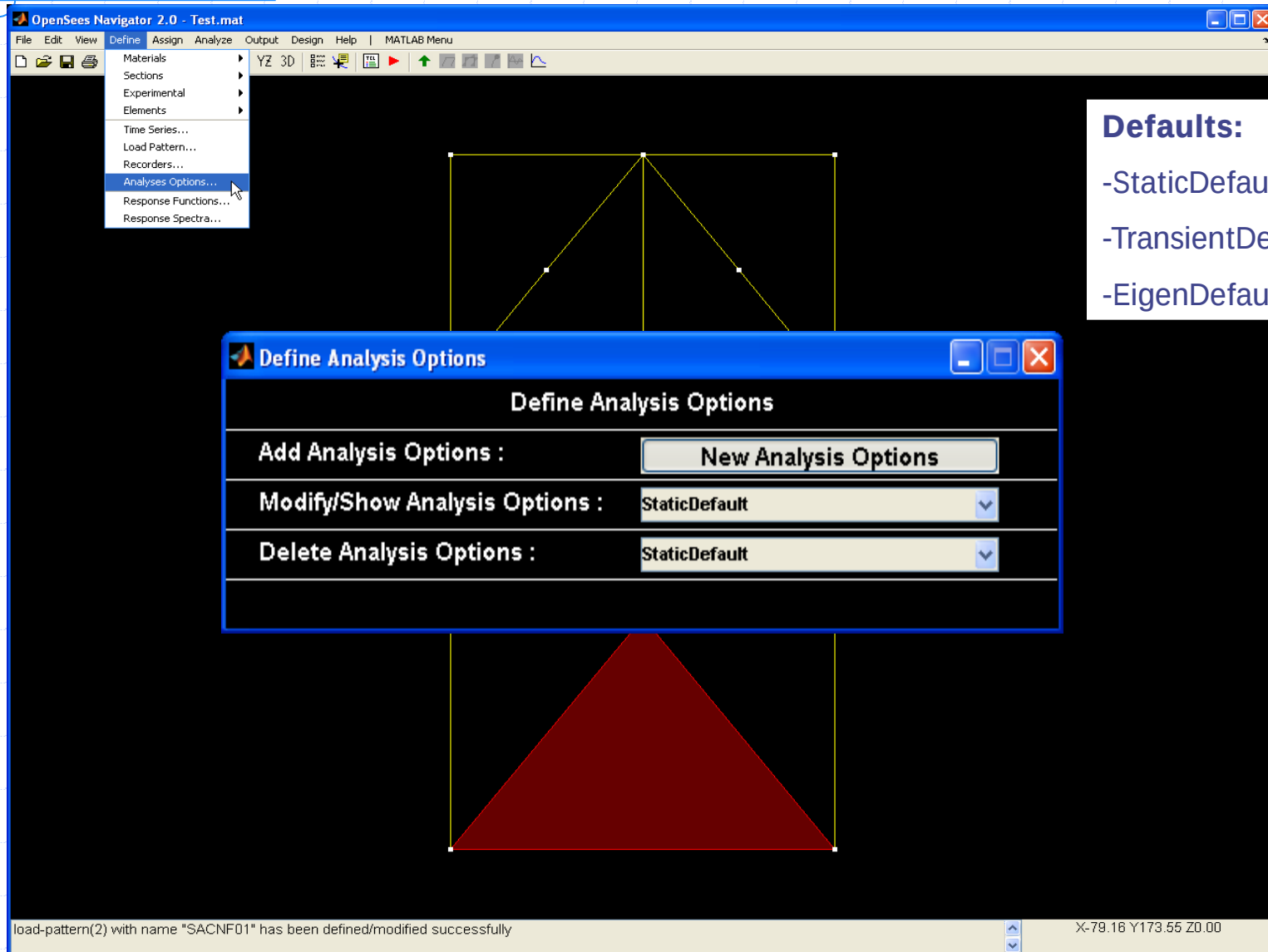
- ☐ Reaction Forces Without Inertia
- ☐ Reaction Forces Including Inertia
- ☐ Unbalanced Loads Without Inertia
- ☐ Unbalanced Loads Including Inertia

Define BeamColumn Recorder

Define BeamColumn Element Recorder

Recorder Name :	ElemForces	Add
Element Number(s) :	all	☐ Envelope
Arguments :	☒ Global Resisting Forces ☒ Local Resisting Forces	
Section Response :	☒ Section	
Section Number(s) :	1 5	
Arguments :	☒ Forces ☒ Deformations ☐ Stiffness	
Fiber Response :	☒ Fiber	
Arguments :	☐ Stress/Strain 0. Y-Coor 0. Z-Coor	

Define Analysis Options



Defaults:

- StaticDefault
- TransientDefault
- EigenDefault

Define New Analysis Options

Define New Analysis Options

Analysis Optn Name :	AnalysisOptn01	Add
Analysis Type :	Transient	
Constraint Handler Type :	Plain Constraints	
Integrator Type :	AlphaOS	
Solution Algorithm Type :	Linear	
Convergence Test Type :	Energy Increment	
DOF Numberer Type :	Plain	
System of Equations Type :	BandGeneral	

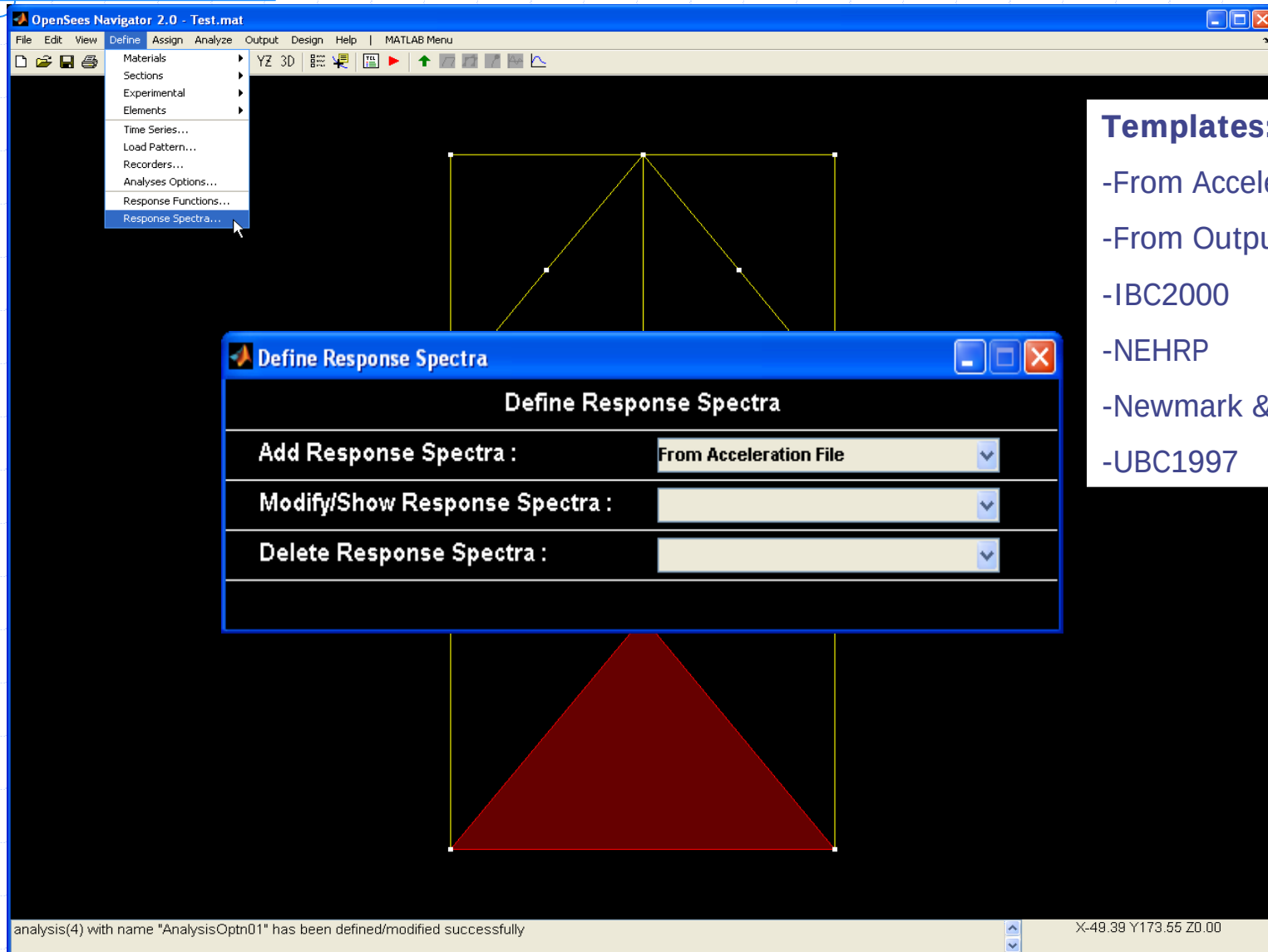
Integrator Type:

For example use AlphaOS Method for Hybrid Simulation

Solution Algorithm:

The AlphaOS Method requires a Linear solution algorithm

Define Response Spectra



Templates:

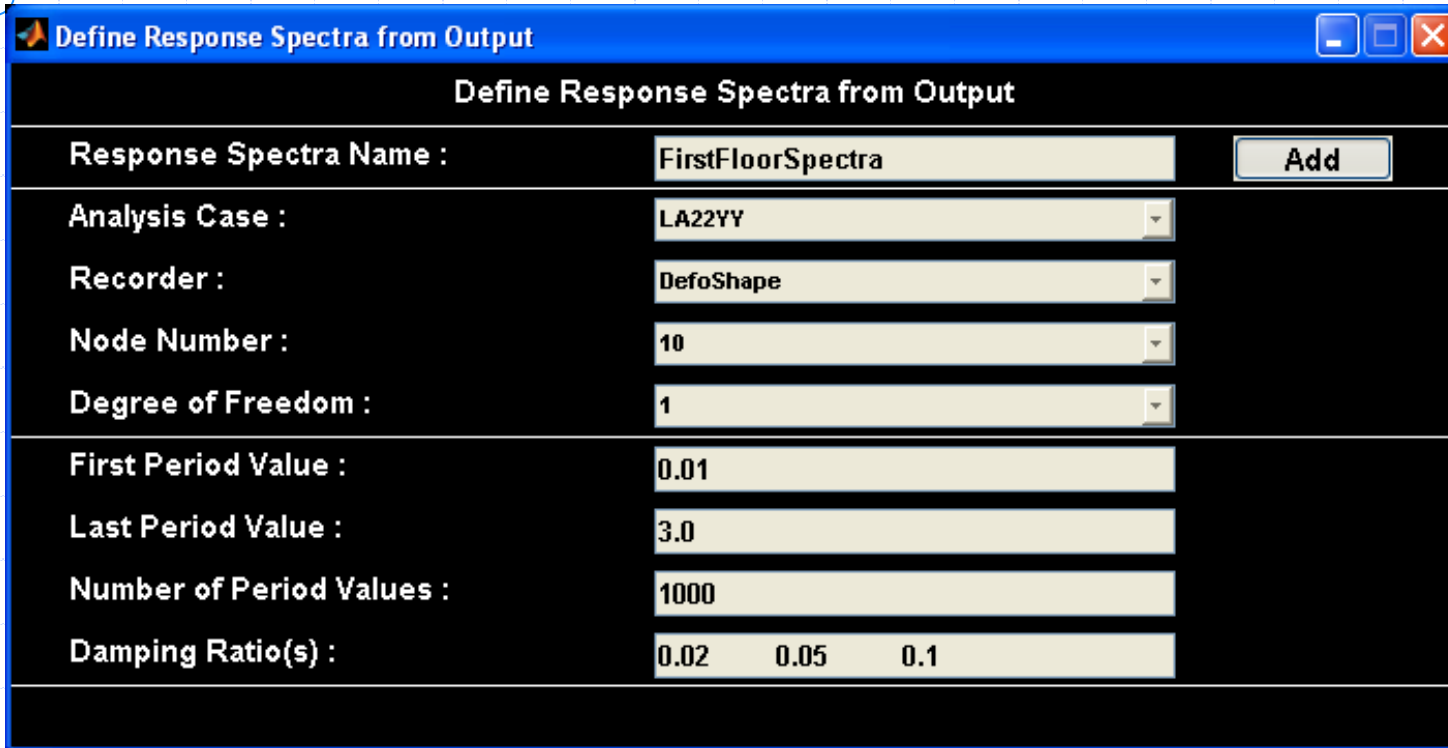
- From Acceleration File
- From Output
- IBC2000
- NEHRP
- Newmark & Hall
- UBC1997

Define From Acceleration File Spectra

Define Response Spectra from Acceleration File

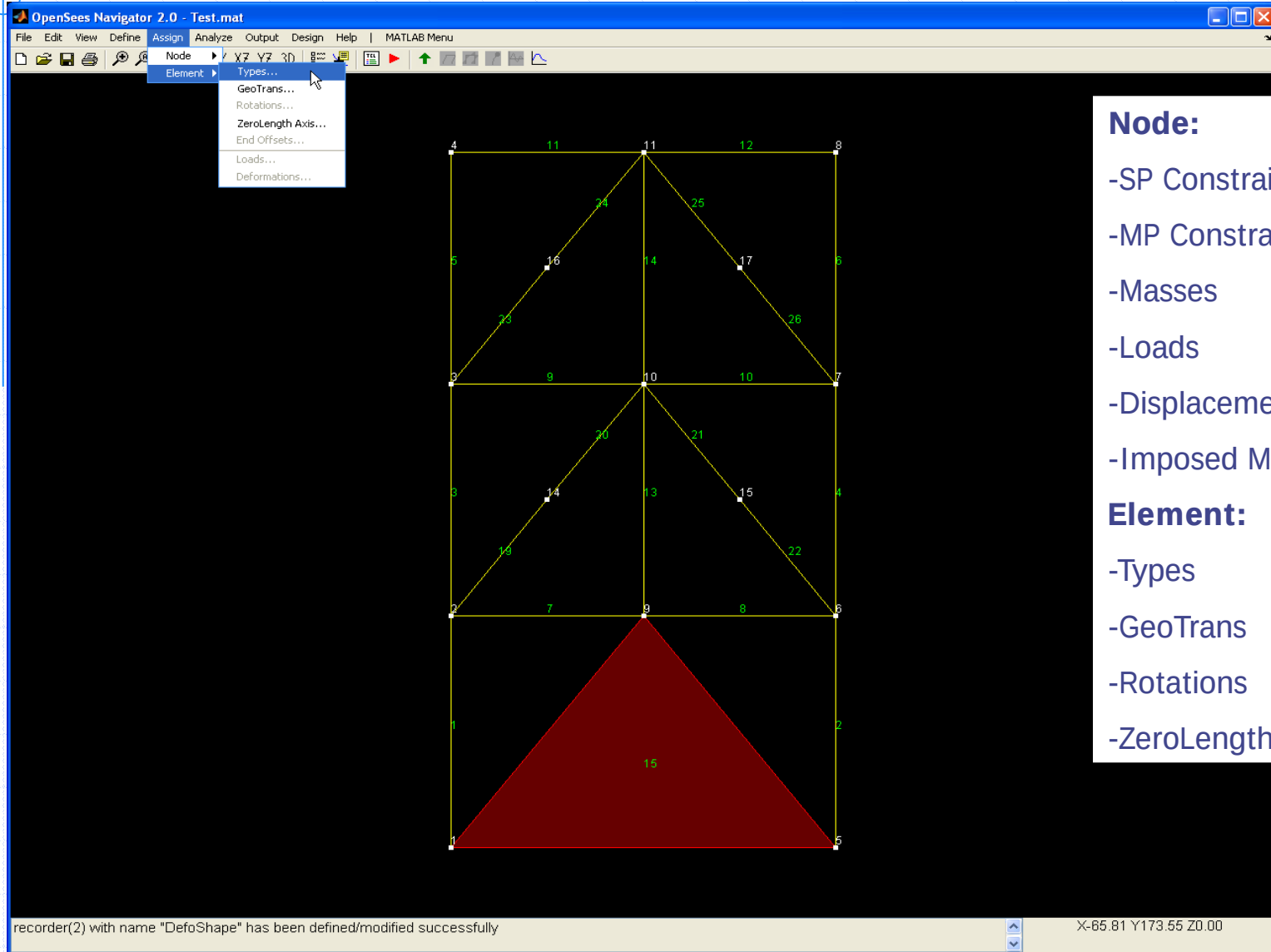
Response Spectra Name :	LA22YY	Add
Time Interval (dt) :	0.008164	
Acceleration File Name (filePath) :	ZipperExpHS\GroundMotions\la22yy.thf	Browse
Load Factor (cFactor) :	386.1	
First Period Value :	0.01	
Last Period Value :	3.0	
Number of Period Values :	1000	
Damping Ratio(s) :	0.02 0.05 0.1	

Define From Output Spectra

A screenshot of a software dialog box titled "Define Response Spectra from Output". The dialog box has a blue title bar with standard window controls (minimize, maximize, close). The main area is black with white text. It contains several input fields and a button. The fields are: "Response Spectra Name" with the value "FirstFloorSpectra" and an "Add" button; "Analysis Case" with a dropdown menu showing "LA22YY"; "Recorder" with a dropdown menu showing "DefoShape"; "Node Number" with a dropdown menu showing "10"; "Degree of Freedom" with a dropdown menu showing "1"; "First Period Value" with a text field showing "0.01"; "Last Period Value" with a text field showing "3.0"; "Number of Period Values" with a text field showing "1000"; and "Damping Ratio(s)" with a text field showing "0.02 0.05 0.1".

Define Response Spectra from Output	
Response Spectra Name :	FirstFloorSpectra Add
Analysis Case :	LA22YY
Recorder :	DefoShape
Node Number :	10
Degree of Freedom :	1
First Period Value :	0.01
Last Period Value :	3.0
Number of Period Values :	1000
Damping Ratio(s) :	0.02 0.05 0.1

Assign Menu



Node:

- SP Constraints
- MP Constraints
- Masses
- Loads
- Displacements
- Imposed Motions

Element:

- Types
- GeoTrans
- Rotations
- ZeroLength Axis

Assign Menu

Assign Nodal Masses

Replace/Add/Delete Masses :

Node Number(s) :

Mass X-dir :

Mass Y-dir :

Mass Moment of Inertia Z-dir :

Select Nodes

X-Coordinate :

Y-Coordinate :

Assign Element Types

Assign Element Types :

Element Number(s) :

Element Type :

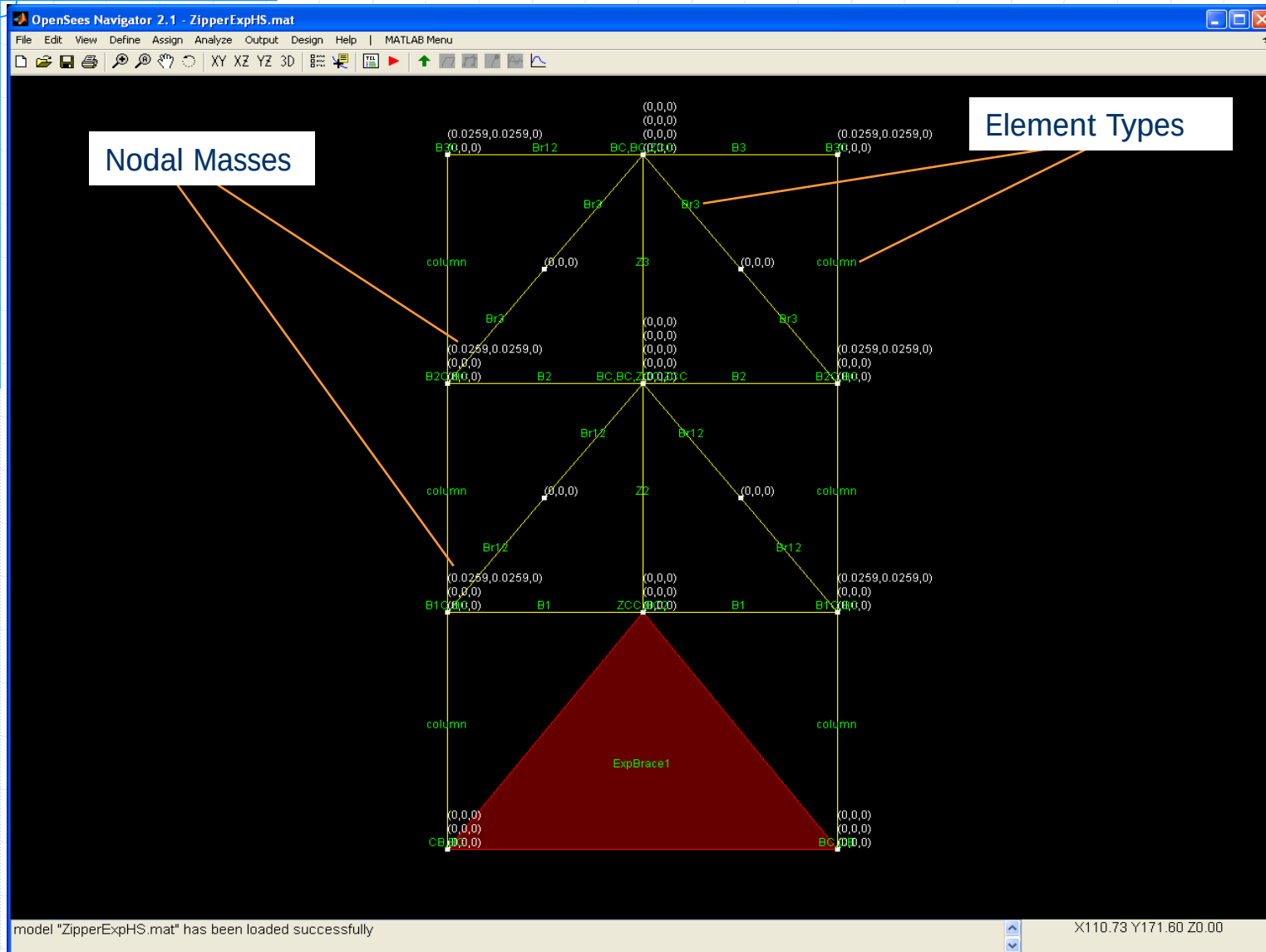
Assign Element Geometric Transformations

Assign Geometric Transformations :

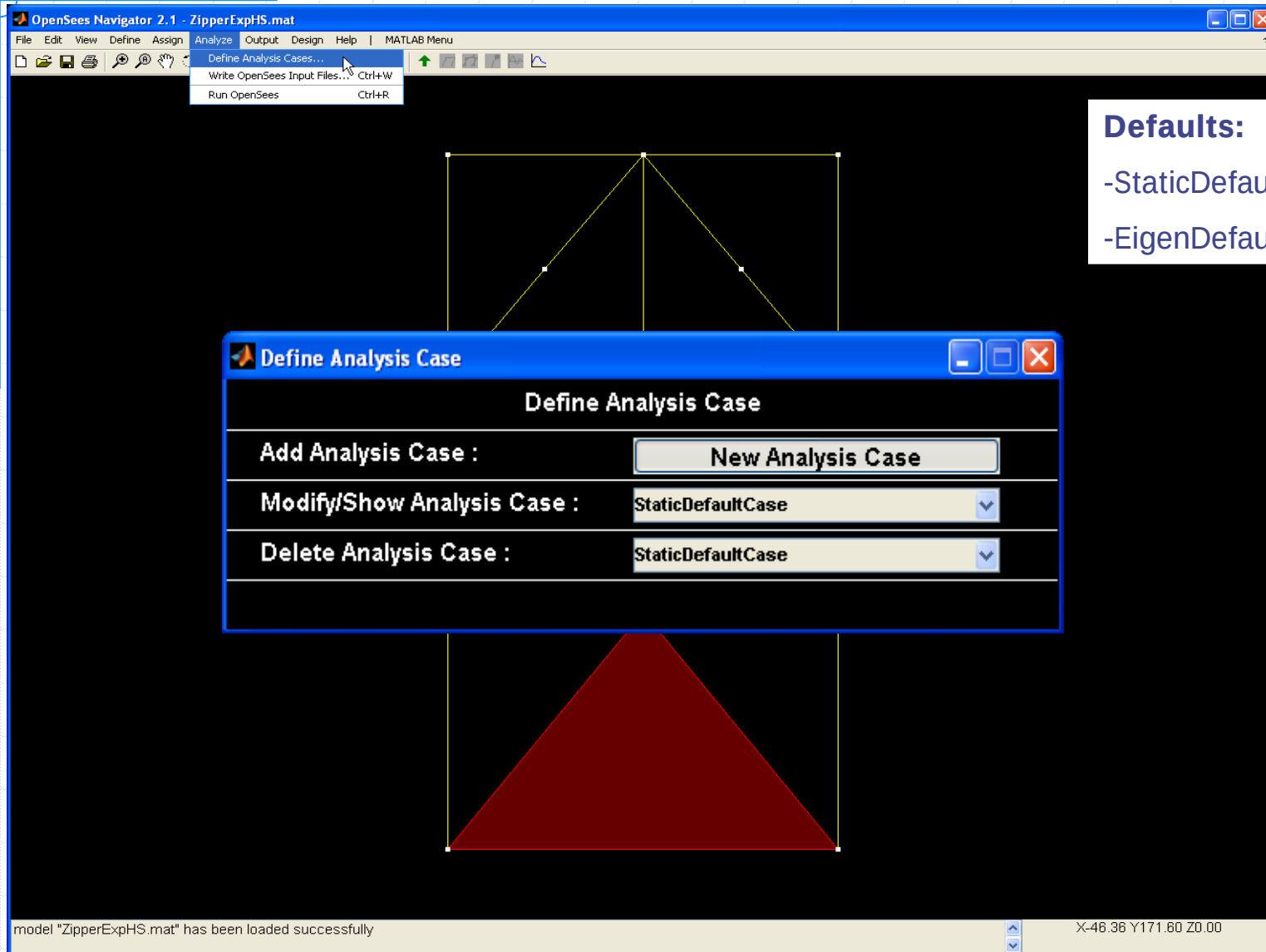
Element Number(s) :

Geometric Transformation :

Assigned Properties



Define Analysis Cases



Defaults:

- StaticDefaultCase
- EigenDefaultCase

Define New Analysis Case

Define New Analysis Case

Analysis Case Name :

Load Pattern Name(s) :
LA22YY5percent
None

Recorder Name(s) :
EigenVector
None

Analysis Options Name :

User Defined Analysis Script :

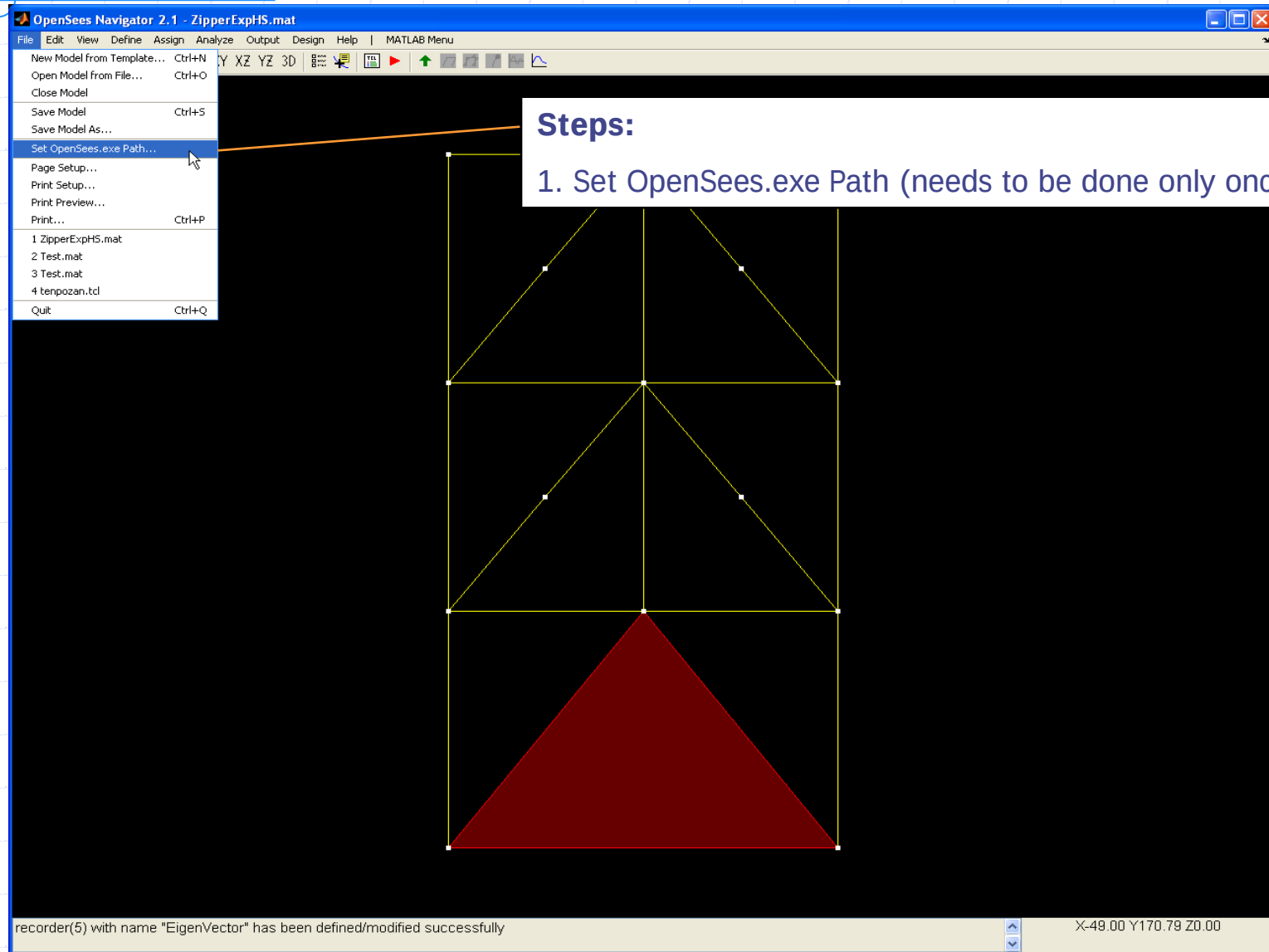
Start from Previous Analysis Case :

Num of Eigenvalues (numEigVal) :

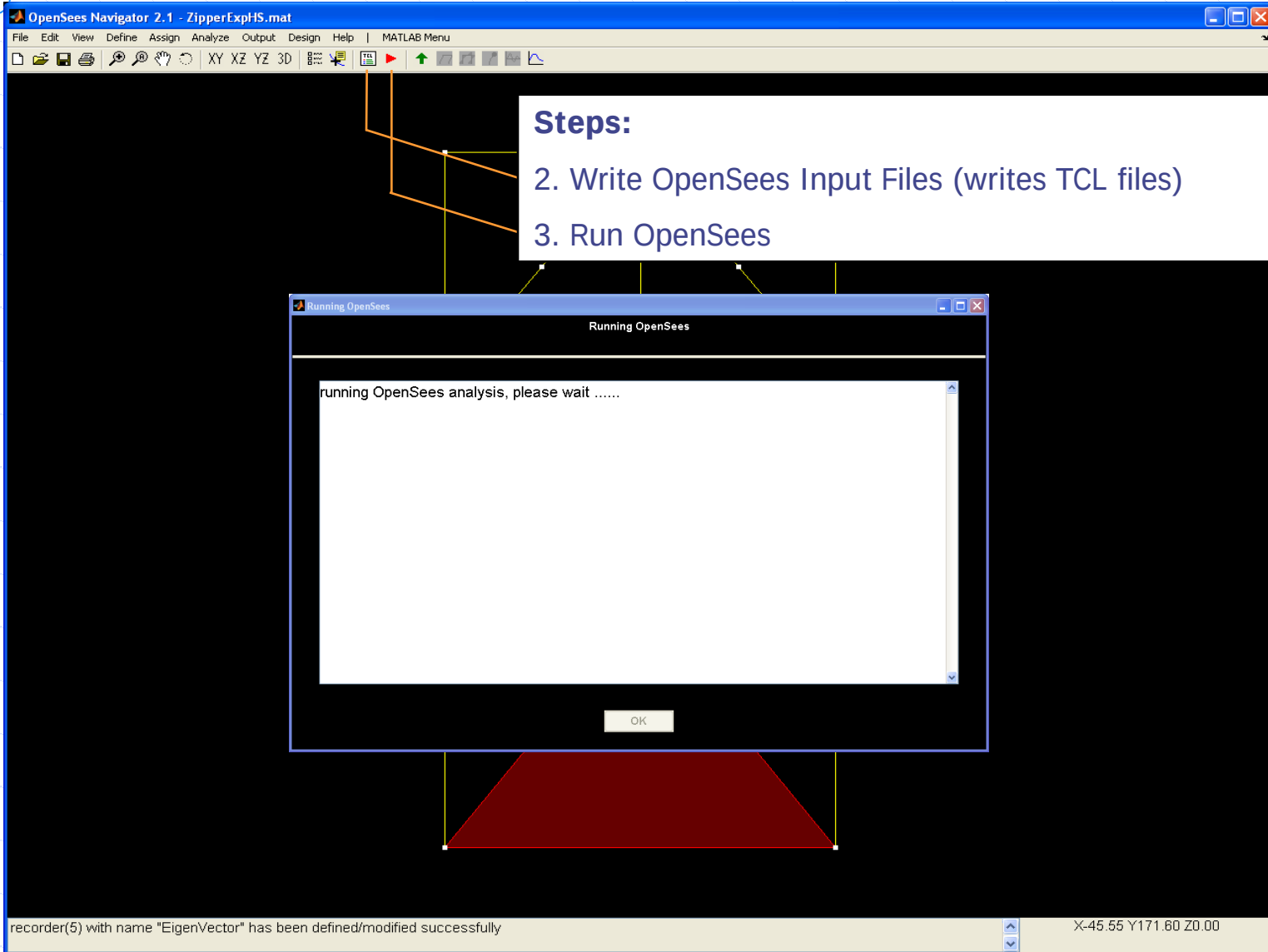
Analysis Type (typeEig) :

For Example:
Periods and Mode Shapes after Time-History Analysis

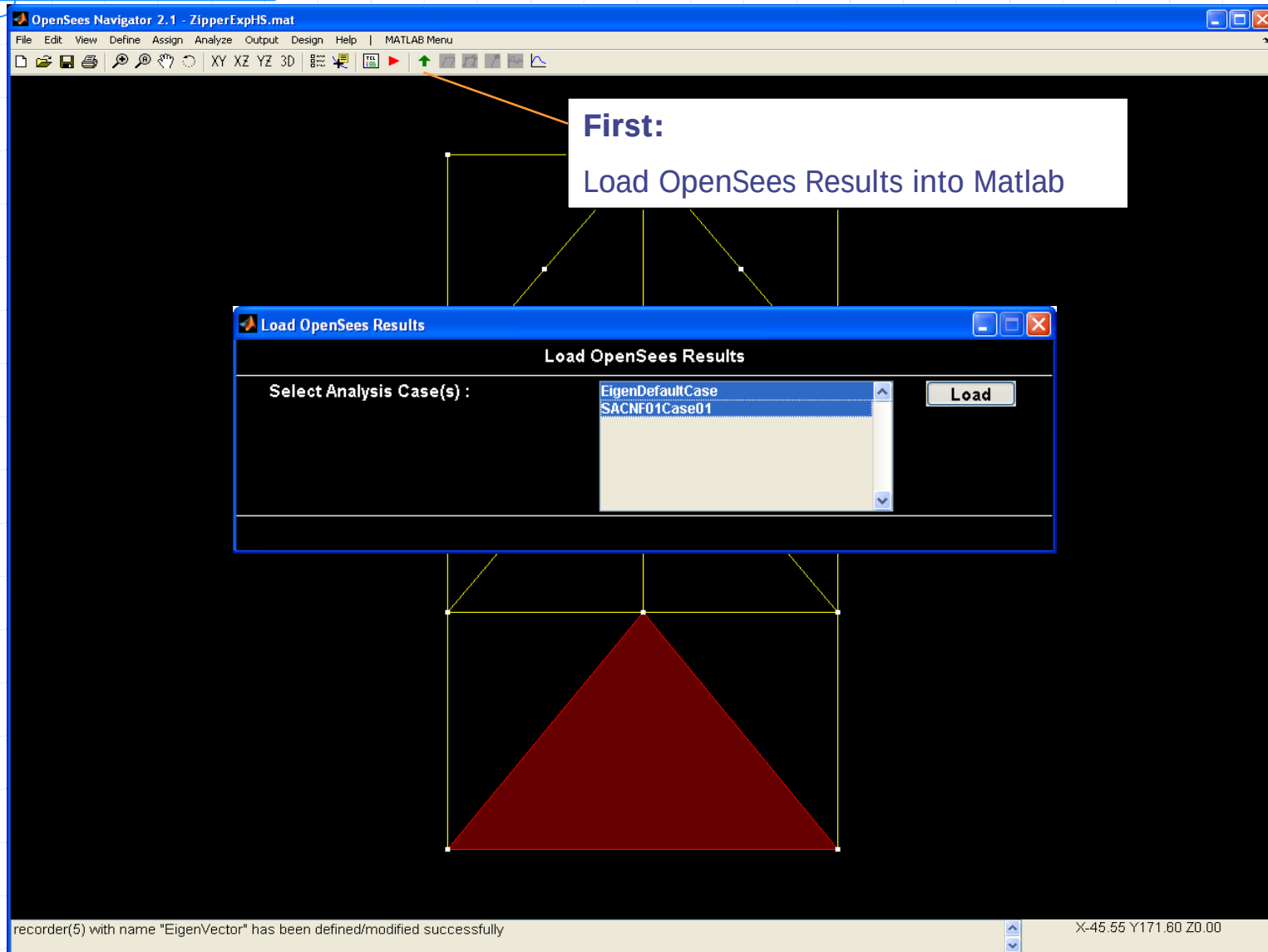
To Run OpenSees



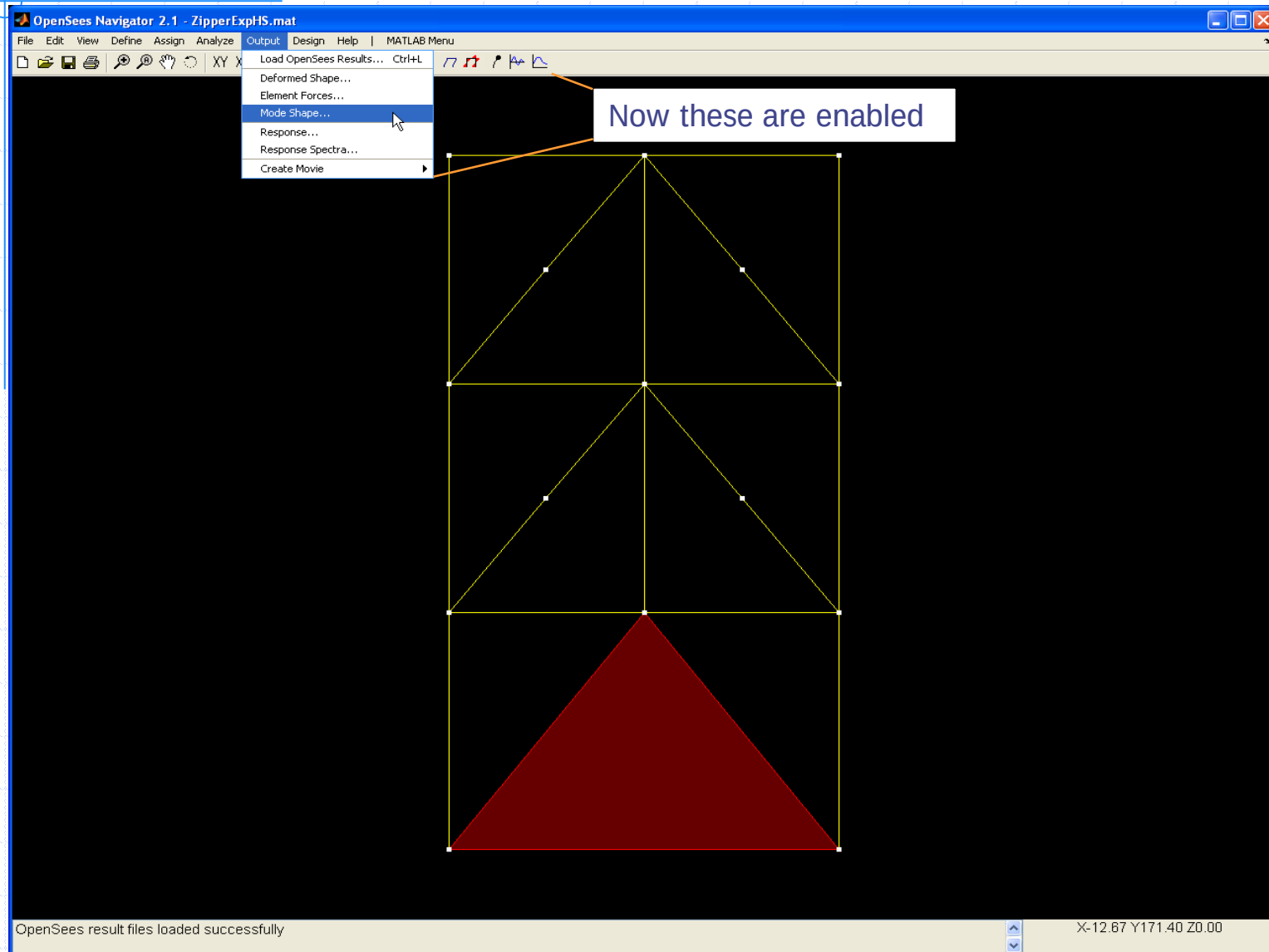
To Run OpenSees



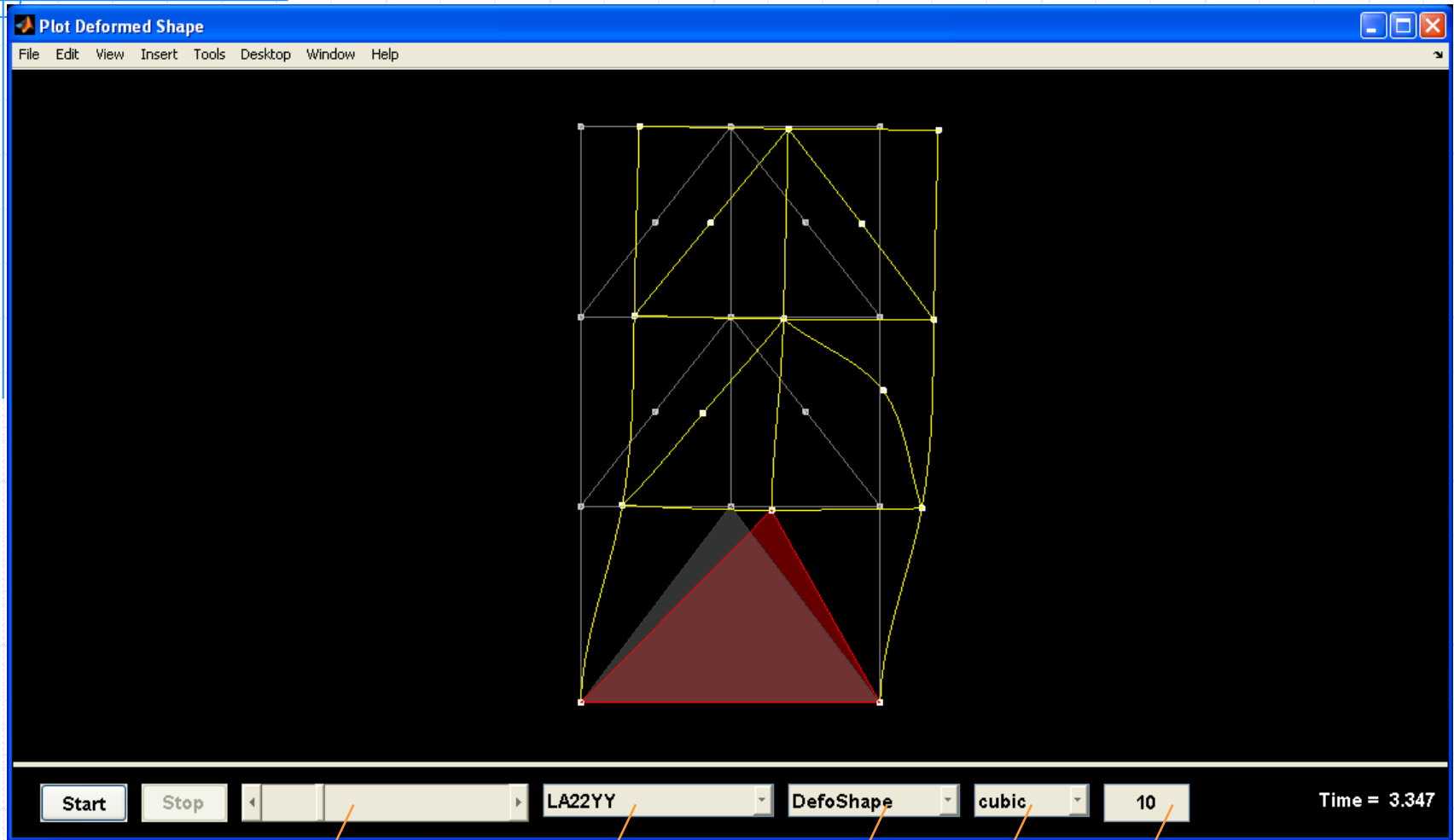
Post-Processing: Output



Post-Processing: Output



Plot Deformed Shape



Time Step

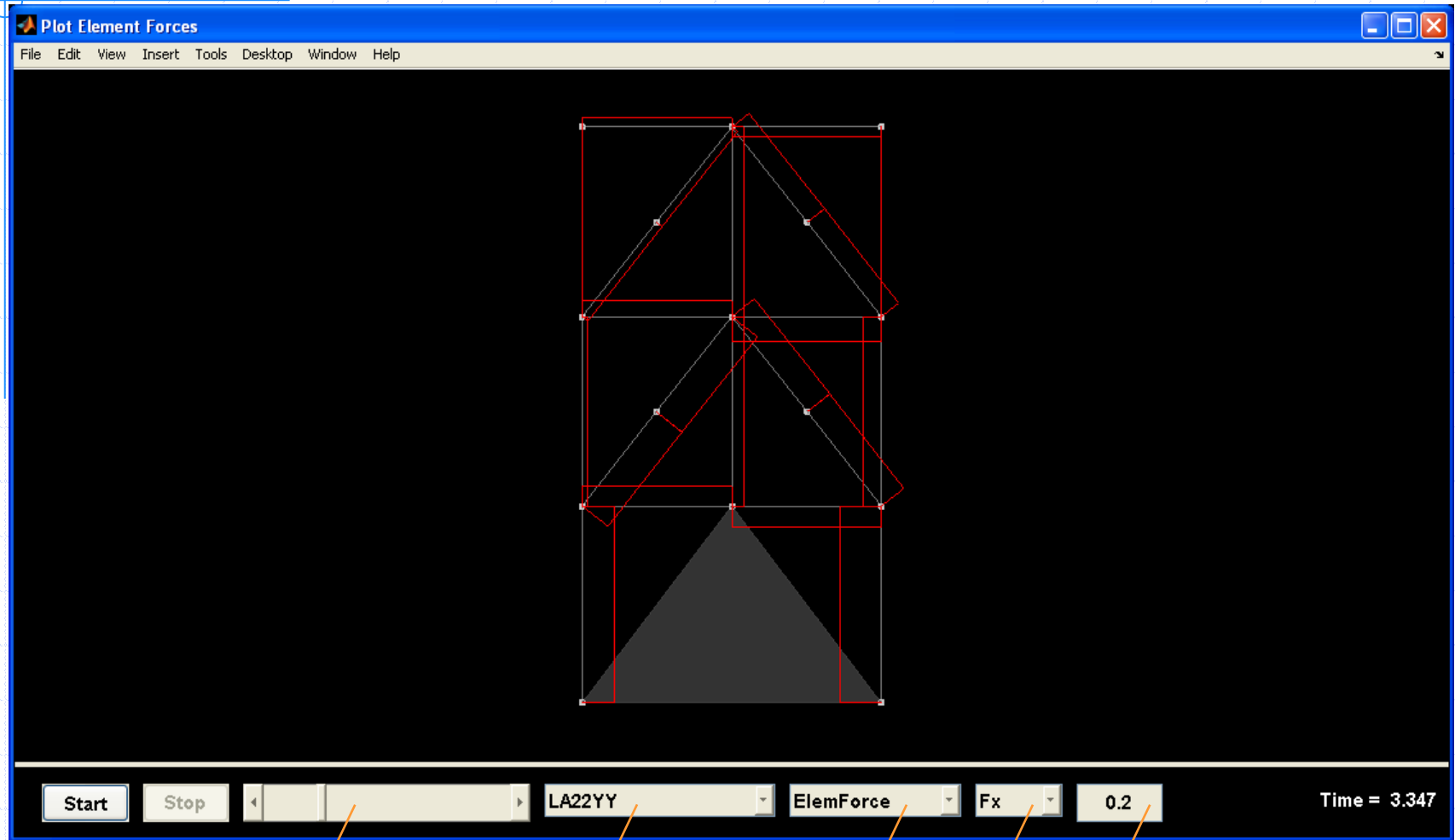
AnalysisCase

Recorder

Order

Magnification

Plot Element Forces: Axial Forces



Time Step

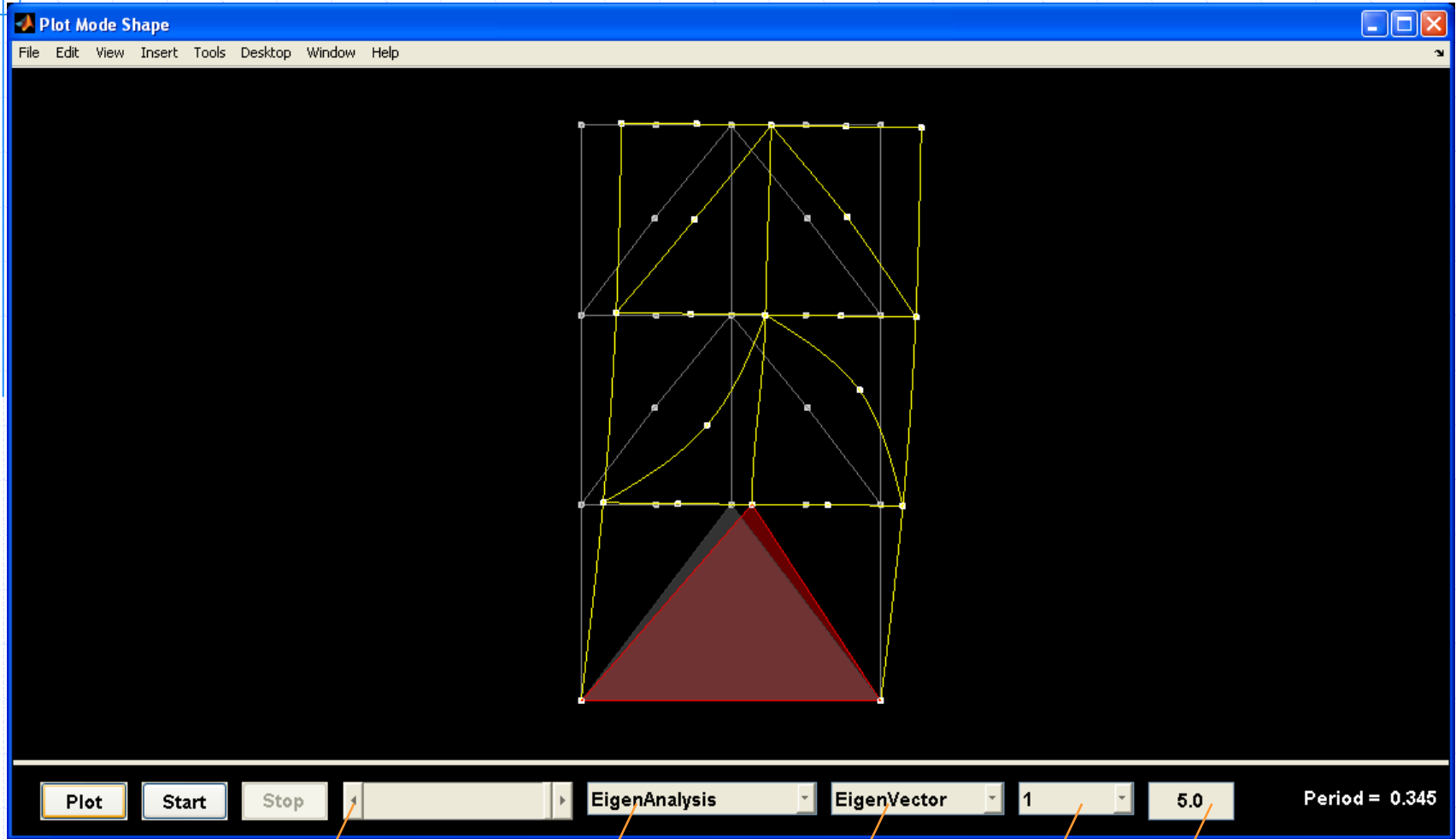
AnalysisCase

Recorder

Response

Magnification

Plot Mode Shape: 1st Mode



Animation Speed

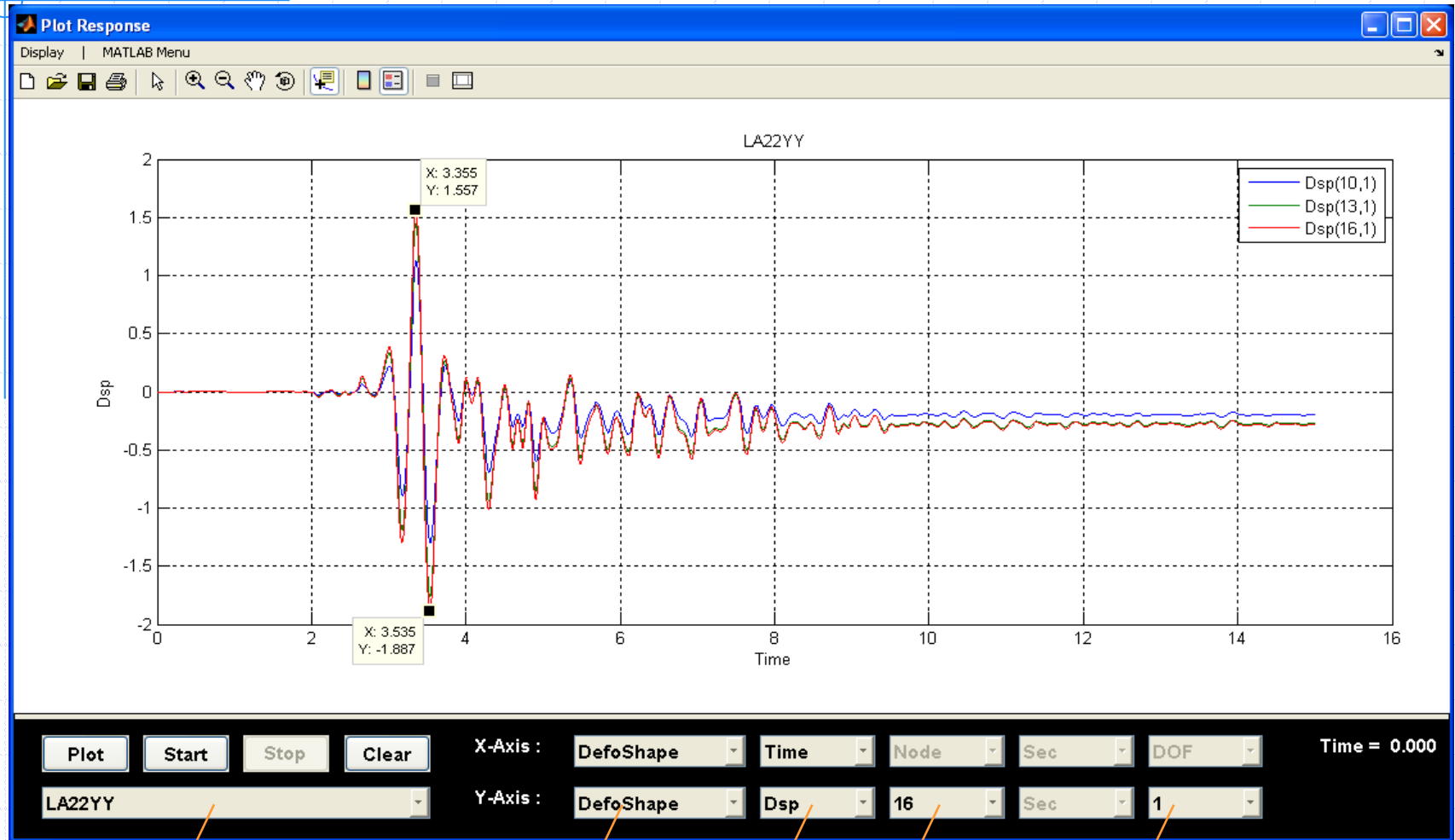
AnalysisCase

Recorder

Mode

Magnification

Plot Response



AnalysisCase

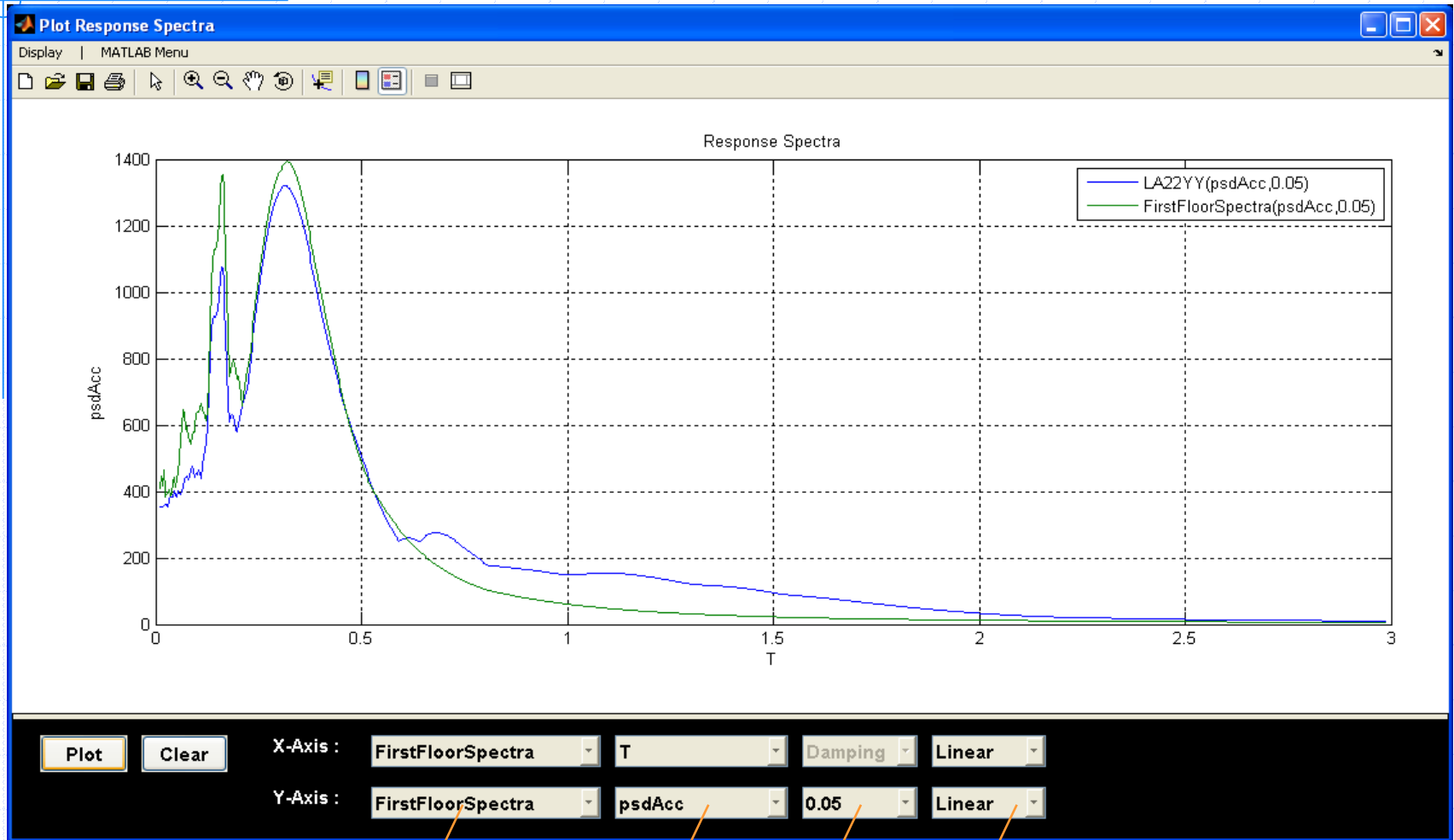
Recorder

Parameter

Node/Element

DOF

Plot Response Spectra



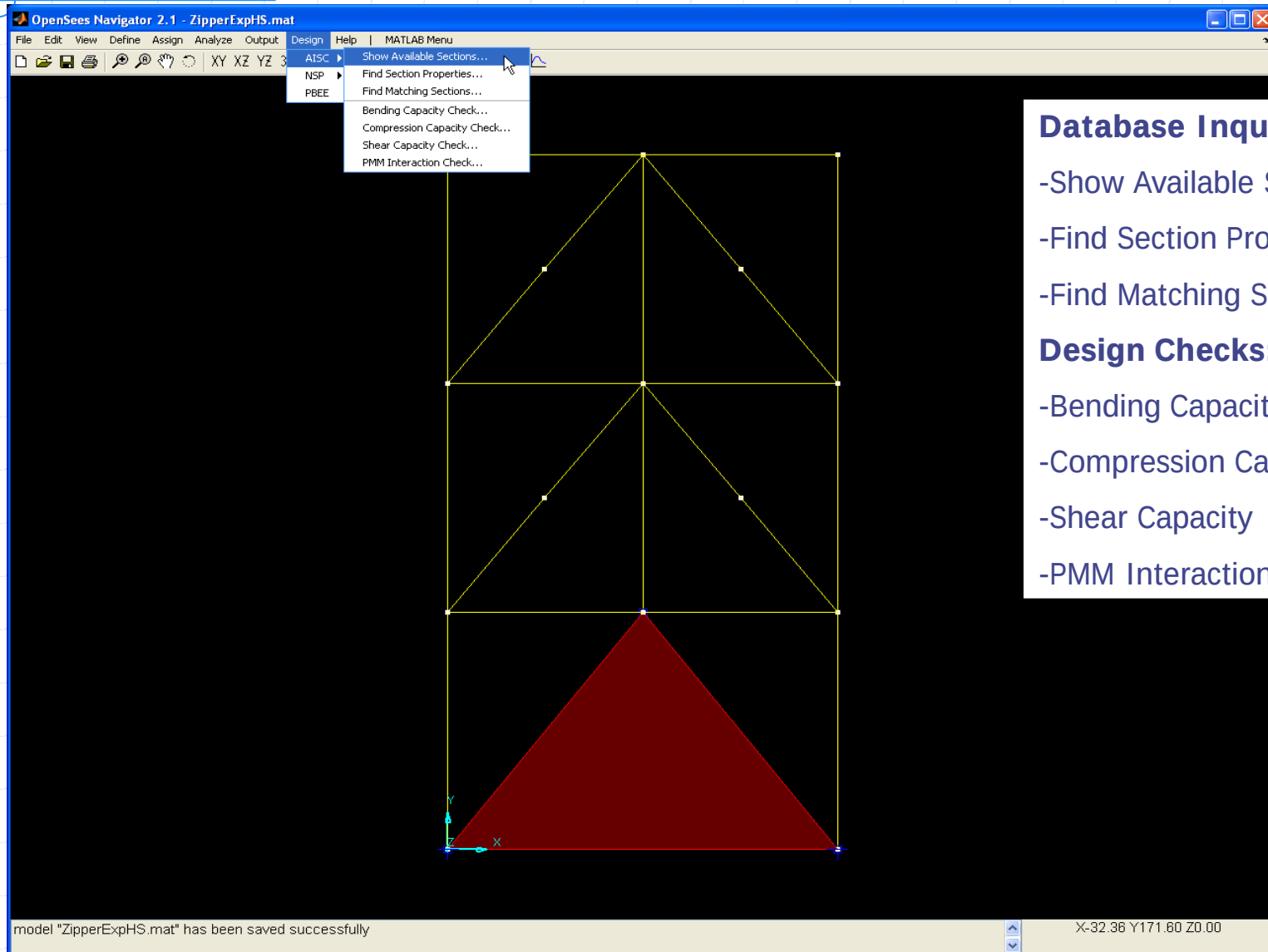
Spectra Name

Response

Damping

Axis Scale

AISC Design Toolbox



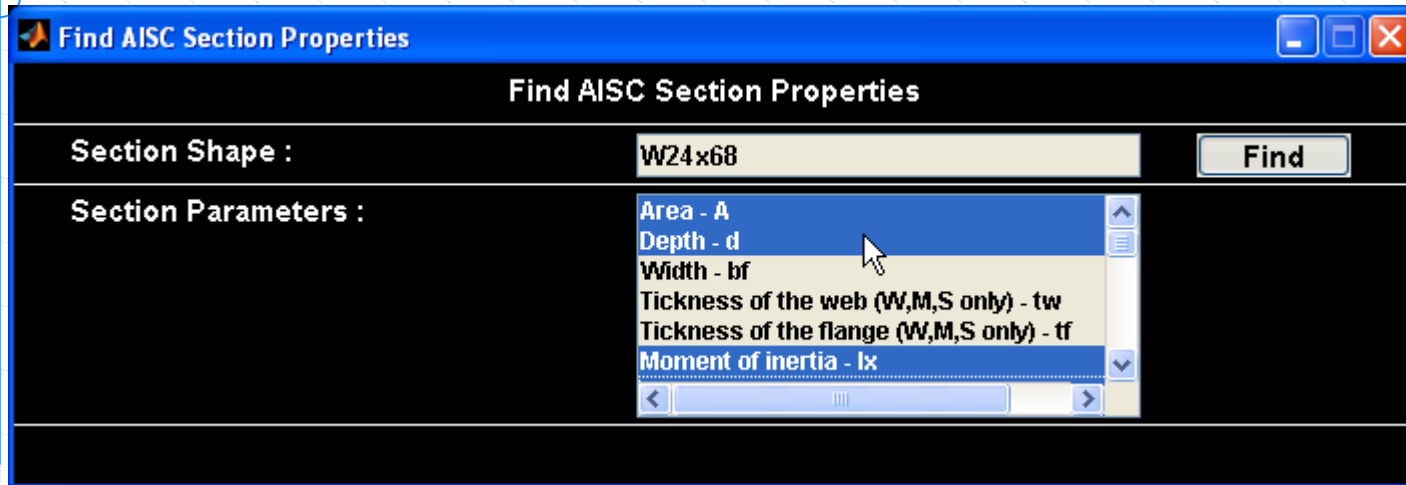
Database Inquiries:

- Show Available Sections
- Find Section Properties
- Find Matching Sections

Design Checks:

- Bending Capacity
- Compression Capacity
- Shear Capacity
- PMM Interaction

AISC Toolbox: Find Section Prop.

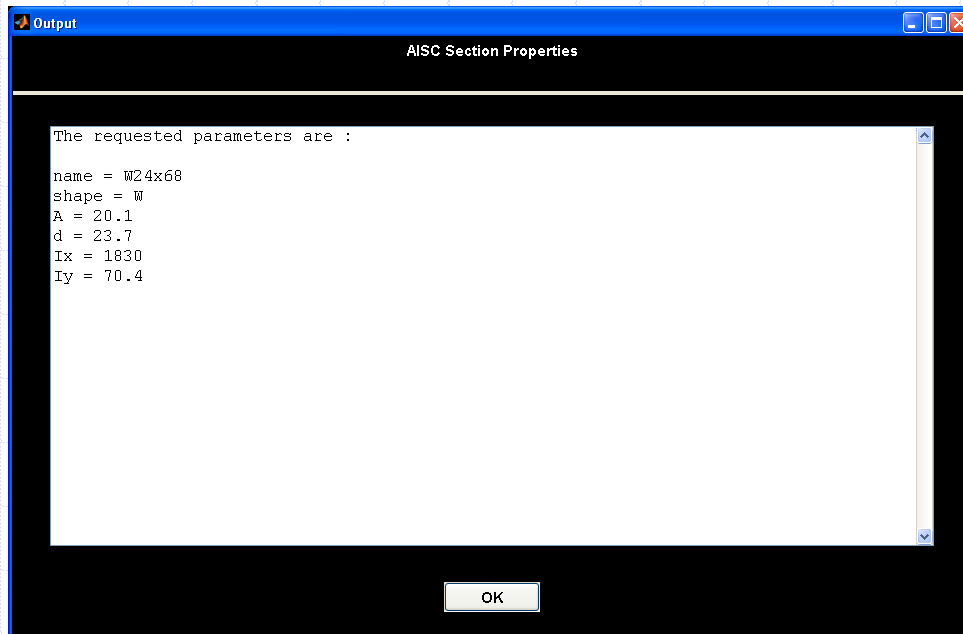


The 'Find AISC Section Properties' dialog box has a blue title bar with standard window controls. The main area is black with white text. It features two input fields: 'Section Shape' containing 'W24x68' and 'Section Parameters' containing a list of properties. A 'Find' button is located to the right of the 'Section Shape' field. The 'Section Parameters' list includes 'Area - A', 'Depth - d', 'Width - bf', 'Thickness of the web (W,M,S only) - tw', 'Thickness of the flange (W,M,S only) - tf', and 'Moment of inertia - Ix'. A mouse cursor is pointing at 'Depth - d'.

Find AISC Section Properties

Section Shape :

Section Parameters :



The 'Output' dialog box has a blue title bar with standard window controls. The main area is black with white text. It displays the results of the search for 'W24x68' section properties. The text is as follows: 'The requested parameters are :', 'name = W24x68', 'shape = W', 'A = 20.1', 'd = 23.7', 'Ix = 1830', and 'Iy = 70.4'. An 'OK' button is located at the bottom center.

Output

AISC Section Properties

The requested parameters are :

name = W24x68
shape = W
A = 20.1
d = 23.7
Ix = 1830
Iy = 70.4

AISC Toolbox: Find Matching Sec.

Find Matching AISC Sections

Select Section Shape :

Add Parameter : Min : Max :

Modify Parameter : Min : Max :

Delete Parameter :

Sort by Parameter :

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

AISC Toolbox: Bending Capacity

AISC Bending Capacity

Bending Capacity of AISC Section

Section Shape :	W24x68	Calculate
Unbraced Length (Lb) :	40	[in]
Bending Coefficient (Cb) :	1	[-]
Yield Stress (Fy) :	50	[ksi]
Modulus of Elasticity (E) :	29000	[ksi]
Direction :	strong	

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

Output

AISC Bending Capacity

```
Bending capacity for section W24x68 :  
With Lb = 120 in  
  Cb = 1  
  Fy = 50 ksi  
  E  = 29000 ksi  
  
phi = 0.9  
Mp = 8850  
Mr = 6160  
Lp = 79.2626  
Lr = 208.7244  
Flange_Compactness = Compact  
Web_Compactness = Compact  
Capacity = 7203.19  
FailureMode = Lateral torsional buckling
```

OK

AISC Toolbox: Compression Capacity

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, Qa(web) = 1)  
Capacity = 661.6242
```

OK

AISC Toolbox: Shear Capacity

AISC Shear Capacity

Shear Capacity of AISC Section

Section Shape :	W24x68	Calculate
Distance between Stiffeners (a) :	24	[in]
Yield Stress (Fy) :	50	[ksi]
Modulus of Elasticity (E) :	29000	[ksi]

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

Output

AISC Shear Capacity

The Shear Capacity parameters are :

```
T_tw_p = 78.2264  
T_tw_r = 97.4274  
T_tw = 49.8795  
phi = 0.9  
Capacity = 265.5585  
FailureMode = Reaching yielding capacity 0.6*Fy
```

OK

AISC Toolbox: PMM Interaction

AISC PMM Interaction Check

PMM Interaction Check of AISC Section

Section Shape :	W24x68	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	[-]

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
- Easy way to study material, section, element or system behavior
- Hybrid Simulation interface
- Many built in post processing toolboxes
- Great tool to visualize structural behavior
- Response Spectra generation
- Graphical user interface for AISC steel manual
- Design toolboxes

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: Downloads

OPENSEES NAVIGATOR

[HOME](#) [INTRODUCTION](#) [MANUALS](#) [TUTORIALS](#) [PRESENTATIONS](#) [DISCUSSION](#) [UPDATES](#) [DOWNLOADS](#) [LINKS](#)

Home> Downloads> Stand-Alone Windows>

MCRInstaller.exe

OpenSeesNavigator.zip

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.

Hit Counter

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

Thank you!

OpenSees Navigator 2.1 is available at
<http://peer.berkeley.edu/OpenSeesNavigator>

Andreas Schellenberg:
andreas.schellenberg@gmail.com

Tony Yang:
yangtony2004@gmail.com



nees@berkeley

The George E. Brown, Jr. Network for Earthquake Engineering Simulation

