

Promoting and Enabling International Collaboration

# **OpenFresco: An open source framework for performing geographically distributed and local hybrid simulations**

**Andreas Schellenberg, Hong Kim  
Stephen A. Mahin, Gregory L. Fenves**  
Department of Civil and Environmental Engineering  
University of California, Berkeley

**Yoshikazu Takahashi**  
Kyoto University

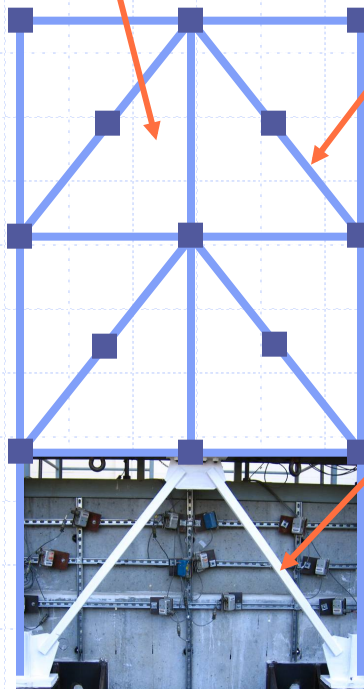




# Hybrid Simulation

$$\mathbf{M} \cdot \ddot{\mathbf{u}} + \mathbf{C} \cdot \dot{\mathbf{u}} + \mathbf{P}_r(\mathbf{u}) = \mathbf{P}(t)$$

Dynamic Loading

- Seismic
- Wind
- Blast/Impact
- Wave
- Vehicle



-  analytical model of structural energy dissipation and inertia
-  physical model of structural resistance

Versatile implementation and support for:

- Slow or fast execution
- Local or geographically distributed
- Displacement, force & mixed control
- Collaborative computing

# OpenFresco

## Open-source Framework for Experimental Setup and Control

Secure, object oriented, network enabled  
“middleware” -- Pairs computer analysis software with laboratory control systems and other software to enable hybrid and collaborative computing:

### ◆ Software

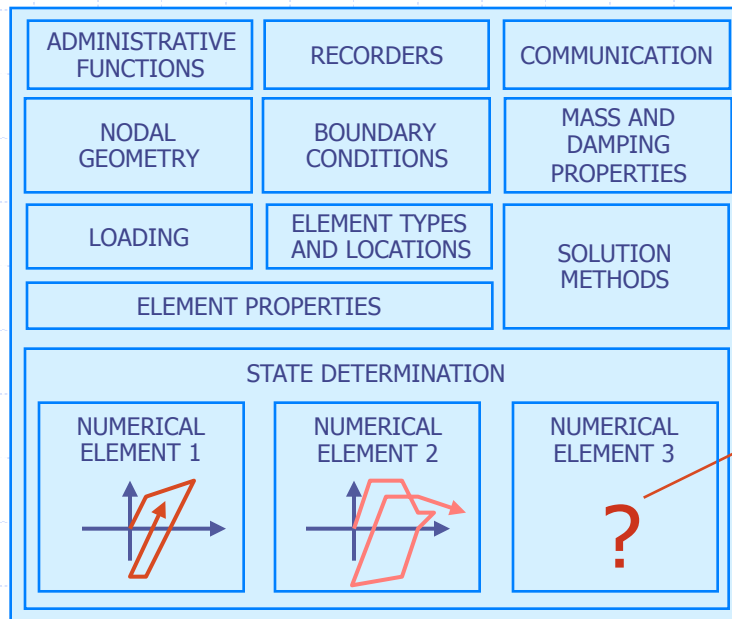
- Abacus
- Ansys (soon)
- OpenFresco Express
- LS-Dyna
- Matlab
- OpenSees
- SimCor
- Simulink

### ◆ Control Systems

- dSpace
- MTS
  - ◆ STS family
  - ◆ Flextest/CSI
  - ◆ Flextest/Scramnet
- National Instruments
- Pacific Instruments
- Shore Western

# Implementation strategy

Embed test specimen(s) in an existing computational framework of users choice

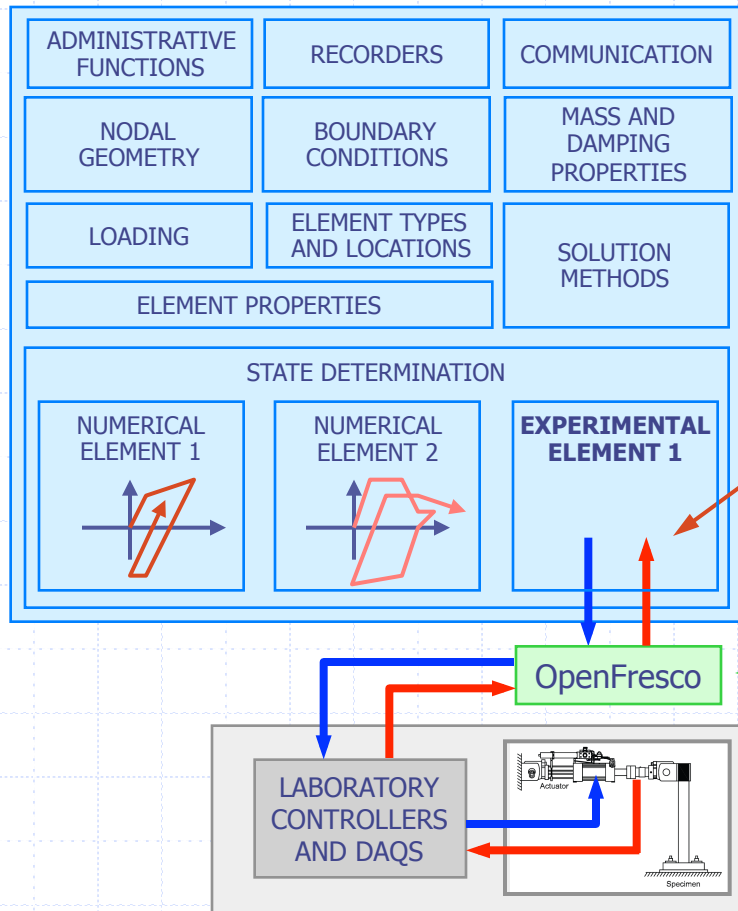


Typical features of an analysis framework

Proper numerical model uncertain

# Implementation strategy

Embed test specimen(s) in an existing computational framework of users choice



Typical features of an analysis framework

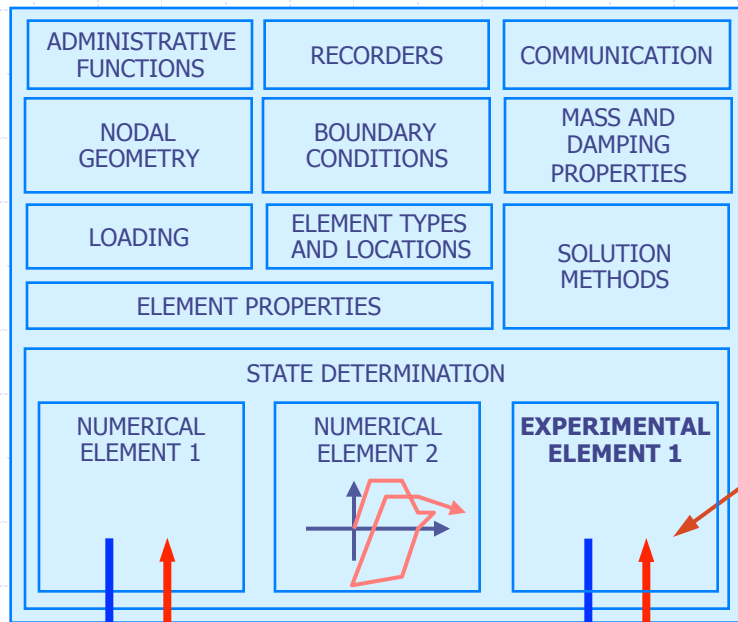
Define element as an "Experimental Element"

OpenFresco

Laboratory

# Implementation strategy

Embed test specimen(s) in an existing computational framework of users choice

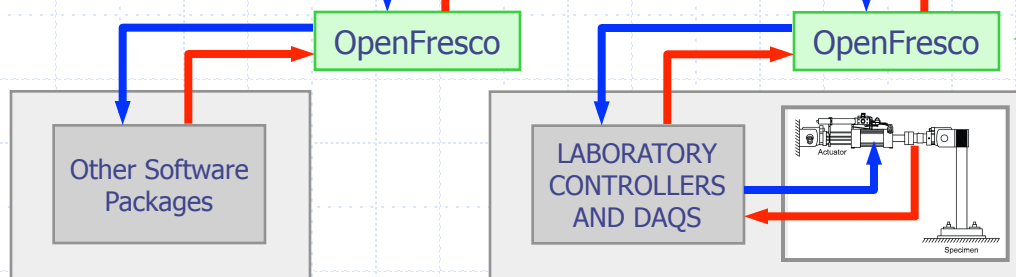


Typical features of an analysis framework

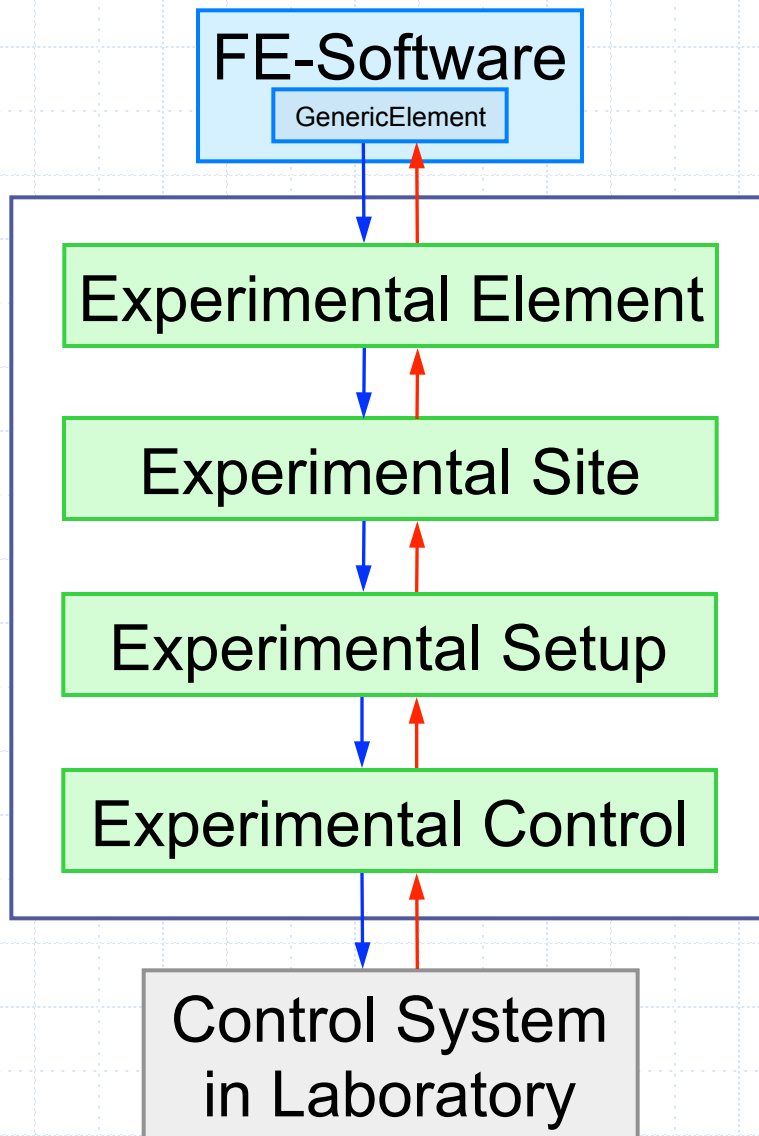
Define element as an “Experimental Element”

OpenFresco

Laboratories



# OpenFresco Components



provides all features of unmodified computational framework, including parallel and network computing

represents the part of the structure that is physically tested and provides the interface between the FE-software and the experimental software framework

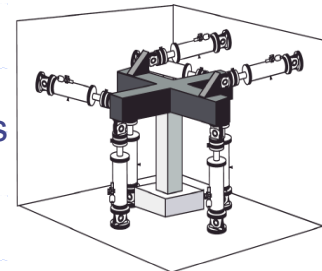
stores data and provides communication methods for distributed testing



transforms between the experimental element degrees of freedom and the actuator degrees of freedom (linear or non-linear transformations)

interfaces to the different control and data acquisition systems in the laboratories

provides control of physical actuators as well as data acquisition using physical instrumentation devices



# OpenSees Navigator

**Andreas Schellenberg**

*Project Engineer*

Rutherford & Chekene Structural and Geotechnical Engineers

**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

**OpenSees Navigator 2.1**

File Edit View Define Assign Analyze Output Design Help | MATLAB Menu

XY XZ YZ 3D

## OpenSees Navigator

Release 2.4.2, December 2007

Developed by

<b>Andreas Schellenberg</b> Dep. of Civil and Env. Eng. University of California, Berkeley	<b>Tony Yang, Ph.D.</b> Dep. of Civil and Env. Eng. University of California, Berkeley	<b>Eiji Kohama, Dr. Eng.</b> Geo. and Structural Eng. Dep. Port and Airport Research Institute
--	--	--

OpenSees Navigator is supported by funds from

The Pacific Earthquake Engineering Research (PEER) Center  
National Science Foundation (NSF)  
The George E. Brown Jr Network for Earthquake Engineering Simulation (NEES)  
Port and Airport Research Institute (PARI)

(C) Copyright 2004, The Regents of the University of California. All Rights Reserved.

Welcome to OpenSees Navigator !

X0.47 Y0.49 Z0.00

# Define geometry: new model template

The screenshot displays the OpenSees Navigator 2.1 software interface. The main window shows a 'Geometry Templates' dialog box with several structural models represented by icons. The models are:

- Stick Model
- Beam Model
- EBF Model (Eclectic Braced Frame)
- Zipper Frame
- Inverted-V Braced Frame
- Moment Frame
- Single Area Mesh

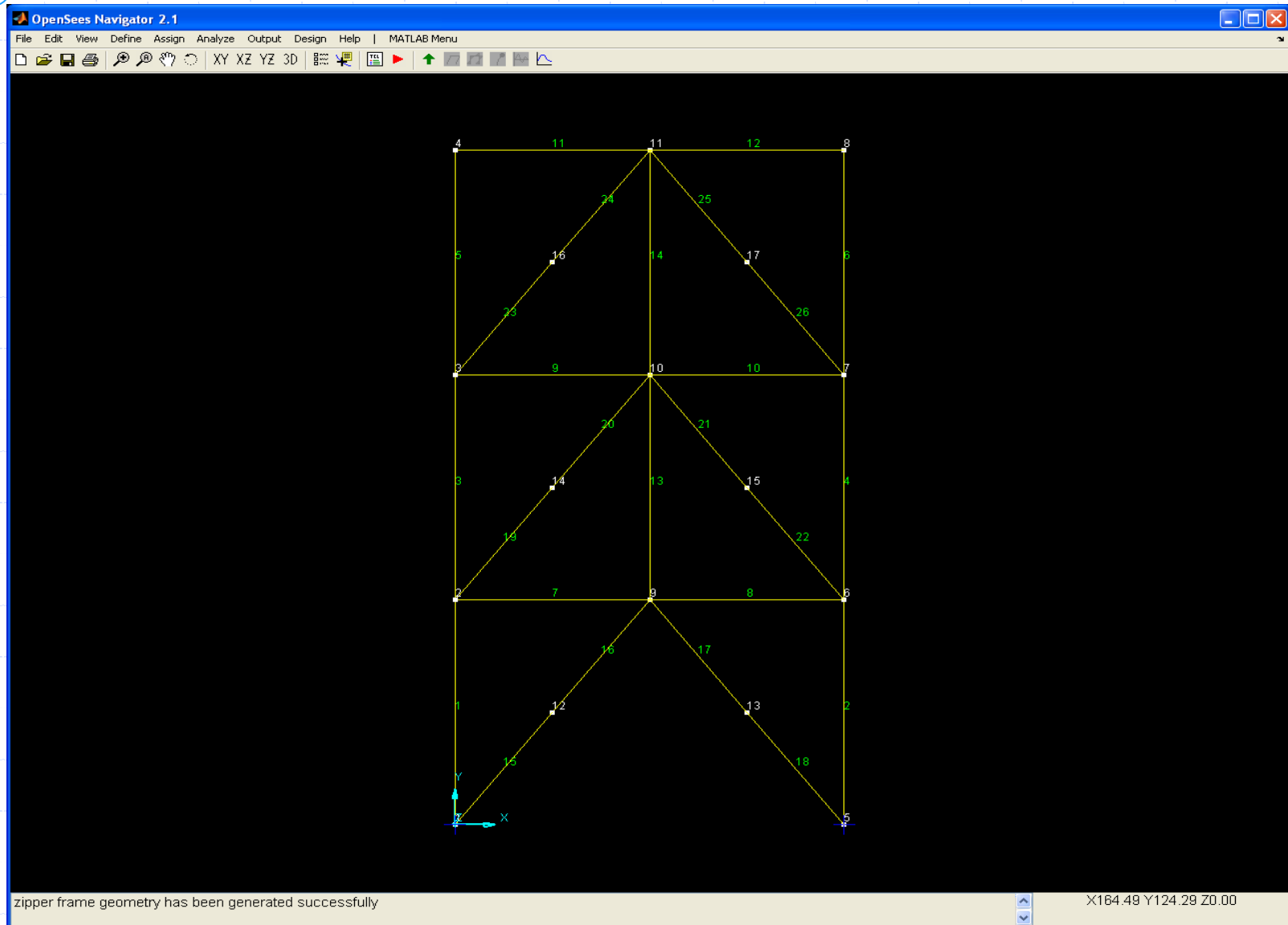
The 'File' menu is open, showing options like 'New Model from Template...', 'Open Model from File...', 'Close Model', 'Save Model', 'Save Model As...', 'Set OpenSees.exe Path...', 'Page Setup...', 'Print Setup...', 'Print Preview...', 'Print...', and 'Quit'. The status bar at the bottom indicates 'Model has been closed and workspace has been reinitialized successfully' and shows coordinates 'X0.00 Y0.00 Z0.00'.

# Define geometry: Zipper braced frame

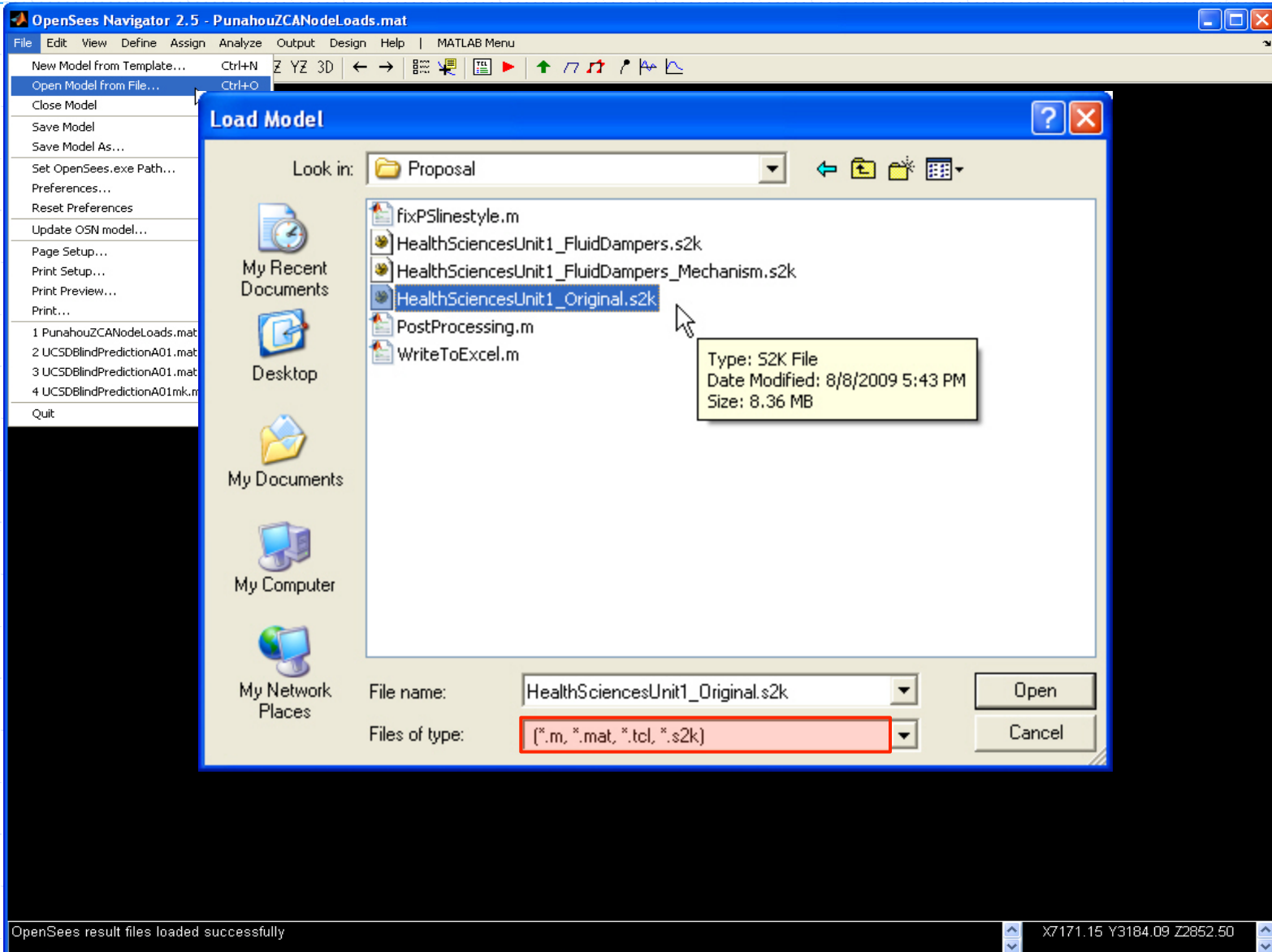
**Define Zipper Frame Geometry**

Dimension (ndm) :	2d	<input type="button" value="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) :	<input type="button" value="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	

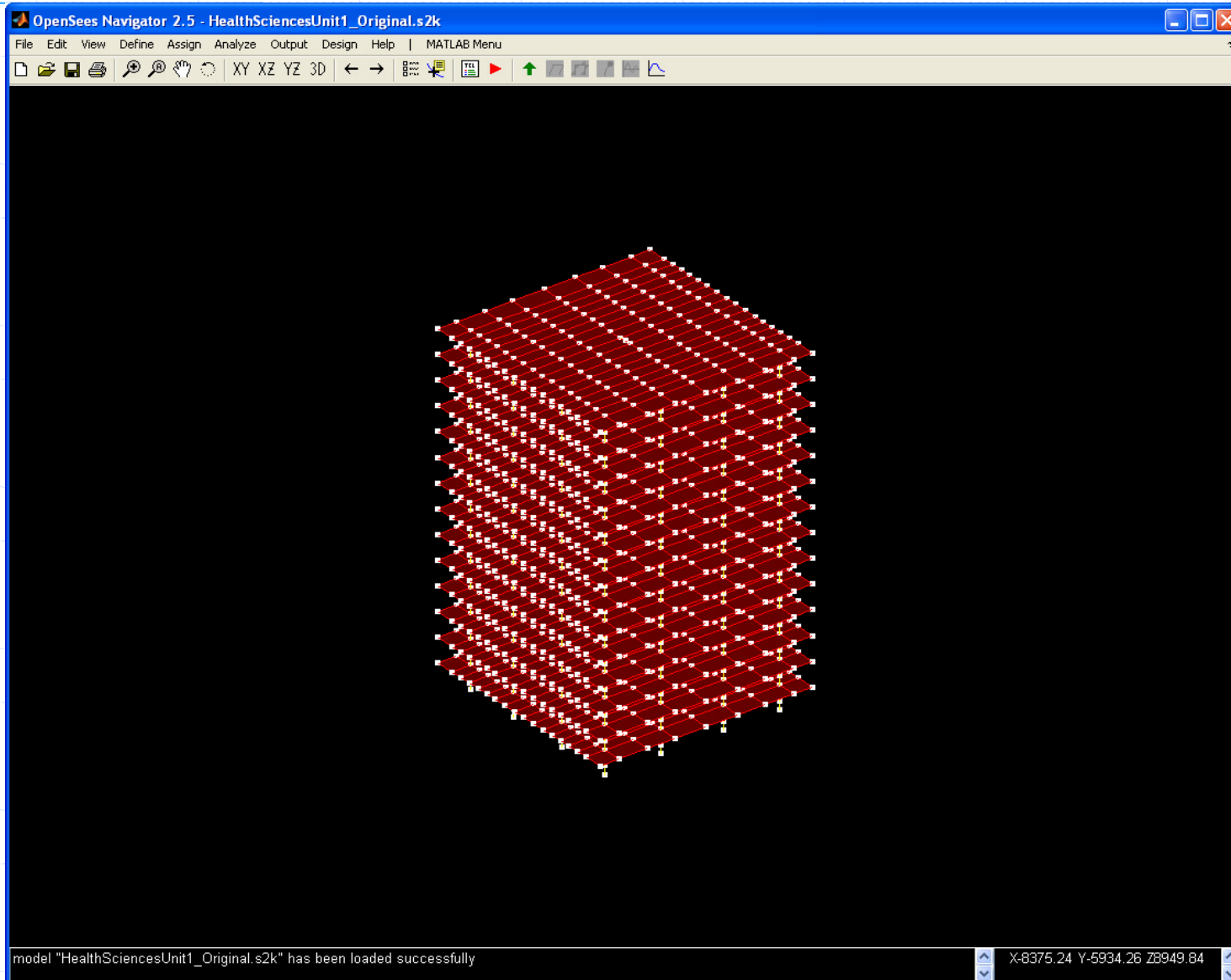
# View geometry: display



# Define geometry: import from SAP2000



# Define geometry: import from SAP2000

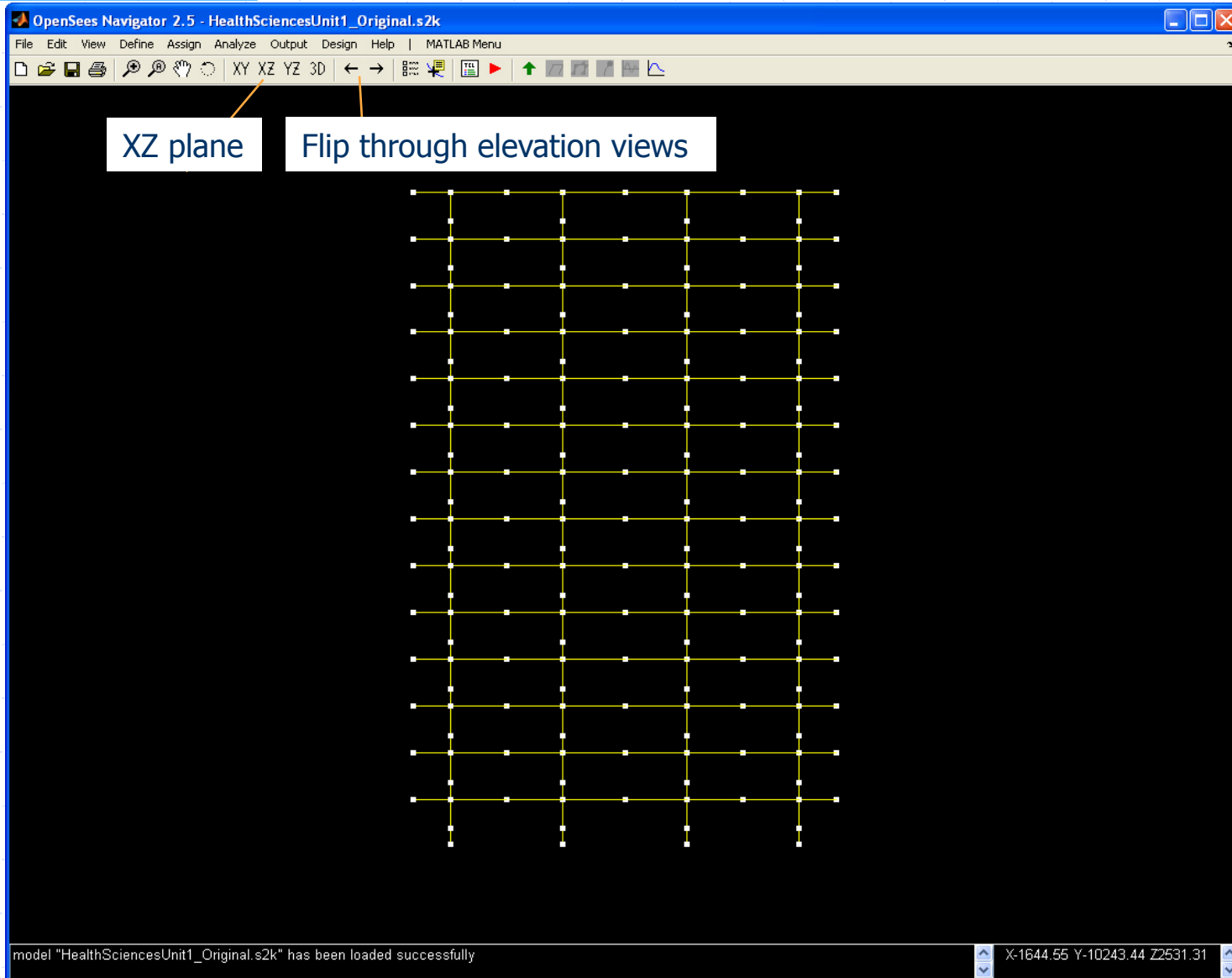




# Define geometry: import from SAP2000



# Define geometry: import from SAP2000



# View geometry: set display options

OpenSees Navigator 2.1

File Edit View Define Assign Analyze Output Design Help | MATLAB Menu

Set Display Options... (Ctrl+E) Z YZ 3D

Set Display Options... (Ctrl+E)

2D XY-Plane  
2D XZ-Plane  
2D YZ-Plane  
3D SW-View  
3D SE-View  
3D NE-View  
3D NW-View  
Set 3D View...

**Set Display Options**

Node	Element	General
<input checked="" type="checkbox"/> Tags	<input checked="" type="checkbox"/> Tags	<input checked="" type="checkbox"/> Model
<input checked="" type="checkbox"/> SP Constraints	<input type="checkbox"/> Types	<input checked="" type="checkbox"/> Global Axes
<input type="checkbox"/> MP Constraints	<input type="checkbox"/> Geo Trans	<input type="checkbox"/> Grid Lines
<input type="checkbox"/> Masses	<input type="checkbox"/> Local Axes	
<input type="checkbox"/> Loads/Displacements	<input type="checkbox"/> Zero Length	
	<input type="checkbox"/> Loads/Deformations	

Select all Clear all OK

zipper frame geometry has been generated successfully X-26.61 Y173.55 Z0.00

## Node:

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

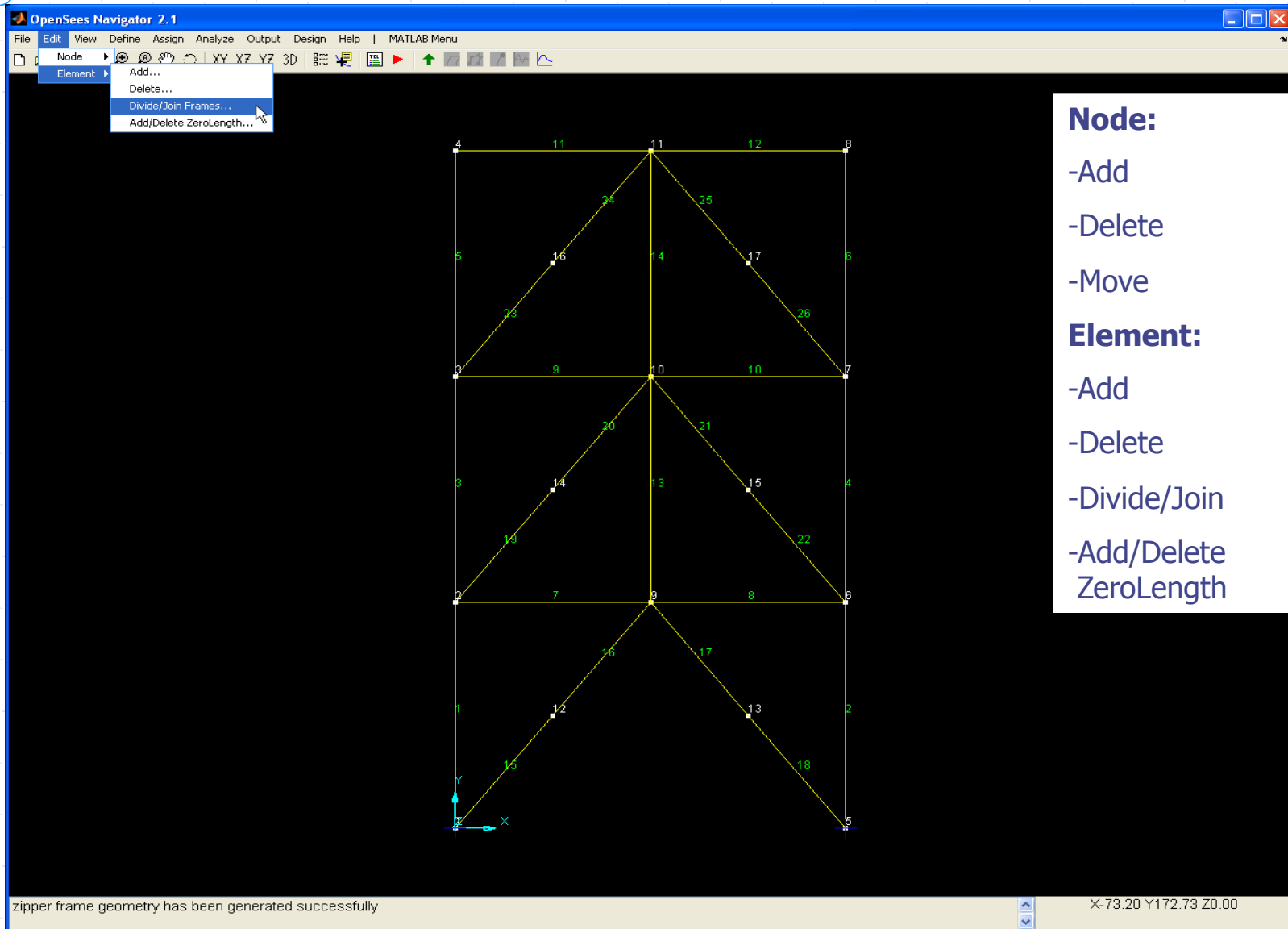
## Element:

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

## General:

- Model
- Global Axes
- Grid Lines

# Edit geometry



The screenshot displays the OpenSees Navigator 2.1 interface. The main window shows a 3D zipper frame geometry with nodes numbered 1 through 17 and elements numbered 1 through 30. A context menu is open over the 'Element' option, listing: Add..., Delete..., Divide/Join Frames..., and Add/Delete ZeroLength... The status bar at the bottom indicates 'zipper frame geometry has been generated successfully' and shows coordinates 'X-73.20 Y172.73 Z0.00'.

## 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

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 Uniaxial Material

Define Uniaxial Material

Add Material : BoucWen

Modify Material : ElasticDefault

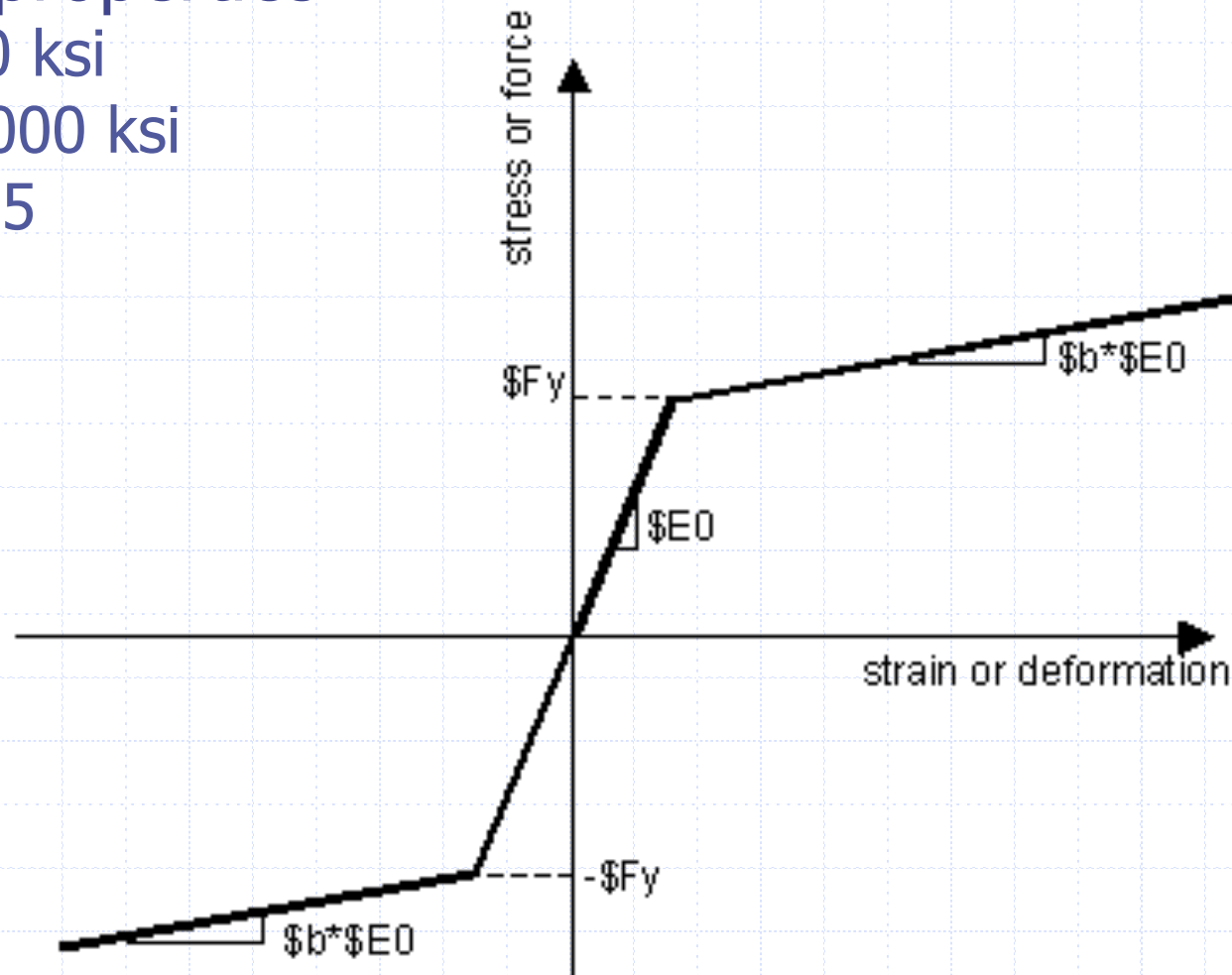
Delete Material : ElasticDefault

zipper frame geometry has been generated successfully

# Define uniaxial material: Steel01

## ◆ Material properties

- $F_y = 50$  ksi
- $E = 29000$  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

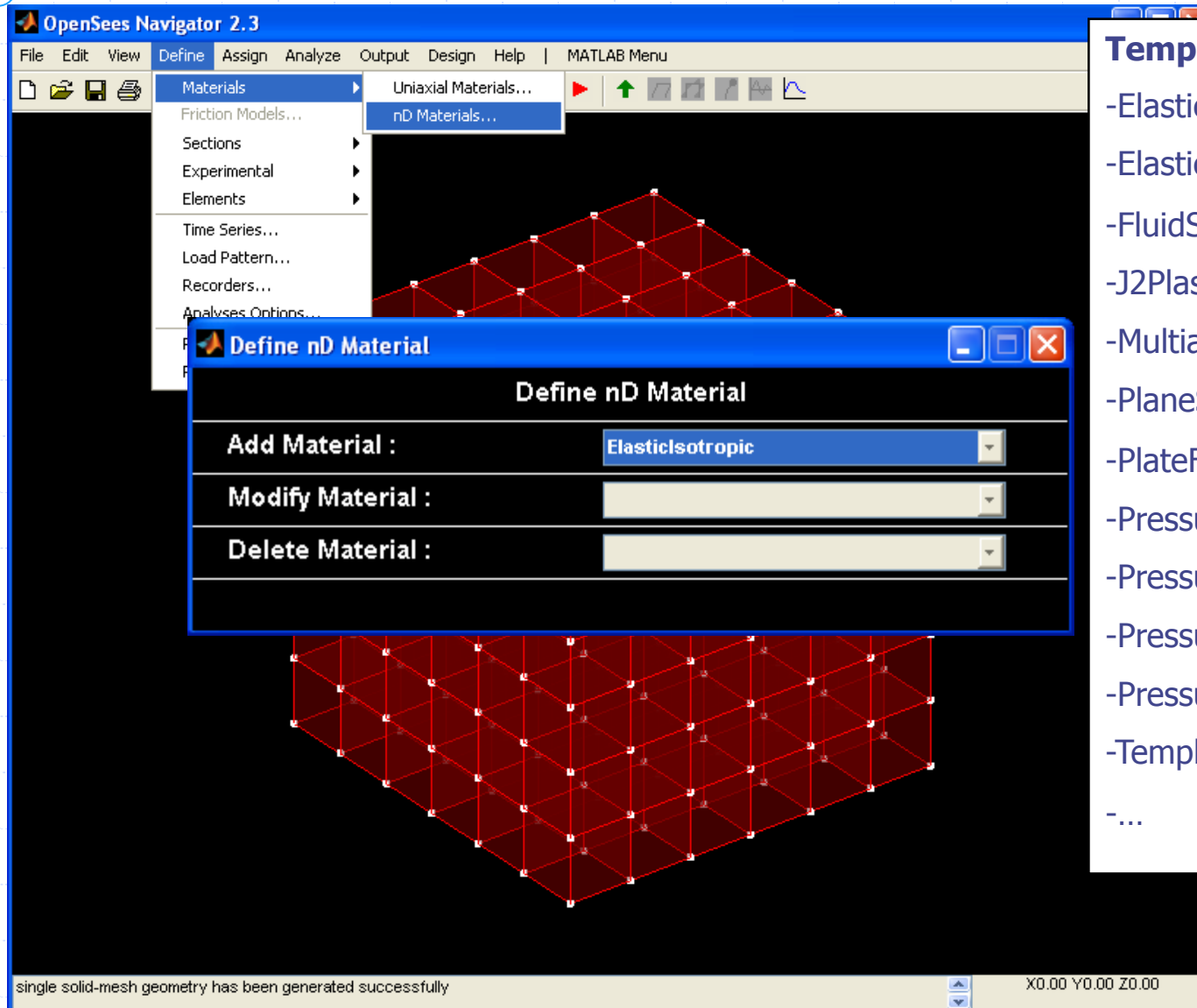
The screenshot displays the OpenSees Navigator 2.1 interface. The main window shows a zipper frame geometry with nodes 4, 11, 8, 5, 16, 14, 17, and 6. A menu is open over the 'Materials' option, with 'Uniaxial Materials...' selected. A dialog box titled 'Define Uniaxial Material' is overlaid on the geometry. The dialog box contains the following fields:

Define Uniaxial Material	
Add Material :	BoucWen
Modify/Show Material :	ElasticDefault
Delete Material :	A50

The status bar at the bottom of the window displays the text: "zipper frame geometry has been generated successfully" and "X-56.17 Y172.73 Z0.00".



# Define material: nD materials



## Templates:

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

# Define section: line sections

The screenshot shows the OpenSees Navigator 2.1 interface. The 'Define' menu is open, with 'Line Sections...' selected. A 'Define Line Section' dialog box is in the foreground, showing three dropdown menus: 'Add Section' (Aggregator), 'Modify Section' (ElasticDefault), and 'Delete Section' (ElasticDefault). The background shows a zipper frame geometry with nodes 4, 11, 8, 5, 16, 14, 17, 26, 23, and 25. A status bar at the bottom indicates 'zipper frame geometry has been generated successfully' and 'X-50.42 Y171.91 Z0.00'.

**Define Line Section**

Add Section : Aggregator

Modify Section : ElasticDefault

Delete Section : ElasticDefault

zipper frame geometry has been generated successfully

X-50.42 Y171.91 Z0.00

## Templates:

- Aggregator
- Elastic
- Fiber
- Uniaxial

# Define line section: elastic section

**Define Elastic Section**

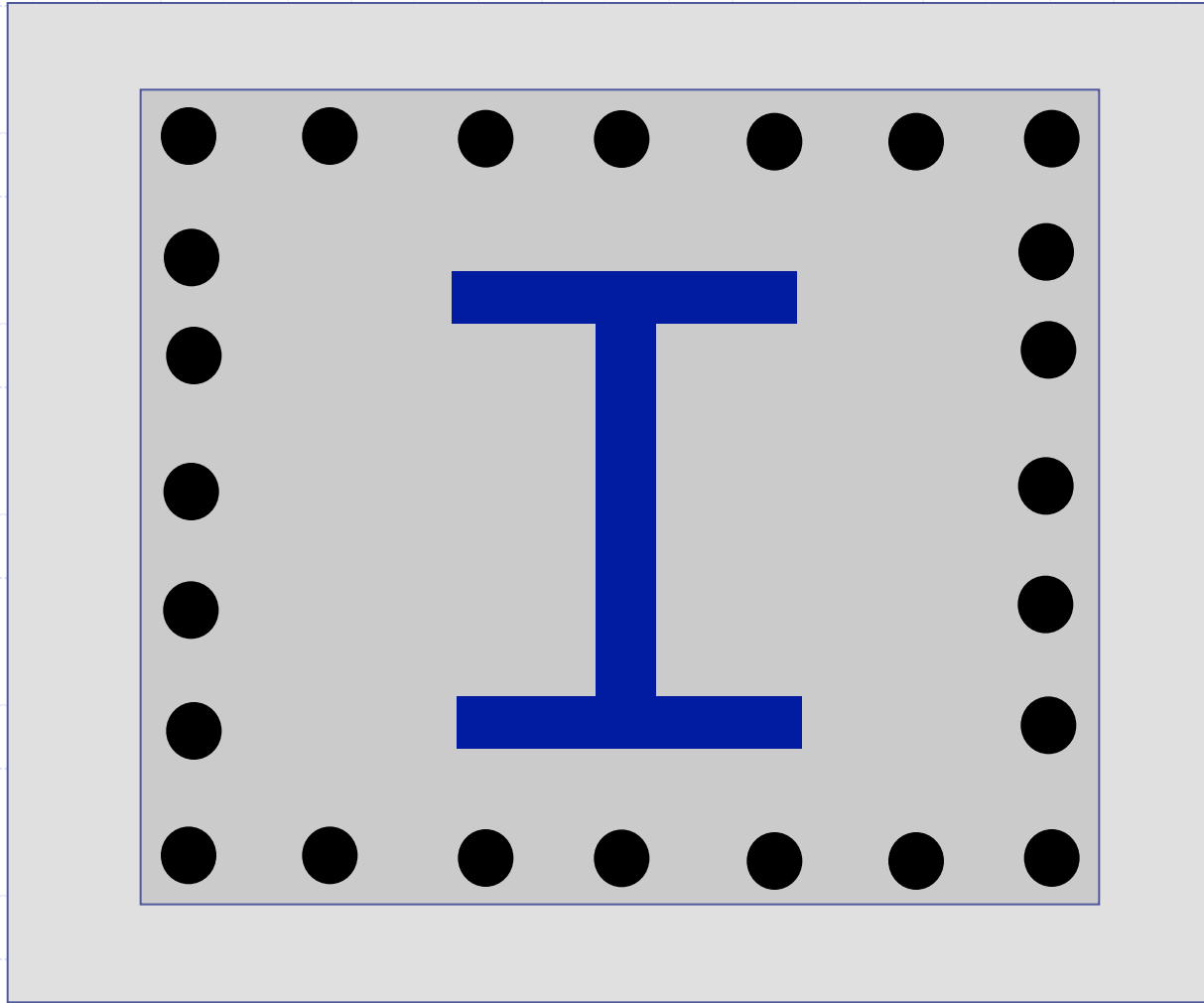
Section Name :	Section01	Add
Modulus of Elasticity (E) :	29000	Database
Cross-Sectional Area (A) :	20	
Moment of Inertia (Iz) :	2000	

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 :** 1stStoryColumn

**Add Fiber :** Fiber

**Modify Fiber :**

**Delete Fiber :**

**Add Patch :** Quadrilateral

**Modify Patch :**

**Delete Patch :**

**Add Layer :** Straight

**Modify Layer :**

**Delete Layer :**

# Define line section: quadrilateral patch

**Define Quadrilateral Patch**

**Define Quadrilateral Patch**

Patch Name :	CoreConcrete	Add
Material Type :	ConfinedConcrete	
Lower Left Corner (yI,zI) :	A50	
Lower Right Corner (yJ,zJ) :	ConfinedConcrete	
Upper Right Corner (yK,zK) :	UnconfinedConcrete	
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

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 section

The screenshot displays the OpenSees Navigator 2.1 interface. The 'Define' menu is open, showing 'Area Sections...' selected. A 'Define Area Section' dialog box is overlaid on the main window, which contains a structural model of a truss. The dialog box has three dropdown menus: 'Add Section' (set to 'Bidirectional'), 'Modify Section', and 'Delete Section'. The structural model shows nodes 4, 11, 8, 5, 16, 14, 17, 23, 24, 25, 26, and 8. A coordinate system (X, Y) is shown at the bottom left. The status bar at the bottom indicates 'section(2) with name "1stFloorBeam" has been defined/modified successfully' and 'X:47.75 Y:173.55 Z:0.00'.

**Define Area Section**

Add Section : Bidirectional

Modify Section :

Delete Section :

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

X:47.75 Y:173.55 Z:0.00

## Templates:

- Bidirectional
- ElasticMembranePlate
- PlateFiber



# Define element: line element

The screenshot shows the OpenSees Navigator 2.1 interface. The 'Define' menu is open, with 'Line Elements...' selected. A 'Define Line Element' dialog box is displayed in the foreground, showing the following settings:

Field	Value
Add Element :	CorotationalTruss
Modify Element :	ElasticDefault
Delete Element :	ElasticDefault

The background shows a truss structure with nodes 4, 5, 8, 11, 16, 17, and 26. A red triangle representing element 15 is highlighted in the foreground, with nodes 1, 2, and 5. A status bar at the bottom left indicates 'element [15] has been added successfully'.

## Templates:

- Actuator
- Adapter
- BeamColumnJoint
- DispBeamColumn
- ElasticBeamColumn
- ElastomericBearing
- FlatSliderBearing
- ForceBeamColumn
- GenericClient
- HingeBeamColumn
- Joint2D
- SingleFPBearing
- Truss
- TwoNodeLink
- ZeroLength
- ...

# Define line element: ElasticBeamColumn

Define ElasticBeamColumn Element

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

Select Section from Database

Database :	AISC	Select
Section Name :	W10X45	
Direction :	strong	

# Define line element: ForceBeamColumn

**Define ForceBeamColumn Element**

Element Name : 1stStoryColumn

Number Intergration Points (NIP) : 5

Section Type : 1stStoryColumn

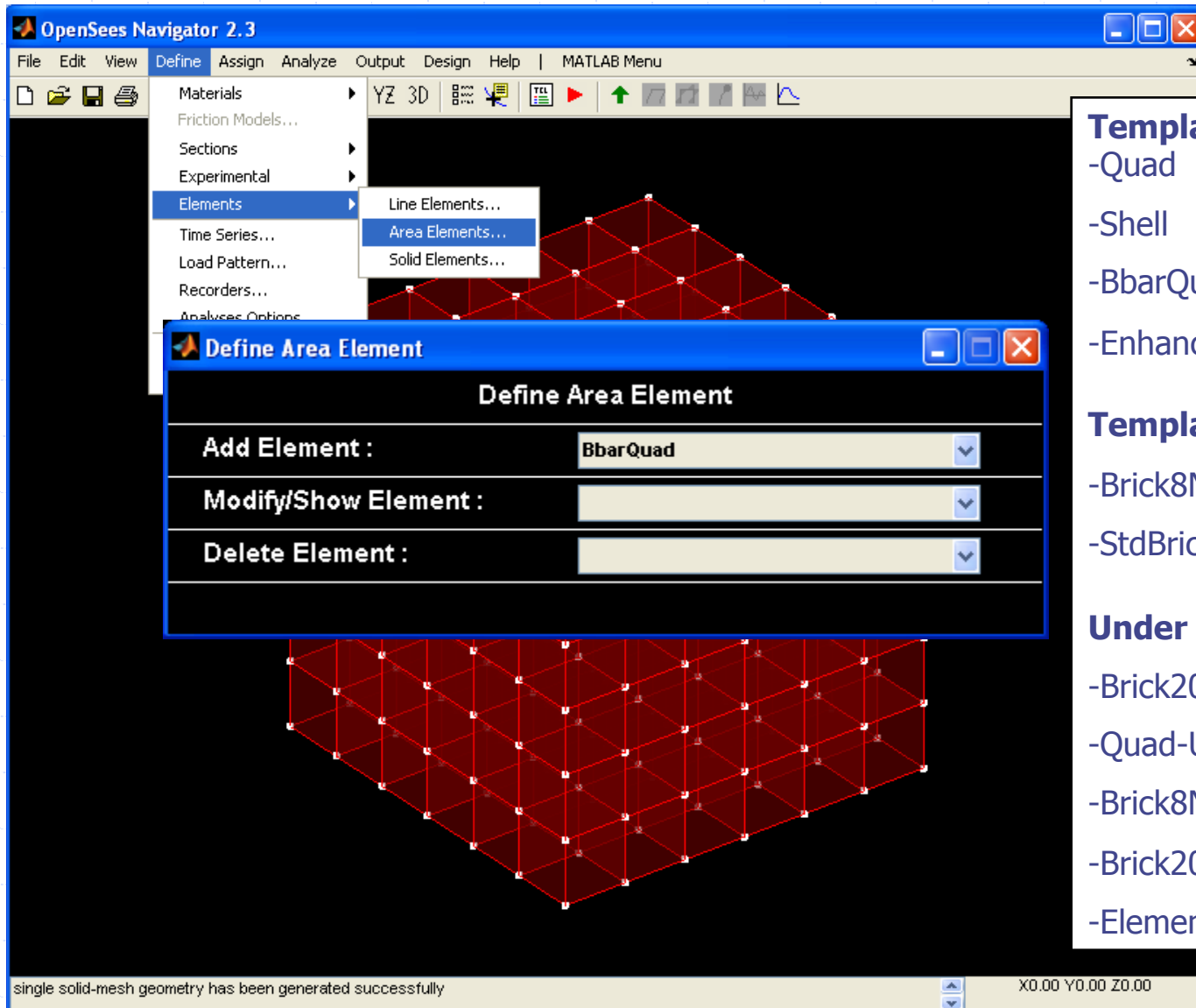
*Optional Arguments :*

Mass Density (massDens) : 0.

Maximum Iterations (maxlters) : 10

Tolerance (tol) : 1E-8

# Define element: area and solid elements



## Templates (area):

- Quad
- Shell
- BbarQuad
- EnhancedQuad

## Templates (solid):

- Brick8N
- StdBrick

## Under development:

- Brick20N
- Quad-UP
- Brick8Nu-p-U
- Brick20Nu-p-U
- ElementsLargeDeformation

# Define TimeSeries:

The screenshot shows the OpenSees Navigator 2.1 interface. The 'Define' menu is open, highlighting 'Time Series...'. A 'Define TimeSeries' dialog box is displayed in the foreground, showing the following options:

Operation	Selected Template
Add TimeSeries :	Constant
Modify/Show TimeSeries :	LinearDefault
Delete TimeSeries :	LinearDefault
Plot TimeSeries :	LinearDefault

The background shows a 3D model of a truss structure. A status bar at the bottom indicates 'element [15] has been added successfully' and 'X:65.20 Y:173.55 Z:0.00'.

## Templates:

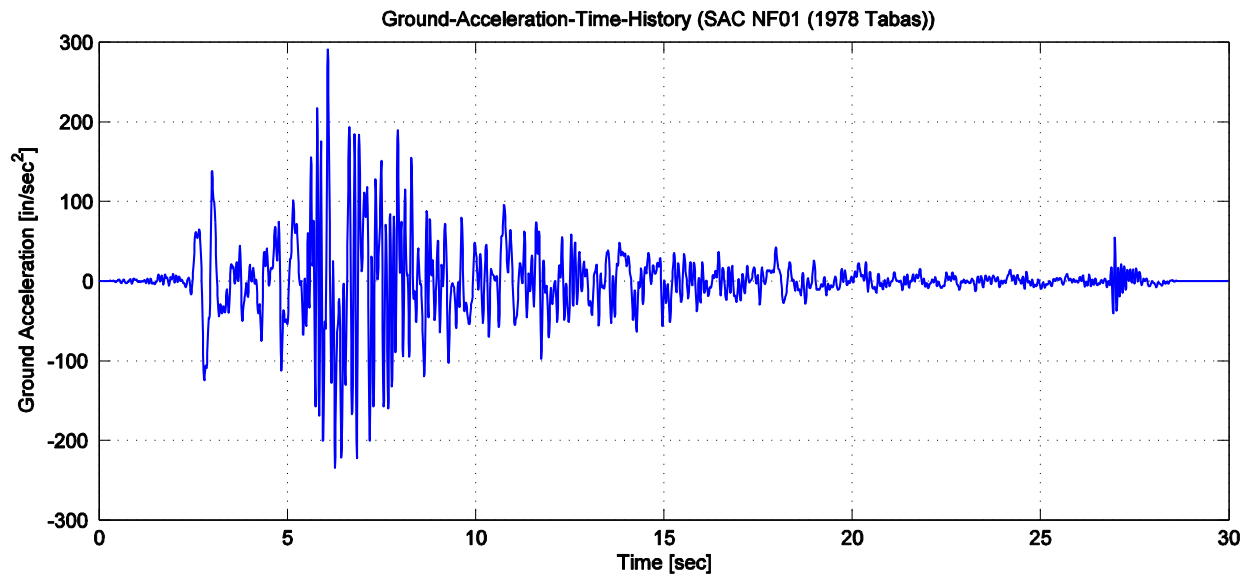
- Constant
- Interpolated GM
- Linear
- PathFile
- PathFileFiltered
- PathValue
- Plain GM
- Pulse
- Rectangular
- Sine
- Triangle

# Define TimeSeries: PathFile

**Define PathFile Time Series**

**Define PathFile Time Series**

TimeSeries Name :	SACNF01	Add
Time Interval (dt) :	0.01	
Time File Name (fileTime) :	<input type="checkbox"/>	Browse
Value File Name (filePath) :	D:\NEES\GroundMotions\SACNF01.txt	Browse
Load Factor (cFactor) :	386.1	



# Define LoadPattern:

The screenshot shows the OpenSees Navigator 2.0 interface. The main window displays a truss structure with a red triangular load pattern applied. A 'Define LoadPattern' dialog box is open, showing the following settings:

Define LoadPattern	
Add LoadPattern :	Plain
Modify LoadPattern :	PlainDefault
Delete LoadPattern :	PlainDefault

The 'Define' menu is open, showing options: Materials, Sections, Experimental, Elements, Time Series..., Load Pattern..., Recorders..., Analyses Options..., Response Functions..., and Response Spectra... The status bar at the bottom indicates: 'time-series(2) with name "SACNF01" has been defined/modified successfully' and 'X:41.59 Y:157.54 Z:0.00'.

- Templates:**
- Plain
  - UniformExcitation
  - MultipleSupport

# Define LoadPattern: UniformExcitation

Define UniformExcitation Load Pattern

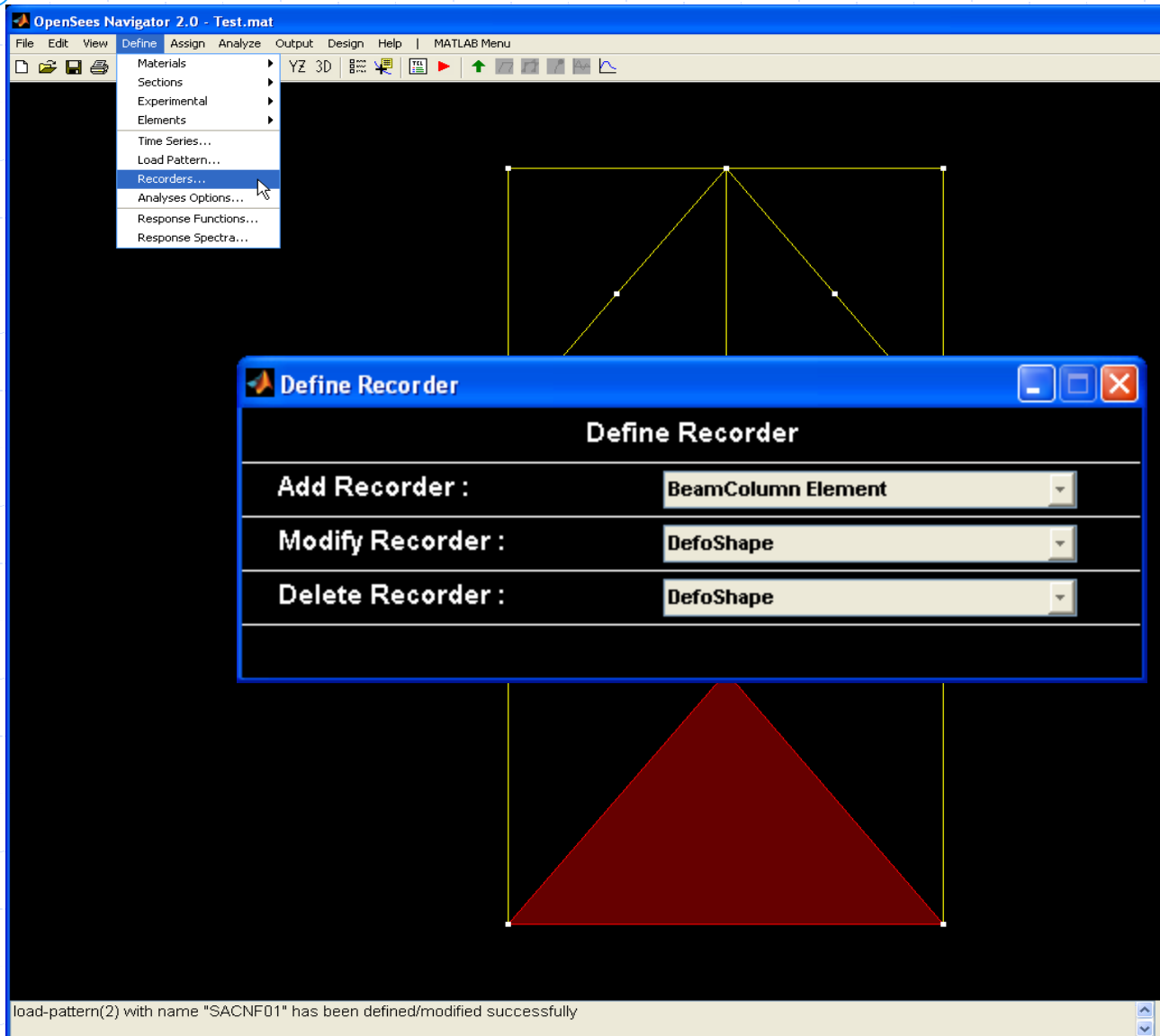
LoadPattern Name : SACNF01

TimeSeries Type : SACNF01

Direction of Excitation (dir) : 1



# 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 Node Recorder**

Recorder Name : DefoShape **Add**

Node Number(s) : all  Envelope

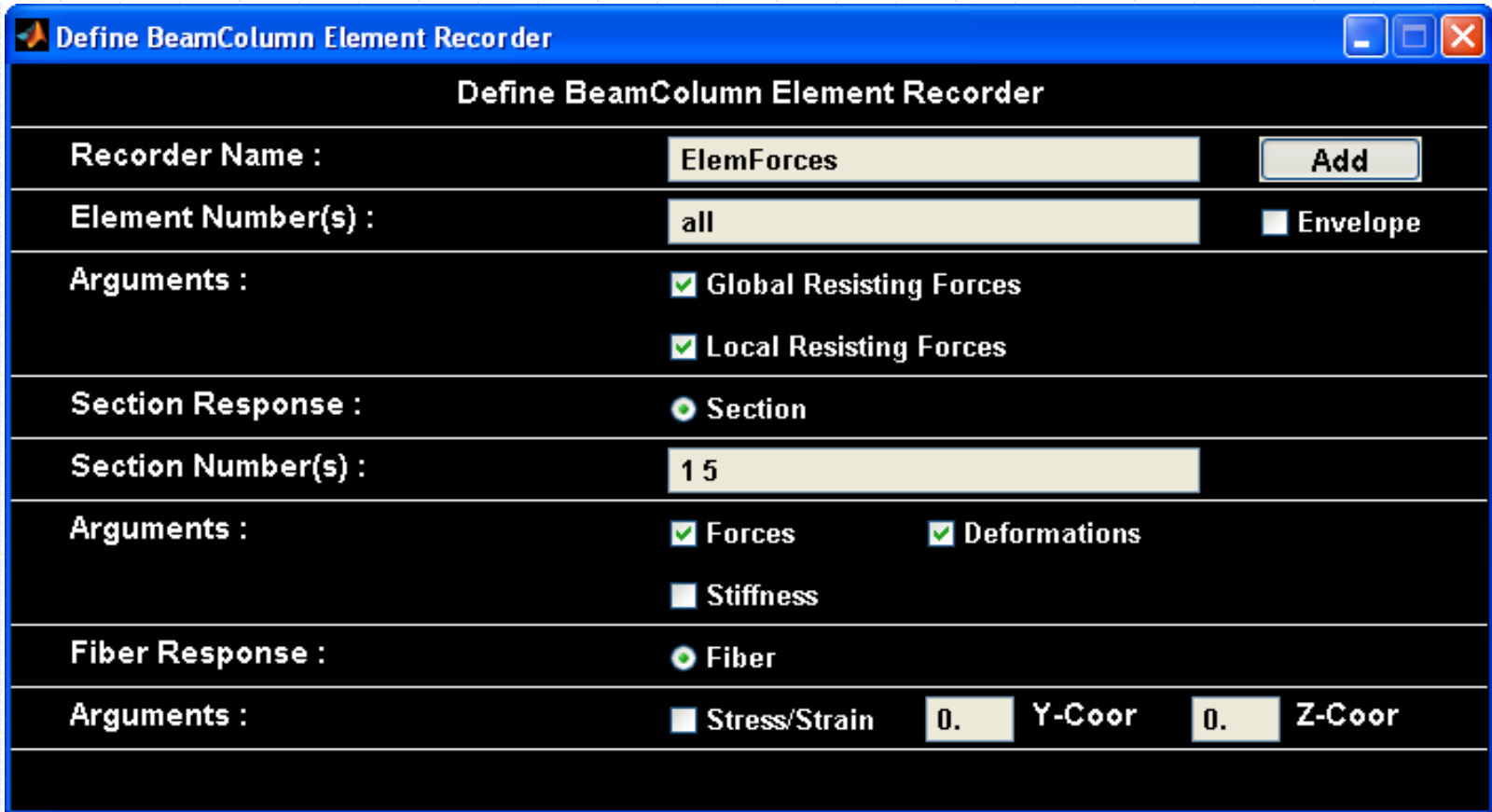
Deformations :

- 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 recorder: 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) :** 15

**Arguments :**  Forces  Deformations  
 Stiffness

**Fiber Response :**  Fiber

**Arguments :**  Stress/Strain 0. Y-Coor 0. Z-Coor

# Define analysis options

OpenSees Navigator 2.0 - Test.mat

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...

**Defaults:**

- StaticDefault
- TransientDefault
- EigenDefault

**Define Analysis Options**

Add Analysis Options :

Modify Analysis Options :

Delete Analysis Options :

load-pattern(2) with name "SACNF01" has been defined/modified successfully

X-79.16 Y173.55 Z0.00

# Define analysis options: new analysis

**Define New Analysis Options**

Analysis Optn Name : AnalysisOptn01

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

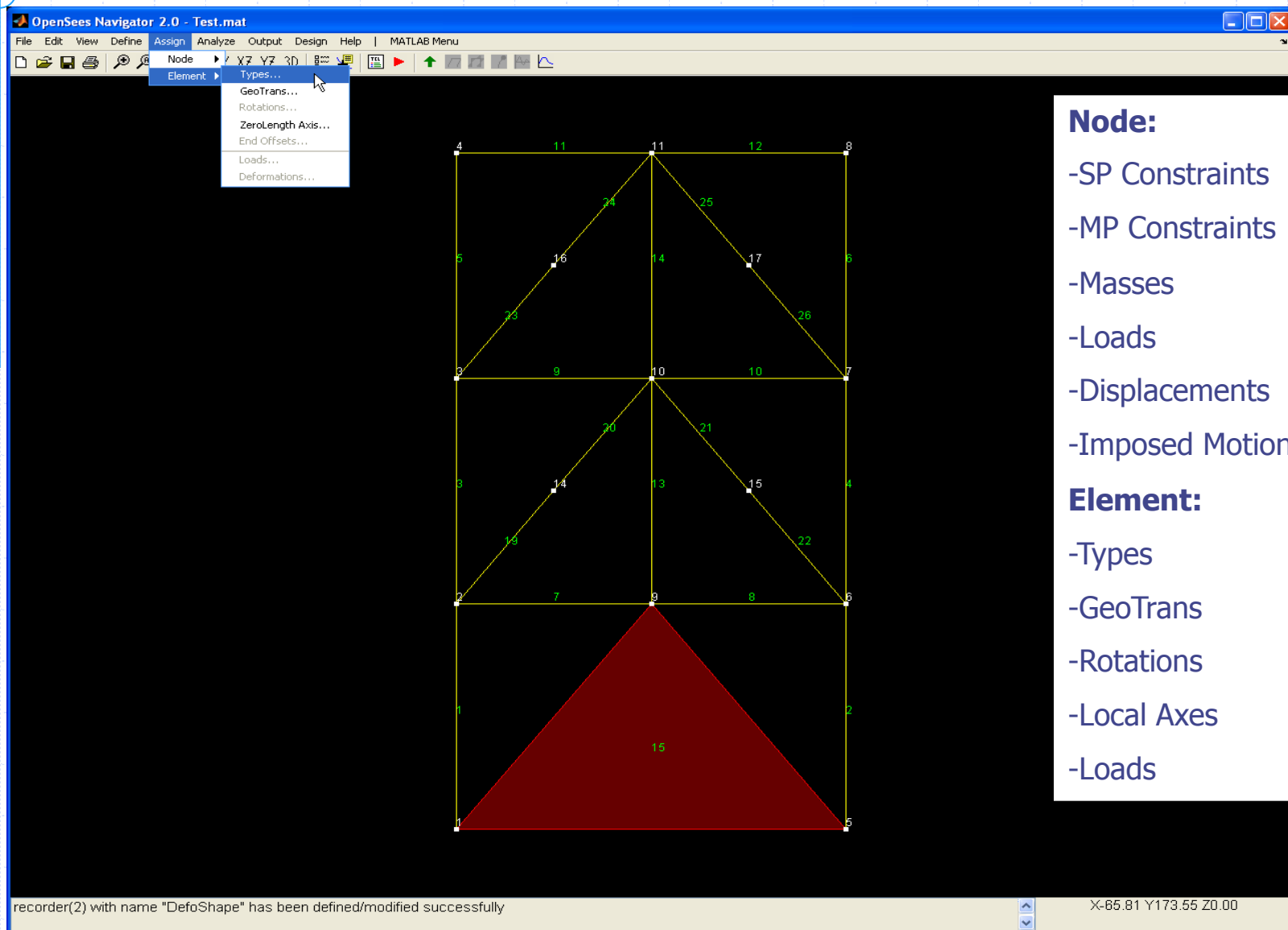
## **Integrator Type:**

For example use AlphaOS Method for Hybrid Simulation

## **Solution Algorithm:**

The AlphaOS Method requires a Linear solution algorithm

# Assign menu



The screenshot displays the OpenSees Navigator 2.0 software interface. The main window shows a 2D mesh structure with nodes numbered 1 through 17 and elements numbered 1 through 30. A red triangle is highlighted at the bottom of the mesh. The 'Assign' menu is open, showing options: Types..., GeoTrans..., Rotations..., ZeroLength Axis..., End Offsets..., Loads..., and Deformations... The status bar at the bottom indicates: recorder(2) with name "DefoShape" has been defined/modified successfully. The coordinates X=65.81 Y=173.55 Z=0.00 are shown in the bottom right corner.

## Node:

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

## Element:

- Types
- GeoTrans
- Rotations
- Local Axes
- Loads

# 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 :

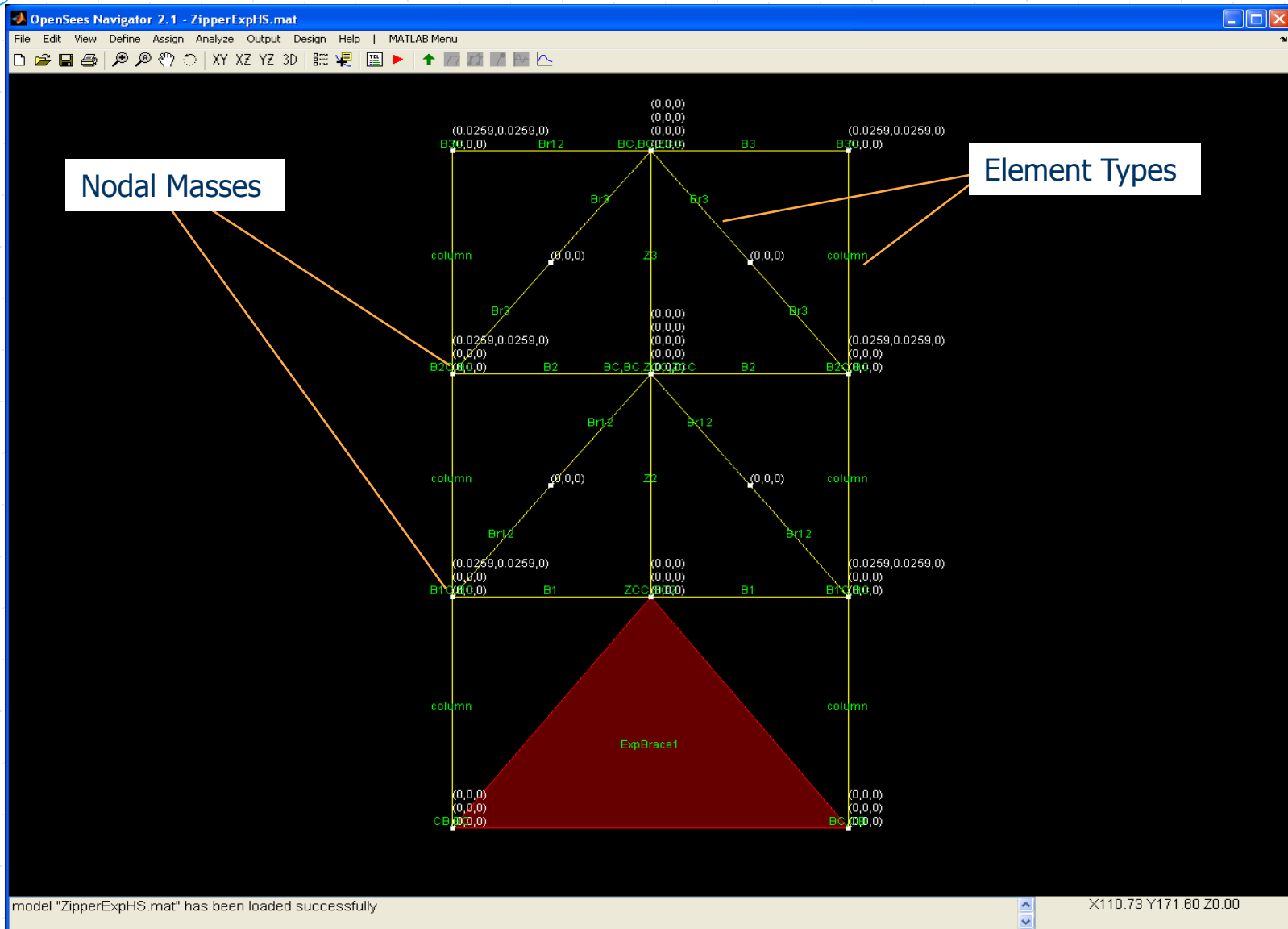
**Assign Element Geometric Transformations**

Assign Element Geometric Transformations :

Element Number(s) :

Geometric Transformation :

# Display assigned properties





# Define analysis case

The screenshot displays the OpenSees Navigator 2.1 interface. The main window shows a structural model with a red triangle and a yellow trapezoid. A 'Define Analysis Case' dialog box is open, showing the following options:

- Add Analysis Case :
- Modify Analysis Case :
- Delete Analysis Case :

To the right of the dialog box, a white box lists the defaults:

**Defaults:**

- StaticDefaultCase
- EigenDefaultCase

The status bar at the bottom of the window indicates: "model 'ZipperExpHS.mat' has been loaded successfully" and "X-48.36 Y171.60 Z0.00".

# Define analysis case: new analysis case

The image shows a software interface for defining a new analysis case. The main dialog box is titled "Define New Analysis Case" and contains the following fields and buttons:

- Analysis Case Name :** EQ1 **Add**
- Number of Load Steps (numIncr) :** 20480
- Time Step Increment (dt) :** 0.00390625

A sub-dialog box titled "Define New Damping Parameters" is open over the main dialog. It contains the following fields and buttons:

- Damping Parameter Set Name :** DampingParam01 **Add**
- Region Defined by :** Nodes
- Node or Element Number(s) :** all
- Mass Prop. Damping ( $\alpha_M$ ) :** 0. **Calculate**
- Kcurr Prop. Damping ( $\beta_K$ ) :** 0.
- Kinit Prop. Damping ( $\beta_{Kinit}$ ) :** 0.
- Kcom Prop. Damping ( $\beta_{Kcom}$ ) :** 0.

# 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]

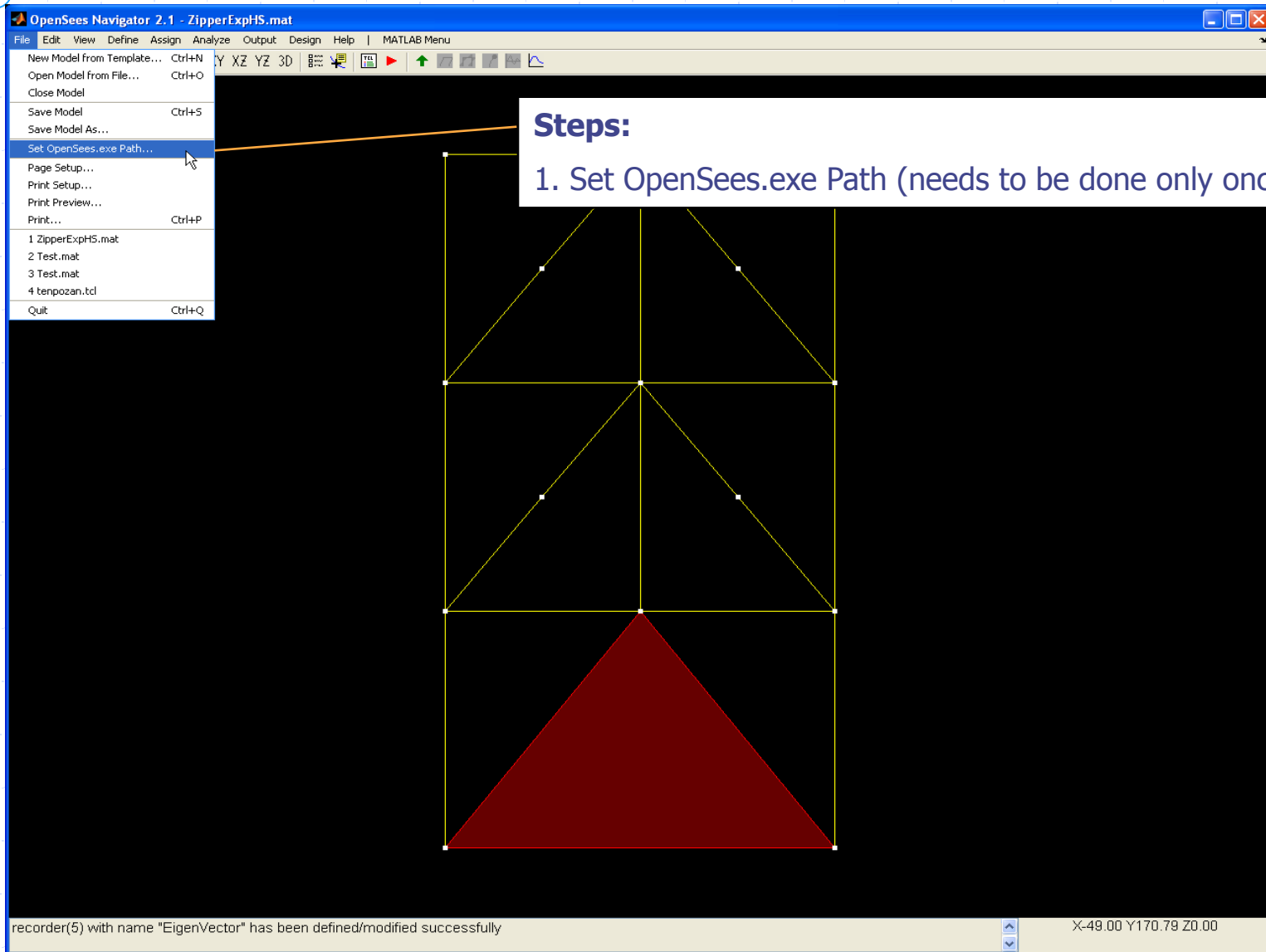
while ($ok == 0 && $tCurrent < $tFinal) {
  if (fmod($tCurrent,1) < 1.0E-16) {
    puts "$i $tCurrent"
  }
  set ok [analyze 1 $dt]

  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



The screenshot shows the OpenSees Navigator 2.1 interface. The 'File' menu is open, and the 'Set OpenSees.exe Path...' option is highlighted. A callout box with the title 'Steps:' contains the following instruction:

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

The main window displays a 3D model of a structure, which is a square frame with a red triangle at the bottom. The status bar at the bottom of the window shows the text: 'recorder(5) with name "EigenVector" has been defined/modified successfully' and 'X-49.00 Y170.79 Z0.00'.

# Run OpenSees: write TCL files

The screenshot displays the OpenSees Navigator 2.1 software interface. The main window title is "OpenSees Navigator 2.1 - ZipperExpHS.mat". The menu bar includes "File", "Edit", "View", "Define", "Assign", "Analyze", "Output", "Design", "Help", and "MATLAB Menu". The toolbar contains various icons for file operations and analysis. A white box with a blue border is overlaid on the interface, containing the following steps:

**Steps:**

2. Write OpenSees Input Files (writes TCL files)
3. Run OpenSees

The "Running OpenSees" dialog box is open, showing a text area with the message "running OpenSees analysis, please wait ....." and an "OK" button. At the bottom of the main window, a status bar displays the text "recorder(5) with name 'EigenVector' has been defined/modified successfully" and the coordinates "X-45.55 Y171.60 Z0.00".

# Post processing: load results

The screenshot displays the OpenSees Navigator 2.1 software interface. The main window shows a structural model of a triangular truss structure. A dialog box titled "Load OpenSees Results" is open, allowing the user to select analysis cases. The dialog box contains a list of cases: "EigenDefaultCase" and "SACNF01Case01". A "Load" button is visible next to the list. A callout box with the text "First: Load OpenSees Results" points to the dialog box. The status bar at the bottom of the main window shows the message: "recorder(5) with name 'Eigenvector' has been defined/modified successfully" and the coordinates "X-45.55 Y171.60 Z0.00".

**First:**  
Load OpenSees Results

Load OpenSees Results

Select Analysis Case(s):

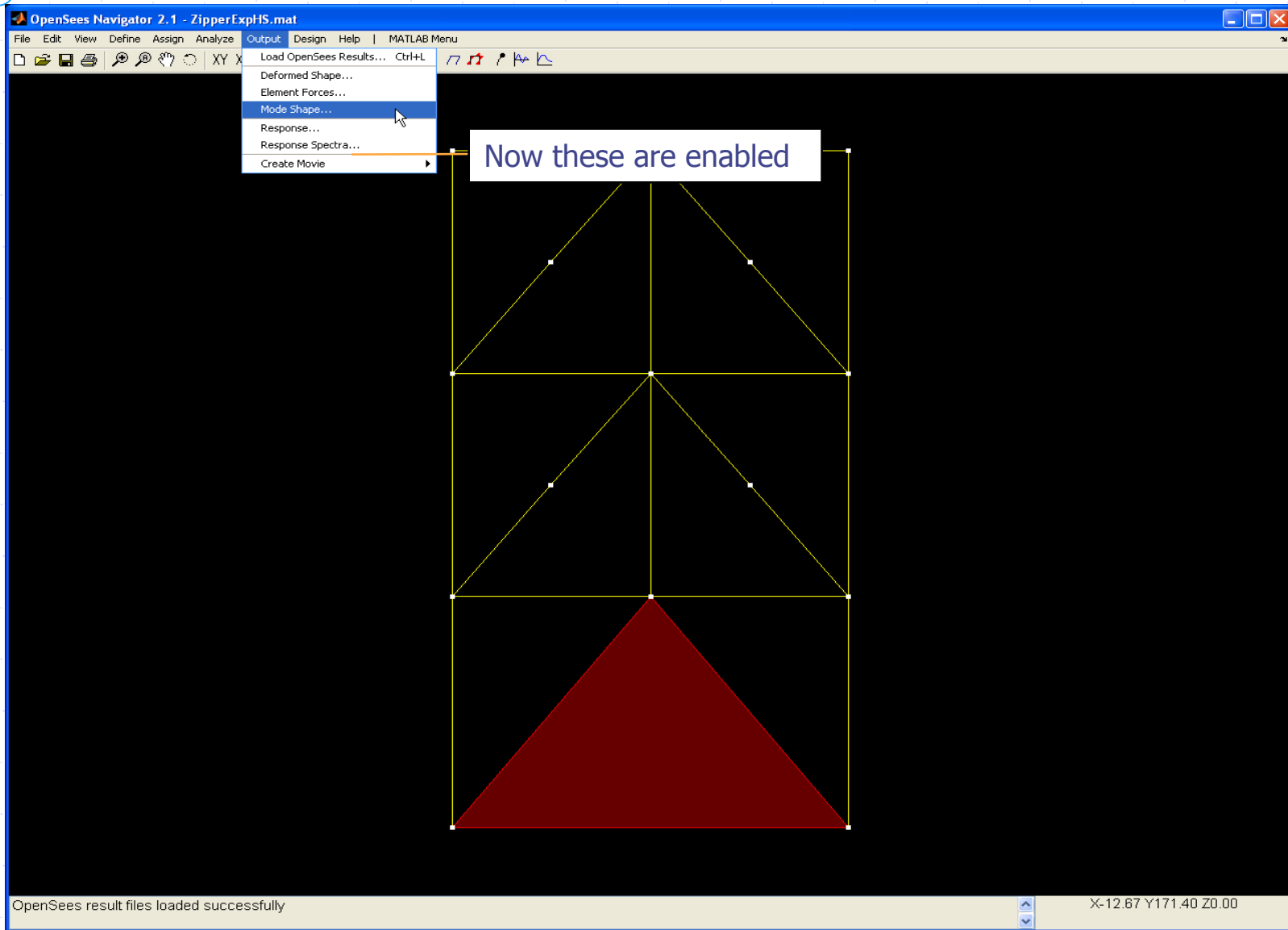
- EigenDefaultCase
- SACNF01Case01

Load

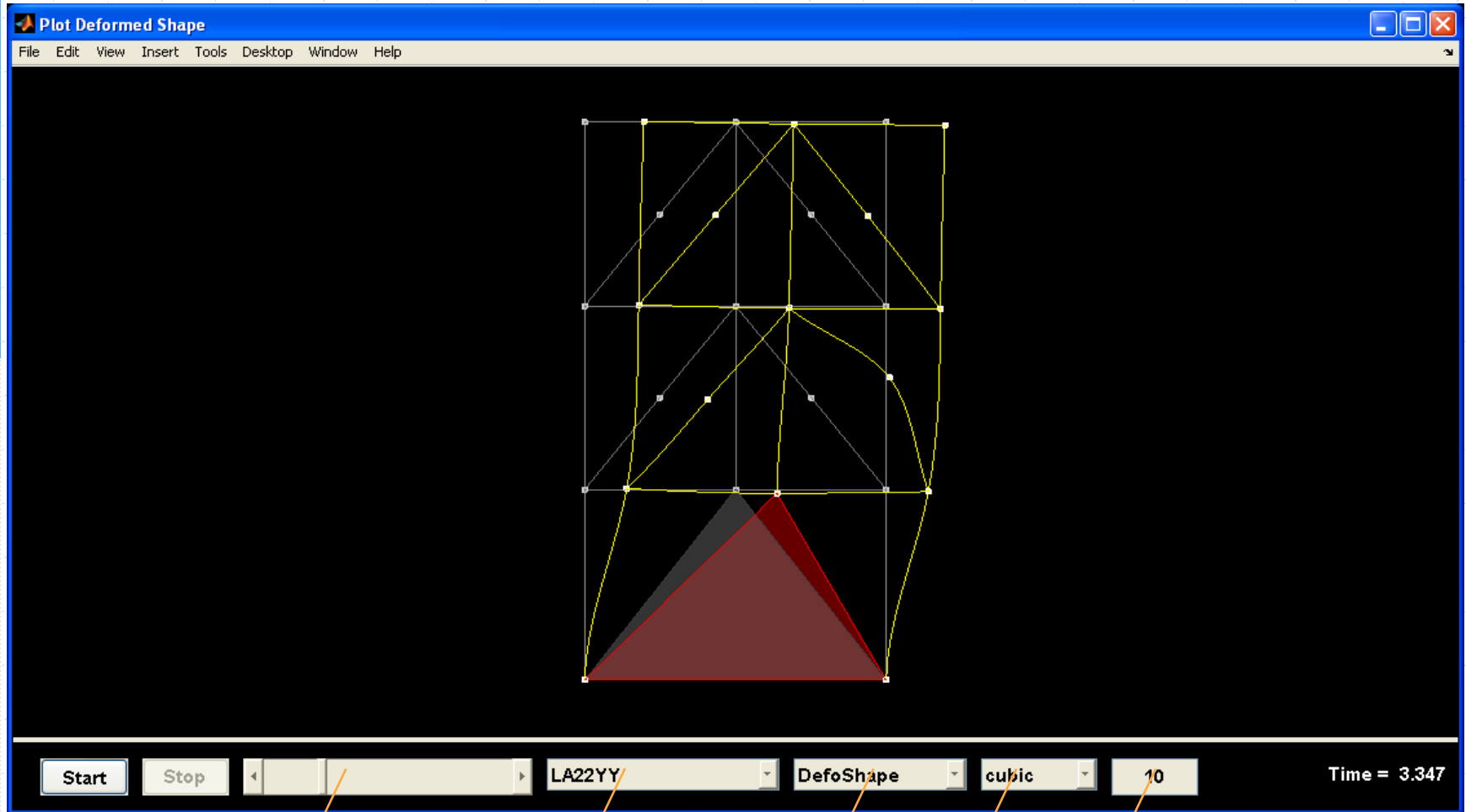
recorder(5) with name "Eigenvector" has been defined/modified successfully

X-45.55 Y171.60 Z0.00

# Post processing: output



# Post processing: plot deformed shape



Time Step

AnalysisCase

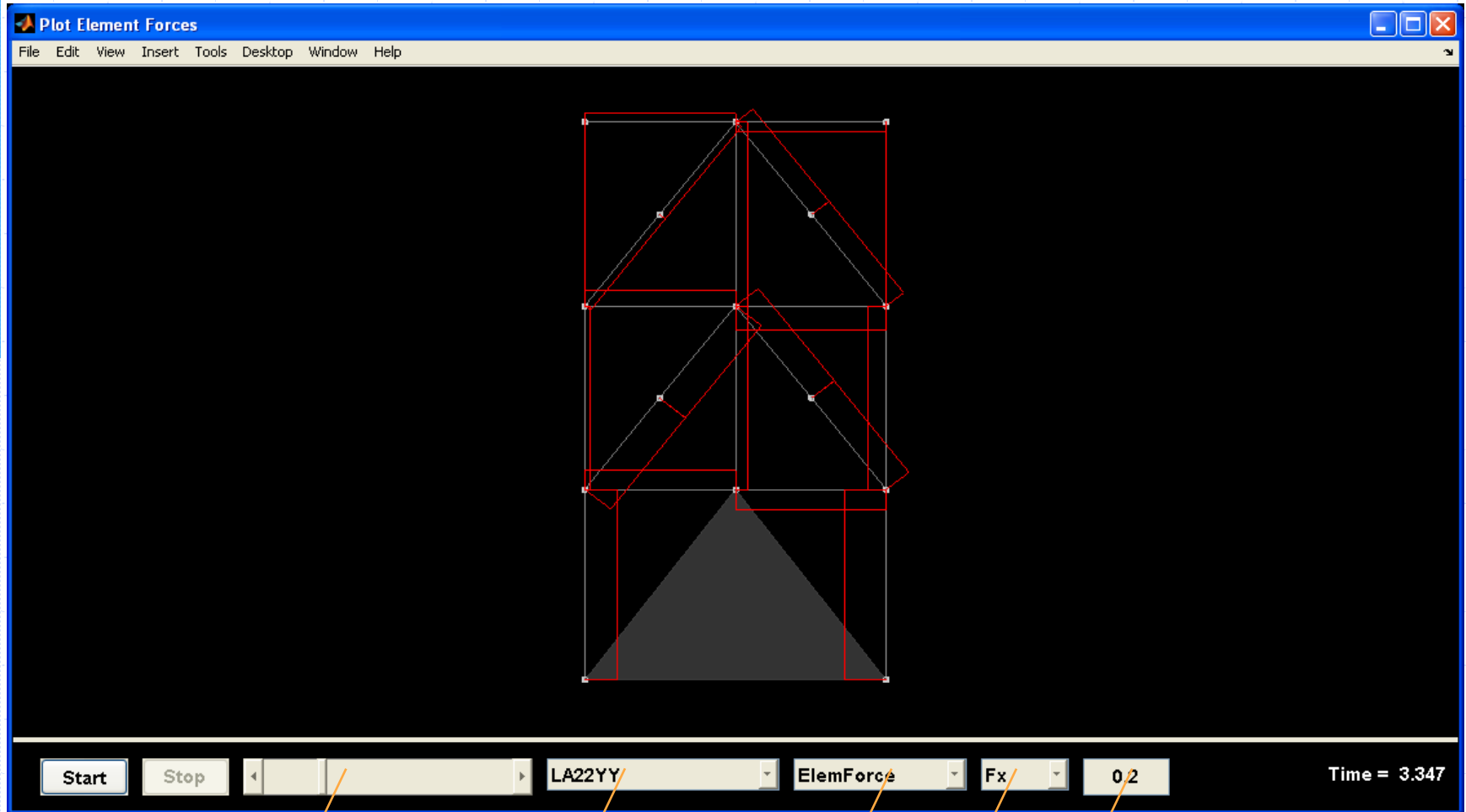
Recorder

Order

Magnification



# Post processing: plot element forces



Time Step

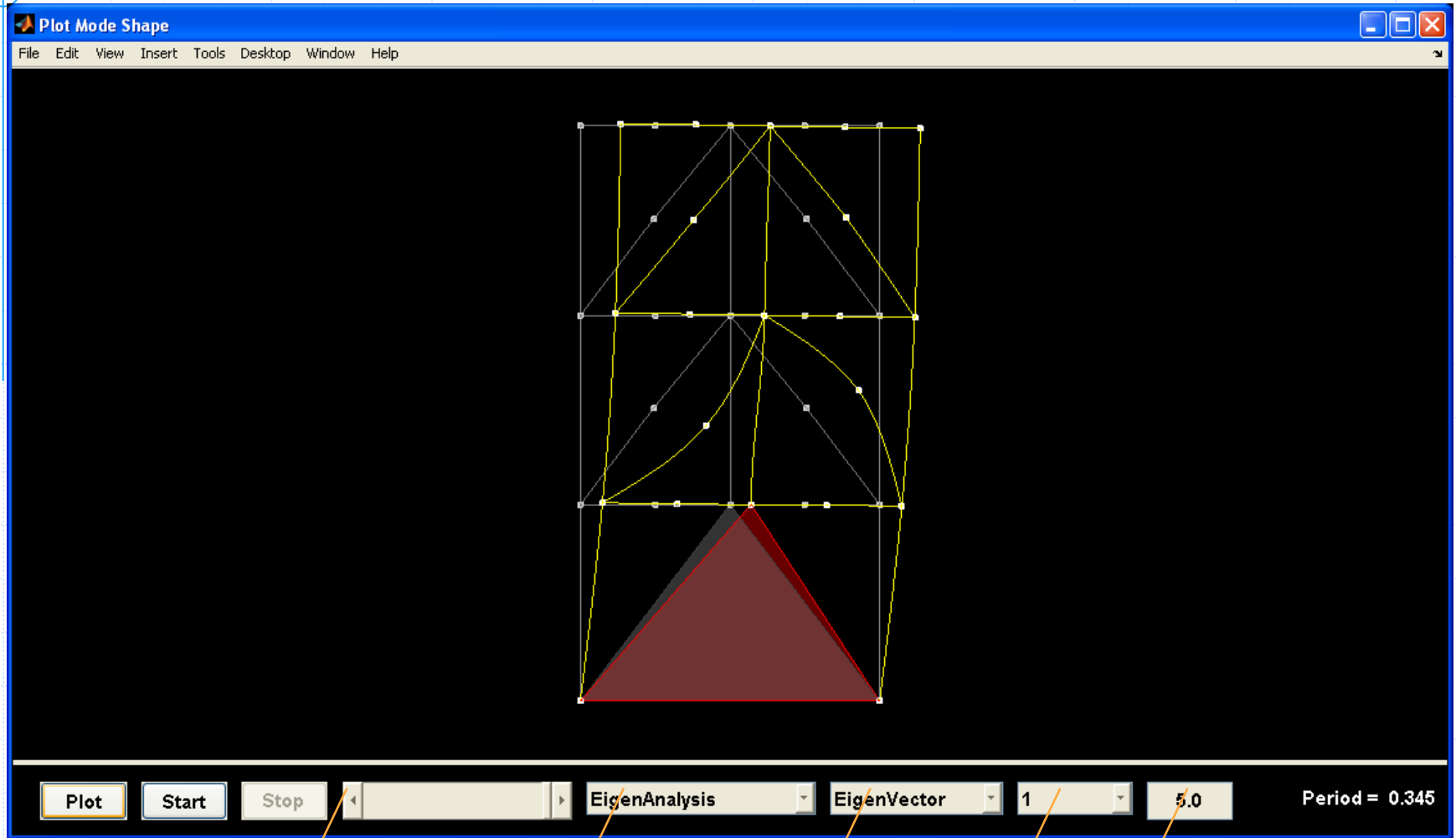
AnalysisCase

Recorder

Response

Magnification

# Post processing: plot mode shape



Animation Speed

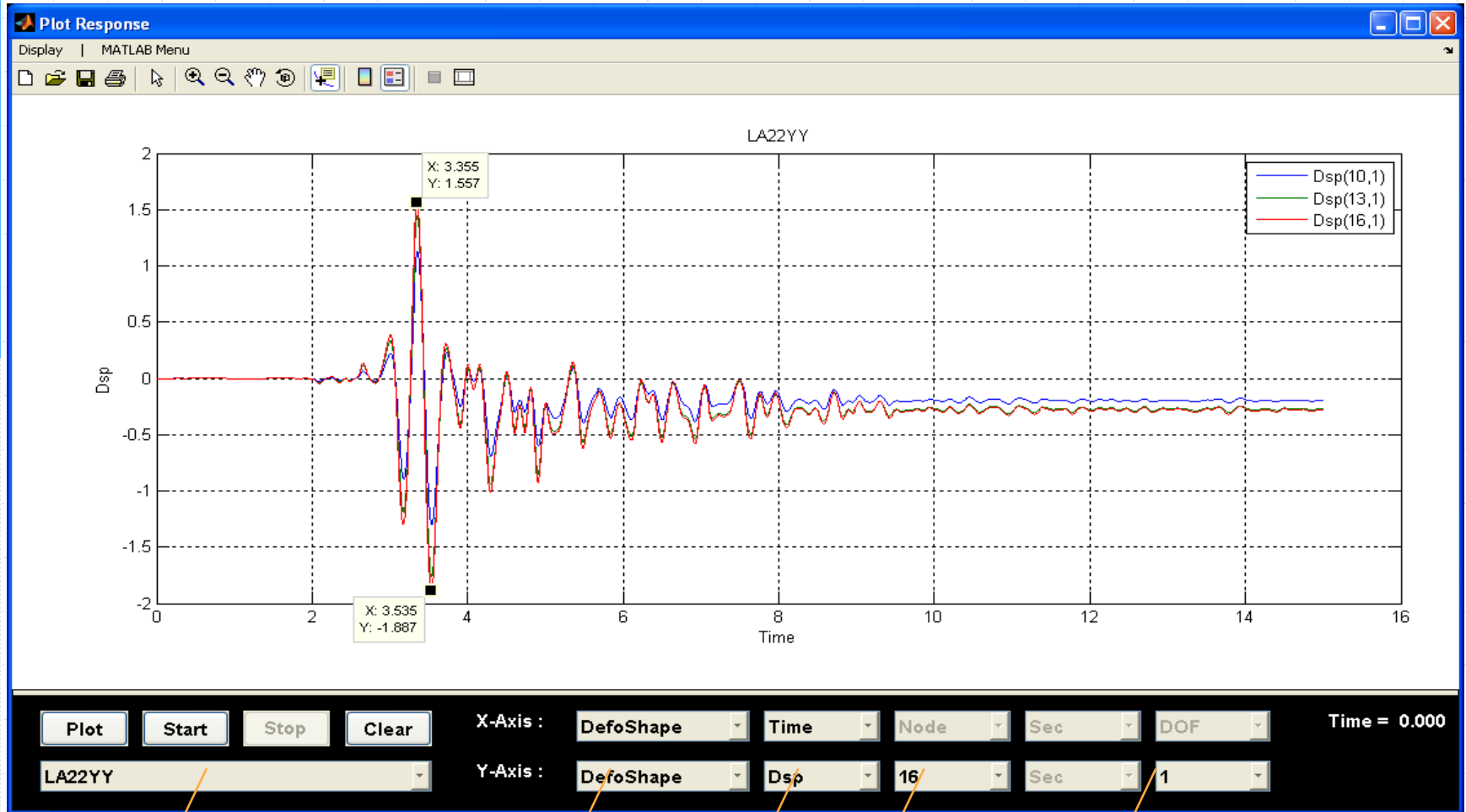
AnalysisCase

Recorder

Mode

Magnification

# Post processing: plot response histories



AnalysisCase

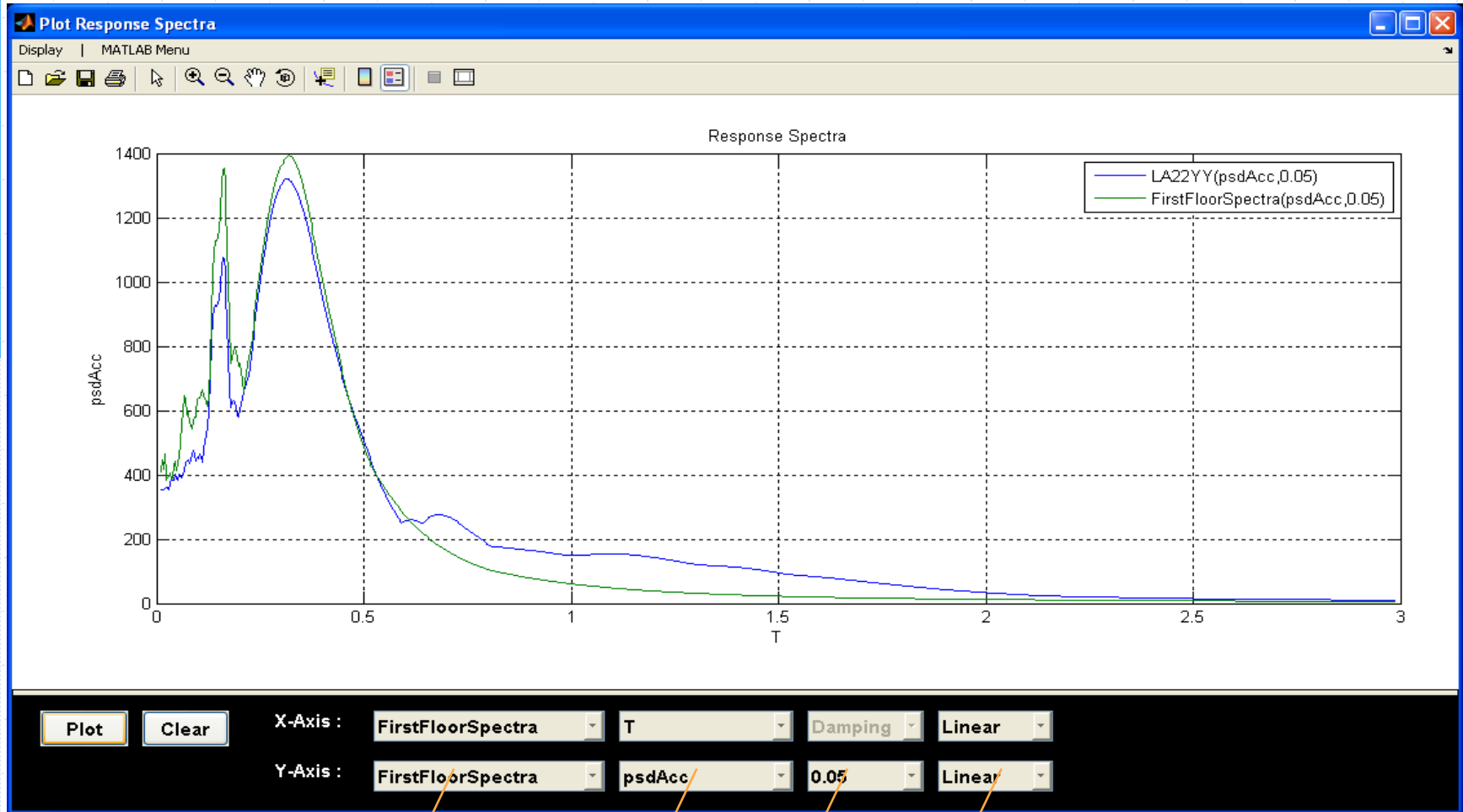
Recorder

Parameter

Node/Element

DOF

# Post processing: plot response spectra



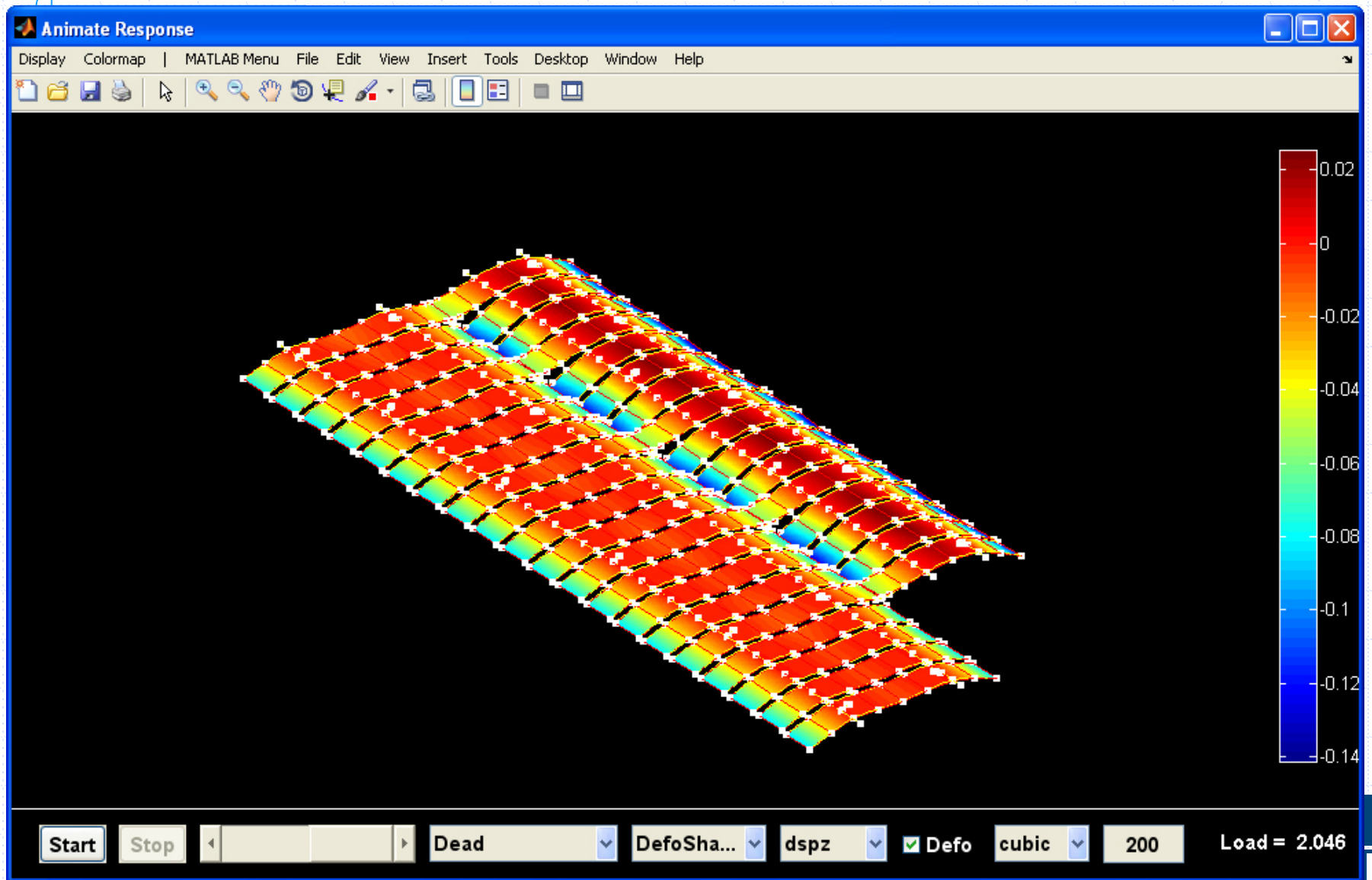
Spectra Name

Response

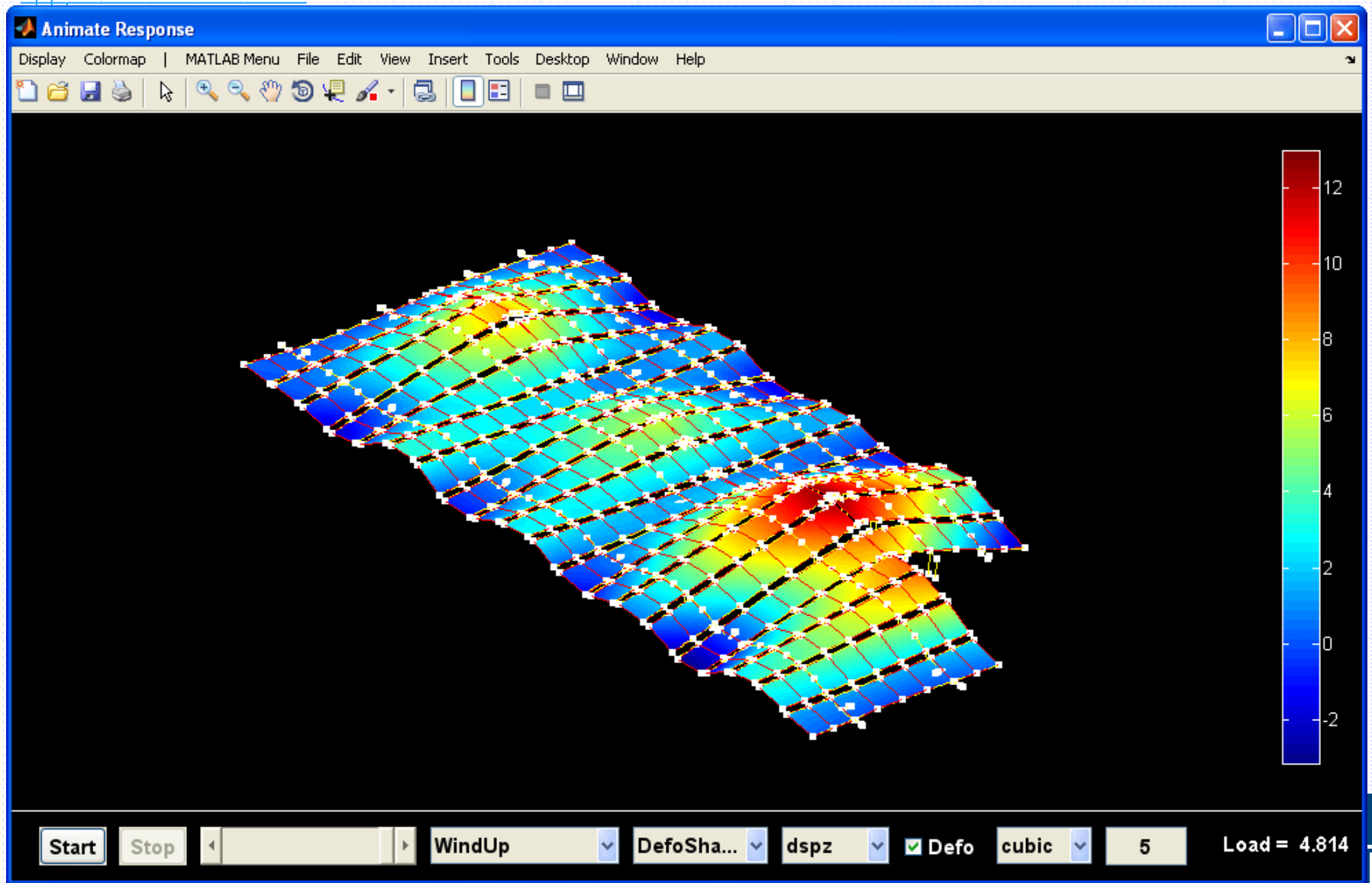
Damping

Axis Scale

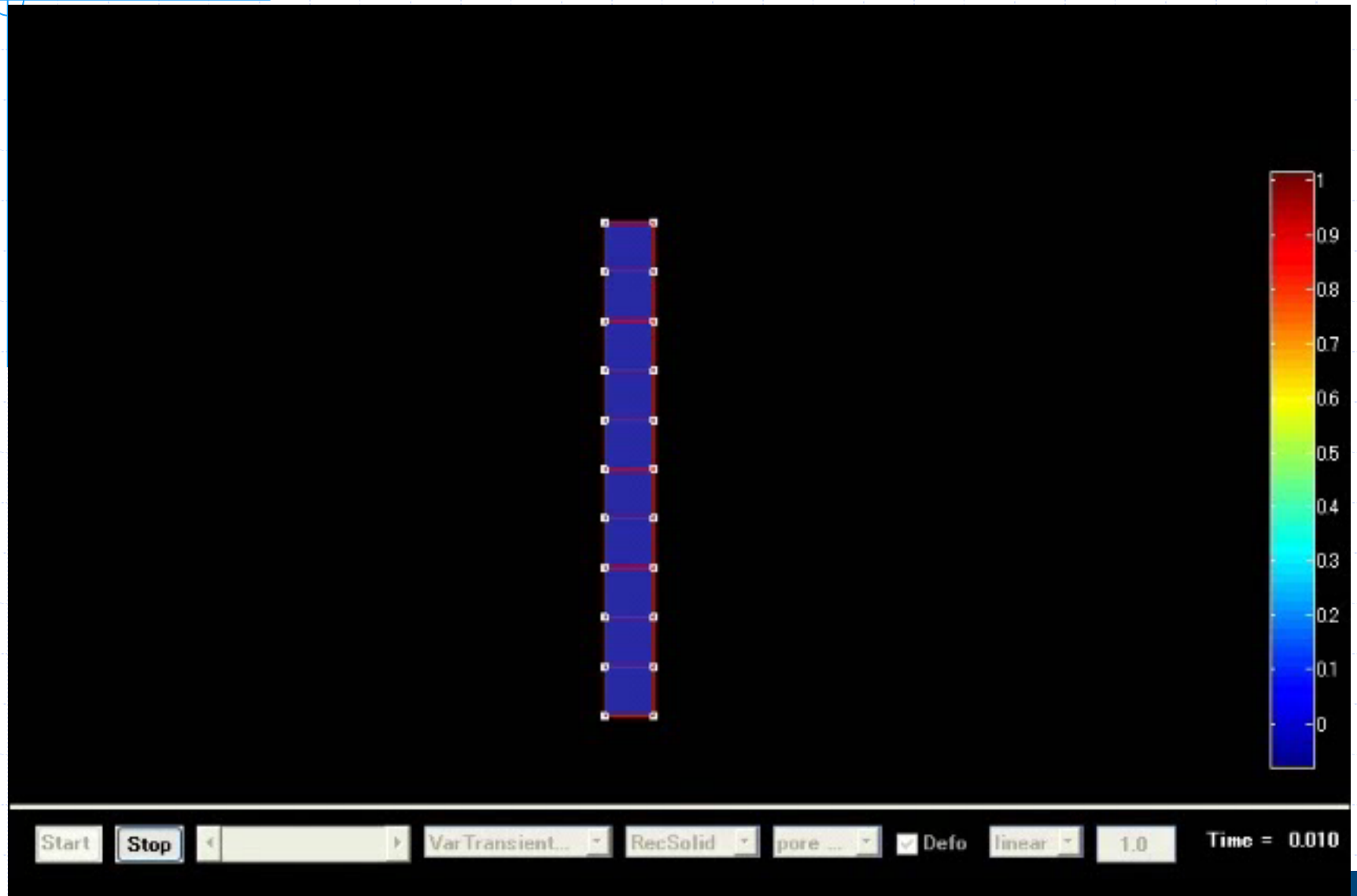
# Post processing: animate response



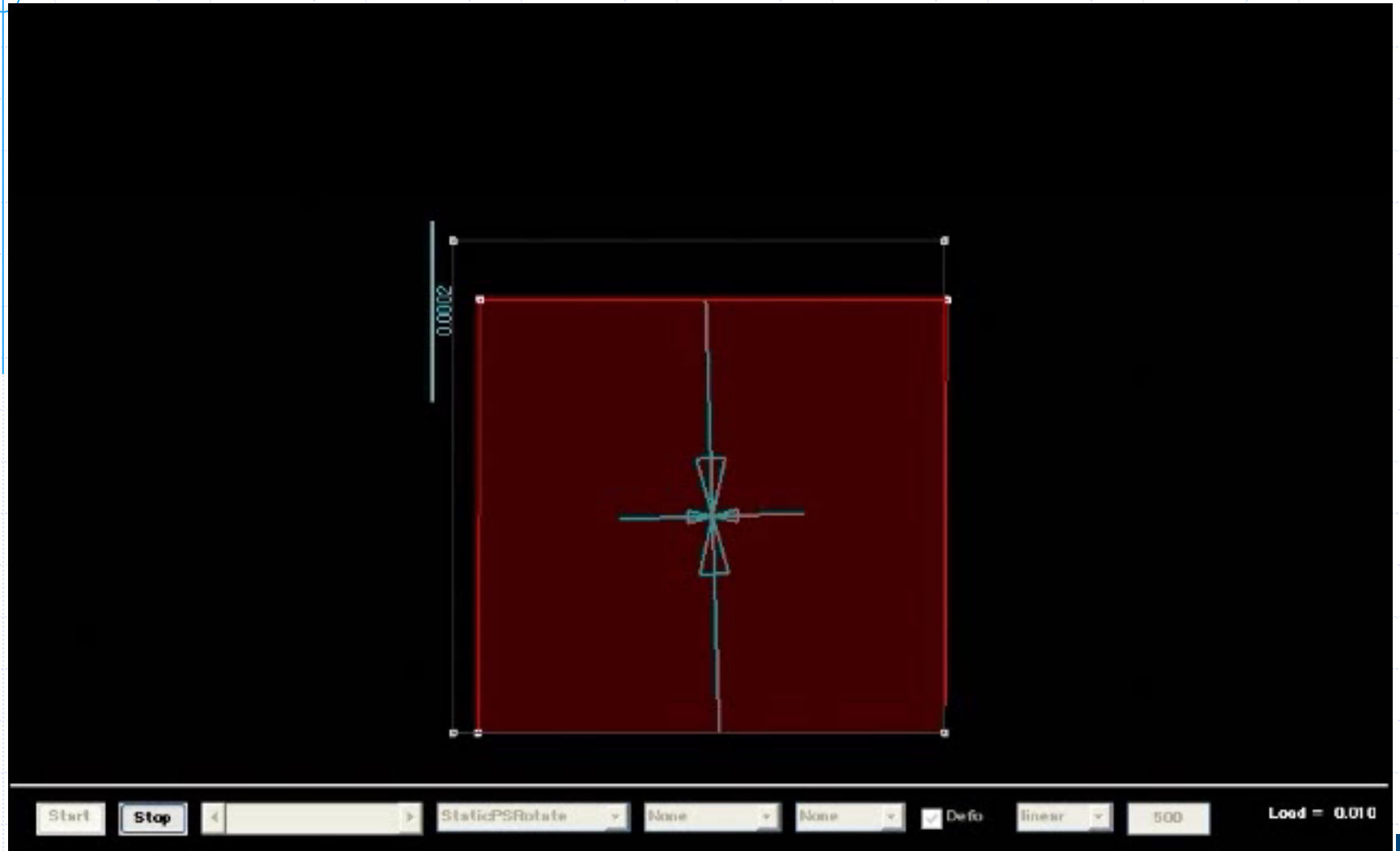
# Post processing: animate response



# Post processing: animate response



# Post processing: principal stress and strain





# Design: AISC design toolbox

The screenshot displays the OpenSees Navigator 2.1 software interface. The title bar reads "OpenSees Navigator 2.1 - ZipperExpHS.mat". The menu bar includes "File", "Edit", "View", "Define", "Assign", "Analyze", "Output", "Design", "Help", and "MATLAB Menu". The "Design" menu is open, showing options: "Show Available Sections...", "Find Section Properties...", "Find Matching Sections...", "Bending Capacity Check...", "Compression Capacity Check...", "Shear Capacity Check...", and "PMM Interaction Check...". The main workspace shows a structural model with a red triangular element at the bottom. A coordinate system with X, Y, and Z axes is visible at the bottom left. The status bar at the bottom indicates "model 'ZipperExpHS.mat' has been saved successfully" and "X-32.36 Y171.60 Z0.00".

## 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

**Find AISC Section Properties**

Section Shape :

Section Parameters :

- Area - A
- Depth - d
- Width - bf
- Tickness of the web (W,M,S only) - tw
- Tickness of the flange (W,M,S only) - tf
- Moment of inertia - Ix

**Output**

AISC Section Properties

The requested parameters are :

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

# AISC design toolbox: matching sections

**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 design toolbox: bending capacity

**AISC Bending Capacity**

**Bending Capacity of AISC Section**

Section Shape :	W24x68	<input type="button" value="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
```

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

OK

# AISC design toolbox: shear capacity

**AISC Shear Capacity**

Shear Capacity of AISC Section

Section Shape :	W24x68	<b>Calculate</b>
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 design toolbox: PMM interaction

**AISC PMM Interaction Check**

**PMM Interaction Check of AISC Section**

Section Shape :	W24x68	<b>Calculate</b>
Yield Stress (Fy) :	50	[ksi]
Modulus of Elasticity (E) :	29000	[ksi]
<b>Demand :</b>		
Applied Axial Force (Pu) :		[kips]
Applied Moment about X axis (Mux) :		[kips - in]
Applied Moment about Y axis (Muy) :		[kips - in]
<b>Compression :</b>		
Effective Length (kLx) :		[in]
Effective Length (kLy) :		[in]
<b>Bending :</b>		
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.
  - 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								
<a href="#">HOME</a>	<a href="#">INTRODUCTION</a>	<a href="#">MANUALS</a>	<a href="#">TUTORIALS</a>	<a href="#">PRESENTATIONS</a>	<a href="#">DISCUSSION</a>	<a href="#">UPDATES</a>	<a href="#">DOWNLOADS</a>	<a href="#">LINKS</a>
Home>		<input type="text"/> <input type="button" value="Search"/>						
<a href="#">Introduction</a>	<p>Dear OpenSees Navigator users,</p> <p>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 <a href="mailto:andreas.schellenberg@gmail.com">andreas.schellenberg@gmail.com</a> or <a href="mailto:yangtony2004@gmail.com">yangtony2004@gmail.com</a> 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.</p> <p>Thank you.</p> <p>Please feel free to visit our websites to discover in what other fun research we are involved:</p> <p>Andreas Schellenberg &amp; <a href="#">Tony Yang</a></p>							
<a href="#">Manuals</a>								
<a href="#">Tutorials</a>								
<a href="#">Presentations</a>								
<a href="#">Discussion</a>								
<a href="#">Updates</a>								
<a href="#">Downloads</a>								
<a href="#">Links</a>								
<input type="checkbox"/> Hit Counter	OpenSees Navigator ©2004-2006 <a href="#">UC Berkeley</a> . All rights reserved. Please <a href="#">contact us</a> with any questions or comments.							

# Website: download

## OPENSEES NAVIGATOR

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

Home > Downloads > Stand-Alone Windows >

<a href="#">MCRInstaller.exe</a>	<b>Installation Instructions:</b> <ol style="list-style-type: none"><li>1. Download the two files on the left.</li><li>2. Install the Matlab component runtime libraries by executing MCRInstaller.exe and following the on screen instructions (this has only to be done once).</li><li>3. Extract OpenSeesNavigator.zip in any folder of your choice and then execute OpenSeesNavigator.exe.</li><li>4. If you like you can create a shortcut to OpenSeesNavigator.exe on your Desktop.</li></ol>
<a href="#">OpenSeesNavigator.zip</a>	

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